Smooth particle hydrodynamics simulation of dam-break impacting different obstacles

Xiewei Tian*
School of Computer Science and Technology, Xi'an University of Science and Technology, Xi'an, China
*Corresponding author e-mail: iamtianxiew@mail.xust.edu.cn

Abstract. Aiming at the problems of smoothed particle hydrodynamics (SPH) method boundary imposing difficulties and particles easily penetrating solid walls, an improved complex boundary processing method is proposed. First, the improved SPH method is applied to numerically simulate the dam-break flow problem, and the numerical results are compared with the experimental results to verify the effectiveness of the method; then, the flux is added to the continuity equation to improve the pressure; finally, Numerical simulation is carried out to study the flow state of the dam-break water under different conditions of impacting on different obstacles, and the obstacles are considered as columnar, columnar with holes, wedge-shaped, semi-circular, 1/4 lower right circle obstacles and 1/4 upper right circle obstacles. The results show that: the stable pressure field can be obtained by adding flux; the improved boundary treatment method can effectively prevent particles from penetrating the solid wall, which is suitable for any complex boundary; SPH method has significant advantages in dealing with strongly nonlinear extreme deformation flow problems such as dam-break.

1. Introduction
Computational Fluid Dynamics (CFD) is the main force in solving fluid dynamics problems. CFD mainly uses numerical methods based on meshing. Numerical methods based on meshing mainly include finite element method, finite volume method and finite difference method, etc. [1] Although this type of method has achieved great success, it is difficult for this type of numerical method to deal with the problems of free surface [2], fluid-structure interaction [3] and large fluid deformation [4].

It is precisely because the numerical method based on meshing has the above defects, the meshless method has begun to develop, among which the Smoothed Particle Hydrodynamics (SPH) method is the most widely used and developed fastest meshless Lagrangian Particle method. The SPH method was first proposed by Lucy [5], Gingold and Monaghan [6] and applied to the field of astronomy. The SPH method uses the kernel function to approximate the physical problem, discretizes the continuous matter in space into a series of movable particles (called smooth particles), and replaces the continuously distributed fluid with discrete particles. Each particle carries information about its location and various properties, such as mass, density, speed, and energy. Therefore, compared with other numerical methods, SPH method does not require mesh at all, and is suitable for simulating free surface, motion boundary and complex flow problems.
As a typical free-surface flow problem, dam-break flow has always been a classic example to verify the numerical simulation method. It contains various complex physical phenomena such as splashing, fusion and near-solid wall flow caused by collision of water current and solid wall, so it has always attracted the attention of scholars at home and abroad. In 1994, Monaghan [7] gave the symmetric discrete scheme of the N-S equation, and successfully simulated the 2D dam-break flow problem using the SPH method. In 2003, Colagrossi et al. [8] proposed a scheme to initialize the periodic density of SPH particles and simulated the 2D dam-break flow problem. In 2008, Lee et al. [9] used the incompressible and weakly compressible SPH method to numerically simulate the 2D dam break problem. In 2014, Leroy et al. [10] introduced the fixed-wall boundary conditions proposed by Ferrand et al. [11] into the incompressible SPH method, and numerically simulated the 2D dam break problem with wedge-shaped obstacles added. In 2020, Krimi et al. [12] added an adaptive numerical dissipation term to the weakly compressible SPH method, which improved the accuracy of the method and simulated the 2D dam-break flow problem.

In this paper, a numerical simulation of the 2D dam-break flow problem is carried out, and the two types of situations where no obstacles are added and a series of obstacles are added are respectively considered. Firstly, the numerical simulation of the dam-break flow problem without obstacles is carried out, and the effectiveness of the proposed SPH method is verified by comparing with the results in the literature. Secondly, in order to effectively prevent particles from penetrating the complex boundary, an improved boundary processing method is proposed. Finally, the improved SPH method is used to numerically simulate the two-dimensional dam-break flow problem with cylindrical, hole-shaped, wedge-shaped, semi-circular, and quarter-semi-circular obstacles. In this way, the dynamic potential of the dam-breaking flow in different situations is studied.

2. Basic equations

2.1. Governing equation

In the Langranian coordinate system, the governing equations of the two-dimensional isothermal, compressible Newtonian viscous fluid are:

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

\[
\frac{dv}{dt} = -\left(\frac{1}{\rho}\right) \nabla p + v \nabla^2 v + g
\]

Where \(\rho\) is the fluid density, \(v\) is the fluid velocity, \(p\) is the fluid pressure, \(\nu\) is the kinematic viscosity, and \(g\) is the acceleration due to gravity.

2.2. Equation of state

In the SPH method, the equation of state is usually used to treat incompressible fluids as slightly compressible fluids. This paper adopts the state equation established by Monaghan [6]:

\[
P(\rho) = P_0 \left(\frac{\rho}{\rho_0}\right)^\gamma - 1
\]

In the formula, \(P_0\) is the reference pressure, and \(\gamma\) can be adjusted. For fluids, \(\gamma = 7\) is usually selected. Both \(\gamma\) and \(P_0\) are used to control the oscillation amplitude of the fluid density near the normal state in the calculation (generally required to be controlled at about 1%). In the actual calculation process, it is necessary to select an appropriate value for \(P_0\), so that the approximate sound velocity \(c\) of the weakly compressible water flow is more than 10 times the maximum flow velocity in the flow.
field to ensure the incompressibility of the simulated flow field. Among them, \( P_0 = \left(100 \rho_0 v_{\text{max}}^2\right)/\gamma \cdot \rho_0 \)
is the initial density of the fluid, and the maximum velocity of \( v_{\text{max}} \).

3. SPH discrete

3.1. Basic principles

The SPH method is a method of difference integration through a kernel function. Constructing the SPH equation is divided into two parts: the kernel approximation process and the particle approximation process [13]. In the SPH method, the integral expression of any function \( f(x) \) in the problem \( \Omega \) is:

\[
\langle f(x) \rangle = \frac{1}{\Omega} \int f(x') W(x - x', h) \, dx'
\]

(4)

In the formula, \( \langle f(x) \rangle \) is the kernel approximation of \( f(x) \), \( W \) is the kernel function or smooth function, its choice directly affects the calculation error and stability, this paper adopts the B-spline kernel function; \( h \) is the smooth length of the kernel function support domain.

The integral form of equation (4) is written as the particle discretization as follows:

\[
\langle f(x_i) \rangle = \sum_{j=1}^{N} m_j f(x_j) W(x_i - x_j, h)
\]

(5)

Where, \( m_j \) represents particle mass, \( \rho_j \) represents particle density, \( W(x_i - x_j, h) \) is an integral smooth function. Equation (5) shows that the value of the field function at particle \( i \) can be obtained by the weighted average of the value of the function of all particles in the compact supported domain of the particle through the kernel function.

3.2. Discrete governing equations

In the SPH method, there are many different ways of discretizing the fluid control equation. For the continuity equation (1), the SPH discrete format selected in this paper is:

\[
\frac{d \rho_i}{dt} = \sum_j m_j v_{ij} \cdot \nabla W_{ij}
\]

(6)

Where \( v_{ij} = v_i - v_j \), \( \nabla W_{ij} \) represents the space derivative of the kernel function with respect to the coordinate of particle \( i \).

For the momentum conservation equation (2), discretize it as:

\[
\frac{dv_i}{dt} = -\frac{1}{m_i} \sum_j (V_{j1}^2 + V_{j2}^2) \widetilde{P}_j \nabla W_{ij} + \frac{1}{m_i} \sum_j \frac{2 \mu_i \mu_j}{\mu_i + \mu_j} (V_{j1}^2 + V_{j2}^2) v_j \left(\frac{1}{r_{ij}} \nabla W_{ij} - r_{ij} \nabla W_{ij}ight) + g
\]

(7)

In the formula, the first term is the pressure term, where:

\[
\widetilde{P}_j = \frac{\rho_i P_j + \rho_j P_i}{\rho_i + \rho_j}
\]

(8)

In Equation (8), \( \widetilde{P}_j \) is the pressure after density weighted average. \( V_i = \frac{m_i}{\rho_i} \) is the volume of the particle \( i \). \( \mu = \rho v \) is the kinematic viscosity coefficient.
3.3. Artificial viscosity

The distortion caused by numerical oscillation often appears in the computational fluid flow problem with SPH method. For this reason, Monaghan proposed to add artificial viscosity term into the momentum conservation equation, and the specific expression is as follows:

\[
\Pi_{ij} = \begin{cases} 
-\alpha_{ij} c_{ij}^2 + \beta_{ij} \phi_{ij}^2, & v_{ij} \cdot x_{ij} < 0 \\
0, & v_{ij} \cdot x_{ij} \geq 0 
\end{cases}
\]  

(9)

Where:

\[
\phi_{ij} = \frac{h v_{ij} \cdot x_{ij}}{|x_{ij}|^2 + \phi^2}
\]  

(10)

\[
\alpha_{ij} = \frac{c_i + c_j}{2}
\]  

(11)

\[
\beta_{ij} = \frac{\rho_i + \rho_j}{2}
\]  

(12)

\[
v_{ij} = v_i - v_j, x_{ij} = x_i - x_j
\]  

(13)

In artificial viscosity, \(\alpha_{ij}\) and \(\beta_{ij}\) are standard constants, and the value is generally about 1.0. The factor \(\phi = 0.1h\) is used to prevent numerical oscillations caused by particles close to each other, and \(c\) and \(v\) represent the speed of sound and the speed of particles. In the equation, the ones related to \(\alpha_{ij}\) include shear viscosity and expansion viscosity; those related to \(\beta_{ij}\) play an important role in preventing non-physical penetration of particles.

4. Density correction

The SPH method is a meshless method in which fluid particles can move freely. This movement leads to too many or too few adjacent particles in the support domain, which causes pressure instability. This is also the main reason why the traditional SPH method cannot obtain a stable and smooth pressure field. In this paper, flux is added to the density equation (6) for density correction. The specific formula is as follows:

\[
\frac{d\rho_i}{dt} = \sum_j m_j \left( v_{ij} + n_{ij} \left( \frac{c_i}{\rho_i} \left( \rho_j - \rho_i - \Delta \rho_j \right) \right) \right) \cdot \nabla W_{ij}
\]  

(14)

Among them, the effective sound velocity value \(c_{ij} = \max\left( c_i, c_j \right)\) involved in flux, the calculation formula is as follows:

\[
c_i = \left[ \frac{\partial P}{\partial \rho_i} \right]^{\gamma-1} = c_0 \left( \frac{\rho_i}{\rho_0} \right)^{\gamma-1}/2
\]  

(15)

In Equation (14), \(n_{ij}\) is the normal vector between particle i and j:

\[
n_{ij} = \frac{x_{ij}}{r_{ij}}
\]  

(16)
Where $\mathbf{r}_{ij} = \mathbf{r}_i - \mathbf{r}_j$, $r_{ij}$ is the distance between two particles. Then add the corrected hydrostatic pressure term to the density difference, the calculation formula is as follows:

$$\Delta \rho_{ij} = \frac{\delta \rho_{ij} + \delta \rho_{ij}}{2}$$

(17)

$$\delta \rho_{ij} = \left( \rho_{i}^{t-1} + \frac{\rho_{i}^{t-1}(\gamma - 1)}{c_s^2} g(y_j - y_i) \right)^{\frac{1}{\gamma - 1}}$$

(18)

5. Boundary processing

In order to ensure that the solid wall boundary meets the impermeability and non-slip conditions, this paper adopts a new processing method. On the solid wall, arrange a layer of virtual particles on the solid wall; on the solid wall, arrange three to four layers of virtual particles outside the solid wall.

Different from the Monaghan method, in order to prevent particles from penetrating the boundary of the solid wall, the virtual particles on the solid wall no longer exert repulsive force on the particles adjacent to the boundary. Instead, it participates in the calculation of speed and pressure in the control equation. During the calculation, the density and position of the virtual particles on the solid wall always remain unchanged. The pressure of the virtual particles on the solid wall is obtained by the weighted average of the pressures of all fluid particles in the support domain, and the SPH regularization interpolation formula is adopted, and the expression is as follows:

$$P_j = \frac{\sum_i (m_i / \rho_i) P_{W_{ij}}}{\sum_i (m_i / \rho_i) W_{ij}}$$

(19)

Where, $P$ represents the pressure, $i$ represents the virtual particle on the solid wall, and $j$ represents the fluid particle adjacent to the particle $i$. The summation in the formula is only for the fluid particle.

The density and position of virtual particles outside the solid wall remain unchanged as the virtual particles on the solid wall. And each virtual particle outside the solid wall has a unique virtual particle on the solid wall linked to it. The value of the pressure of the virtual particles outside the solid wall is the same as the value of the virtual particles on the linked solid wall. The model is shown in the figure below:

![Fig. 1 Solid wall boundary treatment](image)

The above method is improved when dealing with the problem of the solid wall boundary of the complex shape obstacle. At the boundary of the obstacle, the method of demarcation is used to avoid the influence caused by the two solid walls being too close; and the method of dynamically generating virtual particles outside the solid wall is used at the junction. That is, the mirror particle method and the virtual particle method are combined to further consolidate the problem of weak force at the complex boundary and prevent particles from penetrating the boundary.
When approaching the weak boundary, the virtual particles outside the solid wall corresponding to the virtual particles on the solid wall are dynamically generated. The position of the virtual particles outside the solid wall is no longer fixed, and its position is on the line connecting the fluid particles and the virtual particles on the solid wall, the model is shown in Fig. 2.

Fig. 2 Special treatment at complex boundaries

The figure depicts the situation where fluid particles flow through a wedge-shaped obstacle. Dynamic virtual particles are generated instantly at each time step, and the physical quantity they carry is still obtained by the virtual particles on the solid wall. And the dynamic virtual particles generated by each fluid particle do not affect each other. When particles 1, 2, and 3 approach the wedge-shaped solid wall, they will generate corresponding virtual particle groups 1, 2, and 3 respectively, which are located on the line between the current fluid particle and the solid wall particle. The virtual particle groups are independent of each other and do not interfere with each other. They are generated and used immediately only when needed.

At the same time, pressure acquisition methods at complex boundaries also need to be adjusted. If the pressure of the solid-wall particles at the complex boundary is still selected as the weighted average of the pressures of all fluid particles in the entire support domain, when there are fluids on both sides of the complex boundary, the pressure on both sides may be offset, resulting in insufficient repulsive force on the complex boundary. Make a vertical line connecting the current fluid particle and the solid wall particle, and select only the fluid particle on the same side as the current fluid particle. Use the pressure of the fluid particles in this area to do the weighted average to get the pressure of the solid wall particles at the moment.

As shown in Fig. 3, for No. 1 fluid particle, first connect the No. 1 particle and the solid wall particle, make a vertical line connecting the solid wall particle, and then select the fluid particle on the left side of the vertical line to do the weighted average to obtain the solid wall pressure; For No. 2 fluid particles, we select the upper left fluid particle for weighted average; for No. 3 fluid particles, we select the upper fluid particle for weighted average.

Fig. 3 Pressure acquisition at complex boundaries
6. Numerical examples

6.1. Dam-break flow without adding obstacles

First, consider the problem of dam-break flow without adding obstacles. This calculation example was originally proposed by Colagrossi et al. [8] in 2D space, and it has now been used as one of the standard calculation examples for dam-break flow simulation.

The calculation model of the initial state of the two-dimensional dam break is shown in Fig. 4. In order to verify the effectiveness of the meshless SPH particle method in simulating free-surface flow, this paper first conducts a numerical simulation of the dam-break flow problem without adding obstacles. The dimensions of the dam-break flow are consistent with those in the literature [8]. That is, the initial height of the dam is \(H\), and the length is \(L = 2H\). The length and height of the rectangular container are \(d = 5.366H\) and \(D = 3H\), respectively. Points A and B are the monitoring points of the water level height. Point C is the wall pressure monitoring point. In the calculation, the initial spacing of the particles is \(\delta_0 = 0.02H\), and the total number of corresponding fluid particles is 5000 \((50 \times 100)\). The kernel function uses cubic spline function, and the smoothing step is set to \(\delta h = 1.3\delta_0\). In order to ensure the stability of the value, the time step is \(\Delta t = 1.0 \times 10^{-4}\).

![Fig. 4 Calculation model of 2D dam-break flow](image)

Fig. 5 shows the pressure field of the dam-breaking flow at several key moments. It can be seen from the figure that the fluid particles reach the right wall at 0.8s, and the right wall is subjected to a larger impact at this time; it is blocked by the right wall and the fluid rises to the highest point at 1.5s; under the action of gravity, the fluid begins to fall, the falling fluid hits the fluid at the bottom at 1.96s, and the place where it touches the bottom also receives a greater impact. Numerical results show that the improved SPH method can accurately and stably simulate the entire dam break process, and the free surface obtained is consistent with the results of the literature [14], which verifies the effectiveness of the SPH method in simulating the free surface flow problem.

![Fig. 5 Pressure field of dam-breaking flow](image)
Fig. 5 The particle distribution of the dam-breaking flow without adding obstacle at different moments

Fig. 6 further shows the change of the front position of the dam-break flow obtained by the improved SPH method over time. Numerical simulation of 3 different numbers of fluid particles, the total number of fluid particles are 1250 (25×50), 5000 (50×100), 20000 (100×200). Comparing the distribution of three different numbers of fluid particle fronts, it can be seen that the improved SPH particle method used in this paper has good convergence; at the same time, it is compared with the results given in the literature, and it is consistent with the results in the literature [14], which verifies the effectiveness of the improved SPH particle method.

Fig. 6 The change of the front position of the dam-break flow with time

Fig. 7 shows the pressure change diagram at point c (see Fig. 4) over time. Comparing the numerical simulation results in this paper with Colagrossi and JOSEPHINE, it can be seen that it is close to the pressure curve in the literature. The results show that after adding the density correction term, a relatively stable pressure field is obtained.

Fig. 7 Change of pressure on the right wall of the dam-break flow with time
6.2. Dam-break flow with columnar obstacle added
In order to study the application of the improved SPH particle method in practical problems, Fig. 8 shows the process of the dam-break impacting the cylindrical obstacle. Numerical results show that the meshless SPH particle method can simulate the dam break problem of columnar obstacle stably and accurately. The figure shows the fluid pressure field at several key moments. The fluid hits the columnar obstacle in 0.34s, and the bottom of the columnar obstacle receives a greater impact; under the obstruction of the obstacle, the dam-breaking flow splashes upwards, producing water spray, which rises to the highest point in 0.6s; under the action of gravity, it touches the bottom of the container at 1.3s and produces a large impact; then it reaches the highest point of the right wall at 1.96s and produces a large impact on the right wall.

\( t = 0.34 \text{s} \) \( t = 0.6 \text{s} \) \( t = 1.3 \text{s} \) \( t = 1.96 \text{s} \)

Fig. 8 Dam-break flow with columnar obstacle

6.3. Dam-break flow with columnar obstacles with holes added
In order to further study the application of the improved SPH particle method in practical problems, Fig. 9 shows the process of the dam-break impacting the cylindrical obstacle with holes. It can be seen from the figure that the fluid hits the cylindrical obstacle at 0.34s, and a large number of fluid particles pass through the hole at 0.5s. At this time, the pressure is wavy, and the hole receives a larger impact; It touched the bottom of the container at 1.24s and reached the highest point on the right wall at 1.88s. The difference from the cylindrical shape is that when the dam-break stream hits the cylindrical shape with holes, fluid particles begin to overflow from the hole, and the particles flowing out of the hole have greater kinetic energy and reach the right wall first.

\( t = 0.34 \text{s} \) \( t = 0.5 \text{s} \)

Fig. 9 Dam-break flow with columnar obstacles with holes added
6.4. Dam-break flow with wedge-shaped obstacle added

To further investigate the application of the improved SPH particle method in a practical problem, Fig. 10 shows the process of a dam-break striking a wedged barrier. As can be seen from the figure, the fluid impinges on the cylindrical obstacle at 0.34s and produces a large impact force. It rises under the obstruction and reaches the highest point at 0.76s. Under the action of gravity, it touches the bottom of the container at 1.22s and produces a large impact force. It reaches the highest point of the right wall at 1.92s and causes a large impact force on the right wall. In contrast to the column barrier, the dam-break flow flows forward when it hits the blocker barrier as a whole and has reached the right wall at 1.22s. However, when the blocker barrier prevents the flow, the dam-break flow is upward and still has a long way to go from the right wall.

6.5. Dam-break flow with semi-circular obstacle added

In order to further study the application of the improved SPH particle method in practical problems, Fig. 11 shows the process of the dam break hitting the semi-circular obstacle. The fluid hits the columnar obstacle in 0.34s and produces a greater impact on the obstacle; it rushes forward to the right under the obstacle, and rises to the highest point in 0.76s; under the action of gravity, it contacts the container in 1.0s. The bottom has a greater impact on the bottom; it reaches the highest point of the right wall at 1.94s. Similar to the wedge-type obstacle, the fluid front first hits the upper right wall and then merges with the lower fluid.
6.6. Dam-break flow with 1/4 lower right circle obstacle added

In order to further study the application of the meshless SPH particle method in practical problems, Fig. 12 shows the process of the dam-break impacting the 1/4 lower right circle obstacle. The fluid hits the columnar obstacle in 0.4s and produces a small impact; guided by the obstacle, it rises to the highest point in 0.94s; under the action of gravity, it touches the bottom of the container in 1.34s and produces a larger impact. It reached the highest point on the right wall in 2.0s. The shape of this obstacle is similar to the wedge type, which plays a role of drainage when the fluid passes. Compared with the wedge-shaped, this shape has a better drainage effect and the fluid is guided to a higher place.

Fig. 12 Dam-break flow with 1/4 lower right circle obstacle
6.7. Dam-break flow with 1/4 upper right circle obstacle added
To further study the meshless method of SPH particles in the application of practical problems, Fig. 13 shows the dam the impact process of 1/4 upper right circle obstacle. The fluid impinges on the cylindrical barrier at 0.42s and produces a large impact force. Under the obstruction, it rises to the highest point at 1.02s. Because the obstacle has a left semicircular arc, part of the particles are bounced to the left wall. Under the action of gravity, it contacts the bottom of the container at 1.38s and produces a large impact force. The highest point on the right wall is reached at 2.2s. Unlike the lower quarter semicircular barrier, the upper quarter semicircular barrier does not have the effect of drainage and will block the water to the left. Therefore, the fluid reaches the highest point on the right wall at the latest under the action of this barrier.

\[
\begin{align*}
(a) & \quad t = 0.42\text{s} \\
(b) & \quad t = 1.02\text{s} \\
(c) & \quad t = 1.38\text{s} \\
(d) & \quad t = 2.2\text{s}
\end{align*}
\]

Fig. 13 Dam-break flow with 1/4 upper right circle obstacle

6.8. Pressure output comparison
Fig. 14 shows the variation of the right wall pressure of dam-break flow with time under different obstacles, from which the following conclusions can be drawn. First of all, the fluid reaches the right wall fastest is the cylindrical barrier with holes, and the slowest is the 1/4 upper right circle barrier. Secondly, the columnar shape is compared with the columnar shape with holes, the cylindrical shape with holes makes some fluid particles pass through the obstacle first and reach the right wall, which reduces the pressure on the right wall. Finally, the comparison between 1/4 upper right circle and 1/4 lower right circle shows that under the obstruction of the 1/4 upper right circle, the wave peak under the pressure on the right wall is backward, and the peak value is lower than that of the 1/4 lower right circle, which can effectively block the dam break flow. Therefore, in order to alleviate the impact force on the right wall, the obstacles with holes such as the column are preferred. If the dam break flow reaches the right wall at the latest, the obstacles such as 1/4 upper right circle are recommended.

\[
\begin{align*}
(a) & \quad \text{columnar} \\
(b) & \quad \text{columnar with holes}
\end{align*}
\]
Fig. 14 The pressure on the right wall changes with time under different obstacles

7. Conclusion
This paper uses the SPH particle method to simulate the obstacles in seven cases of obstacles. On the dam-break flow problem, the conclusions are as follows:

(1) Adding flux to the continuity equation can effectively improve the pressure of the flow field.

(2) The improved boundary processing method can handle complex boundary problems well and prevent particles from penetrating obstacles. Numerical simulation results show that this method can be applied to obstacles of various shapes and is a robust boundary processing method.

(3) Through the numerical simulation of the dam-break flow, the results show that the change of the front position of the dam-break flow with time is close to the experimental value, which verifies the effectiveness of the improved SPH particle method.

(4) Numerical simulation of dam breaks and impacts on different obstacles. The data obtained from the front of the water flow and the overall movement trend of the water flow can provide a certain scientific basis for coastal disaster prevention and mitigation.

References
[1] P. Lin, P.L.F. Liu. A numerical model for breaking wave in the surf zone, J. Journal of Fluid Mechanics. 359 (1998) 239-264.
[2] Yuehao Tang, Shengyun Chen, Qinghui Jiang. A conservative SPH scheme using exact projection with semi-analytical boundary method for free-surface flows. Applied Mathematical Modelling. 82 (2020) 607-635.
[3] Zhang Chi, Rezavand Massoud, Hu Xiangyu. A multi-resolution SPH method for fluid-structure interactions, J. Journal of Computational Physics, 2020, pp.
[4] H. Karim, Serroukh, M. Mabssout, M.I. Herreros. Updated Lagrangian Taylor-SPH method for large deformation in dynamic problems, J. Applied Mathematical Modelling. 80 (2020).
[5] L.B. Lucy. A numerical approach to the testing of the fission hypothesis, J. Astronomical Journal. 82 (1977) 1013-1024.
[6] R. Gingold, J. Monaghan. Smoothed particle hydrodynamics: theory and application to non-spherical stars, Monthly Notices of the Royal Astronomical Society. 181(1977) 375–398.
[7] J. Monaghan. Simulating free surface flows with SPH, J. Journal of computational Physics. 110(1994) 399-406.
[8] Andrea Colagrossi, Maurizio Landrini. Numerical simulation of interfacial flows by smoothed
particle hydrodynamics, J. Journal of Computational Physics. 191(2003).

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

[10] A. Leroy, D. Violeau, M. Ferrand, C. Kassiotis. Unified semi-analytical wall boundary conditions applied to 2-D incompressible SPH, J. Journal of Computational Physics. 261 (2014).

[11] M. Ferrand, D.R. Laurence, B.D. Rogers, D. Violeau, C. Kassiotis. Unified semi-analytical wall boundary conditions for inviscid, laminar or turbulent flows in the meshless SPH method, J. International Journal for Numerical Methods in Fluids. 71 (2013) 446-472.

[12] Krimi Abdelkader, Ramirez Luis, Khelladi Sofiane, Navarrina Fermin, Deligant Michael, Nogueira Xesús. Improved δ-SPH Scheme with Automatic and Adaptive Numerical Dissipation, J. Water. 12 (2020) 2858-2858.

[13] M.B. Liu, G.R. Liu. Smoothed particle hydrodynamics (SPH): an overview and recent developments, J. Archives of Computational Methods in Engineering. 17 (2010) 25-76.

[14] J.M. Cherfils, G. Pinon, E. Rivoalen. JOSEPHINE: A parallel SPH code for free-surface flows, J. Computer Physics Communications. 183 (2012) 1468–1480.