Simulation of a Ship with Partially Filled Tanks Rolling in Waves by Applying Moving Particle Semi-Implicit Method

Jen-Shiang Kouh
Department of Engineering Science and Ocean Engineering, National Taiwan University, Taipei, Taiwan, R.O.C.
kouhjsh@ntu.edu.tw

Hung-Pin Chien[1], Chun-Chung Chang[2], Yen-Jen Chen[3]

Abstract **- Free surface behavior is a common phenomenon in the nature. It combines spray creation, large deformation and breakup of water. Due to its unfixed computational boundary, the boundary position needs to be solved as a portion of result. However, it is a challenge to give sufficient spatial resolutions along the time-dependently changing free surface. In this paper, a mesh free method, moving particle semi-implicit method (MPS), based on the "particle mechanics" will be used and applied to analyze the phenomena of a ship with partially filled tanks rolling in regular waves. Thereby, not only the external flow around the hull but also the free surface flow inside partially filled tanks can be simulated together by using a single algorithm based on this method. The influences of the sloshing effect resulted from the free surface flow to the rolling of a ship will be discussed in this paper. Some case studies will be chosen for demonstrating results thanks to the advantage of this method.**

Key Words- Free surface flow, Mesh-free method, Moving particle semi-implicit method, Sloshing effect.

INTRODUCTION

Numerical simulation is difficult to calculate fluid fragmentation on the free surface. Another difficulty of numerical simulation exists in handling complex geometry. However, in the field of naval architecture, the topic of simulating free surface has been concerned all along. In general, MAC method (Marker and Cell) [1] and VOF method (Volume of Fluid) [2] are introduced popularly in solving the free surface problem. Recently, the meshfree methods have been put emphasis on simulating free surface. In this paper, a meshfree method, moving particle semi-implicit method (MPS) [3], based on the "particle mechanics" will be introduced and applied to analyze the unsteady free surface flow. The MPS has the capability to analyze more complex geometry and physics than grid methods. In the method, it is unnecessary to generate mesh and the numerical diffusion doesn't need to be taken into account, because the convection is embedded in the motion of particles. The accuracy of result can be controlled by the number of particles. The information of computational domain can be obtained more detail by means of putting more particles. The simulation of two-dimensional fluid fragmentation is presented in this paper.

NUMERICAL METHOD

The moving particle semi-implicit method (MPS) was a meshless method for incompressible flow with free surfaces. It was advocated by Koshizuka et.al [4] in 1995. The basic concept comes from solving the discrete particles distributed in the computational and boundary domain. Each particle has corresponding information of coordinate, velocity and pressure. The kernel function is employed to simulate a particle interacts with its neighboring particles. By tracing the motion of every particle in Lagrangian coordinate, the information of particles is obtained then.

I. Governing Equations

Governing equations are the mass, momentum conservation equations for Newtonian, incompressible flows as follows:

$$\frac{\partial \rho}{\partial t} = 0 \tag{1}$$

$$\frac{Du}{Dt} = -\frac{1}{\rho}\nabla P + f \tag{2}$$

Equation (1) is mass conservation and (2) is momentum conservation with convection terms. The left hand side of (2) denotes the convection terms in Lagrangian time differentiation and describes the particle motion. The right hand side consists of pressure gradient and force terms. All differential terms are discretized to simulate the interaction between particles.

II. Kernel Functions

The continuous flow is supposed to consist of particles with coordinate, velocity and pressure information. A particle interacts with its neighboring particles in a specific area. The kernel function $\omega(r)$ is employed to describe the interaction

[1] Hung-Pin Chien, Department of Engineering Science and Ocean Engineering, National Taiwan University, Taipei, Taiwan, R90525026@ntu.edu.tw
[2] Chun-Chung Chang, Department of Engineering Science and Ocean Engineering, National Taiwan University, Taipei, Taiwan, R93525021@ntu.edu.tw
[3] Yen-Jen Chen, Department of Multimedia and Game Science, Lunghwa University of Science and Technology, Taoyuan, Taiwan, clive_chen@yahoo.com

between particles. The kernel function employed in this research is as follows:

$$\omega(r)=\begin{cases}\frac{r_e}{r}-1 & (0\le r<r_e)\\ 0 & (r\le r_e)\end{cases} \tag{3}$$

In (3), r is a distance between two particles and r_e is the radius of the interaction area. The interaction can be neglected at $r \ge r_e$. However, the kernel function is infinity at $r = 0$. It means the overlap between particles can be avoided [5].

III. Particle Number Density

Particle number density n_i is defined by (4) where particle i is located at coordinate r_i.

$$n_i=\sum_{j\neq i}\omega\left(\left|r_j-r_i\right|\right) \tag{4}$$

Equation (4) can be expressed as the contribution from particle i itself. It also denotes the concentrated extent of neighboring particles around particle i. The number of particles in a unit volume can be written as the relation between n_i and N_i:

$$N_i=\frac{n_i}{\int_V \omega(r)dv} \tag{5}$$

The dominator of (5) is the integral of the kernel function in the whole region.

$$\rho_i=mN_i=\frac{mn_i}{\int_V \omega(r)dv} \tag{6}$$

Equation (6) shows that the fluid density is proportional to the particle number density where m denotes mass of a particle. For incompressible flow, the particle number density keeps a constant as

$$n^0=\sum_{i\neq j}\omega\left(\left|r_j^{\,0}-r_i^{\,0}\right|\right) \tag{7}$$

IV. Modeling of Gradient

A gradient vector between two particles i and j can be defined as $(\psi_j-\psi_i)(r_j-r_i)/\mid r_j-r_i\mid^2$. Therefore, the gradient vector at r_i is obtained by a weighted average of these vectors.

$$\nabla\phi_i=\frac{d}{n^0}\sum_{i\neq j}\frac{\phi_j-\phi_i}{\left|r_j-r_i\right|^2}\left(r_j-r_i\right)\omega\left(\left|r_j-r_i\right|\right) \tag{8}$$

where d is the number of space dimensions. In this paper, (9) is used instead of (8). The modeling of gradient is introduced to compute the pressure gradient term in governing equation. Because of (8) is not sensitive to absolute pressure, it can be arranged as follows:

$$\nabla\phi_i=\frac{d}{n^0}\sum_{i\neq j}\frac{\phi_j-\phi_i^{\,'}}{\left|r_j-r_i\right|^2}\left(r_j-r_i\right)\omega\left(\left|r_j-r_i\right|\right) \tag{9}$$

$\psi_i^{\,'}$ is computed from (10)

$$\phi_i^{\,'}=\min(\phi_j) \tag{10}$$

V. Modeling of Laplacian

The operator $\nabla^2\phi$ is obtained by introducing the diffusion equation. The modeling of Laplacian is presented as follows:

$$\nabla^2\phi_i=\frac{2d}{n^0\lambda}\sum_{i\neq j}\left(\phi_j-\phi_i\right)\omega\left(\left|r_j-r_i\right|_i\right) \tag{11}$$

$$\lambda=\frac{\sum_{i\neq j}\omega\left(\left|r_j-r_i\right|\right)\left|r_j-r_i\right|^2}{\sum_{i\neq j}\omega\left(\left|r_j-r_i\right|\right)} \tag{12}$$

where λ is introduced so that the variance increase is equal to the analytical solution.

VI. Boundary Condition

Actual boundary conditions for the MPS method are simple handling than traditional numerical method. Because there are no particles outside of the free surface, the particle number density on the free surface is smaller than in the liquid. If a particle satisfies

$$n_i^*<\beta\cdot n_0 \tag{13}$$

It will be defined as free surface. Thus, it is not necessary to define free surface specifically. Here β is a parameter which is smaller than 1. In this paper, β is chosen to be 0.97.

VII. Calculation Parameters

The distance between two neighboring particles l_o is 0.008 m. The density of liquid ρ is 1000 kg . m^3 and the gravity g is 9.8 m/s^2. Koshizuka [4] advocated $r_e = 2.1l_o$ when calculating the particle number density and the modeling of gradient. On the other hand, use $r_e = 4.0l_o$ for the modeling of Laplacian. Each time step is considered according to the Courant equation.

$$\Delta t=\min\left(\alpha\, l_0/v_{\max},1.0\times10^{-3}\right) \tag{14}$$

α is chosen as 0.1 and v_{max} is the temporal maximum velocity of particles.

CALCULATION RESULTS

Calculations of a ship rolling in waves are presented. The effects of viscosity and surface tension are neglected. The total number of particles used for the simulation is about 7,000. The initial particle configuration is shown in Figure 1. The breadth of the tank is 0.133 m, the depth is 0.073 m and the draft is 0.042 m. The waves are generated by a piston type wave generator. The period of wave maker is 0.35 sec. Four different cases as below are calculated.

- Without flooding water in the ship.
- With flooding water in the ship.
- Without flooding water in the ship and without stabilizer.
- With flooding water in the ship and with stabilizer.

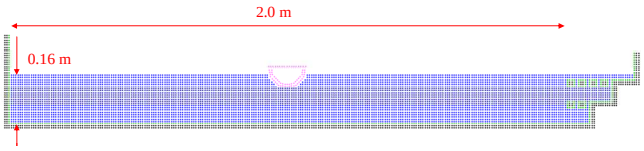

FIGURE 1
GEOMETRY CONFIGURATION FOR COMPUTATION.

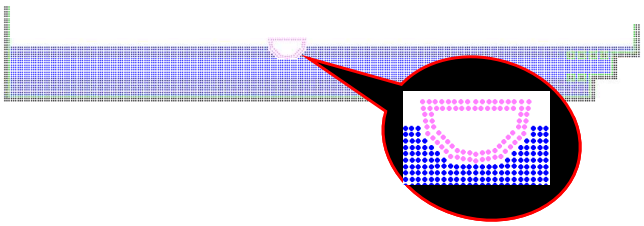

FIGURE 2
A SHIP WITHOUT STABILIZER.

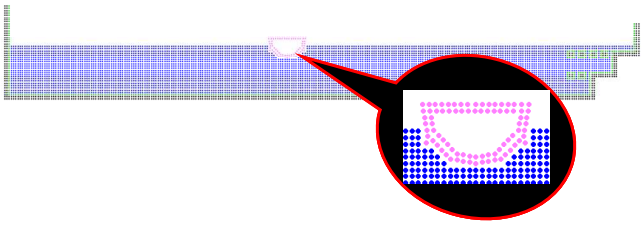

FIGURE 3
A SHIP WITH STABILIZER.

I. Without Stabilizer

Theoretically, the free-surface effect may induce a large roll angle and then reduce the stability of the ship. In order to investigating the free-surface effect when the water is partially filled in the tank, a tank without stabilizer is selected for numerical simulation. Figure 4 and 6 illustrates the condition of a ship with and without flooding water in waves. Figure 5 and 7 demonstrates the time histories of the sway and heave motion of the ship.

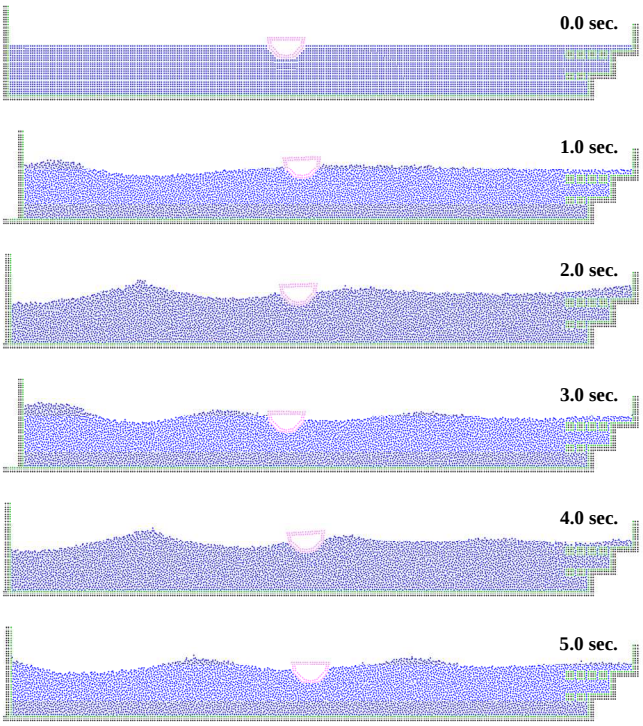

FIGURE 4
SIMULATION OF FREE MOTIONS OF A SHIP WITHOUT FLOODING WATER IN WAVES.

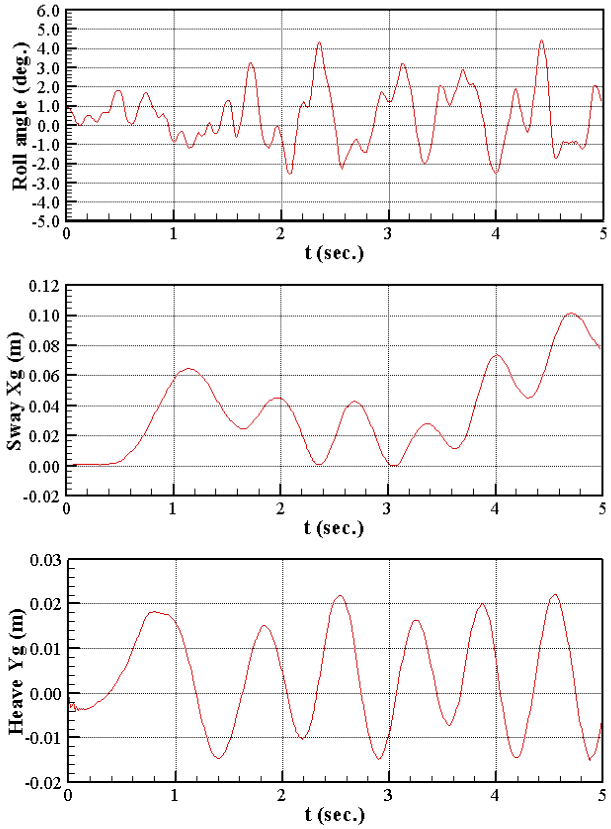

FIGURE 5
THE FREE MOTIONS OF A SHIP WITHOUT FLOODING WATER.

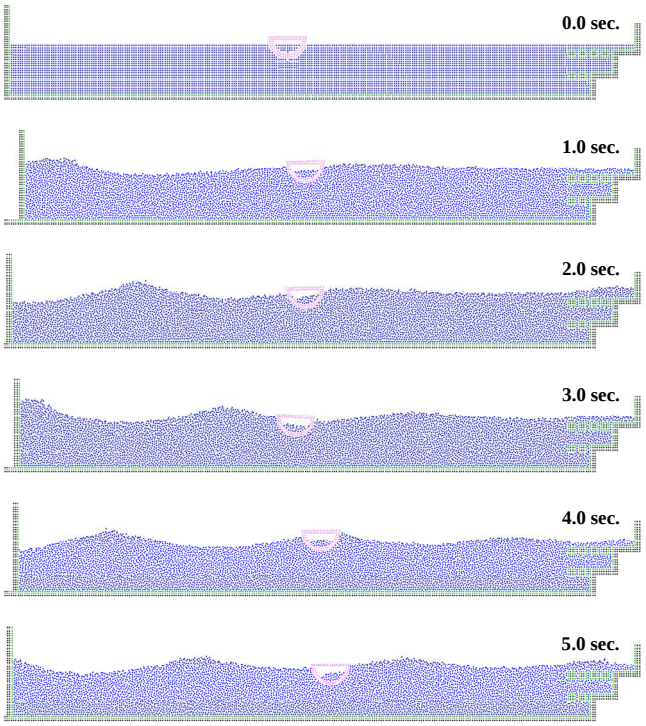

FIGURE 6
SIMULATION OF FREE MOTIONS OF A SHIP WITH FLOODING WATER IN WAVES.

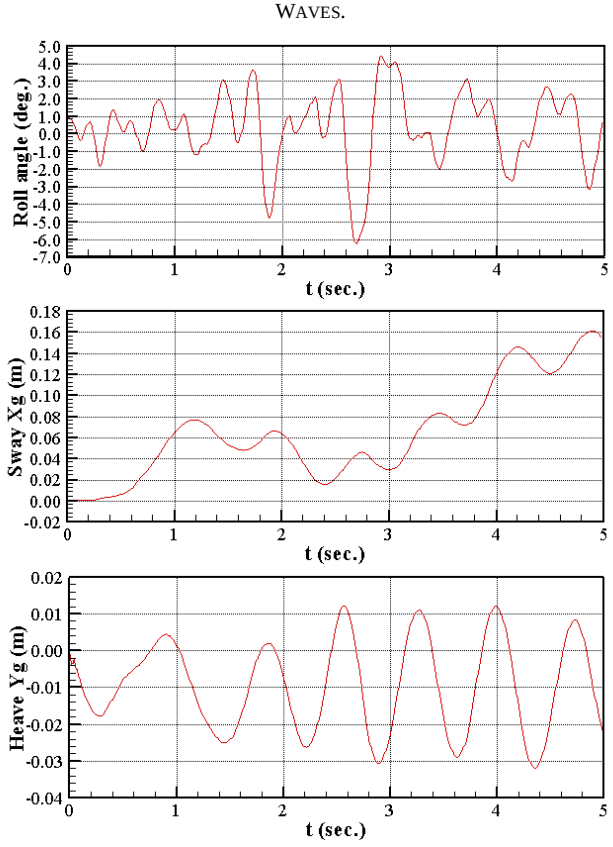

FIGURE 7
THE FREE MOTIONS OF A SHIP WITH FLOODING WATER.

II. With Stabilizer

Simulate the effects of the stabilizer about the former two cases.

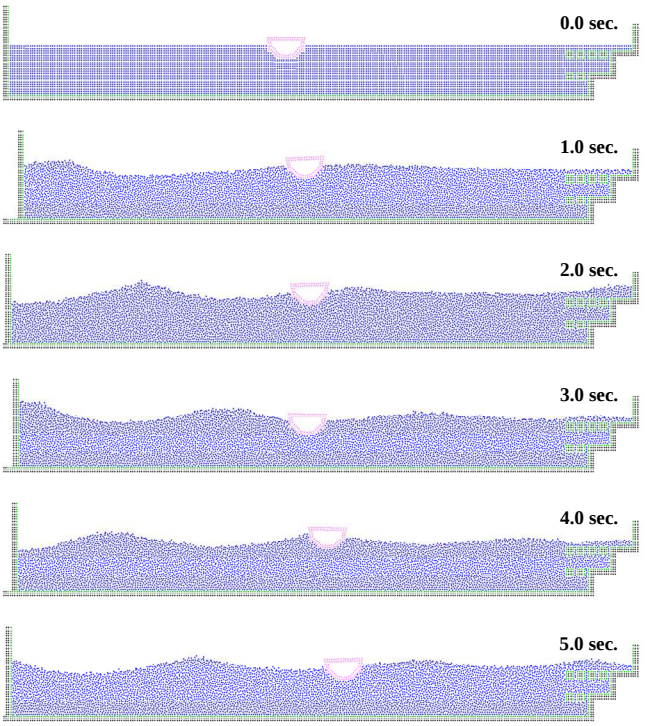

FIGURE 8
SIMULATION OF FREE MOTIONS OF A SHIP WITH STABILIZER IN WAVES.

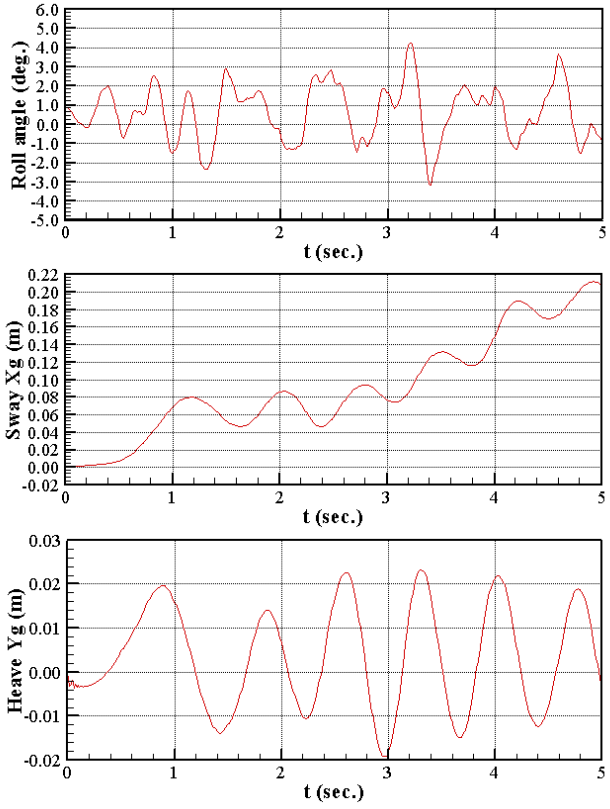

FIGURE 9
THE FREE MOTIONS OF A SHIP WITH STABILIZER (WITHOUT FLOODING WATER).

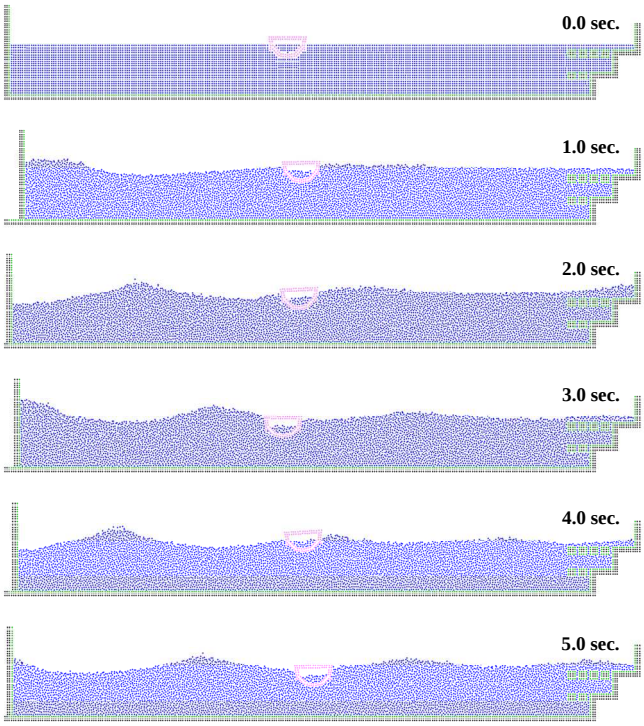

FIGURE 10
SIMULATION OF FREE MOTIONS OF A SHIP WITH FLOODING WATER AND STABILIZER IN WAVES.

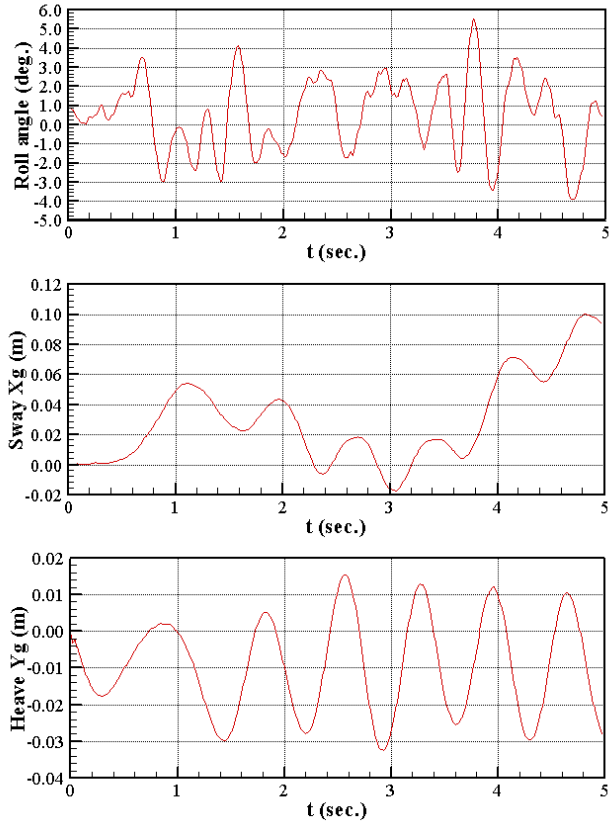

FIGURE 11
THE FREE MOTIONS OF A SHIP WITH STABILIZER (WITH FLOODING WATER).

III. Results

The figures below show the change of rolling angle and the angular velocity, the strength of waves, of the ship. From figure 12 and 13, it is the difference between the ship with and without flooding water. The results show the inner free-surface effect influence the stabilization of a ship seriously. If the waves become violent, the ship may have a risk to capsize. In addition, figure 14 and 15 tell the difference between installing the stabilizer or not when the ship is without flooding water. Figure 16 and 17 demonstrate the difference between installing the stabilizer or not when the ship is with flooding water. From figure 14 and 16, the results show the average rolling frequency which is decreased when the stabilizer installed on the ship. From figure 15 and 17, the max intensity of waves was decreased by the stabilizer. Therefore, it is obviously that the stabilizer can reduce the rolling frequency of the ship and the intensity of waves acting on the ship.

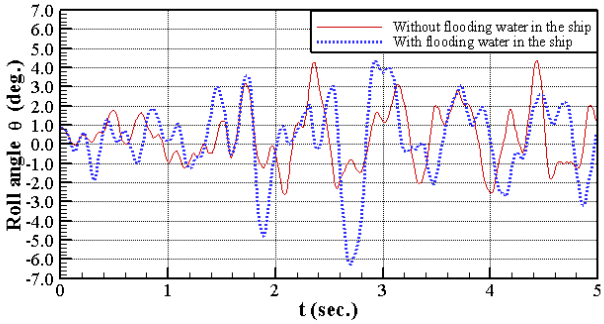

FIGURE 12
THE EFFECT OF THE INTERNAL FREE SURFACE FLOW.

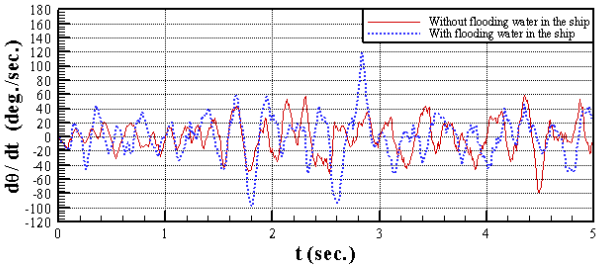

FIGURE 13
THE EFFECT OF THE INTERNAL FREE SURFACE DUE TO FLOODING WATER.

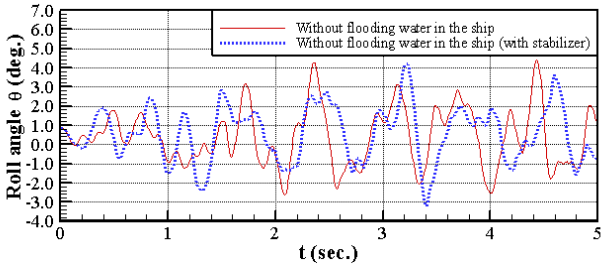

FIGURE 14
THE EFFECT OF THE STABILIZER (WITHOUT FLOODING WATER).

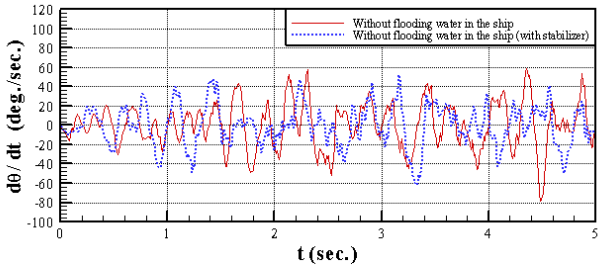

FIGURE 15
THE EFFECT OF THE STABILIZER (WITHOUT FLOODING WATER).

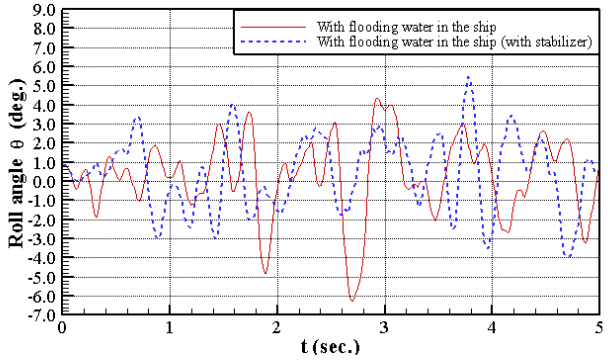

FIGURE 16
THE EFFECT OF THE STABILIZER (WITH FLOODING WATER).

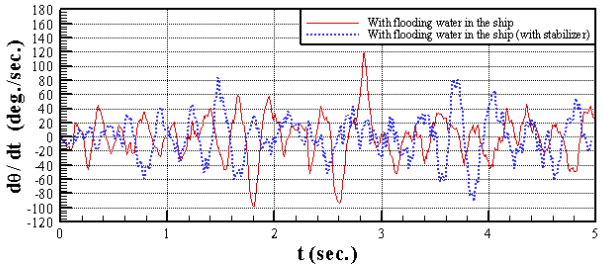

FIGURE 17
THE EFFECT OF THE STABILIZER (WITH FLOODING WATER).

CONCLUSIONS

This paper presents 2D results from the numerical simulation of a ship rolling in waves. The internal flow of flooding water inside the ship and the external flow about the ship can be calculated simultaneously. In contrast to the traditional mesh methods, the MPS method provides obvious advantage for dealing with free-surface flow. Without striving for extra arrangement for the moving grids and using any complicated algorithms, the simulation can be carried out directly. Not only the time and effort in grid generation can be saved, but also the phenomenon of the free-surface flow can be fully captured.

It can be seen that flooding water in the ship can produce a strongly negative effect on the rolling motion of the ship. Therefore, it is necessary to avoid the effect of free-surface flow for safety consideration. The stabilizer can be used to prolong the rolling period effectively by referring to the numerical results shown above.

At present, the simulation does not include the surface tension and the violent wave generation. Further improvements are necessary for computing many other flow problems with multiphase phenomenon.

ACKNOWLEDGMENT

The authors would like to thank the National Science Council R.O.C. for the finial support of this research under the grant NSC 94-2611-E-002-016.

REFERENCES

[1] Harlow F. H., Welch J. E., "Numerical calculation of time dependent viscous incompressible flow with the free surface", *The Physics of Fluids*, Vol. 8, 1965, pp. 2182-2189.

[2] Hirt C. W., Nicholls B. D., "Volume of fluid (VOF) method for dynamics of free boundaries", *Journal of Computational Physics*, Vol.39, 1981, pp. 201-221.

[3] Koshizuka S., Oka Y., "Moving particle semi-implicit method for fragmentation of incompressible fluid", *Nuclear Science and Engineering*, Vol. 123, 1996, pp. 421-434.

[4] Koshizuka S., Tamako H. and Oka Y., "A particle method for incompressible viscous flow with fluid fragmentation", *Comput. Fluid Dyn. J.* Vol. 4, 1995, pp. 29–46.

[5] Koshizuka S., Oka Y., "Numerical analysis of breaking waves using the moving particle semi-implicit method", *Int. J. Numer. Meth. Fluids*, Vol. 26, 1998, pp. 751–769.