Available online at http://ijim.srbiau.ac.ir/
Int. J. Industrial Mathematics (ISSN 2008-5621)
Vol. 7, No. 1, 2015 Article ID IJIM-00513, 8 pages
Research Article

Incompressible smoothed particle hydrodynamics simulations on free surface flows

A. Mahmoud Aly *†

Abstract

The water wave generation by wave paddle and a freely falling rigid body are examined by using an Incompressible Smoothed Particle Hydrodynamics (ISPH). In the current ISPH method, the pressure was evaluated by solving pressure Poisson equation using a semi-implicit algorithm based on the projection scheme and the source term of pressure Poisson equation contains both of divergence free velocity field and density invariance condition. Here, the fluid-structure interaction is introduced in free surface flows and the structure is taken as a rigid body motion. In this study, we generated the water waves using the Scott Russell wave generator, in which the heavy box sinking vertically into water. Also, the solitary wave is generated by using the wave paddle and the generated solitary wave profiles are compared with the available results with a good agreement. Free falling of torpedo over the water in tank was simulated by using 3D-ISPH method.

Keywords : Circular cylinder; ISPH; Free surface flow; Scott Russell; Torpedo; Wave paddle.

1 Introduction

The water entry of a body is an interesting topic in the naval hydrodynamics and the interaction of solitary wave and marine structures are considering a fundamental problem in ocean engineering. Numerous experimental, theoretical and numerical studies have been performed to study the water entry problems. Greenhow and Lin [1] conducted a series of experiments to show the considerable differences in the free surface deformation for the entry and exit of a circular cylinder. Zhao et al. [2] used both the experiment and the potential flow theory to investigate the water entry of a falling wedge. Kleefsman et al. [3] and Panahi et al. [4] computed the water entry of a cylinder by solving the NS equation with a volume-of-fluid surface tracking using a finite volume formulation. Lin [5] used the concept of a locally relative stationary in his Reynolds-averaged NS (RANS) modeling to study the water entry of a circular cylinder with prescribed falling velocity. Most of the previous techniques are capturing the free-surface on grid system. However, there is a different approach without grid system, the so-called particle methods, which provides a robust numerical tool to simulate the complicated interactions between the flow and a solid body. Owing to the mesh-free nature, the breakup and reconnection of the free surfaces can be easily realized in a particle method without the sophisticated mesh management as required in a grid method.

The smoothed particle hydrodynamic (SPH) method was originally proposed by Lucy [6] and further developed by Gingold and Monaghan [7]

*Corresponding author. Tel: 001003122251, e-mail: abdelreheam.abdallah@sci.svu.edu.eg
†Department of Mathematics, Faculty of Science, South Valley University, Qena, Egypt.

for treating astrophysical problems. The basic idea of SPH is to use the collective motions of large number of particles to represent a flow in a Lagrangian way rather than Eulerian way. In a particle approach, the governing equations are discretized and solved with respect to the individual particles filled within the computational domain. 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 modeling flows with a complex geometry or flows where large deformations or the appearance of a free surface occurs. Oger et al. [8] employed the 2D SPH model with a fluidsolid coupling technique to study the water entry of a wedge with different degrees of freedom. The numerical model used a highly robust spatially varying particle resolution to improve the computational accuracy and efficiency. Recently, Liu et al. [9] implemented the two phase SPH model to simulate water entry of a wedge. A two-dimensional SPH model is implemented to study the water entry problem of a wedge entering the free surface as Kai et al. [10].

The SPH is originally developed in compressible flow, and then some special treatment is required to satisfy the incompressible condition. A divergence-free condition in projection based incompressible SPH was initially proposed by Cummins and Rudman [11]. Shao and Lo [12] introduced an incompressible version of the SPH. In incompressible SPH method, the pressure is implicitly calculated by solving a discretized pressure Poisson equation at every time step. Recently, an incompressible SPH model had been widely used to simulate free surface flows for incompressible fluids [13, 14, 15, 16, 17, 18, 19, 20]. Lee et al. [14] presented comparisons of a semi-implicit and truly incompressible SPH (ISPH) algorithm with the classical WCSPH method, showing how some of the problems encountered in WCSPH have been resolved by using ISPH to simulate incompressible flows. Khayyer et al. [15, 16] proposed a corrected incompressible SPH method (CISPH) derived based on a vibrational approach to ensure the angular momentum conservation of ISPH formulations. In the incompressible SPH approach, some progress has been made through correcting the kernel function or employing a higher order PPE source term, e.g., Khayyer et al. [16]. A stabilized incompressible SPH method by relaxing the density invariance condition is proposed by Asai et al. [18]. In addition, Aly et al. [19, 20] applied the stabilized version of ISPH method to simulate both of fluid-fluid interactions and fluid-structure interactions. More recently, Liu et al. [25] developed an incompressible smoothed particle hydrodynamics (ISPH) model for simulation of fluid-structure coupling problems, especially for moving structures. In their model, the mirror particle method is employed for a moving boundary. The surface force integration and force-motion algorithms are presented to solve the translation and rotation of structure body. In the most recent work of Koh et al. [26], 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 this study, ISPH method is introduced to simulate wave generation by using both of the Scott Russell wave generator and wave paddle generator. The generated solitary wave by using the wave paddle is compared with the available results corresponding to Maiti and Sen [24]. Free falling of torpedo over water in tank is simulated using the current ISPH method, in which both of the torpedo and fluid are modeled by using ISPH method.

2 MATHEMATICAL ANALYSIS

2.1 Governing equations

The mass and momentum equations of the flows are presented as:

$$\frac{D\rho}{Dt} + \rho\nabla\cdot\mathbf{u} = 0, \tag{2.1}$$

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla P + \nu_0\nabla^2\mathbf{u} + g, \tag{2.2}$$

where, ρ and ν_0 are density and kinematic viscosity of the fluid, $\mathbf{u}$ and P are the velocity vector and pressure of fluid respectively, t indicates time.

In the most general incompressible flow approach, the density is assumed by a constant value with its initial value ρ^0. The main concept in an incompressible SPH method is solving a discretized pressure Poisson equation at every time step to get the pressure value. In this paper, we used the following equation:

$$< \nabla^2 P_i^{n+1} >= \frac{\rho^*}{\Delta t} < \nabla \cdot \mathbf{u}_i^* > + \alpha \frac{\rho^0 - < \rho_i^n >}{(\Delta t)^2} \tag{2.3}$$

where, α is relaxation coefficient, $\mathbf{u}^*$ is temporal velocity, triangle bracket $<>$ means SPH approximation and subscript i means particle index.

2.2 SPH formulation

The fundamental basis of the SPH method is the interpolation theory. The method allows any function to be expressed in terms of its values at a set of disordered points representing particle positions using kernel function. A physical scalar function $A(r)$ at a certain position r can be represented by the following integral form:

$$A(r) = \int_V A(r') W(r - r', h) \, dr', \tag{2.4}$$

where, V represents the solution space and the smoothing length h represents the effective width of the kernel. The properties of the kernel function should satisfy the following two conditions for mass and energy conservation:

$$\int_V W(r - r', h) \, dr' = 1, \tag{2.5}$$

$$\lim_{h \to 0} W(r - r', h) = \delta(r - r'), \tag{2.6}$$

For SPH numerical analysis, the integral Eq. 2.4 is approximated by a summation of contributions from neighbor particles in the support domain as:

$$A(r_i) \approx < A_i >= \sum_j \frac{m_j}{\rho_j} W(r_{ij}, h) A_j(r_{ij}), \tag{2.7}$$

where, the subscripts i and j indicate positions of labeled particle, and m_j means representative mass related to particle j. The density $< \rho_i^n >$ in SPH form is defined by:

$$\rho(x_i) \approx < \rho_i >= \sum_j m_j W(r_{ij}, h), \tag{2.8}$$

The gradient of the scalar function can be assumed by using the above defined SPH approximation as follows:

$$\nabla A(r_i) \approx < \nabla A_i >= \frac{1}{\rho_i} \sum_j m_j (A_j - A_i) \nabla W(r_{ij}, h), \tag{2.9}$$

Also, the other expression for the gradient can be represented by:

$$\nabla A(r_i) \approx < \nabla A_i >= \rho_i \sum_j m_j \left(\frac{A_j}{\rho_j^2} + \frac{A_i}{\rho_i^2} \right) \nabla W(r_{ij}, h), \tag{2.10}$$

In this paper, quintic spline function is utilized as a kernel function.

$$W(R, h) = \alpha_d \begin{cases} (3-R)^5 - 6(2-R)^5 + 15(1-R)^5 & 0 \preceq R \prec 1 \\ (3-R)^5 - 6(2-R)^5 & 1 \preceq R \prec 2 \\ (3-R)^5 & 2 \preceq R \prec 3 \\ 0 & R \succeq 3 \end{cases} \tag{2.11}$$

where, α_d is $120/h$, $7/478\pi h^2$ and $3/359\pi h^3$, in one, two and three dimensions, respectively. Here, from our checked results, the smoothing kernel function h is chosen around $(1.2 \sim 1.3)d_o$, where d_o is the initial particle distance. In the current incompressible SPH method, the gradient of pressure and the divergence of velocity are approximated as follow:

$$\nabla P(r_i) \approx < \nabla P_i >= \rho_i \sum_j m_j \left(\frac{P_j}{\rho_j^2} + \frac{P_i}{\rho_i^2} \right) \nabla W(r_{ij}, h), \tag{2.12}$$

$$\nabla \cdot \mathbf{u}(r_i) \approx < \nabla \cdot \mathbf{u}_i > = \frac{1}{\rho_i} \sum_j m_j (\mathbf{u}_j - \mathbf{u}_i) \cdot \nabla W(r_{ij}, h), \tag{2.13}$$

Although the Laplacian could be derived directly from the original SPH approximation of a function in Eq. 2.13, this approach may lead to a loss of resolution. Then, second order approximation for the Laplacian terms in this research is utilized as.

$$\nabla \cdot (\nu \nabla \cdot \mathbf{u})(r_i) \approx < \nabla \cdot (\nu \nabla \cdot \mathbf{u}_i) >= \sum_j m_j \Big(\frac{\rho_i \nu_i + \rho_j \nu_j}{\rho_i \rho_j} \frac{\mathbf{r}_{ij} \cdot \nabla W(|r_i - r_j|, h)}{r_{ij}^2 + \eta^2} \Big) \mathbf{u}_{ij}, \tag{2.14}$$

where η is a parameter to avoid a zero dominator, and its value is usually given by $\eta^2 = 0.0001h^2$. For the case of $\nu_i = \nu_j$ and $\rho_i = \rho_j$, the Laplacian term is simplified as:

$$< \nabla \cdot (\nu \nabla \cdot \mathbf{u}_i) >= \frac{2\nu_i}{\rho_i} \sum_j m_j \Big(\frac{\mathbf{r}_{ij} \cdot \nabla W(|r_i - r_j|, h)}{r_{ij}^2 + \eta^2} \Big) \mathbf{u}_{ij}, \tag{2.15}$$

Similarly, the Laplacian of pressure in pressure Poisson equation (PPE) is given by:

$$\nabla^2 P(r_i) \approx < \nabla^2 P_i) >= \frac{2}{\rho_i} \sum_j m_j \Big(\frac{P_{ij} \mathbf{r}_{ij} \cdot \nabla W(|r_i - r_j|, h)}{r_{ij}^2 + \eta^2} \Big). \tag{2.16}$$

The PPE after SPH interpolation is solved by a preconditioned (diagonal scaling) Conjugate Gradient (PCG) method with a convergence tolerance $(= 1.010 - 9)$.

2.3 Treatment of moving rigid body

Koshizuka et al. [21] 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 as follows: Assuming that, the number of solid particles is n with location $\mathbf{r}_k$ for each particle, the center of the solid object at $\mathbf{r}_c$, the relative coordinate of a solid particle to the center $\mathbf{q}_k$ and the moment of inertia I of the solid object are calculated by:

$$\mathbf{r}_c = \frac{1}{n} \sum_{k=1}^{n} \mathbf{r}_k, \tag{2.17}$$

$$\mathbf{q}_k = \mathbf{r}_k - \mathbf{r}_c, \tag{2.18}$$

$$I = \sum_{k=1}^{n} |\mathbf{q}_k|^2, \tag{2.19}$$

The translational velocity $\mathbf{T}$ and rotational velocity $\mathbf{R}$ of solid object are calculated by:

$$\mathbf{T} = \frac{1}{n} \sum_{k=1}^{n} \mathbf{u}_k, \tag{2.20}$$

$$\mathbf{R} = \frac{1}{I} \sum_{k=1}^{n} \mathbf{u}_k \times \mathbf{q}_k, \tag{2.21}$$

Finally, the velocity of each particle in the solid body is replaced by:

$$\mathbf{u}_k = \mathbf{T} + \mathbf{q}_k \times \mathbf{R}, \tag{2.22}$$

From the above rigid body corrections, the motion of free moving object can be tracked as a complete rigid body. Gotoh and Sakai [22] showed that the previous treatment works very well in a stable computation where the Courant condition is satisfied. In addition, Shao [23] investigated the water entry of a free falling wedge using an incompressible smoothed particles hydrodynamics (Incom-SPH).

2.4 Treatment of boundary condition

The boundary condition on the rigid bodies has an important role to prevent penetration and to reduce error related to truncation of the kernel function. In the current research, we implemented dummy boundary particles technique to prevent penetration and reduce the error related to the truncated kernel. The dummy particles are regularly distributed at the initial state with zero velocity.

3 RESULTS AND DISCUSSION

In this section, we simulated the solitary wave generation using the vertical wave paddle. The solitary wave profiles are compared with the available results and it showed a good agreement. In the second simulation, the Scott Russell water generator was proposed initially to create a solitary wave by dropping a weighted box vertically at the one end of a long rectangular box. In addition, water entry of circular cylinder in three dimensions has been studied numerically using ISPH method.

3.1 Wave paddle generator

In the first model based on the Maiti and Sen [24] experiment, the length and flat length of the wave tank, water depth and slope of the right side of the tank were considered to be $10m$, $9.7m$, $0.3m$ and 45 deg, respectively. The motion of the wave paddle are considered for several height ratios $H/d = 0.1, 0.2, 0.3, 0.4, 0.5, 0.6$ and 0.7. Fig. 1 show the solitary wave profiles for wave height $H/d = 0.1$ to 0.7. It is clear that, the solitary wave velocity and profile are increase as the wave height ratio increases. The snapshots of the pressure distribution for solitary wave profiles at height H/d=0.1, 0.4 and 0.7, respectively have been shown in figure 2. In this figure, the snapshots show clearly the wave profiles with smoothness pressure distributions. In addition, the comparison of the solitary wave between the current ISPH results and Maiti and Sen [24] for the height $H/d = 0.6$ and 0.7, respectively is introduced in figure 3 and 4. The comparison shows a good agreement for the two cases of solitary wave profiles at height ratios $H/d = 0.6$ and 0.7.

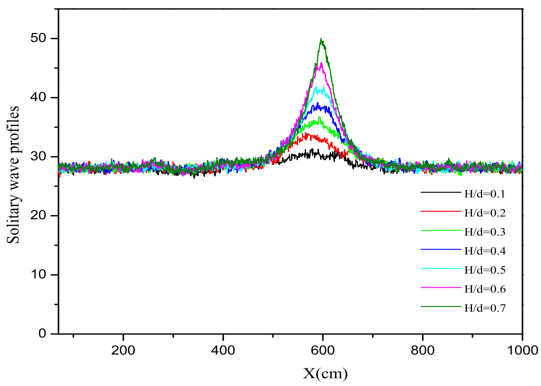

Figure 1: Solitary wave profiles for wave height $H/d = 0.1$ to 0.7. The propagation times are: $H/d = 0.1, 5.68s$; $0.2, 4.80s$; $0.3, 4.34s$; $0.4, 4.04s$; $0.5, 3.82s$; $0.6, 3.62s$; $0.7, 3.47s$, respectively.

3.2 Scott Russell wave generator

In the second simulation, the Scott Russell wave generator was proposed initially to create a solitary wave by dropping a weighted box vertically at the one end of a long rectangular box. Fig. 5 shows the Scott Russell wave generator within several rigid bodies motion over the water waves. Here, Scott Russell wave generator

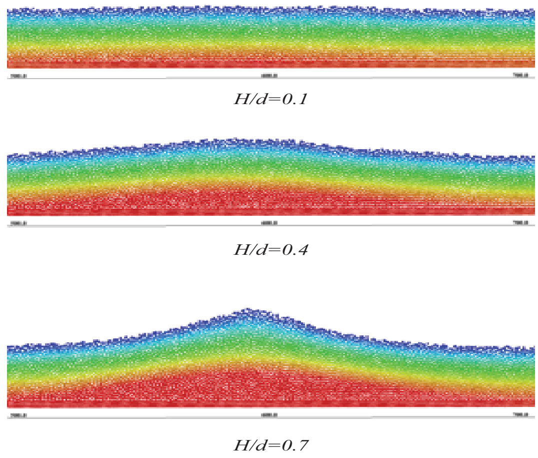

Figure 2: Snapshots of the pressure distribution for solitary wave profiles at height $H/d = 0.1, 0.4$ and 0.7, respectively.

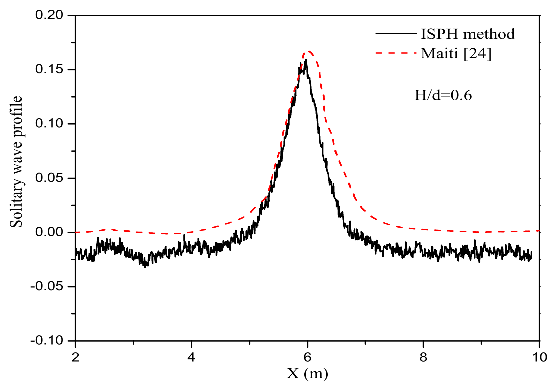

Figure 3: Comparison of solitary wave between the current ISPH results and Maiti and Sen [24] for the height $H/d = 0.6$.

formed a solitary wave and a reverse plunging wave which quickly collapse down with producing a cavity as shown in figure 5. Here, the heavy box, rigid bodies and the fluid are modeled by using ISPH method and the pressure Poisson equation is solved for all solid, fluid and dummy boundary particles. The rigid body motion over the solitary wave is introduced with density ratio between the rigid body and the fluid around 0.5 and with this low density the rigid body is still floating during the whole simulation over the generated solitary wave. Fig. 6 shows the time histories of the pressure distribution for the Scott Russell wave generator. It is observed that, as the heavy box released over the water in tank, the reverse plunging wave is formed directly and also it quickly collapses down with producing a cavity. Also, in

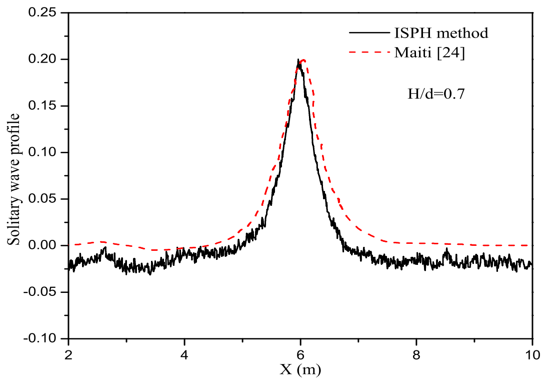

Figure 4: Comparison of solitary wave between the current ISPH results and Maiti and Sen [24] for the height 0.7.

these snapshots, the formation of solitary wave and its movements have been shown clearly with pressure distribution.

Finally, free falling of the circular cylinder over

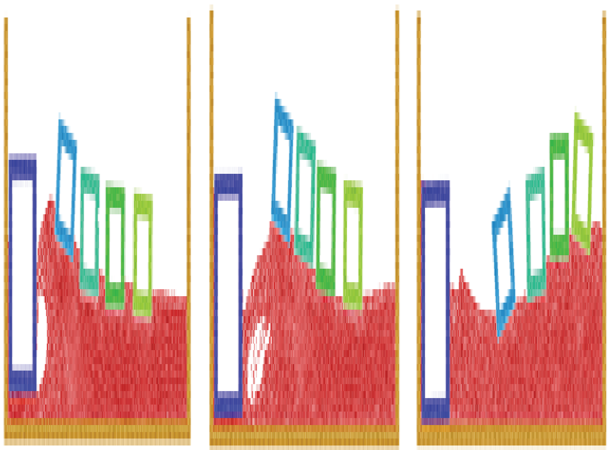

Figure 5: Scott Russell wave generator within several rigid bodies motion over the water waves.

water in tank has been introduced in three dimensions using ISPH method.

3.3 Free falling 3D circular cylinder

Here, we predicted numerically the free falling of torpedo over wave in tank using 3D-ISPH method. The torpedo is taken as a rigid body for simplicity and it modeled by 3D-ISPH method. The snapshots for water entry of freely falling 3D circular cylindrical body over water in tank has been shown in figure 7. In this figure, after the circular cylinder impacts the water, high pressure was generated at the head region of the body and

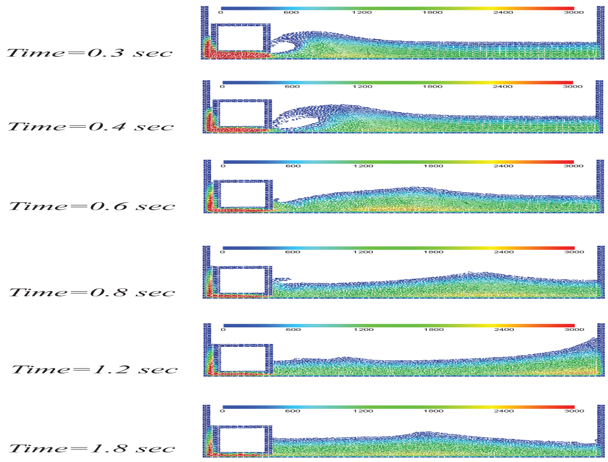

Figure 6: shows the time histories of the pressure distribution for the Scott Russell wave generator.

splashes with highly nonlinear free surface profile were observed.

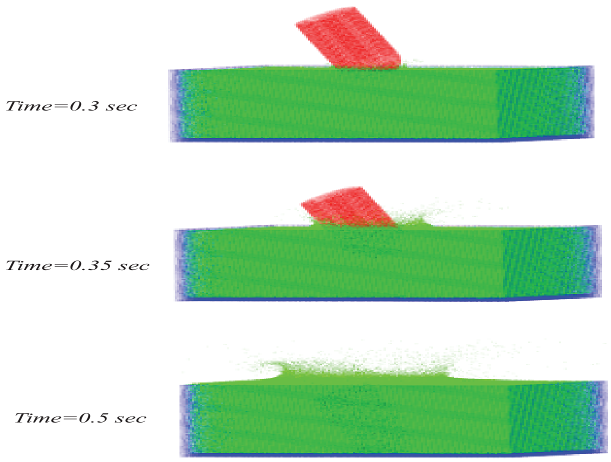

Figure 7: Time histories for water entry of freely falling 3D circular cylindrical body over water in tank.

4 Conclusion

Numerical simulations for three models of nonlinear free surface flows, which are wave paddle generator, Scott Russells wave generator and water entry model of free falling 3D circular cylindrical body, were conducted by using Incompressible Smoothed Particle Hydrodynamics(ISPH) method. The generated solitary wave by using the wave paddle is compared with the available results with a good agreement. Scott Russell water generator was proposed initially to create a solitary wave by dropping a weighted box vertically at the one end of a long rectangular box.

Free falling of torpedo over water in tank is simulated using the current ISPH method, in which both of the torpedo and fluid are modeled by using ISPH method. The current ISPH method can model the complicated free surface flow with low computational cost and reasonable accuracy.

References

[1] M. Greenhow, W. Lin, *Non-linear free surface effects: experiments and theory*, Report Number 83-19, Department of Ocean Engineering, Massachusetts Institute of Technology, (1983)

[2] R. Zhao, O. Faltinsen, J. Aarsnes, *Water entry of arbitrary two-dimensional sections with and without flowseparation*, Twenty-first Symposium on Naval Hydrodynamics, Trondheim, Norway (1997) 408423.

[3] K. M. T. Kleefsman, G. Fekken, A. E. P. Veldman, B. Iwanowski, B. Buchner, *A volume-of-fluid based simulation method for wave impact problems*, Journal of Computational Physics 206 (2005) 363393.

[4] R. Panahi, E. Jahanbakhsh, M. S. Seif, *Development of a VoF-fractional step solver for floating body motion simulation*, Applied Ocean Research 28 (2006) 171181.

[5] P. Z. Lin, *A fixed-grid model for simulation of a moving body in free surface flows*, Computers and Fluids 36 (2007) 549561.

[6] L. B. Lucy, *A numerical approach to the testing of the fusion process*, Astron J. 88 (1977) 101324.

[7] R. A. Gingold and J. J. Monaghan, *Smoothed particle hydrodynamics: theory and application to non-spherical stars*, Mon Not R Astron Soc. 181 (1977) 37589.

[8] G. Oger, M. Doring, B. Alessandrini, P. Ferrant, *Two-dimensional SPH simulations of wedge water entries*, Journal of Computational Physics 213 (2006) 803822.

[9] H. Liu, K. Gong, B. L. Wang, *Modeling water entry of a wedge by multiphase SPH method*, SPHERIC Newsletter, No. 9, 2009.

[10] G. Kai, L. Hua and W. Ben-long, *Water entry of a wedge based on SPH model with an improved boundary treatment*, Journal of Hydrodynamics 21 (2009) 750-757.

[11] S . J. Cummins and M. Rudman, *An SPH projection method*, Journal Computational Physics 152 (1999) 584607.

[12] S. D. Shao, E. Y. M. Lo, *Incompressible SPH method for simulating Newtonian and non-Newtonian flows with a free surface*, Advances in Water Resources 26 (2003) 787800.

[13] M. Ellero, M. Serrano, P. Espanol, *Incompressible smoothed particle hydrodynamics*, Journal of Computational Physics 226 (2007) 17311752.

[14] E. S. Lee, C. Moulinec, R. Xu, D. Violeau, D. Laurence and P. Stansby, *Comparisons of weakly compressible and truly incompressible algorithms for the SPH mesh free particle method*, Journal of Computational Physics 227 (2008) 84178436.

[15] A. Khayyer, H. Gotoh and S. Shao, *Corrected incompressible SPH method for accurate water-surface tracking in breaking waves*, Coastal Engineering 55 (2008) 236250.

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

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

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

[19] A. M. Aly, M. Asai, Y. Sonoda, *Modeling of surface tension force for free surface flows in ISPH method*, International Journal of Numerical Methods for Heat and Fluid Flow, 2013.

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

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

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

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

[24] S. Maiti and D. Sen, *Computation of solitary waves during propagation and run up on a slope*, Ocean Engineering 26 (1999) 1063-1083.

[25] X. Liu, P. Lina and S. Shao, *An ISPH simulation of coupled structure interaction with free surface flows*, Journal of Fluids and Structures 48 (2014) 46-61.

[26] C. G. Koh, M. Luo, M. Gao, and W. Bai, *Modeling of liquid sloshing with constrained floating baffle*, Comput. Struct. 122 (2013) 270-279.

Abdelraheem Mahmoud Aly received his M. Sc. (Mathematics) from South Valley University, Egypt in 2008 and he received his Ph.D. degree from the Kyushu University, Japan in 2012. Dr. Abdelraheem employed as a postdoctoral fellow in Ulsan University, South Korea at 2013 and also he worked in Kyushu University, Japan from $(2014-2015)$. Currently, he is an assistant Professor in Department of Mathematics at South Valley University, Qena, Egypt. His research interests include Computational Fluid Dynamics, Smoothed Particle Hydrodynamics, Heat Transfer and Mathematical Modeling of Multi Phase Flow through Porous Media.