NUMERICAL SIMULATION USING PARTICLE AND BOUNDARY ELEMENT METHOD FOR HIGHLY NONLINEAR INTERACTION PROBLEMS BETWEEN WAVES AND FLOATING BODY

Makoto SUEYOSHI

Research Institute for Applied Mechanics, Kyushu University, Kasuga, Japan

Hajime KIHARA

National Defense Academy, Yokosuka, Japan

Masashi *KASHIWAGI*

Research Institute for Applied Mechanics, Kyushu University, Kasuga, Japan

ABSTRACT

A numerical technique based on a moving particle semi-implicit (MPS) method was developed to solve wave-body interaction problems. The numerical method is targeted on to treat extremely nonlinear phenomena like breaking waves and large amplitude of motions of a floating body.

It is well known that particle methods consume larger computational resources than usual numerical methods using some grid systems. Therefore the numerical examples had been clustered in the transitional problems in a short period of time. On the other hand, steady oscillation problems for the motions of a floating body in regular waves are weak assignments for the particle methods. For these problems requires long time simulations and large computation domain.

The present method is a kind of domain decomposition techniques, which divide the computation area into two regions. One is the upper region including free surface and floating bodies. The other is the lower region including almost of the computation area and the bottom. The nonlinear phenomena are dominant in the limited upper region and treated by the MPS method. In the lower region, the flow is regarded as linear potential flow and treated by a boundary element method. The interface between these two regions is treated as special boundaries in each region to exchange the information of the flow field. The present method can reduce the computation time successfully and provide reasonable results for numerical simulation of waves.

In this paper, details of the numerical method are presented and several numerical examples are demonstrated. The examples include extremely nonlinear phenomena like green water on deck and some of them are compared with experimental results.

KEY WORDS: Hybrid method, particle method, boundary element method, wave-body interaction, deck wetness

INTRODUCTION

The interaction problem between waves and a floating body is an important topic in the field of naval and ocean engineering. There are many numerical study cases from various approaches for this problem. The most traditional approach is to use the potential theory in frequency domain. Recent years, time domain computation by using potential theory or some CFD scheme becomes popular. But for highly nonlinear problems, including violent free surface flow, are considered as one of the challenging topics. The difficulty in treatment of the large deformation of free surface is too complicated to use some grid or mesh systems. Therefore, some new numerical schemes are required for such problems. One of the hopeful solutions is the particle method, which use moving particles as proxy as a conventional grid system. The disadvantage of the particle method is that it consumes larger computation time and resources than conventional mesh type methods. It is a serious problem for numerical simulation of interaction between waves and floating bodies since they usually require larger computational domain than closed domain problems like sloshing. But generally, there is no necessity to apply the particle methods to whole of the computational domain. The highly nonlinear phenomena like breaking waves are observed in limited region around free-surface and floating body.

We have developed a hybrid scheme between particle and boundary element methods to reduce computational effort. The computational area is divided into two regions, upper region and lower one. In the upper region including floating bodies and the free-surface, the present scheme employs Moving Particle Semi-implicit(MPS) method (Koshizuka et al.(1996)) as the particle method. In the lower region, a boundary element method is employed as a linear potential flow solver. This combination is inspired by Iafrati's study(2003), which employs a conventional grid type CFD scheme and a linear potential solver. Naito et al.(2003) develop a treatment of waves in deep water area with particle method and potential theory. But their technique, which employs analytical solution of wave problem as flow field in the outer region, cannot treat complicated condition of waves. The present hybrid method is evolved from their method and has capability to treat more complicated situations.

NUMERICAL IMPLEMENTATION

In the present hybrid scheme, the computational domain is divided into the upper and lower regions. We simulate 2D wave channel tank for first step. Fig.1 shows the schematic view of the concept to divide the whole computation domain. The upper region includes all floating bodies and free surface and some part of vertical walls of the tank. The lower region includes the lower part of the vertical walls and bottom of the tank. The interface boundary between two regions is horizontal line in the water. Along the interface boundary, the upper and the lower regions exchange information of the each flow field to keep consistency as a whole flow field. In this procedure, we need some special treatment to keep robust and fast computation.

The MPS method is a kind of fully Lagrangian method by using moving particles. It is similar to the most popular particle method, SPH(Smoothed Particle Hydrodynamics) method(Monaghan (1994)), although there are some differences in the concept of formulation. The MPS method was proposed for complicated incompressible flow in the field of nuclear engineering by Koshizuka et al.(1996). It employed original form of spatial discretization, which is called as particle interaction models. For instance, gradient of some scalar quantity F at particle i is described as

$$\nabla F_i = \frac{d}{\sum_{j\neq i}^{N} w(r_{ij})} \sum_{j\neq i}^{N} w(r_{ij}) \frac{F_j - F_i}{r_{ij}} \frac{\mathbf{r}_j - \mathbf{r}_i}{r_{ij}} \quad (1)$$

where $\mathbf{r}$ is position vector of particle i; r_{ij} is distance between particle i and j; and $w(r_{ij})$ is a weight function of r_{ij}; d is dimension of computational domain.

We can apply this discrete form to various first order differential operator. The second order differential, Laplacian F is described as

$$\Delta F_i = \frac{2d}{\sum_{j\neq i}^{N} w(r_{ij}) r_{ij}^{2}} \sum_{j\neq i}^{N} w(r_{ij})(F_j - F). \quad (2)$$

In equation (1) and (2), they don't require information of connectivity among particles. It requires only position vector of each particle. By using equation (1) and (2), equations of fluid motion can be handle without any grid or mesh systems.

The other feature of the MPS method is that it employs semi-implicit algorithm to compute the pressure field. Fig.2 shows the sequence of time marching of the MPS method. In this sequence, the pressure computation is to solve a Poisson's equation of the pressure P, which is described as

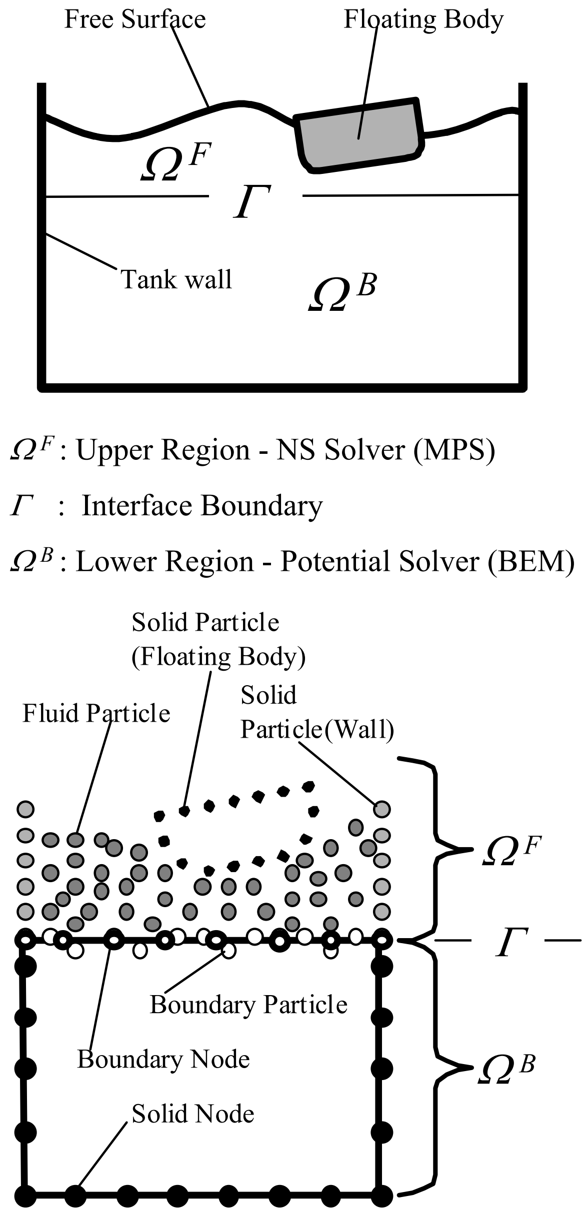

Fig.1 Schematic view of concept of computational domain for the present hybrid method.

$$\nabla^2 P^{n+1} = -\frac{\rho}{\Delta t^2} \frac{(n^* - n_{const})}{n_{const}} \quad (3)$$

where n_{const} is a constant value calculated from initial particle arrangement and ρ is density of the fluid.

Thanks to this semi-implicit process, we can use relatively larger time stepping than explicit pressure computation methods for incompressible flows.

In this study our numerical implementation is almost same as Koshizuka et al.(1998). The more details of the MPS method are presented in references.

In the present hybrid method, boundary condition on free surface is that the pressure is constant. On the solid sidewalls, velocity is zero and on the floating body surfaces, it is given from motions of the body. On the moving interface boundary, the velocity is given from the solution of the potential flow solver.

In the lower region, a boundary element method is employed as the potential flow solver. The Linear solver has no need to compute repeatedly the influence coefficient matrix depend on the position of each node point in time domain computation due to the spatial fixed mesh system. The linear system has also no need to solve repeatedly. As the result, the computational time for the time domain linear potential solver is relatively much smaller than one for the particle method.

The potential value, which is computed from the pressure of the upper region, is given as the boundary condition on the interface boundary. The velocity potential Φ_i at the particle i on the moving interface boundary is described as

$$\Phi_i(t) = \int_0^t \left(-\frac{P_i}{\rho} - g z_i \right) dt \tag{4}$$

where g is acceleration of gravity; z_i is the vertical position of the particle i; and P_i is calculated in the upper region at the particle i. The calculated Φ_i is distributed to the each node point of the BEM solver with 3rd order Lagrange interpolation. In this procedure of interpolation, we apply a simple running average to remove numerical oscillation of

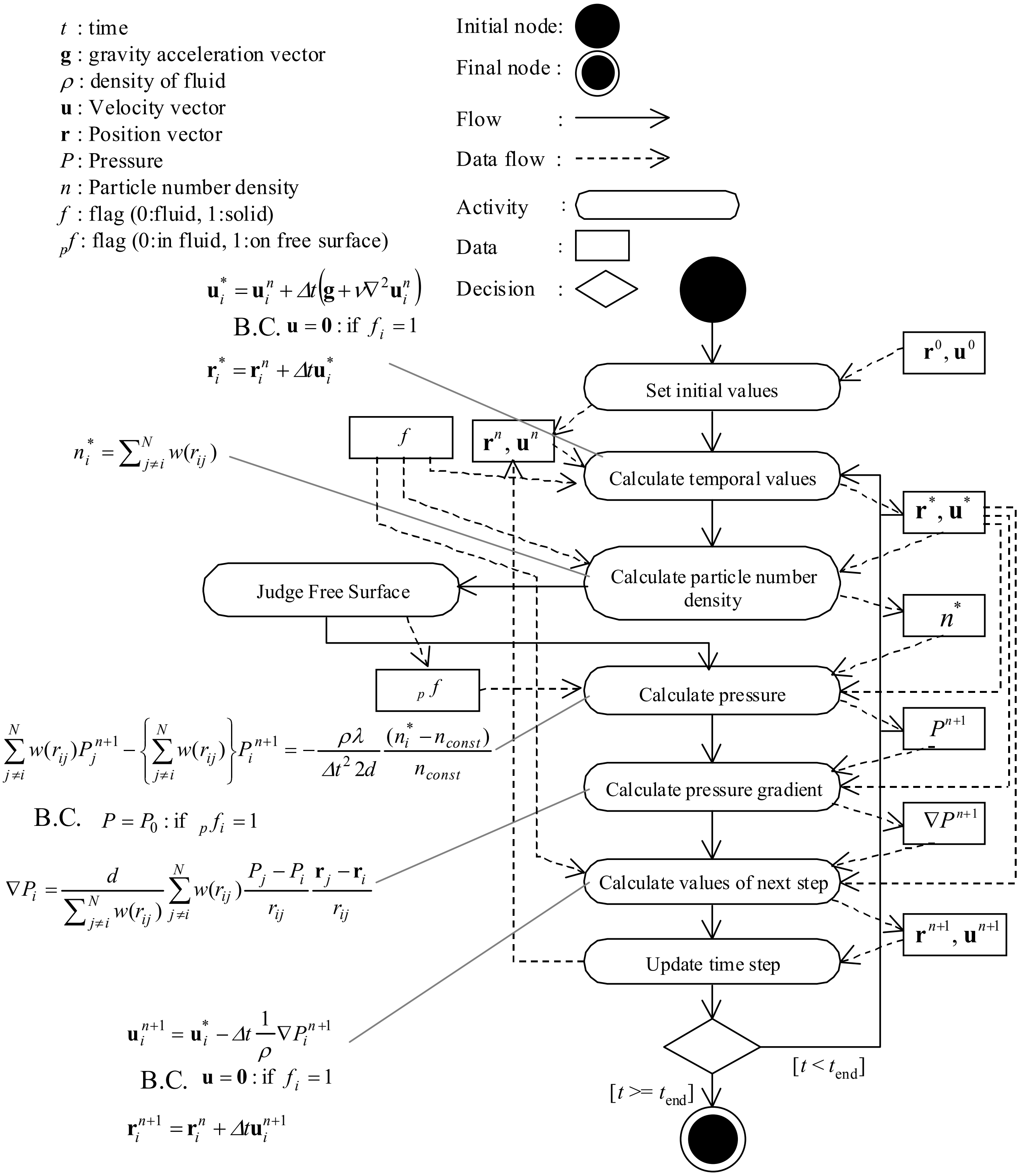

Fig.2 diagram of time marching procedure for MPS method

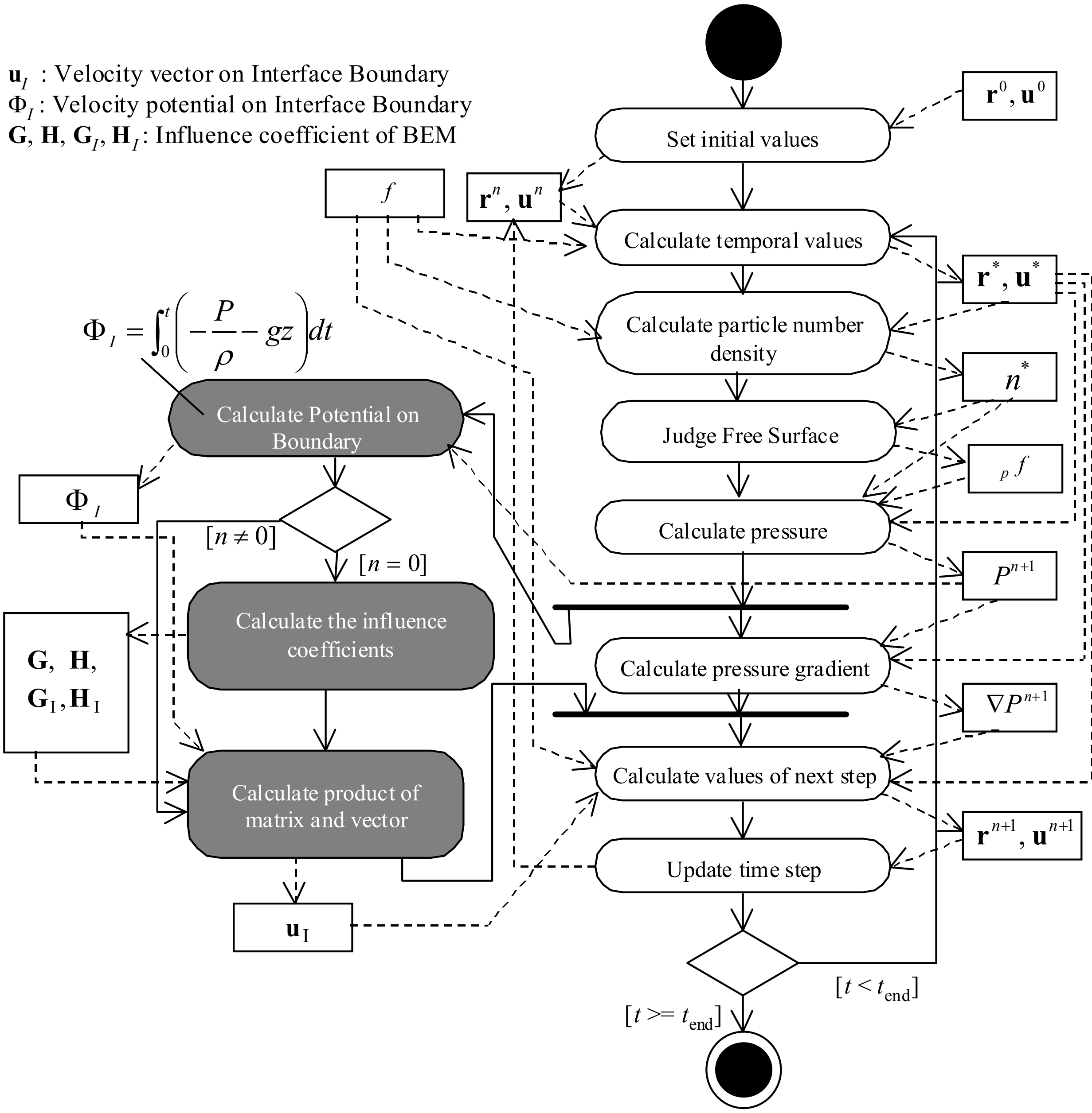

Fig.3 diagram of the data exchange sequence between MPS and BEM in the present hybrid method

spatial distribution of the velocity potential. It has already known that there is some numerical oscillation of the pressure calculated by the MPS method. On the other hand, in the BEM solver, such oscillation is harmful to robust computation. Therefore some numerical noise reduction procedure like the running average technique is required for the present hybrid method.

Fig.3 shows a diagram of the computational sequence to exchange information of the flow field between the MPS procedure and BEM procedure. In Fig.3, The right side white round shapes indicates computation of the original MPS method and the left side gray colored shapes indicates additional process for the present hybrid method.

NUMERICAL EXAMPLES

In order to confirm the availability of the present method for wave problems, a simple rectangular shape 2D wave channel is simulated. Fig.4 shows the arrangement for the numerical simulation. Both longitudinal ends of the 2D wave channel are enclosed by solid vertical walls and there are no wave absorbing beach. The bottom is completely flat and gives a constant depth. At the left side of the tank, a triangle shape plunger type wave maker is arranged. When waves are generated, the wave maker is forcedly oscillated in vertical direction. The motion is sinusoidal motion with transient increase in amplitude at the beginning.

This computational domain is divided into the upper and lower regions at z = - d(z = 0: Calm water level). On this line z = - d, there are an array of particles and nodes points of BEM solver for the moving interface boundary. These array and nodes is put on same straight line at the beginning of computation. As time marching proceeds, arrayed particles for the upper region move in vertical direction and show a wavy shape.

Fig.5 shows snapshots of particles' arrangement in the upper region. In this case, period of heaving motion of the plunger type wave maker is 1.0 sec. and the amplitude is 2.5 cm. The arrayed particles on the interface boundary moves

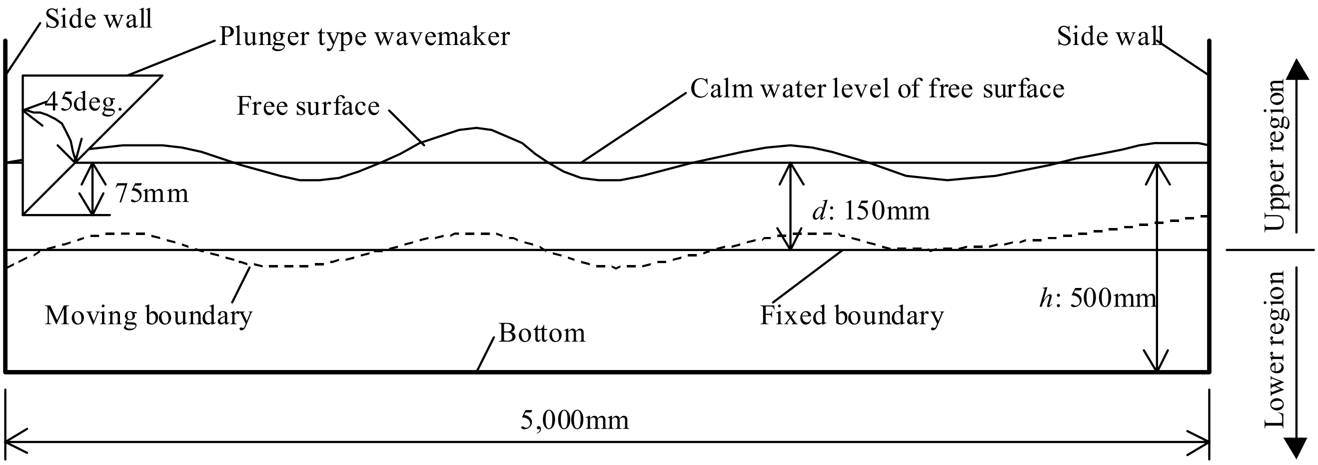

Fig.4 Setup for 2D wave channel simulation

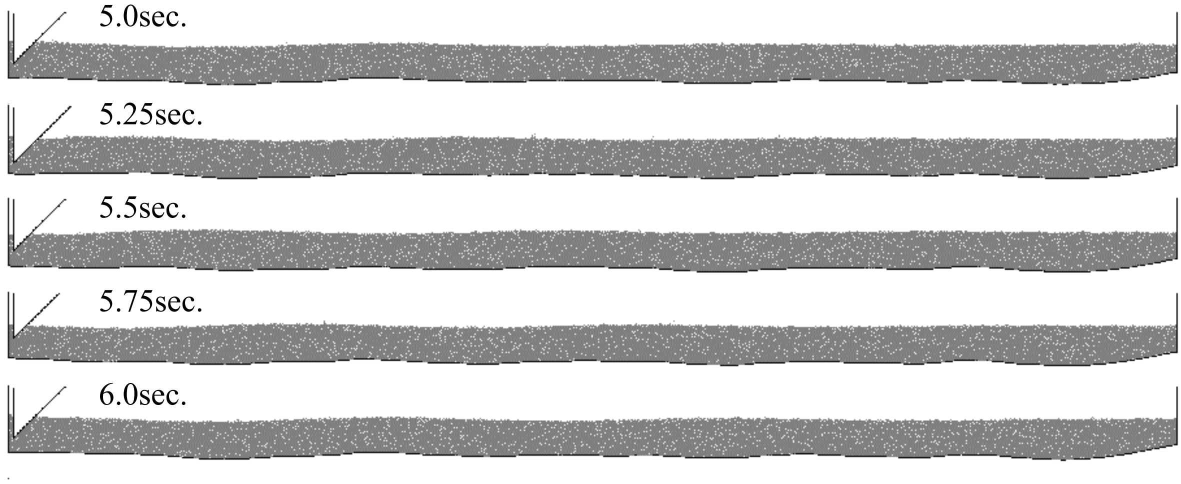

Fig.5 Snapshots of particles' arrangement in upper region

with the propagation of waves. The numerical computation itself is stable and waves propagate from the wave maker.

In order to confirm whether the present method has capability to simulate water waves, we check the relation between the wavelength and the period. Table 1 shows some comparisons of dispersion relation of the progressive water waves in finite depth between linear theoretical solution and numerical one by the present numerical method. In all cases, the error from the theoretical result is smaller than 2 percent.

Table 1 Comparison of relation between wavelength and wave period

	Linear Solution	Hybrid Method
T: 1.0sec. h: 0.15m d: 0.15m (Fixed bottom)	1.09m	1.08m
T: 1.0sec. h: 0.5m d: 0.15m	1.51m	1.49m
T: 0.7sec. h: 0.5m d: 0.15m	0.76m	0.77m

Some fully nonlinear BEM programs can simulate this wave channel problem. Fig.6 shows comparison of profiles of numerical results between the present method and such fully nonlinear BEM program. The node points of the nonlinear BEM solver fit to the particle arrangement successfully at each moment.

In Fig.7, comparison of the spatial distribution of the velocity potential on the interface boundary is shown. In this case, The point x=0 is left end of the tank (wavemaker side) and x=5.0 is the right end. The amplitude by the present method decreases as the waves propagate from left to right. It suggests that there is some wave damping in the results of the present method. Fig.8 shows the spatial distributions of the velocity potential in some different cases in parameter for running average treatment to reduce numerical oscillation. This results suggests that the numerical treatment at the interface boundary effects on the damping of wave amplitude.

The present method has some problems of wave propagation in the amplitude, however the dispersion

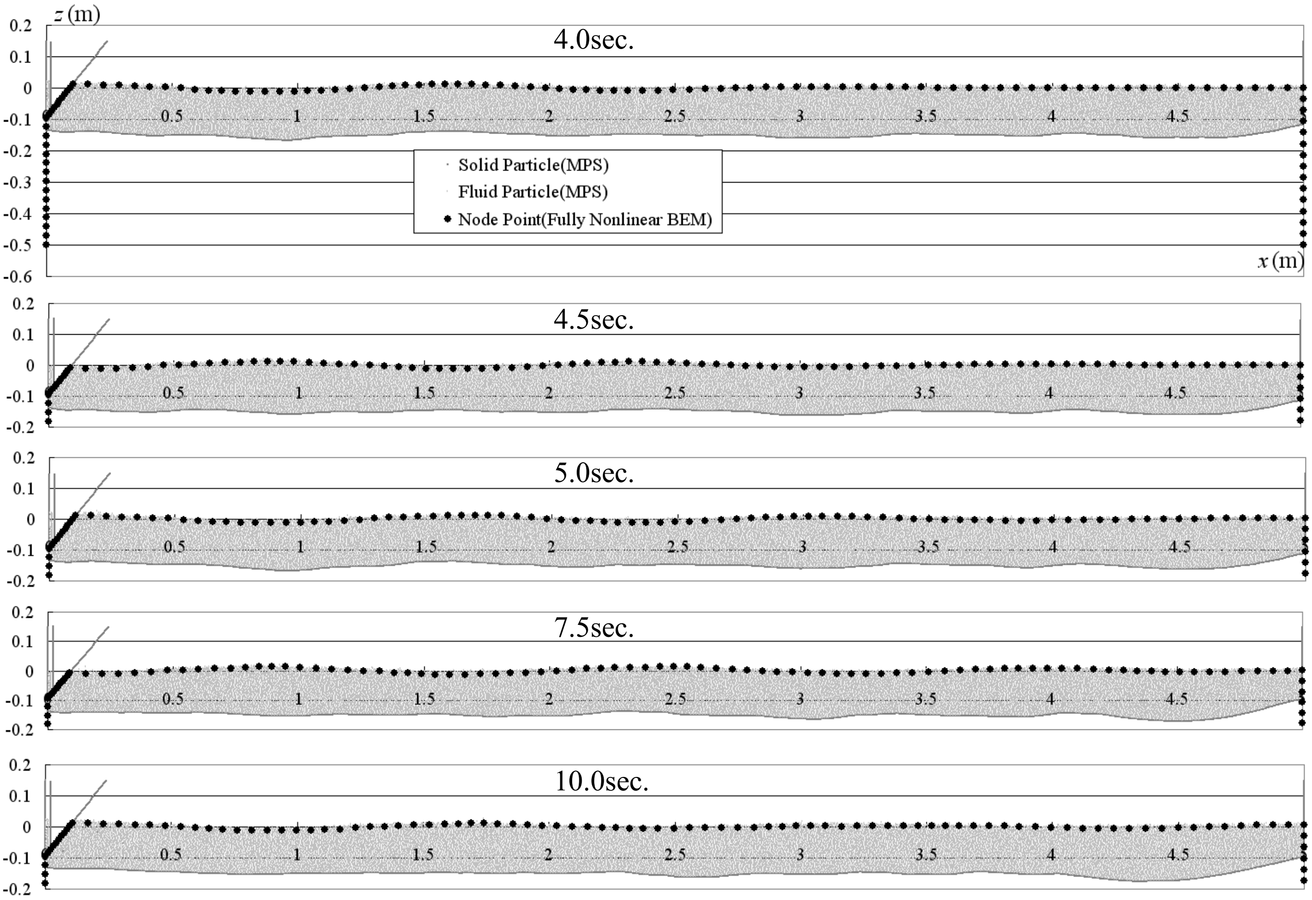

Fig.6 Comparison of profiles of waves between the present hybrid method and fully nonlinear BEM

relation is successfully computed. But consumption of computational resources are drastically decreased. Table 2 shows comparison of the number of particles and computational time for this 2D wave channel problem between the present and original purely particle methods.

The present hybrid method has capability to treat more complicated flow than above-mentioned wave channel problem. In theory, the present method can treat extremely large deformation of free surface like breaking waves and extremely nonlinear phenomena of the body motions like capsizing or flooding. Such complicated wave–floating body interaction problems are computed and compared with experimental results. Fig.9 shows a sketch for setup of numerical simulation and experiment. The floating body has rectangular hull shape and very small freeboard. In some cases of the experiment, we can observe green water on the deck in waves. The floating body is moored with spring tention in horizontal direction through the carriage and slider mechanism.

Fig.10 shows snapshots of a part of particles' arrangement in the upper region and experimental result. The comparison of close up view of the snapshots is shown. On the deck of weather side, water flows onto the deck in both of numerical and experimental results. In such highly nonlinear free surface flow, the computation can be carried out.

In Fig.11, comparisons of time evolutions of the motions of the body are shown. In these numerical results, amplitudes of the body is smaller than experimental ones. The agreement of the phase in each mode is not so bad. The reason of the tendency may be that there is the wave damping in amplitude. In this numerical simulation, the amplitude of wave maker is given as same as the experimental one. In the numerical computation, the wave becomes smaller as distance from wave maker becomes long. Therefore the wave amplitude in front of the body may be smaller than experiment.

CONCLUSIONS

In the present study, we developed a hybrid technique to simulate complicated wave problems and following conclusions are obtained.

1. Characteristics of wave dispersion for finite water depth can be simulated successfully.
2. There is still some numerical damping in the present numerical procedure.
3. Capability of the present method for a complicated condition is demonstrated.

ACKNOWLEDGEMENTS

A part of the present study is supported by joint research projects of Research Institute for Applied Mechanics(Kyushu University) and REDAS(the Shipbuilders' Association of Japan).

REFERENCES

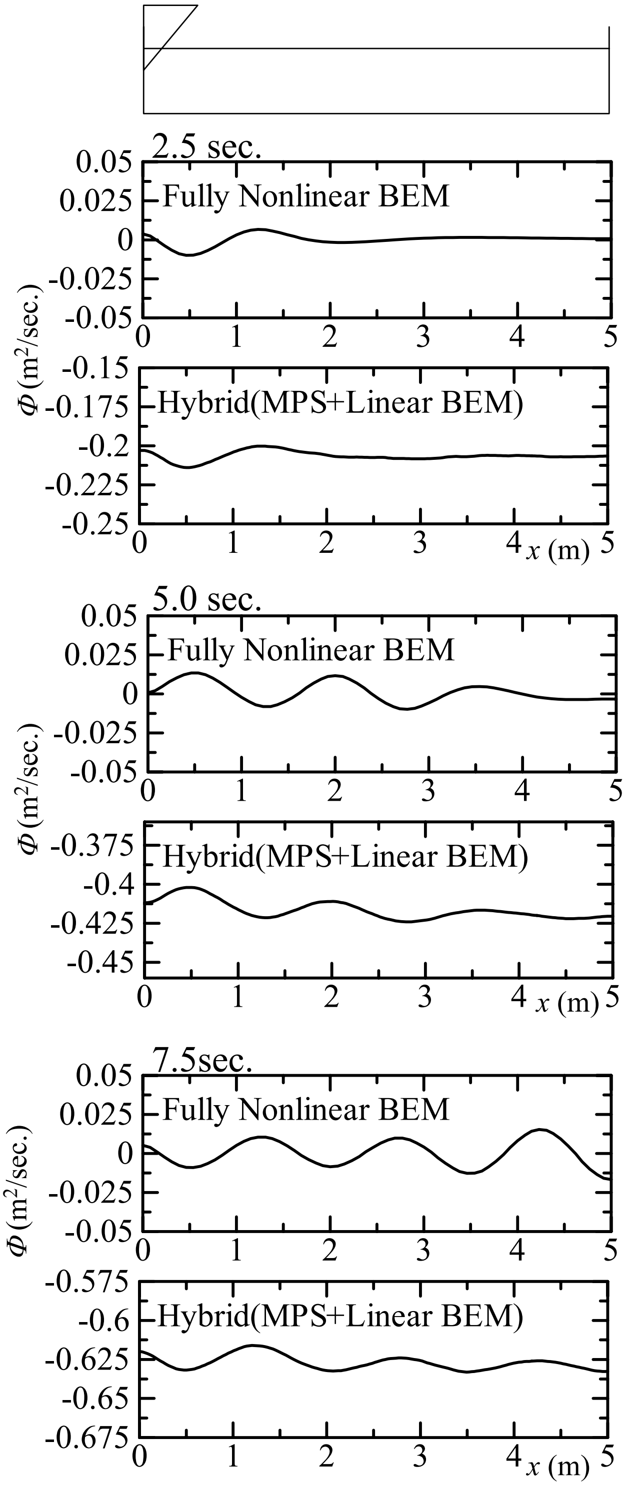

Fig.7 Spatial distributions of velocity potential on the interface boundary(T=1.0(sec.))

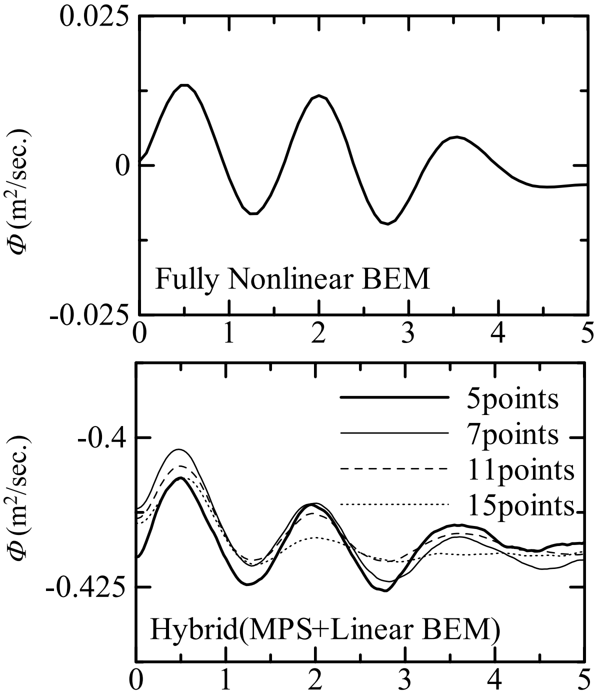

Fig.8 Comparison of spatial distributions of velocity potential on the interface boundary(T=1.0(sec.))

Table 2 Comparison of total number of particles and computation time

	MPS Method	Hybrid Method
Total Number of Particles	101,234	31,204
Computation Time	69h50m	11h40m

Iafrati A., Campana E.F.(2003), "A domain decomposition approach to compute wave breaking(wave breaking flows)," *Int. J. Numer. Meth. Fluids*, 41, pp 419-445.
Koshizuka S., Oka, Y.(1996), "Moving particle semi-implicit method for fragmentation of incompressible fluid," *Nuclear Science and Engineering*, Vol. 123, pp 421-434
Koshizuka S., Nobe A., Oka Y.(1998), "Numerical analysys of breaking waves using the moving particle semi-implicit method," *Int. J. Numer. Meth. Fluids*, 26, pp 751-769
Monaghan JJ(1994), "Simulating free surface flows with SPH," *Jour. of Comp. Phys.*, Vol. 110, pp 399-406
Naito S., Sueyoshi M. and Ohashi T.(2003), "Study on Particle Method with Small Size Calculation Domain," *J.Kansai Soc. N. A.*, *Japan*, No.240, pp109-114

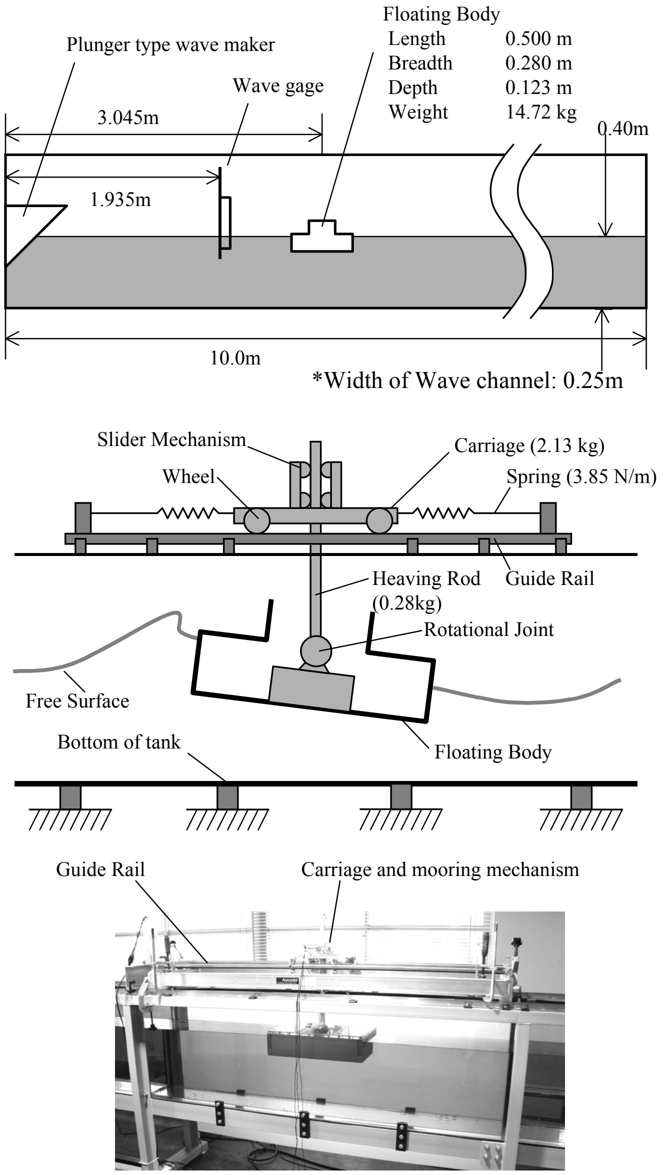

Fig.9 Setup of numerical simulation and experiment for floating body problem in wave

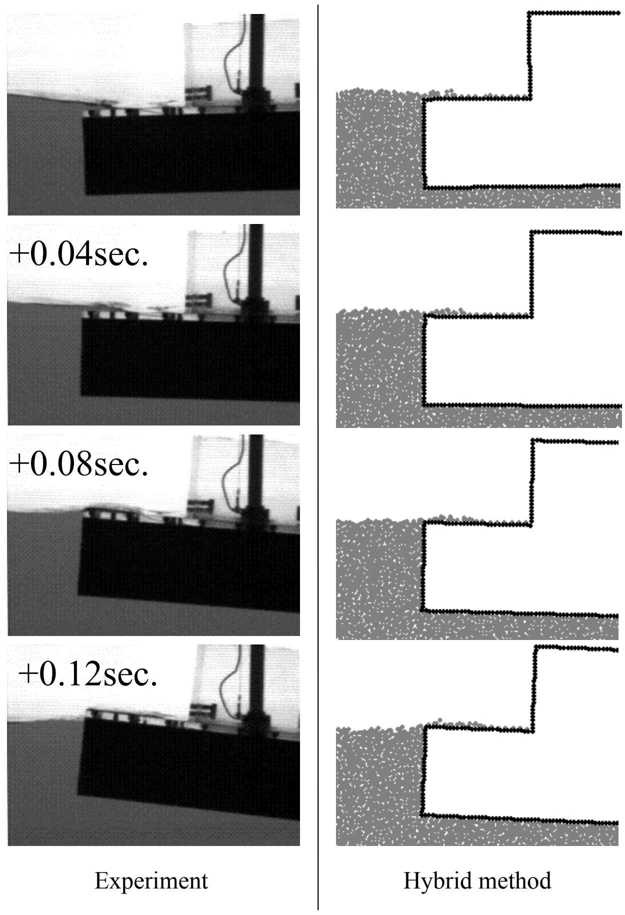

Fig.10 Close up view around floating body

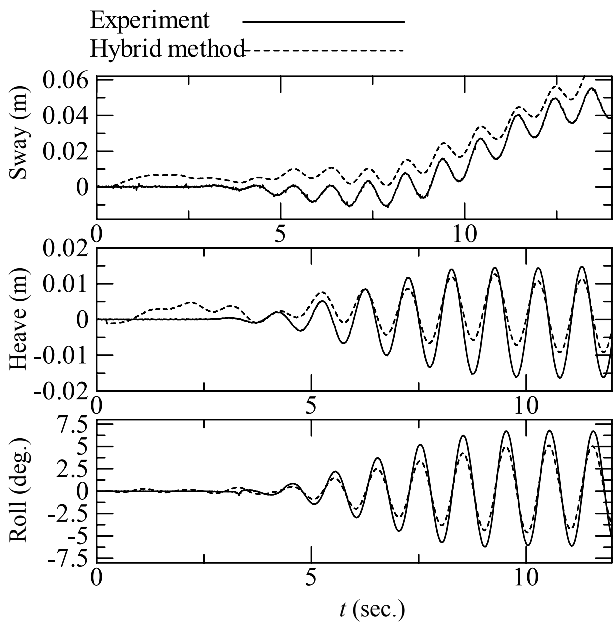

Fig.11 Time evolutions of motions of the floating body