Abstract for the 20th Intl Workshop on Water Waves and Floating Bodies, Spitsbergen (Norway), 29th May - 1st June, 2005

A Particle Method for Impulsive Loads Caused by Violent Sloshing

by Makoto Sueyoshi, Zdravko R. Kishev and Masashi Kashiwagi
Research Institute for Applied Mechanics, Kyushu University
Kasuga Fukuoka 816-8580, Japan

Introduction

It has been difficult to simulate the impulsive pressure by violent sloshing because of extreme deformation of the free surface. Traditional techniques to treat free surface motion are not suitable for such problem due to limitation in the deformation of grid or the numerical diffusion. However, in recent years, some new techniques are available to treat violent motion of moving boundaries. One of such numerical techniques is a particle method. With this method, Landrini et al. [2] show numerical and experimental results of the sloshing problem in a partially filled tank. Their numerical method is the SPH (Smoothed Particle Hydrodynamics) method [1], whereas the present paper uses the MPS (Moving Particle Semi-implicit) method introduced by Koshizuka et al. [3] to simulate violent sloshing.

MPS Method and Its Improvement

The MPS method is similar to the SPH method in that both are based on the Lagrangian co-ordinate system. However the unique spatial discretization and the semi-implicit treatment of pressure-velocity coupling are salient characteristics. The discretization model is called a particle interaction model. The gradient of a scalar Φ is described by

$$\nabla\Phi_i = \frac{d}{\sum_j^N w(r_{ij})} \sum_{j\neq i}^{N} w(r_{ij}) \frac{\Phi_j - \Phi_i}{r_{ij}} \frac{\mathbf{r}_j - \mathbf{r}_i}{r_{ij}} . \tag{1}$$

where d is the number of dimension of computation region, i.e. d =2.0 for 2D computations and d =3.0 for 3D. $\mathbf{r}$ is the position vector of a particle, r_{ij} is the distance between particles denoted by subscript i and j, and $w(r)$ is a kernel function described by

$$w(r) = \begin{cases} \dfrac{r_0}{r} - 1 & : r \le r_0 \\ 0 & : r > r_0 \end{cases} . \tag{2}$$

The gradient of Φ can be described by the weighted average of products of the difference in Φ and the unit vector. This discretized form is simple and does not need any information of connectivity relationship among computation points. In addition to say, it does not depend on whether it is 2D or 3D computation. The discrete form for the diffusion of Φ is described by

$$\nabla^2\Phi_i = \frac{2d}{\lambda} \sum_{j\neq i}^{N} w(r_{ij})(\Phi_j - \Phi_i) . \tag{3}$$

This form is a distribution of Φ from particle i to neighboring particles j. λ is a constant to adjust the distributed quantity to an analytical solution.

The other important feature, semi-implicit velocity-pressure coupling, is similar to a usual fractional step method. The first step of the sequence is the computation of temporal velocity and position vector, $\mathbf{u}^*$ and $\mathbf{r}^*$. They are described by

$$\mathbf{u}^* = \mathbf{u}^n + \Delta t\left(\mathbf{g} + \nu\nabla^2\mathbf{u}^n\right) \quad \text{and} \quad \mathbf{r}^* = \mathbf{r}^n + \Delta t\mathbf{u}^* \tag{4}$$

where $\mathbf{g}$ is the vector of the gravitational acceleration and ν is the kinematic viscosity.

The diffusion term in (4) is discretized by using (3). The next step is the computation of the particle number density n as

$$n_i^* = \sum_{j\neq i}^{N} w(r_{ij}) . \tag{5}$$

Then a Poisson equation of pressure P is solved by an appropriate iterative solver. The Poisson equation in the MPS

method is described with the particle number density by

$$\nabla^2 P_i^{n+1} = -\frac{\rho}{\Delta t^2}\frac{(n_i^* - n_{const})}{n_{const}}. \qquad (6)$$

Last, the pressure gradient is calculated, which corrects the velocity and position vectors at n+1 time step. They are

$$\mathbf{u}^{n+1} = \mathbf{u}^* - \Delta t \frac{1}{\rho}\nabla P_i^{n+1} \quad \text{and}$$

$$\mathbf{r}^{n+1} = \mathbf{r}^n + \Delta t\, \mathbf{u}_i^{n+1}. \qquad (7)$$

These steps are repeated until the end of all computations.

This sequence is quite different from the usual SPH method, which uses the equation of state to compute pressure.

Free surface profiles simulated by the MPS method shows good agreement with experimental results. Figure 1 is an example of comparison of the free-surface shape in a dam-breaking flow between numerical and experimental results. However it has been pointed out that the calculated pressure includes violent fluctuation in time and space. Therefore, to reduce it, an auxiliary computation for the pressure is added to the original MPS method [4]. This auxiliary computation calculates a sub-flow field on the particle arrangement calculated by the original MPS method. The spatial discretization of sub-flow field uses (1), (3) and their combinations. The velocity-pressure coupling scheme is a simple fractional step method similar to conventional methods using grids.

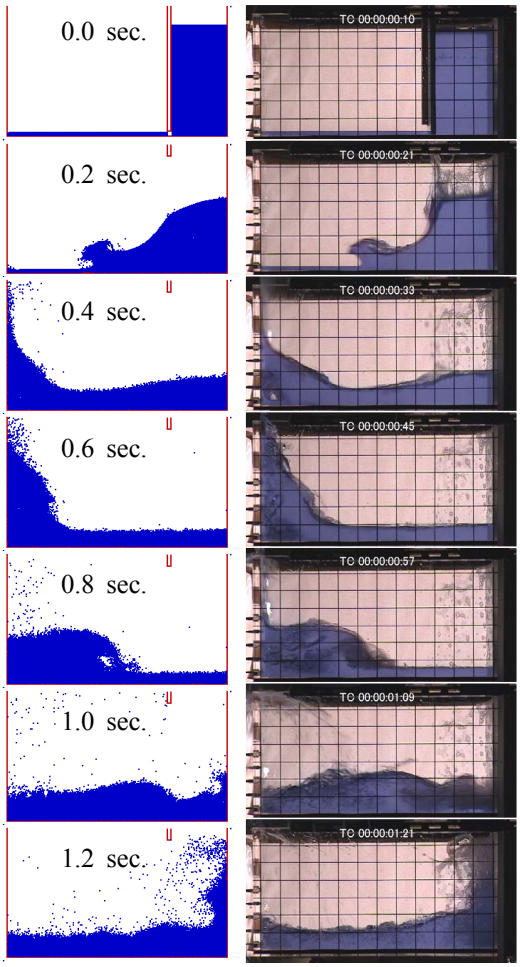

Figure 1 Comparison of free surface profiles between numerical computation(by MPS method) and experimental result.

The first step is the computation of temporal velocity of sub flow field described by

$$\mathbf{u}_{ALE}^* = \mathbf{u}_{ALE}^n + \Delta t \begin{bmatrix} -\{(\mathbf{u}_{ALE}^n - \mathbf{u})\cdot\nabla\}\mathbf{u}_{ALE}^n \\ +\mathbf{g} + \nu\nabla^2\mathbf{u}_{ALE}^n \end{bmatrix}, \qquad (8)$$

where $\mathbf{u}_{ALE}$ is the velocity vector on the sub-flow field, and $\mathbf{u}$ is the velocity vector of particle motion calculated by the original MPS method. Spatially descretized points are treated as moving nodes as in the same in the ALE method. The next step is computation of the divergence of $\mathbf{u}^*_{ALE}$.

$$D_i = \nabla\cdot\mathbf{u}^*_{ALE\,i} \cong \frac{d}{\sum_{j\neq i}^{N} w(r_{ij})}\sum_{j\neq i}^{N} w(r_{ij})\frac{\left(\mathbf{u}^*_{ALE\,j} - \mathbf{u}^*_{ALE\,i}\right)}{r_{ij}}\cdot\frac{\left(\mathbf{r}_j^n - \mathbf{r}_i^n\right)}{r_{ij}}. \qquad (9)$$

Then a Poisson equation for the pressure on sub-flow field P_{ALE} is described by

$$\nabla^2 P_{ALE}^{n+1} = \rho\frac{D_i}{\Delta t}. \qquad (10)$$

Finally, $\mathbf{u}_{ALE}$ at n+1 step is corrected from $\mathbf{u}^*_{ALE}$ as

$$\mathbf{u}_{ALE}^{n+1} = \mathbf{u}_{ALE}^* - \Delta t\frac{1}{\rho}\nabla P_{ALE}^{n+1}. \qquad (11)$$

The sequence is implemented in parallel with computation of the original MPS method shown in Figure 2.

Results and Discussion

Computed results of the pressure by both methods are shown in Figure 3 for a dam breaking flow. The time history of the pressure by the original MPS method includes unrealistic violent fluctuation, whereas the present method using the auxiliary computation shows a much smoother time history.

The present method is applied to the sloshing problem and compared with the experimental result carried out by Hu and Kishev [5] to validate the numerical result. The experimental tank and conditions are shown in Figure 4.

Comparison of free surface profiles in the case of forced oscillation period of 1.3 sec. is shown in Figure 5, which is the most violent case because 1.3sec. is about the resonant frequency. Good agreement is obtained for the free-surface profiles at each time instant.

The most important quantity in this study is the impulsive load caused by violent sloshing. At point P1 in Figure 4, time history of the pressure was experimentally measured and is numerically calculated. Results of two cases are shown in Figures 6 and 7. For the period 1.3 sec., two characteristic peaks appear over one period in both of numerical and experimental results. The time interval between two peaks is almost the same. For the period 1.7sec., a primary sharp peak and secondary peak can be observed. These are also simulated clearly.

As a conclusion, good qualitative agreement is obtained for the time history of the pressure by the present method, even when the flow is extremely violent. Quantitative agreement looks also very promising.

References

1) JJ Monaghan : Simulating Free Surface Flows with SPH, Jour. of Comp. Phys., Vol. 110, 1994, pp.399-406.

2) Murizio Landrini, Andrea Colagrossi, Odd M. Faltinsen : Sloshing in 2-D Flows by SPH Method, Proc. of The 8th

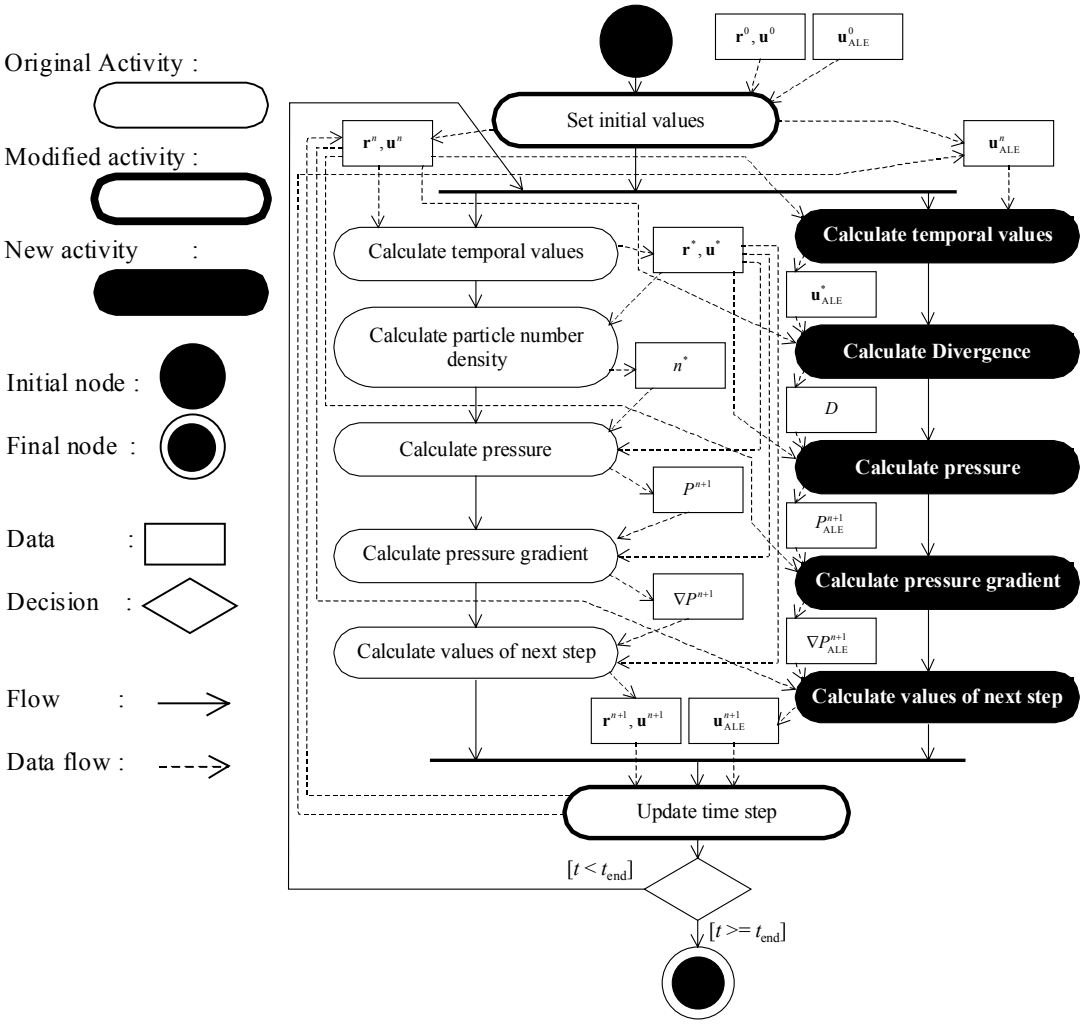

Figure 2 Calculation flow by combination of the MPS method and an auxiliary computation.

3) S. Koshizuka and Y. Oka : Moving-Particle Semi-Implicit Method for Fragmentation of In- compressible Fluid, Nuclear Science and Engineering, 123, pp421-434,1996.

4) M. Sueyoshi, S. Naito : A Numerical Study of Violent Free Surface Problems with Particle Method for Marine Engineering, Proc. of 8th International Conference on Numerical Ship Hydrodynamics, Vol.2, 330-339, 2003.

5) Changhong Hu, Zdravko Kishev : Numerical Simulation of Violent Sloshing by CIP-CSL3 Method, Report of RIAM Symposium No. 15 ME-S7, 2004, pp39-43.

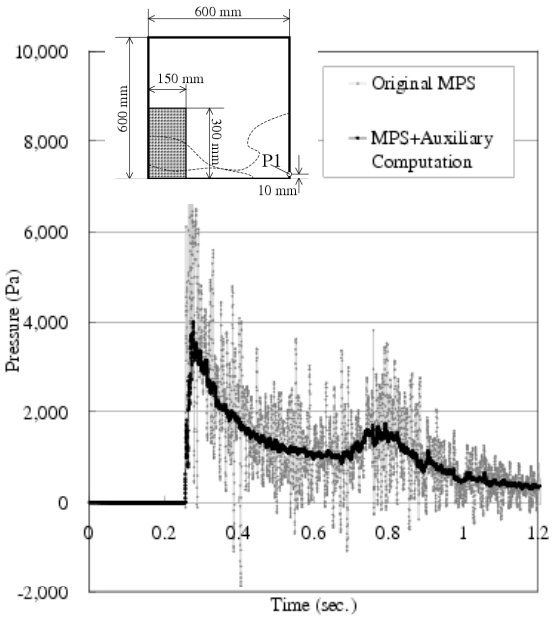

Figure 3 Comparison of time history of the pressure on side wall at P1 between the original and the present MPS methods in a case of dam-breaking flow.

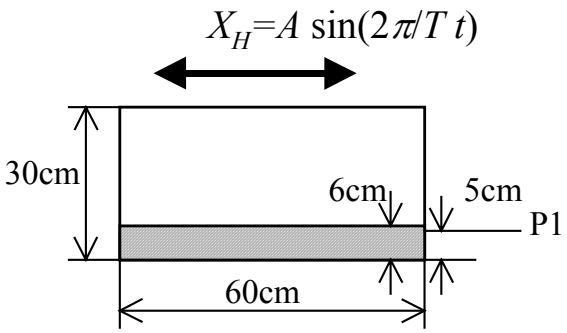

Figure 4 Schematic of the set up for sloshing experiment and computation.

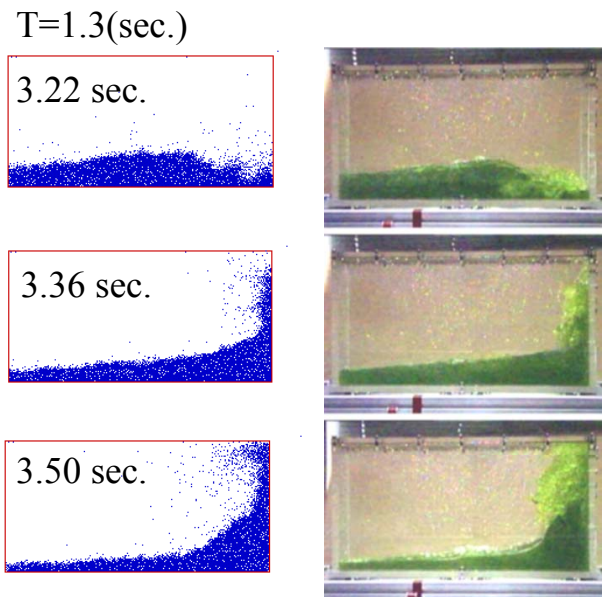

Figure 5 Comparison of free surface profiles at T=1.3(sec.), l_0 = 2.5(mm)

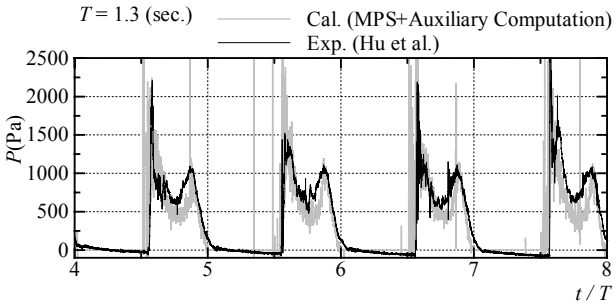

Figure 6 Comparison of time history of the pressure between numerical simulation and experiment (T=1.3(sec.)).

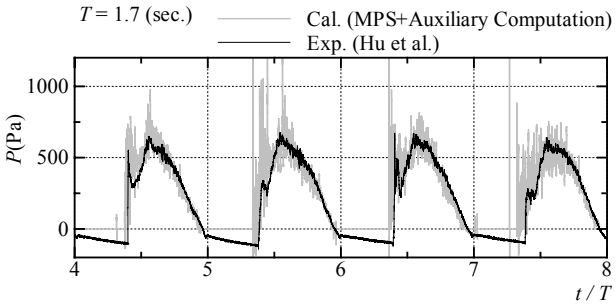

Figure 7 Comparison of time history of the pressure between numerical simulation and experiment (T=1.7(sec.)).