11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

LARGE SCALE SIMULATION OF FLUID-STRUCTURE INTERACTION USING AN INCOMPRESSIBLE SMOOTHED PARTICLE HYDRODYNAMICS

ABDELRAHEEM M. ALY[*,†] **AND MITSUTERU ASAI**[†]

[*] Department of Civil Engineering, Kyushu University, 744 Motooka, Nishi–ku, Fukuoka
819-0395, Japan
e-mail: asai@doc.kyushu-u.ac.jp

[†]Department of Mathematics, Faculty of Science, South Valley University, Qena, 83523, Egypt
email: abdelraheem@doc.kyushu-u.ac.jp

Key Words: *Free surface, Flood disaster, Hydrodynamic forces, ISPH method, Rigid body.*

Abstract. Numerical simulations for free surface flow models, which are water entry of several rigid bodies, fluid tank sloshing and flood disaster over several rigid bodies were conducted by using an Incompressible smoothed particle hydrodynamics (ISPH) method. The governing equations are discretized and solved with respect to Lagrangian moving particles filled within the mesh-free computational domain and the pressure was evaluated by solving pressure Poisson equation using a semi-implicit algorithm based on the projection scheme to ensure divergence free velocity field and density invariance condition. In this study, we modeled the structure as a rigid body motion by two different techniques.

In the first technique, we modelled the rigid body corresponding to Koshizuka et al. [1]. They proposed a passively moving-solid model to describe the motion of rigid body in a fluid. Firstly, both of fluid and solid particles are solved with the same calculation procedures. Secondly, an additional procedure is applied to solid particles.

In the second technique, we compute the motions of a rigid body by direct integration of fluid pressure at the position of each particle on the body surface and the equations of translational and rotational motions were integrated in time to update the position of the rigid body at each time step.

The performance of these two techniques was validated through the comparison with experimental results.

1 INTRODUCTION

For fluid-structure interaction problems, the structures are usually described by a

Lagrangian formulation, while the fluids are often described with an Eulerian one. The coupling of the two media is generally obtained using a formulation called ALE (Arbitrary Lagrangian-Eulerian) for the fluid. Various problems have been tackled via this formulation, including the study of valve spring Rugonyi and Bathe [2], the interaction of compressible and incompressible fluids with structures Bathe and Zhang [3], or the absorption of hydro-elastic shock Le Tallec and Mouro [4]. Alternatively in the Lagrangian SPH method, the state of a whole system is represented by a set of particles, which possess individual properties and move with the fluid. The SPH technique was originally proposed by Lucy [5] and further developed by Gingold and Monaghan [6] for treating astrophysical problems. Its main advantage is the absence of a computational grid or mesh since it is spatially discretized into Lagrangian moving particles. This allows the possibility of easily modelling flows with a complex geometry or flows where large deformations or the appearance of a free surface occurs. The SPH methods have already produced encouraging results for problems of interaction between solids Gray et al. [7].

The SPH is originally developed in compressible flow, and then some special treatment is required to satisfy the incompressible condition. One approach is to run the simulations in the quasi-incompressible limit, that is, by selecting the smallest possible speed of sound which still gives a very low Mach number ensuring density fluctuations within 1% [8-11]. This method is known as the Weakly Compressible Smooth Particle Hydrodynamics (WCSPH). In the WCSPH, the artificial viscosity, which is originally developed by Monaghan, has been widely used not only for the energy dissipation but also for preventing unphysical penetration of particles.

A proposal for constructing an incompressible SPH model has been introduced, whose pressure is implicitly calculated by solving a discretized pressure Poisson equation at every time step [12-24]. A stabilized incompressible SPH method involving relaxation of the density invariance condition was proposed by Asai et al. [22]. In this technique, the pressure Poisson equation was solved related to velocity divergence-free and density invariance conditions. This formulation leads to a new pressure Poisson equation with a relaxation coefficient, which can be estimated via a pre-analysis calculation. Asai et al. [22] tested the efficiency of the proposed formulation via a numerically-modelled dam-breaking problem, and its effects were discussed using several resolution models with different initial particle distances. Aly et al. [23, 24] applied the stabilized version of the incompressible SPH method to simulate both fluid-fluid and fluid-structure interactions. In the most recent work of Koh et al. [25], the consistent particle method was proposed to eliminate pressure fluctuation in solving large-amplitude, free-surface motion. In this method, which is accompanied with an alternating of the kernel function by the Taylor series expansion-based partial differential operators, a zero-density-variation condition and a velocity-divergence-free condition are also combined in a source term of PPE to enforce fluid incompressibility.

In the current study, we modelled the fluid-structure interaction using stabilized ISPH method [22]. Both of the fluid and structure domains are modelled by ISPH method. The structure is modelled by two different techniques in ISPH method. In the first technique, we modelled the rigid body corresponding to Koshizuka et al. [1]. In the second technique, we compute the motions of a rigid body by direct integration of fluid pressure at the position of each particle on the body surface and the equations of translational and rotational motions were integrated in time to update the position of the rigid body at each time step.

In this study, several different simulations for fluid-structure interaction in large scale domain were performed

2 INCOMPRESSIBLE SMOOTHED-PARTICLE HYDRODYNAMICS (ISPH) FORMULATION

The incompressible SPH formulation is a procedure similar to the moving particle, semi-implicit method (MPS) proposed by Koshizuka et al. [26]. The primary feature is the application of a semi-implicit integration scheme to particle-discretized equations for the incompressible flow problem.

2.1 Governing equation

The flow mass and momentum conservation equations are presented as:

$$\frac{D\rho}{Dt} + \rho\nabla\cdot\mathbf{u} = 0, \tag{1}$$

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla p + \nu_0\nabla^2\mathbf{u} + \mathrm{g} + \frac{1}{\rho}\nabla\cdot\boldsymbol{\tau} \tag{2}$$

where ρ and ν_0 are the density and kinematic viscosity of the fluid, $\mathbf{u}$ and p are the velocity vector and pressure of the fluid respectively, and t indicates time. The turbulence stress $\boldsymbol{\tau}$ is necessary to represent the effects of turbulence with coarse spatial grids. In the most general incompressible flow approach, the density, ρ, is assumed by a constant value, ρ^0, with the initial value. The aforementioned governing equations lead to the following progression as follows,

$$\nabla\cdot\mathbf{u} = 0, \tag{3}$$

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho^0}\nabla p + \nu_0\nabla^2\mathbf{u} + \mathbf{g} + \frac{1}{\rho}\nabla\cdot\boldsymbol{\tau} \tag{4}$$

The primary issue in an incompressible SPH method is solving a discretized PPE at every time step to obtain the pressure value. In this study, the following formulation is mathematically derived and used for pressure calculation.

$$< \nabla^2 p^{n+1} > \; = \frac{\rho^0}{\Delta t} < \nabla\cdot\mathbf{u}^* > \; + \alpha\frac{\rho^0 - < \rho^n >}{\Delta t^2} \tag{5}$$

where, $(0 \leq \alpha \leq 1)$ is the relaxation coefficient, $\mathbf{u}^*$ the temporal velocity, and the triangle bracket $< >$ is the SPH approximation. A detailed procedure for deriving Eq. (5) can be found in Asai et al. [22].

3 TREATMENT OF A MOVING RIGID BODY

In the first technique, Koshizuka et al. [1] proposed a passively-moving solid model to describe the motion of a rigid body in a fluid. According to this study, the treatment of the moving rigid body in the fluid can be divided into two steps. First, both the fluid and solid particles are solved via the same calculation procedures. Secondly, an additional procedure is applied to solid particles.

Considering n solid particles with location, $\boldsymbol{r}_k$, the center of the solid object, $\boldsymbol{r}_c$, and the relative coordinate of a solid particle to the center, $\boldsymbol{q}_k$, the moment of inertia, I , of the solid object can be calculated.

$$\boldsymbol{r_c} = \frac{\boldsymbol{1}}{\boldsymbol{n}} \sum_{\boldsymbol{k=1}}^{\boldsymbol{n}} \boldsymbol{r_k}, \tag{6}$$

$$\boldsymbol{q_k} = \boldsymbol{r_k} - \boldsymbol{r_c}, \tag{7}$$

$$I = \sum_{\boldsymbol{k=1}}^{\boldsymbol{n}} |\boldsymbol{q_k}|^{\boldsymbol{2}}, \tag{8}$$

The translational velocity, T, and rotational velocity, R, of a solid object are calculated as:

$$\boldsymbol{T} = \frac{1}{n} \sum_{k=1}^{n} \boldsymbol{u_k}, \tag{9}$$

$$\boldsymbol{R} = \frac{1}{I} \sum_{k=1}^{n} \boldsymbol{u_k} \times \boldsymbol{q_k}, \tag{10}$$

Finally, the velocity of each particle in the solid body can be expressed as

$$\boldsymbol{u_k} = \boldsymbol{T} + \boldsymbol{q_k} \times \boldsymbol{R}\,. \tag{11}$$

With the above rigid body correction, the motion of a free-moving object can be computed. Gotoh and Sakai [27] and Shao [28] showed that the above treatment can be applied to a free-falling wedge in water, and works well in a stable computation where the Courant condition is satisfied.

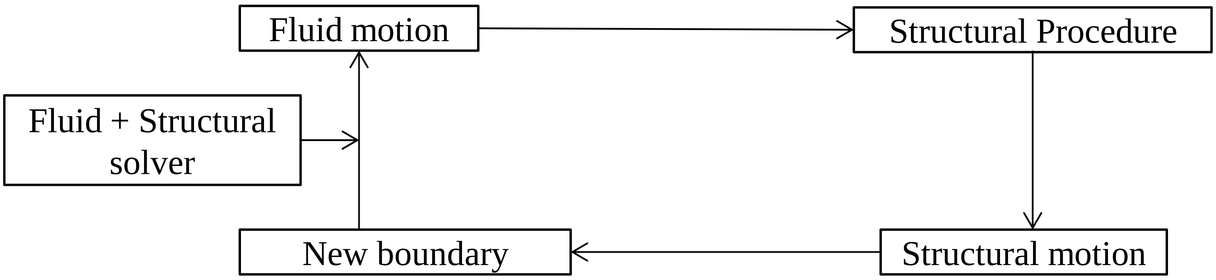
In the second technique, we compute the motions of a rigid body by direct integration of fluid pressure at the position of each particle on the body surface and the equations of translational and rotational motions were integrated in time to update the position of the rigid body surface at each time step. The equations for translation motions are described as:

$$M \frac{d^2 \boldsymbol{r_c}}{dt^2} = M\mathbf{g} + \boldsymbol{F_f} + \boldsymbol{F}_e, \tag{12}$$

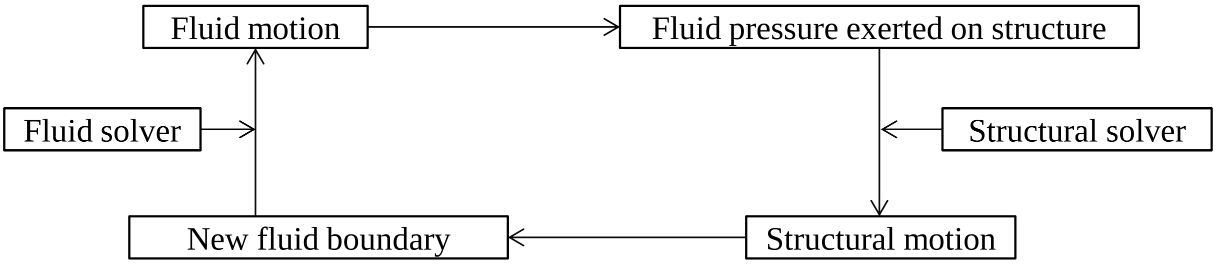
where, M is mass of the body, F_f is the hydrodynamic forces acting on body surface and $\boldsymbol{F_e}$ is the other external forces. The equation of rotational motions are described as:

$$I\ddot{\theta} + \dot{\theta} \times I \cdot \dot{\theta} = \boldsymbol{M_f} + \boldsymbol{M}_e, \tag{13}$$

where, $\dot{\boldsymbol{\theta}}$ is the angular velocity, $\boldsymbol{M_f}$ is the hydrodynamic moment and $\boldsymbol{M_e}$ is the external moment. Here, the hydrodynamic forces and the hydrodynamic moment are calculated as. $\mathbf{F}_f = \sum_i^{NS} p_i \Delta S_i;\ \mathbf{M}_f = \sum_i^{NS} \left(\mathbf{r}_i - \mathbf{r}_c\right) \times p_i \Delta S_i$, NS is the number of body surface particles and ΔS_i is the area of body surface at particle i .



(a) First technique



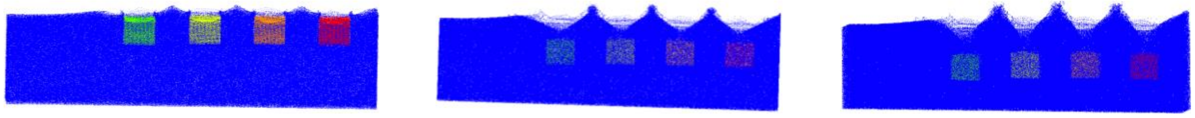
(b) Second technique

Fig. 2 Flow chart of fluid-structure interaction, (a) First technique and (b) Second technique.

4 RESULTS AND DISCUSSION

In this study, several different simulations for the fluid-structure interactions have been introduced and discussed. The structure is taken as a rigid body and is modeled by ISPH method.

Free falling of several rigid body over water in tank is simulated with three diferent densities cases0.5, 1.0 and 1.5, respectively. Fig. 2 shows the snapshots of free falling of several rigid bodies over water in tank at times 0.5 and 1.0 sec for three different density ratios 0.5, 1.0 and 1.5, respectively. The rigid body with small density is still floating over fluid and at the case of similar density, the rigid body is going down to the fluid centre and it still moves inside the fluid, while rigid body with high density is going down directly to the bottom of fluid tank.

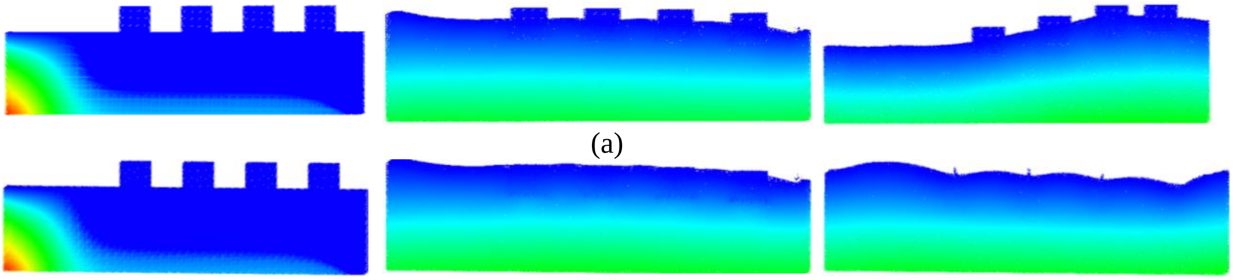


(a) Time 0.5 sec

(b) Time 1.0 sec

Fig. 2 Snapshots of free falling for several rigid bodies over water in tank at times 0.5 and 1.0 sec for three different density ratios 0.5, 1.0 and 1.5, respectively

Fig. 3 shows the snapshots of pressure distribution for fluid tank sloshing. Here, the fluid sloshing problem in a rectangular tank under a sway excitation are introduced with external excitation applied on the tank $x = A(\sin \omega t)$, where A=0.004 m and $\omega = 7.3996$ rad/s. In this figure, the pressure distribution shows the efficiency of the current ISPH method in stabilizing the evaluated pressure and keeping the total volume of fluid during the whole simulation.

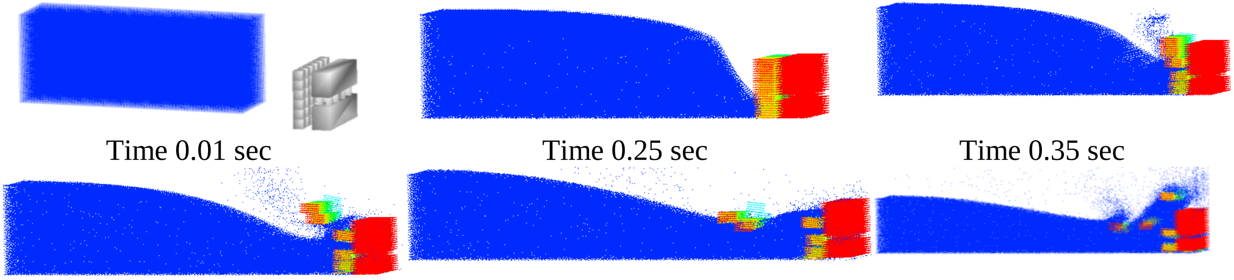

(a)

(b)

Time 0.05 sec Time 1.25 sec Time 2.5 sec

Fig. 3 shows the snapshots of pressure distribution for fluid tank sloshing including rigid body motion with two density ratios (a) 0.5 and (b) 1.0, respectively at times 0.05, 1.25 and 2.5 sec.

The flood disaster over structure is simulated by introducing fluid column collapse impacts several rigid bodies as shown in figure 4.


Time 0.01 sec Time 0.25 sec Time 0.35 sec

Time 0.5 sec Time 0.75 sec Time 1.0 sec

Fig. 4 shows the time histories for the flood disaster impacts several rigid bodies.

For the large scale disaster, we adapted ISPH method to simulate impact flow over structure body. The initial schematic diagram for the current model is introduced in figure 5. The water column has length 50 metre and height 15 metre. The distance between the water column and bridge is 10 mitre. The bridge has height 10.32 mitre with girder, in which the girder is taken as rigid body and the bridge bases are fixed. The snapshots of impact flood disaster over bridge are introduced in figure 6. The water impacts the bridge with high impact force which strongly releases the bridge girder.

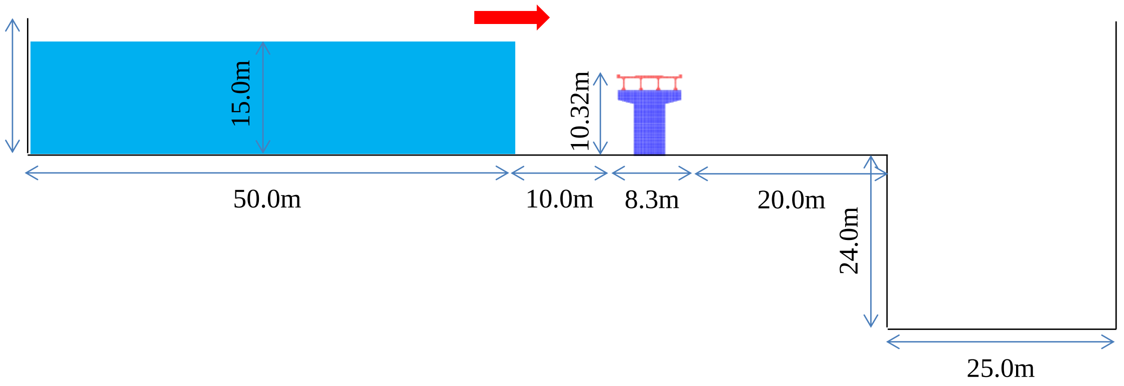
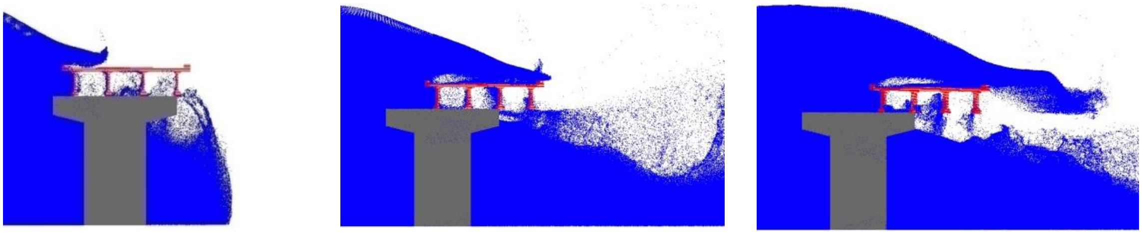
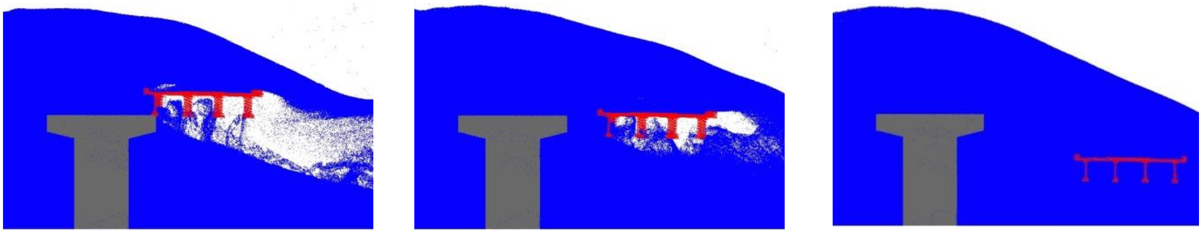


Fig. 5 shows the initial schematic for the large scale disaster model.



Time 1.25 Time 1.75 Time 2.25



Time 2.75 Time 3.25 Time 3.75

Fig. 6 Snapshots of large scale flood disaster over bridge.

CONCLUSIONS

- A stabilized incompressible SPH model with new treatment of boundary condition is successfully adapted to investigate large-scale simulation of fluid-structure interaction. The rigid body is modelled by two different techniques in literatures using ISPH method.
- Numerical simulations of the liquid sloshing with freely floating objects and free falling objects over water tank with varying density were performed.
- The flood disaster in large scale over several rigid bodies and over bridge are simulated using stabilized ISPH method. The bridge girder is taken as rigid body in

large scale simulation.

- The current ISPH method is parallelized to achieve large scale simulation up to 100.000.000 particles.

REFERENCES

[1] Koshizuka, S. Nobe A. and Oka, Y. Nmerical analysis of breaking waves using the moving particle semi-implicit method. *Int. J. Numerical Methods in Fluids,* (1998) **26**: 751-769.

[2] Rugonyi S. and Bathe. K.J. On finite element analysis of fluid flows fully coupled with structural interactions. *Comput Model Eng Sci,* (2001) **2**: 195-212.

[3] Bathe K.J.and Zhang H. Finite element developments for general fluid flows with structural interactions. *Int J. Numer Methods Eng,* (2004) **60**: 213–32.

[4] Tallec P.Le and Mouro J. Fluid structure interaction with large structural displacements. *Comput Methods Appl Mech Eng.* (2000) 1–29.

[5] Lucy L.B. A numerical approach to the testing of the fusion process. *Astron J.* (1977) **88**: 1013–1024,.

[6] Gingold R.A. and Monaghan J.J. Smoothed particle hydrodynamics: theory and application to non-spherical stars. *Mon Not R Astron Soc,* (1977) **181**: 375–89.

[7] Gray, J.P., Monagahn J.J. and Swift R.P. SPH elastic dynamics. *Comp Methods Appl Mech Eng,* (2001) **190**: 6641-6662.

[8] Monaghan. J.J. Simulating Free Surface Flows with SPH. *Journal of Computational Physics,* (1994) **110**: 399-406.

[9] Morris, J.P. Fox P.J. and Zhu Y. Modeling Low Reynolds Number Incompressible Flows Using SPH. *Journal of Computational Physics,* (1997) **136:** 214-226.

[10] Monaghan, J.J. Heat Conduction with Discontinuous Conductivity, *Applied Mathematics Reports and Preprints, Monash University* 1995.

[11] Okahci, N. Hirota, A., Izawa, S. Fukunishi Y. and Higuchi. H. SPH Simulation of Pulsating Pipe Flow at a Junction. in: *Proceedings of 1st International Symposium on Advanced Fluid Information,* (2001) 388-391,.

[12] Cummins S.J. and Rudman. M. An SPH projection method. *Journal Computational Physics,* (1999) **152** (2): 584–607.

[13] Pozorski J. and Wawrenczuk A. SPH computation of incompressible viscous flows. *J. Theor. Appl. Mech.* (2002) **40**: 917-937.

[14] Shao, S. EYM. Lo. Incompressible SPH method for simulating Newtonian and non-Newtonian flows with a free surface. *Advances in Water Resources,* (2003) **26**: 787–800.

[15] Hu X.Y. and Adams N.A. Angular-momentum conservative smoothed particle dynamics for incompressible viscous flows. *Phys. Fluids,* (2006) **18**; 101702-4.

[16] Ellero, M. Serrano M. and Espanol. P. Incompressible smoothed particle hydrodynamics. *Journal of Computational Physics,* (2007) **226**: 1731–1752.

[17] Lee, E-S. Moulinec, C. Xu, R. Violeau, D. Laurence, D. Stansby. P. Comparisons of weakly compressible and truly incompressible algorithms for the SPH mesh free particle method. *Journal of Computational Physics,* (2008) **227**(18): 8417–8436.

[18] Khayyer, A. Gotoh H. and Shao. S. Corrected incompressible SPH method for accurate water-surface tracking in breaking waves. *Coastal Engineering*, (2008) **55**: 236–250.

[19] Khayyer, A. Gotoh, H. Shao. S. Enhanced predictions of wave impact pressure by improved incompressible SPH methods. *Applied Ocean Research*, (2009) **31**: 111-131.

[20] Hu X.Y. and Adams N.A. An incompressible multi-phase SPH method. *Journal of Computational Physics*, (2007) **227**: 264-278.

[21] Hu X.Y. and Adams N.A. A constant-density approach for incompressible multi-phase SPH. *Journal of Computational Physics,* (2009) **228**: 2082–2091.

[22] Asai, M. Aly, A.M. Sonoda Y. and Sakai. Y. A Stabilized Incompressible SPH method by relaxing the Density invariance condition. *Journal of Applied Mathematics*, (2012), 24. doi:10.1155/2012/139583.

[23] Aly, A.M. Asai, M. Sonoda. Y. Simulation of free falling rigid body into water by a stabilized incompressible SPH method. *Ocean Systems Engineering, An international journal*, (2011) **1(3)**: 207-222.

[24] Aly, A.M. Asai, M. Sonoda. Y. Modelling of surface tension force for free surface flows in ISPH method. *International Journal of Numerical Methods for Heat & Fluid Flow*, **(**2013**) 23(3)**: 479–498.

[25] Koh CG, Luo M., Gao, M. and Bai, W. Modeling of liquid sloshing with constrained floating baffle. *Comput. Struct*. (2013); http://dx.doi.org/10.1016/j.compstruc.2013.03.018.

[26] Koshizuka S, Nobe A, Oka Y. Numerical analysis of breaking waves using the moving particle semi-implicit method. *Int J Numer Method Fluid*. (1998) **26**:751–69.

[27] Gotoh H and Sakai T. Key issues in the particle method for computation of wave breaking. *Coastal Engineering* (2006) **53**: 171–179.

[28] Shao S. Incompressible SPH simulation of water entry of a free-falling object. *Int. J. Numer. Meth. Fluids* (2009) **59**: 91–115.