Numerical simulation of fluid–structure interaction with SPH method

Yu Yang¹, Jiaru Shao²

¹College of Mechanical Engineering, Chongqing University of Technology, Chongqing 400054, People’s Republic of China
²E-mail: shaojiaru@cqut.edu.cn

Abstract: Smoothed particle hydrodynamics (SPH) is a meshfree, Lagrangian, particle method, which combines the advantages of Euler and Lagrangian method and has special advantages in simulating violent non-linear fluid–structure interaction problems. In this study, an improved SPH method is used to simulate four fluid–structure interaction cases. Firstly, the violent flow in the mould filling process is simulated to validate that the SPH model can obtain accurate flow patterns with any complex solid boundaries. Secondly, the dam-break flow against a vertical wall is simulated, and the pressure curves are compared with the experiment. Then liquid sloshing problems under different external excitations are simulated, and the complex coupling characteristics can be captured. Finally, the interactions between fluid and floating bodies are researched. The obtained numerical results show good agreement with the results from other sources, and clearly demonstrate the effectiveness of the presented SPH method in modelling fluid–structure interaction problems.

1 Introduction

Fluid–structure interaction problems are a very important topic in daily life, mechanical design, coastal and offshore, and many other domains. Fluid–structure interaction involves the change of free surfaces and the strong coupling between fluid and solid bodies, which involve fix solid boundary, moving boundary and moving body. Typical examples include movement of the torpedo, liquid flow in the turbine, fluid–solid coupling in the pump. These phenomena include similar characteristics, such as violent free-surface flow, the evolution of the cavity and violent impact load. Therefore, researching fluid–structure interaction problems is of great significance in theory and practice.

Many scholars have conducted theoretical, experimental and numerical simulation researches in the area. Theoretical researches are usually used for linear or weakly non-linear fluid dynamics. Some experiments are generally expensive, and in many cases, a physical model cannot be scaled in a practical experimental model. During the last decades, numerical simulation methods have been used to resolve the fluid–structure interaction problems [1–3], and smoothed particle hydrodynamics (SPH) [4–6] is very popular. In SPH, the fluid and solid are represented by many different particles, and these particles can move according to control equations. Therefore, the history of fluid motion can be obtained naturally, and the material interfaces can easily be tracked.

There are some literatures addressing the application of the SPH method. Oger et al. [7] simulated water entry phenomenon of with weakly compressible SPH (WCSPH) model. Shao [8] simulated water entry of cylinder, and track the change of free surface. Marrone et al. [9] presented an SPH model to research the wave, which was generated by a ship. Xu and Liu [10] researched the slamming phenomenon of a 2D bow flare section. Gong et al. [11] presented an improved boundary treatment method, and obtained good results on the fluid–solid interface. Yang et al. [12] present a LANS turbulence model to describe the complex fluid domain. Delorme et al. [13] and Rhee and Engineer [14] tracked the liquid sloshing loads with an SPH model. Iglesias et al. [15] researched the amplitude in sloshing type problems. Anghileri et al. [16] and Meringolo et al. [17] investigated the fluid–structure interaction of water-filled tanks during the impact with the ground. Carlos et al. [18] simulated flow through a constricted channel, which validate the adaptability for a complex boundary. Alessandra et al. [19] simulated the turbulent flow in a circular pipe. These works have validated the ability of the SPH method in simulating fluid–structure interaction problems. However, there are still some problems that need to be resolved. The computational accuracy of SPH is sensitive to the particles distribution [20–23]. The boundary treatment method, which was usually used, cannot satisfy high precision, high efficiency and universal applicability [24–27].

In this paper, an SPH model is used to simulate the fluid–structure interaction problem. This model behaved well in dealing with large density difference, no penetration and complex interface.

2 SPH methodology

The governing motion can be described by the flowing control equations [28]

\[
\frac{d\rho}{dt} = - \rho \nabla \cdot \mathbf{v},
\]

\[
\frac{d\mathbf{v}}{dt} = \frac{1}{\rho} \rho_0 \nabla \rho + \frac{\mu}{\rho} \nabla \mathbf{v} + g + \frac{1}{\rho} \mathbf{V}(\rho R),
\]

where \( \rho \) is the fluid density, \( \mathbf{v} \) is the velocity vector, \( P \) is pressure, \( \mu \) is the dynamic viscosity, \( g \) is the gravitational acceleration, and \( R \) is the Reynolds stress tensor. To solve the equations of motion, an artificial compressibility technique is used, and the artificial equation of state [29] can be described as

\[
P = C_v (\rho - \rho_b),
\]

where \( \rho_b \) is the initial density of the water, \( C_v \) is the virtual sound speed.

By kernel and particle approximation, a field function and its derivatives can be described as following forms:

\[
\frac{d\rho_i}{dt} = \sum_j m_j v_{ij} \nabla_i W_{ij},
\]
\[
\frac{d\rho_i}{dt} = \sum_{j=1}^{N} \left( \frac{m_j}{\rho_j} \right) \frac{1}{\rho_i} \left( \frac{p_i + \rho_i}{\rho_i} \right) \cdot \nabla W_{ij} + \sum_{j=1}^{4m} \left( \frac{\mu_i + \mu_j}{\rho_i + \rho_j} \right) \left( \frac{\mathbf{x}_j - \mathbf{x}_i}{\Delta t} \right) \cdot \nabla W_{ij} + \sum_{j=1}^{m_b} \left( \frac{\mathbf{R}_j - \mathbf{R}_i}{\rho_i \rho_j} \right) \cdot \nabla W_{ij} + g. \tag{5}
\]

The SPH equations of motion for a rigid body can be described as

\[
\frac{da_i}{dt} = \frac{\sum m_a i_j}{M} + g, \tag{6}
\]

\[
\frac{d\mathbf{b}_i}{dt} = \frac{\sum m_i \mathbf{r}_j - \mathbf{R}_i}{I_i}, \tag{7}
\]

where \(\mathbf{a}_i\) is the velocity of the centre of mass, \(\mathbf{r}_j\) is the vector from the centre of mass to point \(j\), and \(I_i\) is the moment of inertia.

In this work, an improved fluid–structure interface treatment (IFSIT) algorithm is used. The solid boundary is constructed by three kinds of ghost particles, as shown in Fig. 1.

The pressure of the ghost particles can be obtained as follows:

\[
P_i = \sum_j p_j W_{ij} \frac{m_j}{\rho_j}, \tag{8}
\]

where \(W^d\) is the improved kernel function [30]. Considering the difference between fluid and solid, the density change rate can be corrected by the following equation:

\[
\frac{d\rho_i}{dt} = \sum_j m_j \rho_j \nabla W_{ij} \frac{\rho_j}{\rho_i}. \tag{9}
\]

Take non-slip boundary condition, for example the variables of the solid particles can be obtained by equations

\[
\rho_i = \sum_j \rho_j W_{ij} \frac{m_j}{\rho_j} = \sum_j m_j W_{ij}, \tag{10}
\]

\[
v_i = - \sum_j v_j W_{ij} \frac{m_j}{\rho_j}. \tag{11}
\]

For particles near the free surface, the support domains are usually cut off. To ensure the free surface, a coefficient \(k_i\) is given

\[
k_i = \sum_j W_{ij} \frac{m_j}{\rho_j}. \tag{12}
\]

If the value in particle \(i\) is <0.95, the particle is treated as the free surface particle, and the pressure will be equal to atmospheric pressure.

3 Numerical examples

3.1 Violent flow within mould filling model

In this section, the SPH model is implemented to simulate violent flow within a fixed boundary. Firstly, a mould filling model for a disc tube is constructed. Fig. 2 shows the geometry of the model, which is similar to what Schmid and Klein provided [31]. The diameter of the outer ring is 135 mm, the diameter of the inner ring is 45 mm, the width of the entrance is 45 mm, and the distance between the centre and entrance is 115 mm. The density of the fluid is \(\rho = 1000 \text{ kg/m}^3\), the kinematic viscosity is 0.01 \text{ m}^2 \text{ s}^{-1}, the inlet velocity of the fluid is 18 m/s. In the SPH simulation, the time step is taken as \(10^{-7}\) s; the particle spacing is 0.00075 m. The number of boundary particles is about 3000.

Fig. 3 shows the results of the mould filling pattern numerically predicted by the SPH_DTKGC method [32], the SPH model of this paper, and the experimental results of Schmid and Klein [31] at different times. The fluid first impacts the inner ring, and flows toward two different directions under the block of the inner ring. After impacting the outer ring, the fluid flows along the outer wall and envelops the entire outer ring gradually. Both sides of the fluid meet at the top of the outer ring and move toward the inner ring. Finally, the fluid fills the entire cavity. All of the results in Fig. 3 had shown similar flow characteristics. However, observing the results, it can be found that the SPH model of this paper can obtain more accurate results. Take Fig. 3c as example, in the lower half of the outer ring, the fluid has enveloped the entire outer ring, which can also be observed in the experimental results. Fig. 4 shows the velocity vectors and the particle distribution at 9.5 ms. Near the solid boundary, the distribution of the fluid particles is very regular, while in many traditional SPH methods, there is often a large gap between the fluid and boundary, which mean a stronger numerical oscillation.

3.2 SPH modelling of dam-break flow

In this section, the SPH model is implemented to simulate dam-break flow against a vertical wall. The geometry of this example is shown in Fig. 4, and the data is similar to what Buchner provided [33]. To quantitatively validate the pressure load on the solid boundary, two probe points \(P_1\) and \(P_2\) are set, as shown in Fig. 5, and \(OP_1 = 0.16\) m, \(OP_2 = 0.584\) m. In the simulation, the time step is taken as \(5 \times 10^{-6}\) s, the particle spacing is 0.0025 m. The number of boundary particles is about 110,000.

Fig. 5 shows the flow history of the dam break problem at 0.37, 0.75, 1.41 and 1.61 s. It can be observed that with the development of the dam-breaking process, water front impacts against the front vertical wall, generates a bounce-back flow pattern after a short period of interaction with the vertical wall, and finally forms a cavity in the right-bottom corner area. Fig. 6 shows the enlarged view of the pressure field in the lower right corner. It can be found that the pressure field is very smoothed, and near the solid wall, no obvious numerical oscillation can be found. The particle distribution is still regular, and there is no particle aggregation phenomenon.
Fig. 7 shows an enlarged view of horizontal velocity $u$ of the fluid tongue just before its impact against the wall. This tongue has a triangular shape and the angle is about $10^\circ$. When the waterfront impacts the vertical wall, the maximum velocity is about 4.7 m/s.

3.3 SPH modelling of sloshing in a rotating liquid tank

In this section, the liquid sloshing in a rotating tank is investigated under different circular frequencies. Fig. 8 shows the geometry of the liquid sloshing system, which is similar to what Iglesias provided [15], i.e. $L = 0.64$ m, $H = 1.15$ m; the water depth is $h_w = 0.03$ m, and the centre of rotation is 0.1 m below the baseline.

The external excitation can be described as $\theta = \theta_0 \sin(\omega t)$, where $\theta_0$ is the angular displacement, $\omega$ is the circular frequency of the rotating motion. In the left wall, probe points are employed, and in this paper, the pressure curve in probe point $P_i$ is predicted ($OP_i = 0.03$ m). The fluid is water. To investigate the free surface evolution and the pressure load on the tank under different circular
Fig. 5 Snapshots of the dam-break flow against a vertical wall
(a) $T = 0.37$ s, (b) $T = 0.75$ s, (c) $T = 1.41$ s, (d) $T = 1.61$ s

Fig. 6 Enlarged view of the pressure field in the lower right corner
Left: $T = 1.14$ s; Right: $T = 1.61$ s

Fig. 7 Enlarged view of the horizontal velocity of the fluid tongue at $t = 0.585$ s

Fig. 8 Illustration of liquid sloshing in a rotating tank
frequencies, the sloshing models are set as $\theta_0 = 6^\circ$, $\omega = 1.0$ and 4.34 rad/s, respectively. The particle spacing is 0.001 m, and about 40,000 particles are used in the simulation.

When $\omega = 1.0$ rad/s, the period of the external excitation is 6.28 s. Fig. 9 shows the corresponding liquid sloshing mode and free surface evolution of the liquid sloshing system in a period. In this case, the sloshing is not very violent, and the free surface does not break up. Though there are non-linear phenomena, the linear wave theory still applies. In the whole process of the sloshing, the water waves are traveling waves. Fig. 10 is the pressure history at $P_i$. It shows that the water impact the solid wall periodically.

Increasing the value of $\omega$ to 4.34 rad/s, the moving condition in Fig. 11 can be obtained. The sloshing is still violent and the water wave has two main wave peaks. The front one is higher than the later one. These two surge fronts with two water wave peaks then impact the solid wall (left and right walls) twice, as shown in Fig. 12. The first impact is more violent with an instantaneous pressure peak. One notable observation is that in this case, the strongest wave impact can happen on the left-top and right-top corners.

### 3.4 Interaction between fluid and floating bodies

In this section, the SPH model is implemented to simulate the interaction between fluid and moving bodies. A simplified model of flood impact on the floating body is constructed. Fig. 13 shows the geometry of the interaction model. The length of the container is 4 m, the height of the left part water is 1.2 m, and the depth of the right part water 0.3 m. Three square boxes (0.2m $\times$ 0.2 m) float on the water, and the distances from the centre of the boxes to the right wall are 1.85, 1.35, and 0.85 m separately. The density of the water is $1000 \text{ kg/m}^3$, and the density of the box is $500 \text{ kg/m}^3$. The particle spacing is 0.005 m, and about 98,000 particles are used in the simulation.

Fig. 14 shows the flow patterns and pressure field distributions at different time. At the beginning of the simulation, the boxes float on the water. Then under the force from dam break, the boxes began to move toward the right wall. Driven by the waves, box B even jumped out of the water. With the increasing of the speed of boxes A and B, they surpassed box c, and hit the right solid wall finally. Under the interaction between water and structures, the reflection of pressure wave can be found obviously. However, the pressure fields are still smoothed relatively. Fig. 15 shows the particles distribution near the floating box C at $t = 0.7$ and 0.9 s. Near the interface, the distribution of the fluid particles is still very regular, and no particle aggregation phenomenon can be found.
This means that the SPH model in this paper is robust in dealing with the interaction between fluid and moving structures.

4 Conclusion

In this paper, an SPH model is used to simulate the fluid–structure interaction problem. An IFSIT algorithm is used to simulate the interaction between fluid and solid bodies. The computation of the fluid–structure interface does not need second particle research, and it will not increase large computation cost. This SPH model can also deal with large density difference problem, guarantee no penetration near the interface, and suitable for the boundary with any shape. Four cases are simulated to validate the effectiveness of this algorithm in dealing with fixed boundary, moving boundary and moving rigid body. In the mould filling model, the accurate flow patterns of violent flow are predicted, which agree well with experiment and other SPH results. In the simulation of dam-break flow, through a comparison with the experiment, it is validated that the SPH can predict pressure between fluid and structures accurately. The model performs exceptionally well when simulating moving boundary problems. It can predict the complex phenomenon in the liquid sloshing. This model can also resolve the fluid–structure interface; even this is a large density difference.
Acknowledgments
This work was supported by the National Natural Science Foundation of China (11602045), Technological Innovation and Application Development Project of Chongqing (cstc2019jscx-zdztzxX0028, cstc2019jscx-zdztzxX0020).

References
[1] Tien, C.M.T., Cong, D.N., Mai, D.N., et al.: ‘High-order fluid solver based on a combined compact integrated RBF approximation and its fluid structure interaction applications’, Comput. Fluids, 2016, 131, (5), pp. 151–168
[2] Panagiotou, P.A., Ioannis, A., Neophytos, L., et al.: ‘FEM approach for diagnosis of induction machines’ non-adjacent broken rotor bars by short-time Fourier transform spectrogram’, J. Eng., 2019, 17, pp. 4556–4570
[3] Peng, W., Wei, Q.: ‘Solving 2-D water entry problems with a CIP method and a parallel computing algorithm’. Mar. Syst. Ocean Technol., 2016, 11, (1-2), pp. 1–9
[4] Gingold, R.A., Monaghan, J.J.: ‘Smoothed particle hydrodynamics-theory and application to non-spherical stars’, Mon. Not. R. Astron. Soc., 1997, 181, pp. 375–389
[5] Lucy, L.B.: ‘A numerical approach to the testing of the fission hypothesis’, Astron. J., 1997, 82, pp. 1013–1024
[6] Liu, G.R., Liu, M.B.: ‘Smoothed particle hydrodynamics: a meshfree particle method’ (World Scientific, Singapore, 2003)

Fig. 14 Pressure field at different time
(a) $T = 0.1$ s, (b) $T = 0.3$ s, (c) $T = 0.5$ s, (d) $T = 0.7$ s, (e) $T = 0.9$ s, (f) $T = 1.1$ s

Fig. 15 Particles distribution near the floating box C at $T = 0.7$ and $0.9$ s
[7] Oger, G., Doring, M., Alessandrini, B., et al.: ‘Two-dimensional SPH simulations of wedge water entries’, *J. Comput. Phys.*, 2006, **213**(2), pp. 803–822

[8] Shao, S.D.: ‘Incompressible SPH simulation of water entry of a free-falling object’. *Int. J. Numer. Methods Fluids*, 2009, **59**(1), pp. 91–115

[9] Marrone, S., Colagrossi, A., Antuono, M., et al.: ‘A 2d δ SPH model to study the breaking wave pattern generated by fast ships’, *J. Fluid Struct.*, 2011, **27**(8), pp. 1199–1215

[10] Xu, F., Liu, H.: ‘Numerical simulation of 2-D bow flare slamming using SPH method’, *Chin. J. Hydrodyn.*, 2013, **28**(5), pp. 585–590

[11] Gong, K., Shao, S.D., Liu, H., et al.: ‘Two-phase SPH simulation of fluid-structure interactions’, *J. Fluid Struct.*, 2016, **65**, pp. 155–179

[12] Yang, X.F., Peng, S.L., Liu, M.B.: ‘Simulation of water entry with smoothed particle hydrodynamics method’, *Chin. J. Comput. Phys.*, 2011, **28**(4), pp. 523–528

[13] Delorme, L., Iglesias, A.S., Perez, S.A.: ‘Sloshing loads simulation in LNG tankers with SPH’. Int. Conf. on Computational Methods in Marine Engineering, Barcelona, 2005

[14] Rhee, S.H., Engineer, L.: ‘Unstructured grid based Reynolds-averaged Navier-Stokes method for liquid tank sloshing’, *J. Fluids Eng.*, 2005, **127**(4), pp. 572–582

[15] Iglesias, A.S., Delorme, L., Perez-Rojas, L., et al.: ‘Liquid moment amplitude assessment in sloshing type problems with smooth particle hydrodynamics’, *Ocean Eng.*, 2006, **33**(11-12), pp. 1462–1484

[16] Anghileri, M., Castelletti, L.M.L., Tirelli, M.: ‘Fluid structure interaction of water filled tanks during the impact with the ground’, *Int. J. Impact Eng.*, 2005, **31**(3), pp. 235–254

[17] Meringolo, D.D., Colagrossi, A., Marrone, S., et al.: ‘On the filtering of acoustic components in weakly-compressible SPH simulations’, *J. Fluid Struct.*, 2017, **70**, pp. 1–23

[18] Carlos, E., Alvarado-Rodriguez Jaime, K., Leonardo, D.G., et al.: ‘Nonreflecting outlet boundary conditions for incompressible flows using SPH’, *Comput. Fluids*, 2017, **133**, pp. 177–188

[19] Alessandra, M., Massimiliano, M., Enrico, N.: ‘Inflow/outflow pressure boundary conditions for smoothed particle hydrodynamics simulations of incompressible flows’, *Comput. Fluids*, 2017, **159**, pp. 9–22

[20] Monaghan, J.J.: ‘Smoothed particle hydrodynamics’, *Rep. Prog. Phys.*, 2005, **68**(8), pp. 1703–1759

[21] Colagrossi, A., Landrini, M.: ‘Numerical simulation of interfacial flows by smoothed particle hydrodynamics’, *J. Comput. Phys.*, 2003, **191**(2), pp. 448–475

[22] Liu, M.B., Shao, J.R., Li, H.Q.: ‘An SPH model for free surface flows with moving rigid objects’, *Int. J. Numer. Methods Fluids*, 2014, **74**, pp. 684–697

[23] Liu, M.B., Liu, G.R.: ‘Smoothed particle hydrodynamics (SPH): an overview and recent developments’, *Archives Comput. Methods Eng.*, 2010, **17**(1), pp. 25–76

[24] Shao, J.R., Li, S.M., Liu, M.B.: ‘Numerical simulation of violent impinging jet flows with improved SPH method’, *Int. J. Comput. Methods*, 2016, **13**(p. 1641001)

[25] Gomez-Gesteira, M., Dalrymple, R.A.: ‘Using a three-dimensional smooth particle hydrodynamics method for wave impact on a tall structure’, *J. Waterway Port Coastal Ocean Eng.*, 2008, **134**(2), pp. 63–69

[26] Marrone, S., Colagrossi, A., Touze, C.G.L., et al.: ‘δ-SPH model for simulating violent impact flows’, *Comput. Methods Appl. Mech. Eng.*, 2011, **200**(3), pp. 1526–1542

[27] Shao, J.R., Li, H.Q., Liu, G.R., et al.: ‘An improved SPH method for modeling liquid sloshing dynamics’, *Comput. Struct.*, 2012, **100-101**, pp. 18–26

[28] Monaghan, J.J., Mériaux, C.A.: ‘An SPH study of driven turbulence near a free surface in a tank under gravity’, *Eur. J. Mech. B. Fluids*, 2018, **68**, pp. 201–210

[29] Morris, J.P.: ‘Analysis of smoothed particle hydrodynamics with applications’. PhD thesis, Monash University, 1996

[30] Bonet, J., Kulasegaram, S.: ‘Correction and stabilization of smooth particle hydrodynamics methods with applications in metal forming simulations’, *Int. J. Numer. Methods Eng.*, 2000, **47**(6), pp. 1189–1214

[31] Schmid, M., Klein, F.: ‘Fluid flow in die cavities - experimental and numerical simulation’. Int. Die Casting Congress and Exposition (NADCA 18), Indianapolis, IN, USA, 1995, pp. 93–99

[32] Ren, J.L., Ouyang, J., Jiang, T., et al.: ‘Simulation of complex filling process based on the generalized Newtonian fluid model using a corrected SPH scheme’, *Comput. Mech.*, 2011, **49**, pp. 643–665

[33] Buchner, B.: ‘Green water on ship-type offshore structures’. PhD thesis, Delft University of Technology, 2002