Jul 12th, 2:30 PM - 2:50 PM

# Hydraulics simulation based on the two-phase SPH model

Hang Wan
*State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University*, 2015323060020@stu.scu.edu.cn

Huixia Yang
*State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University*

Ran Li
*State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University*, liran@scu.edu.cn

Ruifeng Liang
*State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University*

*International Environmental Modelling and Software Society (iEMSs)*
*8th International Congress on Environmental Modelling and Software*
*Toulouse, France, Sabine Sauvage, José-Miguel Sánchez-Pérez, Andrea Rizzoli (Eds.)*
*http://www.iemss.org/society/index.php/iemss-2016-proceedings*

# Hydraulics simulation based on the two-phase SPH model

**Hang Wan[1], Huixia Yang[1], Ran Li* , Ruifeng Liang[1]**

1. State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University, Chengdu 610065, Sichuan, E-mail：2015323060020@stu.scu.edu.cn

* State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University, Chengdu 610065, Sichuan, E-mail：liran@scu.edu.cn

**Abstract:** The two-phase flow of air and water is a common problem in the hydraulics simulation, which is difficult to track the deformations of interface with conventional numerical methods based on Eulerian grid-based approaches. As a Lagrangian mesh-less particle method, the Smooth Particle Hydrodynamics (SPH) model has a great advantage in modelling the interface flow. Due to the limitation of the single phase SPH model based on the platform of DualSPHysics, a two-phase SPH model is developed in the paper to investigate the mechanics of aerated flow with large density differences. However, the Mach number appears different in each phase, and the time step must be adjusted to the maximum velocity, which may result much more complexity. At the same time, the existence of interphase forces causes the interaction between different phases and increases energy loss subsequently. In order to improve the simulation precision and track a smoother interface, the SPH method developed in the paper is based on the two-phase weakly-compressible SPH (2P CMS) model with corrected momentum equations, which combines with the particle pair search method and separate neighbour particle lists method. To validate the new SPH model, the dam break experiment conducted by Zhou in 1999 is employed. The comparison shows that the corrected SPH model can track the deformations of interface in the two-phase flow easily. It demonstrates that the two-phase SPH model performs more accurately than the single phase SPH model.

**Keywords:** SPH; Lagrangian; two-phase flow; dam break

## 1. Introduction

Violent interactions between gas and water lead to obvious changes in the velocity and pressure fields which is difficult to simulate because the interface boundaries move and deform with time and exhibit arbitrary properties. Finite Volume Method (FEM) and Finite Difference Method (FDM) are adopted to deal with the interface of different phases, which are based on grid methods, such as Politano et al （2003）. and Huai et al（2006）. However, because of its own defects, grid-based methods are difficult to track interface fluids with large deformations and breakings in the numerical simulation.

It was shown by Liu et al. (2003) that the Smooth Particle Hydrodynamics (SPH) model has various advantages when dealing with large deformations and breakings flows. With an approximate weight kernel function, Newtonian mechanics equations can be solved easily by the SPH model. Monaghan (1994) simulated the free surface flows with SPH. López et al. (2010) developed a single-phase SPH model to simulate hydraulic jumps. Some researches about multiphase fluids can also be solved with the SPH model. Monaghan (1999) demonstrated that SPH can be used with small density differences in multiphase flows. Sun et al. (2010) researched wave impact on a horizontal deck with a two-phase SPH method that did not consider the difference between the air and water phases. Colagrossi et al. (2003) proposed a multiphase model to simulate the air entrapment with a surface tension correction.

At present, more SPH models are applied to simulate fluid mechanics problems. In order to improve the accuracy about the simulation model, this paper develops a corrected SPH model to simulate the two-phase flow with large density differences. Furthermore, the accuracy of the new SPH model is validated through analyzing the calculation example in the end.

## 2. Two-phase Model Description

The kernel approximation and the particle approximation are introduced to establish the 2P CMS SPH model. Some equations about the numerical model are described below.

### 2.1. The Kernel Function

The kernel function determines the performance of the model. In this paper, the Wendland kernel function is chosen as the kernel function, and the support domain is 2*h* (*h*=1.25**dp* where *h* as its smoothing length that is never zero, *dp* is the distance between particles), the kernel function is equal to *W* (*r*, *h*) = *f* *$\alpha$, where

$$f_w(q)=\begin{cases}(1-\frac{q}{2})^4(1+2q) & 0\le q\le 2\\ 0 & 2<q\end{cases} \tag{1}$$

The normalization constants $\alpha_{W,D}$ as below ( $D$ is associated with the dimension):

$$\begin{aligned}\alpha_{W,1}&=\frac{3}{4h}\\ \alpha_{W,2}&=\frac{7}{4\pi h^2}\\ \alpha_{W,3}&=\frac{21}{16\pi h^3}\end{aligned} \tag{2}$$

### 2.2. The Governing Equations

The SPH form of the continuity equation is:

$$\left\langle\frac{d\rho_a}{dt}\right\rangle=\rho_a\sum_b\frac{m_b}{\rho_b}(\mathbf{u}_a\text{-}\mathbf{u}_b)\nabla_a W_{ab} \tag{3}$$

The delta-SPH method proposed by Molteni and Colagrossi (2009) is introduced to reduce the noise wave :

$$\begin{aligned}\left\langle\frac{d\rho_a}{dt}\right\rangle=&\rho_a\sum_b\frac{m_b}{\rho_b}(\mathbf{u}_a\text{-}\mathbf{u}_b)\nabla_a W_{ab}+2\delta h\sum_b m_b\bar{c}_{ab}\\ &\times\left(\frac{\rho_a}{\rho_b}-1\right)\frac{1}{\mathbf{r}_{ab}^2+\eta^2}\nabla_a w_h(r_{ab})\end{aligned} \tag{4}$$

Where $\bar{c}_{ab}=0.5(c_a+c_b)$, $\eta^2=0.01h^2$, and $\delta$ is a coefficient recommended to be 0.1.

The momentum equation for the water phase is:

$$\left\langle\frac{d\mathrm{u}_a}{dt}\right\rangle=-\sum_b m_b\left(\frac{p_a}{\rho_a^2}+\frac{p_b}{\rho_b^2}+\Pi_{ab}\right)\nabla_a w_{ab}+\mathrm{g} \tag{5}$$

While the momentum equation for the gas phase is:

$$\left\langle\frac{d\mathbf{u}_a}{dt}\right\rangle=-\sum_b m_b\left(\frac{p_a}{\rho_a^2}+\frac{p_b}{\rho_b^2}+\Pi_{ab}\right)\nabla_a w_{ab}+\mathbf{g}-\alpha\rho_{ab}^2\sum_b m_b\nabla_a w_{ab} \tag{6}$$

Where $\alpha$ is a coefficient of losses, $\rho_{ab}=\rho_a-\rho_b$. The correction term makes the interface between gas and liquid smoother. In SPH theory, Monaghan (1992) artificial viscosity, $\Pi_{ab}$, is used extensively.

$$\Pi_{ab}=\begin{cases}\dfrac{-\alpha\bar{c}_{ab}\mu_{ab}+\beta\mu_{ab}}{\bar{\rho}_{ab}}, & \mathbf{u}_{ab}\cdot\mathbf{r}_{ab}<0\\ 0, & \mathbf{u}_{ab}\cdot\mathbf{r}_{ab}>0\end{cases} \tag{7}$$

Where the sound speed c is obtained from the state equation and the kinematic viscosity $\mu_{ab}$ is obtained from the following equation:

$$\mu_{ab} = \frac{h\mathbf{u}_{ab}\mathbf{r}_{ab}}{r_{ab}^2 + \eta^2} \tag{8}$$

The constants $\alpha$ (related to bulk viscosity) and $\beta$ (related to preventing penetration when the Mach number is high) can be valued at approximately 0.1 proposed by Monaghan (1988). $\eta = 0.1 * h$ is used to prevent numerical divergence when the particles are too close to each other.

### 2.3. The State Equation

Because of the mutual influence between the pressure and density, this paper adopts the EOS (equation of state) method which proposed by Monaghan for free surface flow:

$$p = B\left[\left(\frac{\rho_i}{\rho_0}\right)^{\gamma} - 1\right] + p_g \tag{9}$$

$$c_s = \sqrt{\frac{\partial p}{\partial \rho}} = \sqrt{\frac{\gamma B}{\rho_0}}\left(\frac{\rho_i}{\rho_0}\right)^{\gamma-1} \tag{10}$$

For the two-phase model in this research, a modified Tait's equation was used:

$$p = B\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right] + p_g - \alpha\rho^2 \tag{11}$$

Where $c_s$ is the sound speed, $B = \rho_0 c_0^{\,2}/\gamma$ proposed by Monaghan (1992). The subscripts $l$ and $g$ are for water and air, respectively. Here $\gamma_l = 7$, $\gamma_g$ =1.4. $\rho_{0l}$=1000 kg/m$^3$, $\rho_{0g}$=1.29 kg/m$^3$. Issa et al. (2004) proved that when gravity becomes a major issue, the sound speed of water can be evaluated by $c_l = 10\max\left(U_{\max}, \sqrt{gL}\right)$, where $U_{\max}$ is the maximum flow velocity and $L$ is the vertical water depth. The sound speed for the gas is $c_g = \sqrt{c_l^2 \gamma_g \rho_{0l} / \left(\gamma_l \rho_{0g}\right)}$. The $\alpha\rho^2$ term can be an effective restraint for the gas phase diffusion.

### 2.4. The Time Step

This paper adopts the Verlet explicit scheme to solve the SPH time integral:

$$\begin{aligned} \mathrm{v}_a^{n+1} &= \mathrm{v}_a^{n-1} + 2\Delta t \frac{d\mathrm{u}_a^n}{dt}; \\ \rho_a^{n+1} &= \rho_a^{n-1} + 2\Delta t \frac{d\rho_a^n}{dt}; \\ \mathrm{r}_a^{n+1} &= \mathrm{r}_a^n + \Delta t \mathrm{v}_a^n + 0.5\Delta t^2 \frac{d\mathrm{u}_a^n}{dt}; \end{aligned} \tag{12}$$

Where $\mathbf{r}_a$, $\rho_a$, and $\mathbf{v}_a^n$ are the position, density and velocity; and $\Delta t$ is the time increment. Verlet determined that every 40 time steps, the variables (position, density or velocity) sum according to:

$$\mathrm{v}_a^{n+1} = \mathrm{v}_a^{n-1} + \Delta t \frac{d\mathrm{v}_a^n}{dt}; \tag{13}$$

The computation of the time step is associated with the viscosity and unit mass force:

$$\Delta t = C_{CFL} \cdot \min(\Delta t_f, \Delta t_\nu) \tag{14}$$

Where $\Delta t_f = \min(\sqrt{h/|f_i|})$, $\Delta t_\nu = \min\left(0.125\frac{h^2}{\nu}\right)$, $C_{CFL}$ is between 0.1 and 0.2, $f_i$ is the acceleration of the unit mass force.

## 3. Simulation Procedure

The simulation of the SPH model developed in the paper consists of three main steps showed in Figure 1: (I) Data Preparation. Before the program is executed, some data should be loaded which include particle properties, execution parameters and properties of calculation domain. (II) Main Loop. After preparatory works are executed, the interaction of particles will be executed with results saved through circular computations. This step is a core part of calculation model which determines the computing time of SPH model. (III) Display and Analysis. Combing with the analysis software, results will be rendered and analyzed.

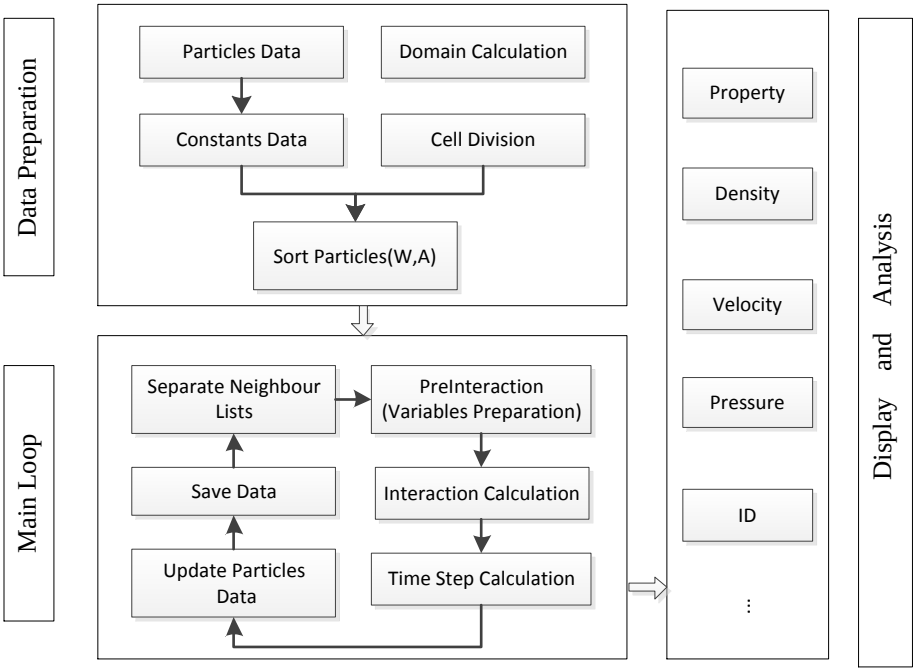



**Figure 1.** Research procedure.

In SPH numerical simulation, all valid particles must to be defined with only Id which have a huge advantages in tracing particles attribute changes including velocity, mass, pressure, density and so on. In the interaction step, each particle will be interacted by other particles in the domain, which will consume a large amount of computation time. In order to improve the computational efficiency, this paper adopts the cell-linked list (CLL) search method proposed by Domínguez (2011) (Figure 2). In the CLL method, the domain is divided into different background cells and all particles are stored into background cells. For 2d coordinates, the cell size is the length of the support domain (2h). All pair-wise distances of particles are less than 2h, which means all adjacent particles can only appear in the cell of the adjacent cell. For a particle located inside a cell, only the interactions with the particles of neighbouring cells need to be considered. Therefore, during the Force Computation, domains without adjacent particles should not be calculated and compared, which improves the computational efficiency greatly.

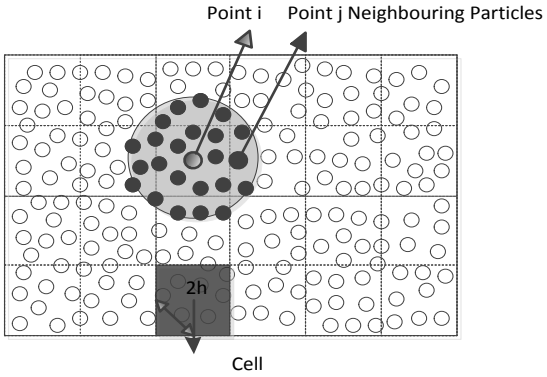



**Figure 2**. Cell-linked list search method.

For particles in the support domain (search radius is 2h), using the antisymmetry of the forces for particle-particle pairs can improve the calculation time further proposed by Harald et al. (1995). With this property, calculation and storage will only calculate one half of all interactions.

## 4. Model validation

To evaluate the advantages of the model, the 2P CMS model for a classic dam break is tested. The experiment of Zhou et al. (1999) is adopted for the validation. The initial configuration of the experiment is shown in Figure 3. The level gauges in Zhou's experiment were located in profiles S1 (992 mm from the right wall). The monitored point P of the pressure was 160 mm above the bottom. The initial particle distance is 0.01 m.

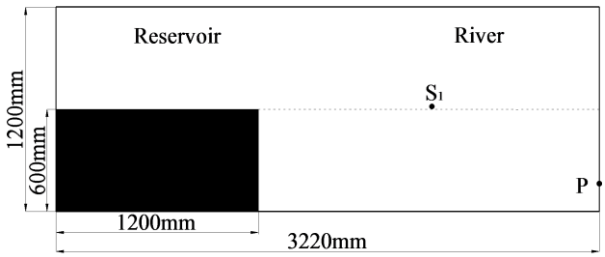
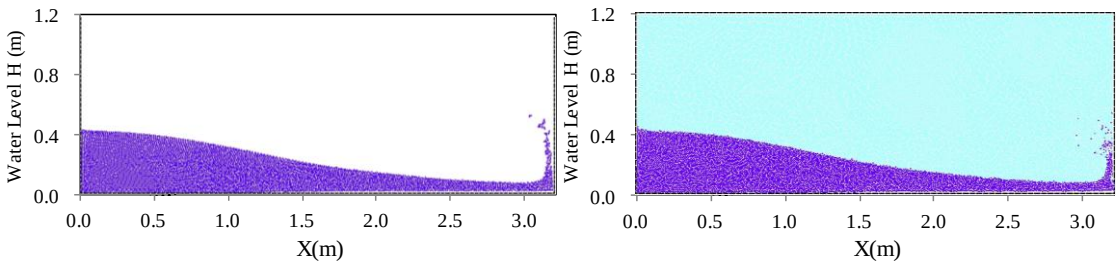



**Figure 3**. Dam break.

Figure 4 shows the simulated profiles at different times including the single phase flow and the two-phase flow. At t = 0.6 s, water hits and climbs up the right wall of the tank. Before the air pocket is entrapped in the flow, the wave profiles simulated by the single and two-phase models are similar. However, after an air pocket is entrapped, some differences emerged. The reason is that the pressure in the air pocket is equal to the free surface, which results in no resistance to the neighbouring water particles. In the two-phase model, the air pocket exists for a period of time due to the air pressure inside, and the pocket shape changes due to the air compressibility.




t = 0.65 s

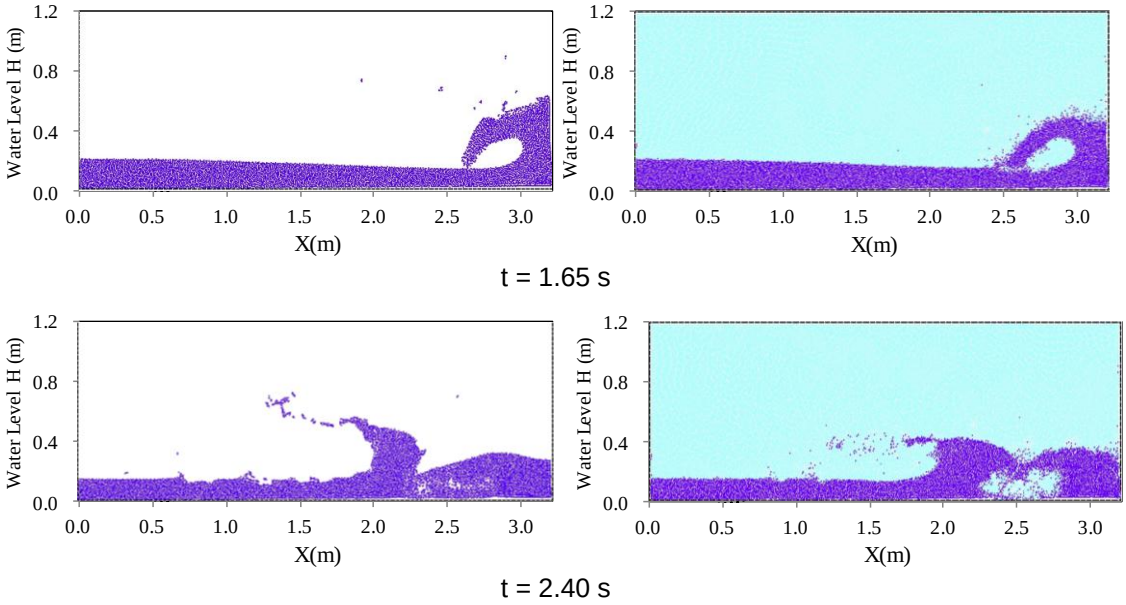



**Figure 4**. Simulation results of profiles at different times by the single phase (left column) and two-phase (right column) SPH models.

As shown in Figure 5, the corrected model appearing in the momentum equation for the gas phase shows smoother than uncorrected model. The corrected SPH algorithm can not only simulate the flow of the dam accurately, but also distinguish the interface of the impermeable two-phase flow clearly.

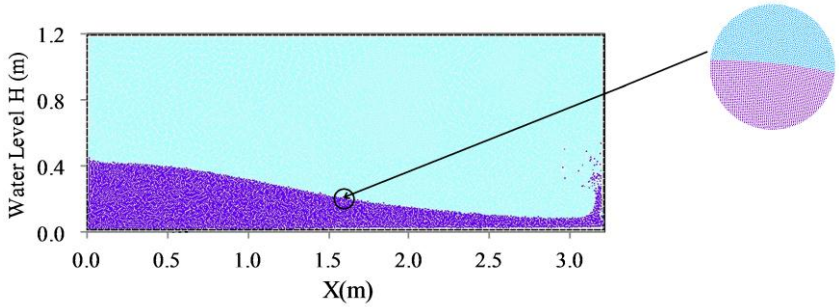



**Figure 5**. The interface of the impermeable two-phase flow.

Figure 6 (left) illustrates the variation of the simulated pressure at the point P (marked as a red line: 2P-SPH by Wan et al.). Comparing with the validation of real flood experiment result presented by Zhou et al. (1999), the two-phase SPH model match the experimental data (5.0 kPa) much better than the single phase SPH model at the point P.

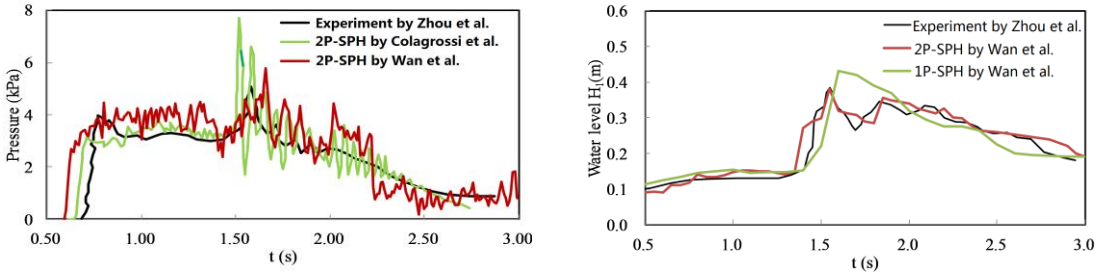



**Figure 6**. Dam break: simulated pressure and the water levels from two two-phase SPH methods compared to the experimental data measured by Zhou et al. (1999).

Figure 6 (right) shows the water levels $S_1$ at profiles 992 mm from the right wall. For the single-phase flow, there is no air pressure on the top of the free surface leading to a higher water level at approximately 1.65 s. The agreement between the simulation results and the experiment fit well for the two-phase simulation results.

## Conclusions

The two-phase SPH model was developed to conduct numerical simulations with respect to the phenomenon of interface flow with large density differences. The model is capable of simulating the dam break issue clearly. The main achievements are summarized as follows: (1) the two-phase model code has been established and the interaction of air and water particles can be correctly reflected; (2) with corrected momentum and state equations, the new two-phase SPH model can get a smoother interface; (3) combining with the particle pair search method and separate neighbour particle lists method, the simulation speed is improved; (4) With validation of real flood experiment result, the two-phase SPH model performs more accurate than the single phase SPH model. However, due to the computational cost, some methods to accelerate the computation speed need to be researched.

## ACKNOWLEDGMENTS

This research is supported by the National Natural Science Foundation of China (Grant No.51279115 and Grant No.51279114).

## References

Colagrossi A., Landrini M., 2003. Numerical Simulation of Interfacial Flows by Smoothed Particle Hydrodynamics. Journal of Computational Physics. 191(2), 448-475.

Domínguez, J.M., Crespo, A.J.C., Gómez-Gesteira, M., Marongiu, J.C., 2011. Neighbour lists in smoothed particle hydrodynamics. International Journal for Numerical Methods in Fluids. 67(12), 2026-2042.

Riffert H., Herold H., Flebbe O., Ruder H., 1995. Numerical aspects of the smoothed particle hydrodynamics method for simulating accretion disks. Computer Physics Communications. 89(1), 1-16.

Huai W., Fang S., 2006. Numerical Simulation of Obstructed Round Buoyant Jets in a Static Uniform Ambient. Journal of Hydraulic Engineering. 132 (4), 428-431.

Issa R., Lee E.S., Violeau D., Laurence D.R., 2004. Incompressible Separated Flows Simulations with the Smoothed Particle Hydrodynamics Gridless Method. International Journal for Numerical Methods in Fluids. 47 (10-11), 1101-1106.

Liu G.R., Liu M.B., Li S., 2004. Smoothed Particle Hydrodynamics: A Meshfree Particle Method. Computational Mechanics. 33(6), 491-491.

López D., Marivela R., Garrote L., 2010. Smoothed Particle Hydrodynamics Model Applied to Hydraulic Structure: A Hydraulic Jump Test Case. Journal of Hydraulic Research. 48(1), 142-158.

Monaghan J.J., 1988. An Introduction to SPH. Computer Physics Communications. 48(1), 89-96.

Monaghan J.J., 1992. Smoothed Particle Hydrodynamics. Annual Review of Astronomy and Astrophysics. 30, 543-574.

Monaghan J.J., 1994. Simulating Free Surface Flows with SPH. Journal of Computational Physics. 110, 339-406.

Monaghan J.J., Kos A., 1999. Solitary Waves on a Cretan Beach. Journal of Waterway, Port, Coastal, and Ocean Engineering. 125(3), 145–154.

Molteni D., Colagrossi A., 2009. A Simple Procedure to Improve the Pressure Evaluation in Hydrodynamic Context using the SPH. Computer Physics Communications. 180(6), 861–872.

Politano M., Carrica P.M., Weber L., 2009. A Multiphase Model for the Hydrodynamics and Total Dissolved Gas in Tailraces. International Journal of Multiphase Flow. 11(35), 1036-1050.

Sun J.W., Liang S.X., Sun Z.C., Zhao X.Z., 2010. A Two-phase Simulation of Wave Impact on a Horizontal Deck Based on SPH Method. Journal of Marine Science and Application. 9(4), 372-378.

Zhou Z.Q., Kat J.O.D., Buchner B., 1999. A Nonlinear 3-D Approach to Simulate Green Water Dynamics on Deck. Proceedings 7th International Conference on Numerical Ship Hydrodynamics, Nantes, France.