Mathematical Modeling and Algorithm Application
ISSN: 3006-0842 | Vol. 2, No. 1, 2024

The Application of an Improved SPH Boundary Treatment Model in Rigid-fluid Interaction Problems

Lintao Zhang*, Hongfu Qiang, Dudou Wang, Yuxiang Liu

Xi'an Hi-Tech Institute, Xi'an, China

* Corresponding author

Abstract: As a truly Lagrange meshfree method, smoothed particle hydrodynamics (SPH) faces the obstacles of rigid-fluid interaction problems. In this paper, the application of SPH methodwith an improved SPH boundary treatment model in solving rigid-fluid interaction problems is studied. Two rigid-fluid interaction problems of eccentric falling block and two dimensional cylinder entry water are simulated. Results of simulationare compared with the experimental results and another simulation results, respectively. Research of this paper expands application of SPH method in solving rigid-fluid interaction problems, and with the help of this method, the stable wrinkled ow ruined eld, smooth velocity and pressure ruined elds could be obtained.

Keywords: Smoothed Particle Hydrodynamics; Solid Boundary; Rigid-fluid Interaction.

1. Introduction

The interaction between fluid and rigid body is a widespread problem in production and application. Although Euler method based fluid dynamics simulation has been developed for 20 years, there are still some problems in solving such problems, such as free surface boundary constraints and large deformation of fluid. smoothed particle hydrodynamics (SPH) method, as a pure Lagrange meshless method, was proposed by Lucy in 1977[1]and has been perfected for nearly 40 years. Now it is widely used in various fields of fluid mechanics. Compared with the traditional mesh method, SPH method has unique advantages[2] in solving the problems of free surface, deformation and moving boundary and large deformation.

However, due to the nature of Lagrange particle method, when solving the problem of fluid-solid interaction, due to the lack of fluid particles outside the boundary, the interpolation near the boundary will cause errors, and the boundary conditions cannot be imposed as strictly as the mesh method. It is a difficult problem that hinders the development of SPH. At present, the application methods of SPH boundary action are mostly extended by solid wall boundary application method. At present, the solid wall boundary application methods are mainly divided into boundary force method and virtual particle method. In 1994, Monaghan[3][4]proposed the boundary force method for the first time to deal with the fluid-solid coupling problem, constructing boundary particles and applying boundary forces through boundary functions. Later, Monaghan, Liu[5] and Han [6] carried out solid wall boundary treatment by improving potential functions and proposing new penalty functions. However, the parameters of the boundary force method are difficult to select. If the boundary force is too strong, it is easy to produce numerical oscillation near the boundary, and if the boundary force is weak, it will cause the problem of penetrating the boundary.

The core of virtual particle method is to use multiple layers of virtual particles as the mirror image of fluid particles, and apply the action to the fluid particles through interpolation. In 1997, Morris[7] began to use the virtual particle method for boundary processing. Marrone et[8] al. proposed and developed a method that combines internal interpolation points with fixed virtual particles, but it is difficult to apply to the boundary of complex shape. In 2015, Liu et al. [9]proposed a new SPH solid wall boundary application model, in which virtual particles are regarded as an extension of fluid with constant density, and physical parameters such as pressure and velocity of virtual particles are obtained by interpolating fluid particles in neighboring domains. In the calculation process, the virtual particles act on the density and pressure of the fluid in the flow field through the governing equations of fluid mechanics, and implicitly apply the wall repulsion force to prevent the particles from penetrating.

2. SPH Calculation Method

2.1. Governing equations and equations of state

Fluid dynamics equations based on Lagrange method are adopted in this paper:

$$\frac{d\rho}{dt} = -\rho\nabla\cdot\boldsymbol{v} \tag{1}$$

$$\frac{d\mathbf{v}}{dt} = -\frac{1}{\rho}\nabla p + \frac{1}{\rho}\nabla\cdot\boldsymbol{\tau} + \mathbf{g} \tag{2}$$

Where d/dt represents the global derivative, ρ, p , v are the density, pressure and velocity of the fluid respectively, τ are the shear stress, g is the volume force per unit mass.

The core of SPH is interpolation. Any physical quantity in SPH is discre $f(\boldsymbol{r})$[3] .The fluid space is discretized by using cubic spline form kernel function.

$$\frac{d\rho_i}{dt} = \sum_{j=1}^{N} m_j v_{ij}\cdot\nabla_i W_{ij} \tag{3}$$

$$\frac{d\boldsymbol{v}_i}{dt} = -\sum_{j=1}^{N} m_j\left(\frac{p_i}{\rho_i^2}+\frac{p_j}{\rho_j^2}+\Pi_{ij}\right)\nabla_i W_{ij} + \boldsymbol{g} + \frac{\boldsymbol{f}_{\mathrm{r}\to\mathrm{f}(i)}}{\rho_i} \tag{4}$$

Π_{ij} is the artificial viscosity, whose expression is shown in reference[10] .

In this paper, Weakly Compressible equations[10] of state were used to calculate the liquid phase. Weakly compressible equations of state look at all theoretically incompressible fluids as weakly compressible. They can explicitly solve the flow field pressure through density changes. The most commonly used form of state equations is:

$$p = p_0[(\frac{\rho}{\rho_0})^{\gamma} - 1] \tag{5}$$

In Eq. (5), ρ_0 is the initial density of the fluid, p_0 is the reference pressure. γ is a constant. $p_0 = c_s^2 \rho / \gamma$ is the reference pressure, c_s is the speed of sound in the flow field, generally selected as about 10 times the maximum speed of the flow field.

2.2. Rigid body motion equation

The rigid body is solved by the centroid dynamics equation. This layer of particles on the surface of the rigid body is used to transfer the contact effect between the fluid and the rigid body. On this basis, the momentum and moment of momentum equations of the rigid body can be discretized as:

$$m\frac{d\boldsymbol{v}_C}{dt} = \sum_j \boldsymbol{f}_{\mathrm{f}\to\mathrm{r}(j)} + m\boldsymbol{g} \tag{6}$$

$$J_C\frac{d\boldsymbol{\omega}}{dt} = \sum_j (\boldsymbol{r}_j - \boldsymbol{R}) \times \boldsymbol{f}_{\mathrm{f}\to\mathrm{r}(j)} \tag{7}$$

Where c represents the center of mass of the rigid body, j represents the surface particle of the rigid body, $\boldsymbol{f}_{\mathrm{f}\to\mathrm{r}(j)}$ represents the resultant force of all neighboring fluid particles of the rigid body boundary particle j on its force, $\mathbf{r}_j$ is the position vector of the particle j , the speed of the rigid $\mathbf{v}_j$ body surface particle can be written as:

$$\mathbf{v}_j = \mathbf{v}_C + \boldsymbol{\omega} \times (\mathbf{r}_j - \mathbf{R}) \tag{8}$$

The following relationship exists between simultaneity $\boldsymbol{f}_{\mathrm{f}\to\mathrm{r}(j)}$ and $\boldsymbol{f}_{\mathrm{r}\to\mathrm{f}(i)}$ action

$$\sum_{i\in\Omega} \boldsymbol{f}_{\mathrm{r}\to\mathrm{f}(i)} + \sum_{j\in\beta} \boldsymbol{f}_{\mathrm{f}\to\mathrm{r}(j)} = 0 \tag{9}$$

2.3. Apply rigid body to solid wall boundary

In order to effectively realize the coupling calculation between solid and fluid and effectively solve the problem of boundary application, an improved solid wall boundary application model[8] based on virtual particle method is adopted to effectively solve the problem that SPH algorithm may produce particle penetration or numerical oscillation on the solid wall boundary.

The fluid-solid wall interaction equation is as follows:

$$\left(\frac{d\rho_i}{dt}\right)_b = \begin{cases} m_b \boldsymbol{v}_{ib} \cdot \nabla_i W_{ib}, \text{ if } \boldsymbol{v}_{ib} \cdot \boldsymbol{n}_b \neq 0 \\ 0 \quad , \quad else \end{cases} \tag{10}$$

$$\left(\frac{d\boldsymbol{v}_i}{dt}\right)_b = \begin{cases} \left[-m_b\left(\frac{p_i + p_b}{\rho_i \rho_b}\right)\boldsymbol{n}_b \cdot \nabla_i W_{ib}\right]\boldsymbol{n}_b, \text{ if } (p_i + p_b) > 0 \\ 0 \quad , \quad else \end{cases} \tag{11}$$

$\left(\frac{d\rho_i}{dt}\right)_b$、$\left(\frac{d\boldsymbol{v}_i}{dt}\right)_b$ respectively represents the density and speed increment generated by the action of the boundary virtual particle b of the fluid particle i , and represents the unit normal of the virtual particle, $\boldsymbol{v}_{ib} = \boldsymbol{v}_i - \boldsymbol{v}_b$.

2.4. Time integration

In this paper, the leap-frog method[10]is used to solve the SPH discrete equation:

$$\phi_i(t + \frac{\Delta t}{2}) = \phi_i(t - \frac{\Delta t}{2}) + \phi_i(t)\Delta t \tag{12}$$

$$r_i(t + \frac{\Delta t}{2}) = r_i(t) + v_i(t + \frac{\Delta t}{2})\Delta t \tag{13}$$

Where, ϕ_i denotes the density ρ or velocity v of the particle i .

3. Numerical Example

3.1. Eccentric square into the water

Eccentric square into water is a classic example of the falling block problem, simulating the rotation and sinking process of the rigid box in the viscous fluid, is a completely free moving solid and fluid interaction problem, such problems need to solve the coupling of fluid dynamics and solid dynamics and the problem of free surface flow. The test model is shown below and compared with Barcarolo[11]'s calculations using the finite volume particle method (FVPM).

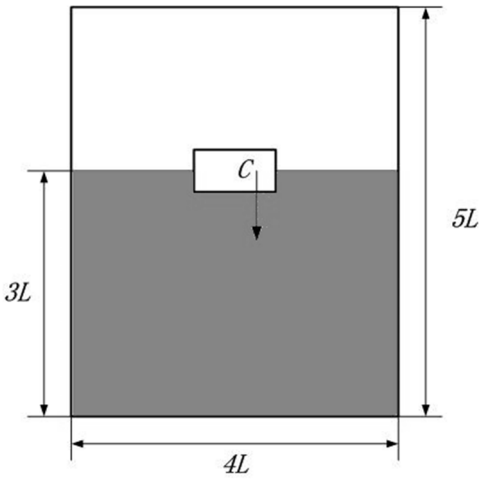
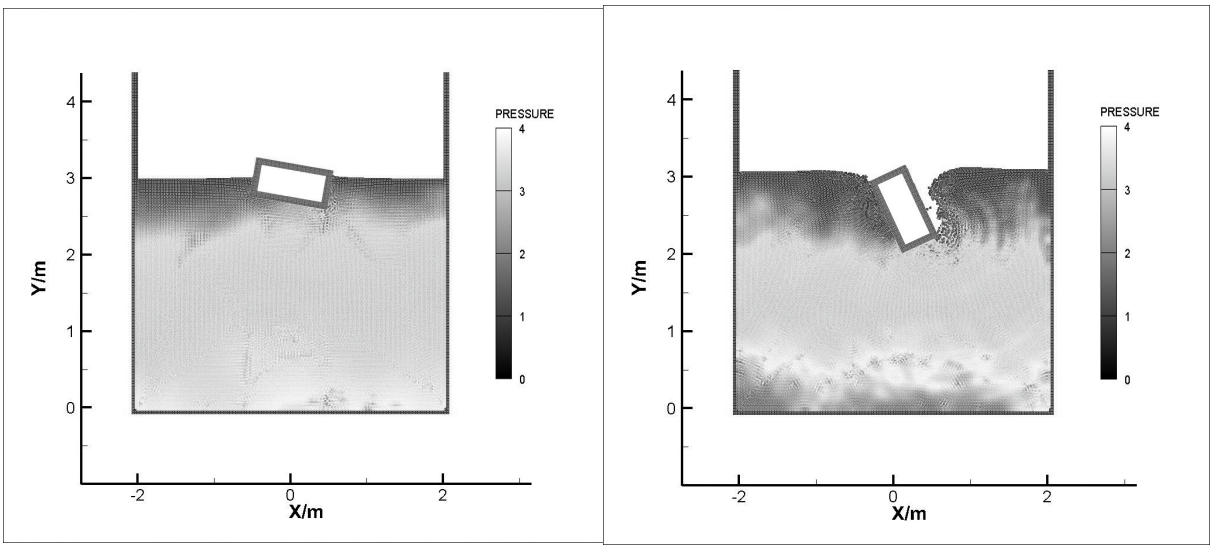


Figure 1. Schematic diagram of the eccentric block model

In Fig.1, the model setup is the same as Barcarolo, the size of the block is 1.0m×0.5m, and the center of gravity point C is initially located at (0.25m, 3m). Mass of the block $M = 1kg$, moment of inertia $I = 0.083kg \cdot m^2$, acceleration of gravity $g = 1m/s^2$. Fluid density $\rho_0 = 1kg/\mathrm{m}^3$, viscosity $\upsilon = 2 \times 10^{-3} m^2/s$, and the block is initially at rest. In the numerical calculation, the total number of particles is 32718, including 29349 fluid particles and 4 layers of particles set at each boundary. The total number of Dummy virtual particles is 3369. The particle spacing $\Delta l_b = 0.02m$, time step $\Delta t = 5.0 \times 10^{-4} s$, sound velocity $c_s \approx 24.5m/s$, the flow field morphology and pressure distribution calculated at different times of the falling block and its surroundings are shown in Fig.2.

As can be seen from Fig.2(a), since the center of gravity of the block is not in the geometric center, the block begins to tilt gradually under the action of gravity. At the same time, since the density of the block is greater than that of the fluid, the block begins to sink gradually. In Fig.2(b) 1.50s, a cavity gradually formed on the right side of the block, after which the block continued to rotate and fall, the cavity gradually closed, and the block was impacted by the right side fluid. In this method, the wall pressure distribution was normal, and no pressure shock or particle penetration occurred. After Fig. 2 (c) 2.0s, due to the pressure of the fluid on the left side, the rotation speed of the block gradually decreases to 0 and continues to fall to the bottom at a rotation Angle.

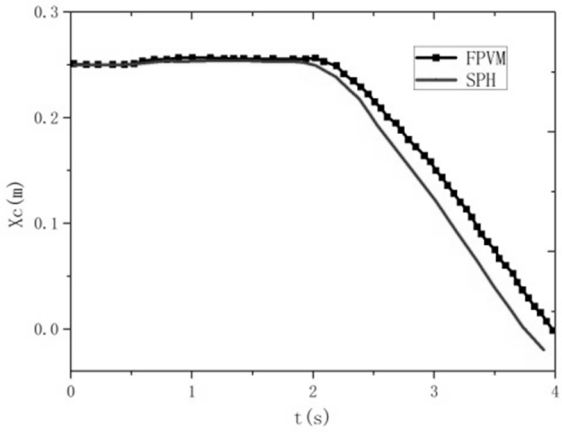


(a) t=0.70s (b) t=1.50s

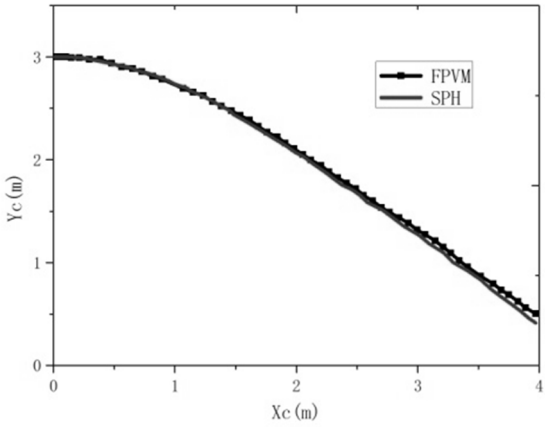
(c) t=2.00s (d) t=2.50s

Figure 2. Calculation results of flow field shape and pressure distribution around eccentric falling block at different time

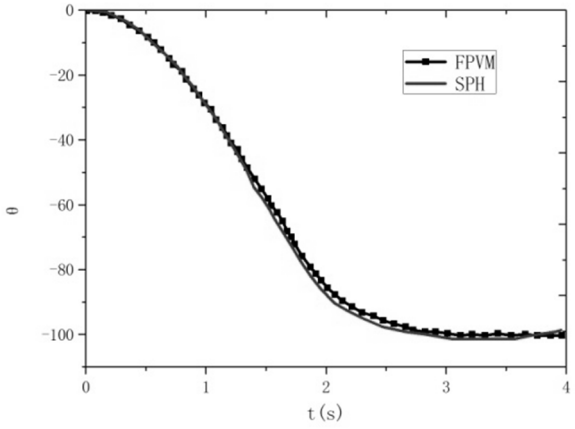
The position of the center of gravity and the deflection Angle of the block were compared with the results of FVPM [11], and it was found that the calculated results obtained by SPH method and FVPM method were similar, as shown in Fig.3. As shown in Fig.3(a), the slight deviation in X direction between SPH and FVPM at the later stage of movement may be caused by the pressure wave echo caused by the weakly compressible algorithm. However, compared with FVPM method, SPH can save a lot of computation.



(a) The change of the abscissa Xc of the center of gravity over time



(b) Yc curve with time



(c) Block deflection Angle variation over time diagram

Figure 3. Comparison of position parameters of different methods for simulating falling block motion

3.2. Two-dimensional cylinder entering water

Two-dimensional cylinder entry into water is also a typical fluid-solid interaction problem, and Greenhow et[12]al.in 1983, conducted relevant experimental research. The water entry model of a two-dimensional cylinder is shown in the figure below. The radius of the cylinder R=5.5×10-2m, the width and height of the pool are 2m and 0.5m respectively, and the parameters of the liquid are the same as those in the previous section. In the experiment, the cylinder was dropped from a height of 0.5m. At the beginning of the simulation in this paper, the cylinder just touched the water surface. According to Greenhow's experiment, the velocity of the cylinder $v_0 = 2.955m/s$ when it entered the water. In calculation, velocity of sound $c_s = 10v_0 = 29.55m/s$, artificial viscosity $\alpha_{av} = 0.05$. Particle spacing $\Delta l_b = 0.02m$, starting time is the time when the sphere touches the water surface, time step $\Delta t = 5.0 \times 10^{-4} s$. The total number of particles is 43058, among which 4 layers of

Dummy virtual particles are arranged at each boundary, and the total number is 3226.

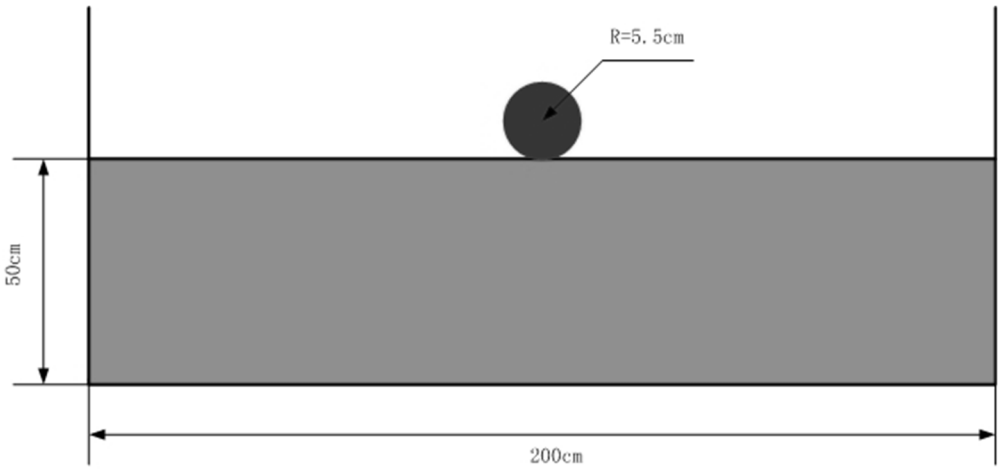
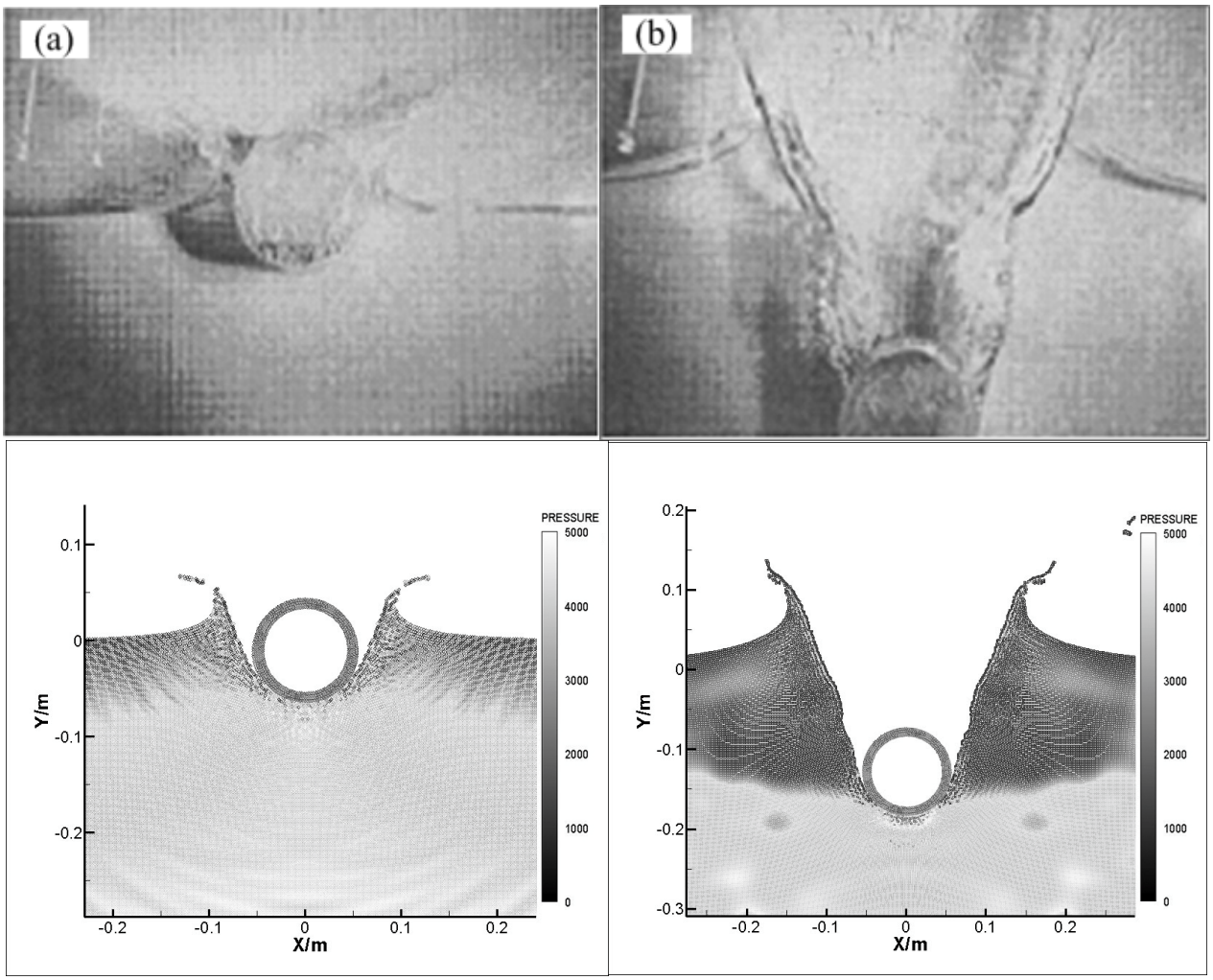


Figure 4. Schematic diagram of a two-dimensional cylinder entering water model

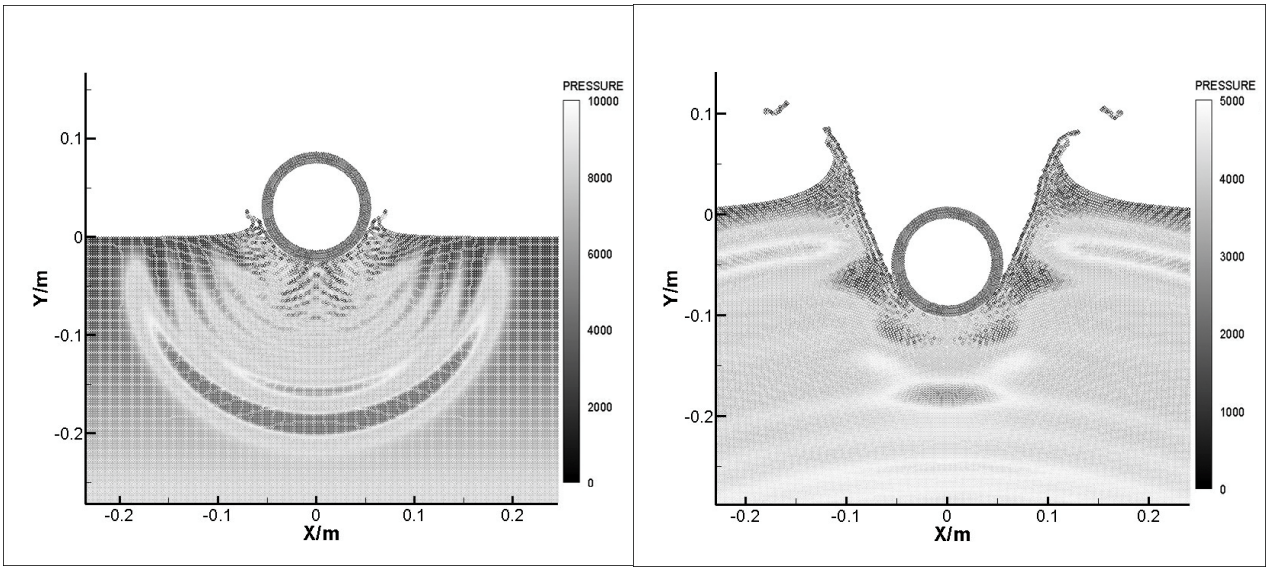
Compared with the experimental results, it can be seen from Fig. 5 that the simulation results (c) (d) of SPH are very close to the experimental results (a) (b) of Greenhow, and because the particle density adopted by the model is large, it can better catch the splash of droplets and the change of free surface.



(c) 30ms (d) 100ms

Figure 5. Comparison of SPH method simulation and experiment for two dimensional cylinder entering water at different time

Meanwhile, as can be seen from Fig. 6, when the cylinder impinges into water, there is no particle penetration and numerical instability. 10ms after the ball just touches the water surface, a compression wave will be generated and transmitted in the liquid with normal pressure distribution. At 50ms, the flow field around the ball is affected by the compression wave reflected back from the wall.



(a) 10ms (b) 50ms

Figure 6. Pressure distribution diagram of flow field around 2D cylinder at different time

4. Conclusions

In this paper, the SPH method is analyzed to solve the problem of fluid-solid interaction, and the numerical simulation of two fluid-solid interaction problems is carried out. The numerical example results show that the proposed method can effectively apply wall action and carry out fluid-solid coupling calculation, and the calculation amount is low, and it is easy to deal with the complicated boundary fluid-solid interaction problem. The application of Liu's improved solid-wall boundary application model in the fluid-solid interaction is extended. At the same time, further expanding the application of the proposed method to the three-dimensional fluid-solid interaction problem is the direction of our next work.

Acknowledgment

This paper was supported by National Natural Science Foundation of China (NSFC) under Grant No. 91952110 for this work.

References

[1] Lucy L B. A Numerical Approach to the Testing of the Fission Hypothesis[J]. The Astronomical Journal, 1977,82:1013-1024.

[2] Qiang Hongfu, Shi Chao, Chen Fuzhen, et al. Numerical simulation of two dimensional droplet collision based on SPH method of multiphase flow with large density difference [J]. Acta Physica Sinica, 2013,62(21):214701.

[3] Monaghan J J. Simulating Free Surface Flows with SPH[J]. Journal of Computational Physics, 1994,110(2):399-406.

[4] Monaghan, J.J. and J.B. Kajtar, SPH particle boundary forces for arbitrary boundaries [J]. Computer Physics Communications, 2009. 180(10): p. 1811-1820.

[5] Liu, M., J. Shao and J. Chang, On the treatment of solid boundary in smoothed particle hydrodynamics[J]. Science China Technological Sciences, 2012. 55(1): p. 244-254.

[6] Han Ya Wei, Qiang Hongfu, Zhao Jiuling, et al. A new repulsive force model for solid wall treatment using smooth particle fluid dynamics [J]. Acta Physica Sinica, 2013, 62(4):44702.

[7] Morris J P, Fox P J, Zhu Y. Modeling Low Reynolds Number Incompressible Flows Using SPH[J]. Journal of Computational Physics, 1997136 (1) : 214-226.

[8] Marrone S, Bouscasse B, Colagrossi A, et al. Study of ship wave breaking patterns using 3D parallel SPH simulations[J]. Computers & Fluids, 2012. 69: p. 54-66.

[9] Liu Hu, Qiang Hongfu, Chen Fuzhen, Han Yawei, Fan Shujia. A novel solid boundary application model for smooth particle dynamics [J]. Acta Physica Sinica, 2015(09):384-397.

[10] Monaghan J J. Smoothed Particle Hydrodynamics[M] World Scientific, 2003:543-574.

[11] Barcarolo, D.A., et al., Adaptive particle refinement and derefinement applied to the smoothed particle hydrodynamics method[J]. Journal of Computational Physics, 2014. 273: p. 640-657.

[12] Greenhow M, Lin W M. Nonlinear-Free Surface Effects: Experiments and Theory[J]. 1983.