DEVELOPMENT OF PARTICLE METHOD REPRESENTING FLOATING BODIES WITH HIGHLY NON-LINEAR WAVES

Tsukamoto, Marcio Michiharu
Universidade de São Paulo
michiharu@tpn.usp.br

Nishimoto, Kazuo
Universidade de São Paulo

Asanuma, Takayuki
NMRI - National Maritime Research Institute

Abstract. *A numerical method called Moving Particle Semi-implicit (MPS) method was developed to analyze incompressible non-viscous flow. It is a particle method using a lagrangean representation without any grid. The governing equations are the Euler equation and the continuity equation. Most of the computational fluid dynamics (CFD) methods are based on eulerian representation and use grids to describe the geometry of the simulated domain. These differences make the MPS method easier to analyze highly nonlinear phenomena as free surface with wave breaking, sloshing, slamming, etc. In previously published articles, results of physical experiments had shown good agreement with the numerical results obtained by the MPS method. In the present work, results of exciting forces were compared with the results obtained by a validated program called Wave Analysis MIT (WAMIT). It had a good agreement of results between these two programs. The optimization of the linear system solver was made implementing in the developed source code a parallelized public code called Portable, Extensible Toolkit for Scientific Computation (PETSc) providing a good performance profit.*

Keywords: *particle method, MPS method, CFD, parallel computing, free surface*

1. Introduction

Particle methods are computational fluid dynamics (CFD) methods that can analyze complex geometries and fluid phenomena. Using this kind of method, analysis of phenomena as wave braking, slamming, green water and capsize of floating structures problems are easier when compared to grid methods. Smoothed Particle Hydrodynamics (SPH - Monaghan and Gingold (1983)) is a particle method that has been applied to compressible flow. The method used in the present work is another newer particle method called Moving Particle Semi-implicit (MPS - Koshizuka *et al.* (1995)) developed to analyse incompressible flow with free surface. In the MPS method, fluids are represented by particles without any grid. Only properties, initial positions and initial velocities of the particles are required. Governing equations are described by particle interaction model. A semi-implicit algorithm is employed to simulate incompressibility. Two-dimensional problems were solved to verify the validity of the particle interaction models used in the MPS method.

The firsts works about the method had been developed in nuclear energy applications. Later, some published works were focused in naval application (Sueyoshi and Naito (2002) and Naito and Sueyoshi (2002)) showing the method has potential to be a powerful program to analyse complex fluid phenomena interacting with walls and floating objects.

The MPS method still consumes big computacional time. One of the functions that consumes most of the calculation time in the MPS method program is the linear system solver that is required at each time step. The optimization of the linear system solver was made implementing in the developed source code a parallelized public code called Portable, Extensible Toolkit for Scientific Computation (PETSc). Tests results showed a good performance improvement.

2. Moving Particle Semi-Implicit Method

A outline of the MPS method is presented in this section. Details of the method are explained in Koshizuka *et al.* (1995) and Koshizuka and Oka (1996).

Conservation laws of mass and momentum for imcompressible non-viscous flows are the governing equations of the MPS method:

$$\frac{D\rho}{Dt} = -\rho\,(\nabla \cdot u) = 0 \tag{1}$$

and

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

Where, ρ is density, u velocity, P pressure, and f external force.

The continuity equation used in the MPS method is written using density, while velocity divergence is usually used in grid methods. The left side of the Eq. (2) denotes the Lagrangian differentiation including convection terms. The convection term is directly calculated by tracing particle motion. The right side consists of pressure gradient and external forces.

Interaction between a particle with its neighboring particles is determinated by a weight (or kernel) function $w(r)$, where r is the distance between particles. Equation (3) shows the weight function used in the MPS method, Fig. 1 represents the weight function graphically and Fig. 2 shows a simplified scheme how the MPS method understands a problem:

$$w\left(r\right) = \begin{cases} \dfrac{r_e}{r} - 1 & para \quad (0 \leq r \leq r_e) \\ 0 & para \quad (r_e \leq r) \end{cases} \tag{3}$$

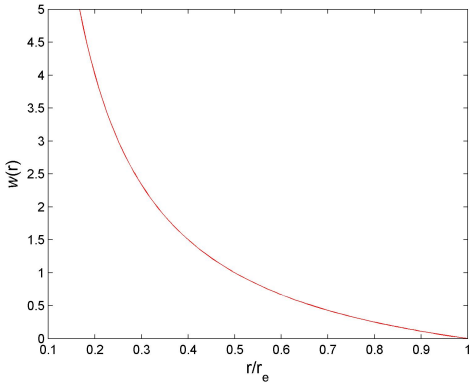

Figure 1. Kernel function

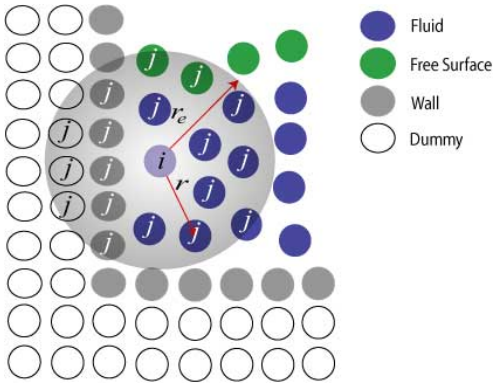

Figure 2. A simplified scheme of the MPS method

Since the area that is covered by this kernel function is bounded, a particle interacts with a finite number of neighboring particles. The radius of the interaction area is determined by parameter r_e (/refFigure:Scheme).

Particle number density at coordinate r_i, where particle i is located, is defined by:

$$\left\langle n\right\rangle_i = \sum_{j \neq i} w\left(\left|r_j - r_i\right|\right) \tag{4}$$

In this equation, the contribution from particle i itself is not considered. When the number of particles in a unit volume is denoted by $\left\langle N\right\rangle_i$, the relation between $\left\langle n\right\rangle_i$ and $\left\langle N\right\rangle_i$ is written by:

$$\left\langle N\right\rangle_i = \frac{\left\langle n\right\rangle_i}{\displaystyle\int_V w\left(r\right) dv} \tag{5}$$

The denominator of Eq. (5) is the integral of the kernel in the whole region, excluding a central part occupied by particle i. Assuming that the particles have the same mass m, the fluid density is proportional to the particle number density:

$$\left\langle \rho\right\rangle_i = m\left\langle N\right\rangle_i = m\frac{\left\langle n\right\rangle_i}{\displaystyle\int_V w\left(r\right) dv} \tag{6}$$

Thus, the continuity equation is satisfied when the particle number density $\left\langle n\right\rangle_i$ is constant. This constant value is denoted n^0.

A gradient vector between two particles i and j having scalar quantities ϕ_i and ϕ_j at coordinates r_i and r_j is simply defined by $\frac{(\phi_j - \phi_i)(r_j - r_i)}{\left|r_j - r_i\right|^2}$. The gradient vectors between particle i and its neighboring particles j are weighted with kernel function and averaged to obtain a gradient vector at particle i:

$$\left\langle \nabla\phi\right\rangle_i = \frac{d}{n^0}\sum_{j \neq i}\left[\frac{\phi_j - \phi_i'}{\left|r_j - r_i\right|^2}\left(r_j - r_i\right) w\left(\left|r_j - r_i\right|\right)\right] \tag{7}$$

Where d is the number of space dimensions and $\phi_i' = min\left(\phi_j\right)$ for any j satisfying $w\left(\left|r_j - r_i\right|\right) \neq 0$. This model is applied to the pressure gradient term in this method.

Laplacian is represented by Eq. (8):

$$\left\langle \nabla^2 \phi \right\rangle_i = \frac{2d}{pnd^0 \lambda} \sum_{j \neq i} \left(\phi_j - \phi_i\right) w\left(|r_j - r_i|\right) \tag{8}$$

Where:

$$\lambda = \frac{\int_V w\left(r\right) r^2 dv}{\int_V w\left(r\right) dv}$$

The continuity equation requires that the fluid density should be constant. This is equivalent to the particle number density being constant, n^0. When the particle number density explicitly calculated (n^*) is different to n^0, it is implicitly corrected to n^0 by:

$$n^* + n' = n^0 \tag{9}$$

Where n' is the correction value. This is related to the velocity correction value u' through the mass conservation.

$$\frac{1}{\Delta t} \frac{pnd'}{pnd^0} = -\nabla \cdot u' \tag{10}$$

The velocity correction value is derived from the implicit pressure gradient term as following.

$$\frac{u_i^{**} - u_i^*}{\Delta t} = -\frac{1}{\rho} \left\langle \nabla P^{n+1} \right\rangle_i \tag{11}$$

Where:

- $u_i' = u_i^{**} - u_i^*$;
- u_i^{**} is the particle's velocity at the next time step;
- u_i^* is the first guess of the particle's velocity calculated explicitly.

With Eq. (9), Eq. (10) and Eq. (11), a Poisson equation of pressure is obtained:

$$\left\langle \nabla^2 P^{n+1} \right\rangle_i = -\frac{\rho}{\Delta t^2} \frac{\left\langle n^* \right\rangle_i - n^0}{n^0} \tag{12}$$

The right side is represented by the deviation of the particle number density from the constant value, while it is usually velocity divergence in grid methods. The left side of Eq. (12) is described by the Laplacian model Eq. (8).

3. Examples

Some cases were simulated to check the validity of the method. Two of these cases are shown in Fig. 3 and 4. Figure 3 shows the simulation of a ship in extremely high waves and Fig. 4 shows a simulation of the collapse of a water column.

The results showed a realistic behaviour.

4. Validation

Exciting forces of a fixed body in regular waves were calculated using MPS method to compare with the program WAMIT to verify its accurancy.

Figure 5 shows the model that was used to get the results to compare with WAMIT.

Figures 6 and 7 show the comparison between these two methods for vertical forces and horintal forces, respectively.

These results of MPS method and WAMIT presented the same tendency. There were a difference between the results values because the program using the MPS method used a bidimensional model and WAMIT used a tridimensional model and there were some effects that the developed program was not considering.

It is important to say that the model used in the MPS method can be discretized with smaller particles to obtain more accurate results. Creating a model with bigger tank dimensions can avoid effects of water reflection.

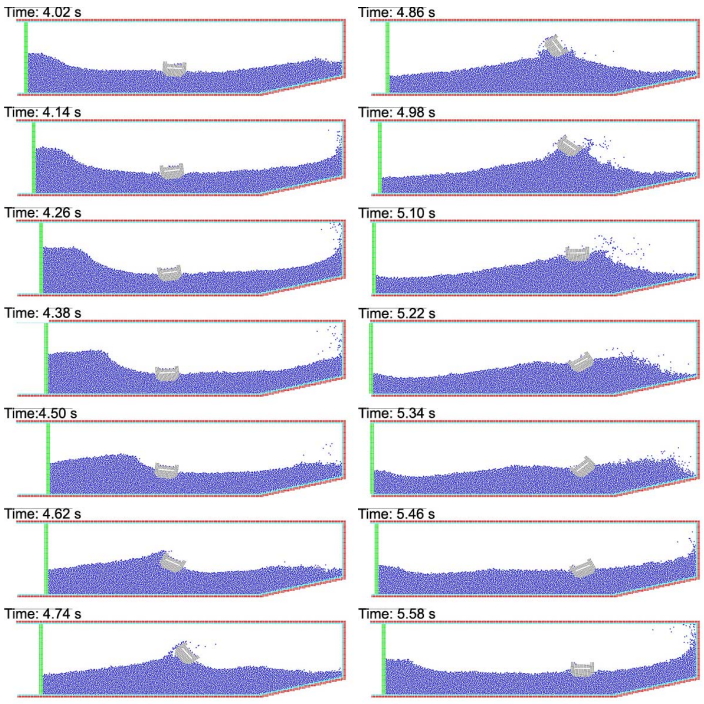

Figure 3. Ship in high waves

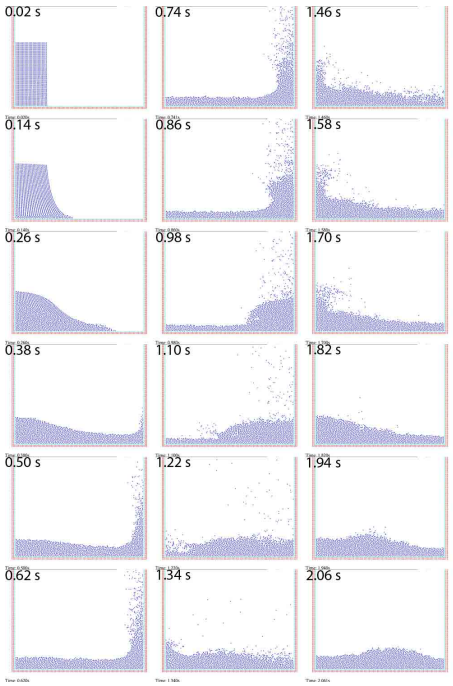

Figure 4. Collapse of a water column

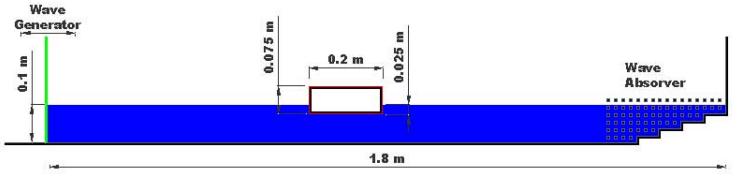

Figure 5. Calculation model

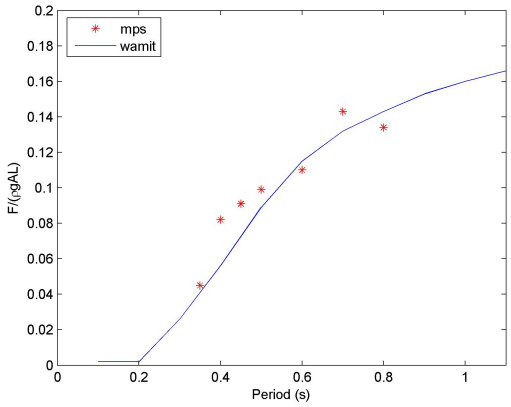

Figure 6. Vertical forces Amplitude (MPS and WAMIT)

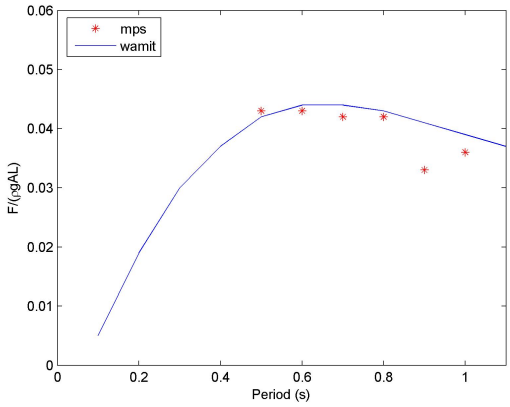

Figure 7. Horizontal forces Amplitude (MPS and WAMIT)

5. Optimization of the linear system solver

The open source created by professor Dr. Seiichi Koshizuka used conjugate gradient method called ICCG (Incomplete Cholesky Conjugate Gradient) to solve the linear system of equations. This method revealed sufficiently robust and fast in cases with less than 100000 particles. When the number of particles was bigger than 100000 the program took too long time to calculate the simulation.

Beacuse of the big computational time, it was necessary to optimize the source code. One of the solutions was the implementation of a package called PETSc (Portable, Extensible Toolkit will be Scientific Computation - Balay *et al.* (2001)) and using a cluster of computers. The PETSc library is a set of functions and structure of data for parallel solution of scientific applications. There are some methods to solve linear systems of equations using a single computer or a cluster of computers.

Some examples were calculated to evaluate its performance.

A group of tests were created to check if the implementation os the package PETSc provided a better performance.

Figure 8 shows the gain of performance using differents numbers of processors for 6 different quantities of particles.

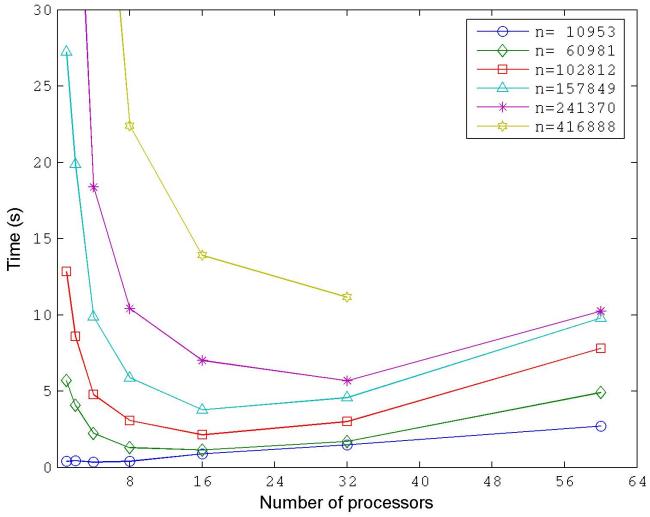

Figure 8. Computational time to solve the linear system for different number of processors with different number of particles

6. Conclusions

A program using the moving particle semi-implicit (MPS) method was developed successfully for simple two-dimensional problems. It can be a very powerful program to calculate highly non-linear problems with fluid-structure interaction. In this study, only linear problems (waves exciting forces) was validated with a good agreement when compared with the program WAMIT.

The linear system solver optimization using a cluster of computers provided a good economy of time. However, It is still necessary to make the program faster to be applicable for three-dimensional problems.

7. References

Balay, S., Buschelman, K., Gropp, W. D., Kaushik, D., Knepley, M. G., McInnes, L. C., Smith, B. F. and Zhang, H. (2001). PETSc Web page. `http://www.mcs.anl.gov/petsc`.

Koshizuka, S. and Oka, Y. (1996). Moving-particle semi-implicit method for fragmentation of incompressible fluid. *Nuclear Science and Engineering*, 123:421–434.

Koshizuka, S., Tamako, H. and Oka, Y. (1995). A paricle method for incompressible viscous flow with fluid fragmentation. *Journal Computational Fluid Dynamics*, 62:29–46.

Monaghan, J. J. and Gingold, R. A. (1983). Shock simulation by the particle method sph. *Journal of Computational Physics*, 52:374–389.

Naito, S. and Sueyoshi, M. (2002). A numerical study on complicated motions of floating bodies. In *Sixth International Ship Stability Workshop*, Glen Cove, New York. Webb Institute.

Sueyoshi, M. and Naito, S. (2002). A numerical study of very large motion of floating body by particle method. In *Asia Pacific Maritime Congress*, pages 146–150, Kobe, Japan. Asia Pacific Workshop on Marine Hydrodynamics.