Air-Water Two Phase Flow Simulation Using Smoothed Particle Hydrodynamics

Fanfan Sun, Mingyi Tan, Jing Tang Xing
University of Southampton
FSI research group
Southampton, UK
ffs1g09@soton.ac.uk

***Abstract*—a new method is proposed for air-water two phase flow simulation using Smoothed Particle Hydrodynamics method: the two different fluid phases are treated separately within the same time step. Air is solved using weakly compressible SPH and water is solved using incompressible SPH. No special treatment is required for the interface. It is the first time these two algorithms are combined together. The dam-break case with air-water two phase fluids is used to demonstrate the performance of the proposed algorithm. Results obtained from single phase flow and air-water two phase flow simulations are compared. It is shown that the consideration of air does not change the water movement significantly.**

I. Introduction

Smoothed particle hydrodynamics (SPH) method is a fully Lagrangian mesh free method first developed for Astrophysics in 1970s [1, 2]. It has been extended into large deformation problems such as free surface flows [3]. For incompressible flows, there are generally two ways to impose incompressibility: one is to assume the fluid to be slightly compressible with a large sound speed to have a small Mach number ensuring the density fluctuation within 1% [3-5]., which is known as weakly compressible SPH (WCSPH); the other one is called truly incompressible SPH (ISPH) in which Poisson's equation needs to be solved for pressure [6-8] . The algorithm in WCPSH is easier to be implemented but very small time step is required and the pressure values obtained are not accurate enough. Pressure values obtained from ISPH method are more accurate and a relatively large time step can be used. Hence, ISPH is preferred for incompressible flows especially for the problems in marine and violent sea environment.

The main aim of this paper is to develop a method to investigate the effect of the entrapped air in water. In marine and coastal engineering field, violent fluid-structure interactions can lead to air entrapment. Simplifying these problems as incompressible fluid interacting with a solid may introduce numerical error. The air phase may have a large influence on the water flow evolution and on subsequent loads on structures [9, 10]. Therefore, in this paper the application of the SPH method on incompressible fluid is extended to two phase flows involving air and water. A variety of methods has been developed to study the multi-phase flows which can be reviewed in [11, 12]. Those methods are classified into two main approaches: one is the Eulerian approach; the other is the Lagrangian approach. The advantage of SPH for multi-phase flow is that each phase of fluid follows its Lagrangian motion hence the material interface are represented self-adaptively without the need for complex interface-capturing or front-tracking algorithms [13].

In the early days, the standard SPH formulation was applied to multi-phase flows with small density differences between the considered media [14, 15]. In the later development, Richie and Thomas [16] assumed that the pressure is constant across the smoothing kernel and the particle density is calculated from the equation of state to handle large density gradients. To avoid the numerical instability caused by large density differences across the interface, Colagrossi and Landrini [9] modified the particle approximation form of the pressure gradient; Hu and Adams [5] developed another particle approximation form in which the neighbouring particles only contribute to the specific volume but not density; Grenier and his colleagues [10, 17] developed a similar formation by using a Shepard kernel instead of the conventional kernel. These methods, however, are based on the WCSPH formulation which is lack of accuracy in liquid pressure calculations. An incompressible SPH multiphase method developed by Hu and Adams [18] treated all the involved fluids as incompressible without the considering the compressible nature of gas. It is ideal to retain the basic nature of the fluid, which means that the liquid should be treated as an incompressible fluid and gas should be treated as compressible. Therefore, two separate algorithms will be used for the different fluids respectively in the same time step. Consequently, the density of the liquid phase can be maintained as constant by using ISPH algorithm. The advantage of using constant density for the liquid phase has been explained in a previous paper [19] and the improvement of boundary treatments for this incompressible SPH method has also been achieved.

Poisson's equation will be solved for the pressure of liquid phase while the equation of state will be used for air phase. And the density of air particles will be updated at each time step. The air-water interface is captured without need for special treatment. The effect of surface tension is not considered.

I. GOVENING EQUATIONS FOR AIR-WATER TWO PHASE FLOW

In SPH methods, fluid field is governed by the classical Navier-Stokes equations including continuity equation and momentum conservative equation. In multi-phase flow, each fluid phase follows this principle correspondingly.

$$\frac{1}{\rho}\frac{d\rho}{dt}+\nabla\cdot\mathbf{v}=0 \tag{1}$$

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

where ρ is the density, $\mathbf{v}$ is the velocity, P is the pressure, $\mathbf{g}$ is the body force (gravity force in this case) and $\boldsymbol{\tau}$ is the viscous stress tensor. For incompressible fluids, the mass density takes a constant, so the continuity equation can be simplified as

$$\nabla\cdot\mathbf{v}=0 \tag{3}$$

The momentum conservation equation is the same for both fluid phases. According to Shao and Lo [7], the viscous stress in the momentum equation can be replaced by divergence of velocity as

$$\frac{1}{\rho}\nabla\cdot\boldsymbol{\tau}=\frac{\mu_{eff}}{\rho}\nabla^2\mathbf{v} \tag{4}$$

where μ_{eff} is the effective viscous coefficient.

In incompressible fluid, the truly incompressible SPH method is applied. The timing algorithm is divided in two steps [20]. The first step calculates an intermediate velocity based on the effect of body force and viscosity.

$$\frac{\mathbf{v}^{n+\frac{1}{2}}-\mathbf{v}^n}{\Delta t}=\mathbf{g}+\frac{1}{\rho}\nabla\cdot\boldsymbol{\tau}^n \tag{5}$$

And this intermediate velocity will be modified by the pressure in the second step.

$$\frac{\mathbf{v}^{n+1}-\mathbf{v}^{n+\frac{1}{2}}}{\Delta t}=-\frac{1}{\rho}\nabla P^{n+1} \tag{6}$$

Applying (3) to (6), the Poisson's equation can be derived

$$\frac{1}{\rho}\Delta P^{n+1}=\frac{1}{\Delta t}\nabla\cdot\mathbf{v}^{n+\frac{1}{2}} \tag{7}$$

This equation will be approximated in SPH form so the pressure at the next time step can be computed. Based on the velocity of the next time step obtained from equation (6) the position of each particle can be updated directly.

For compressible flow, weakly compressible SPH is used so equation (7) is not needed. The pressure can be calculated based on the equation of state

$$P=\frac{c_0^2\rho_0}{\gamma}\left(\left(\frac{\rho}{\rho_0}\right)^{\gamma}-1\right) \tag{8}$$

Where c_0 is the speed of sound in the air, ρ_0 is the reference density, γ is normally chosen to be 1.4 for gas [9]. These parameters and the speed of sound are chosen to limit the threshold of the admissible density variation to 1% [13]

II. SPH FORMULATION

The SPH formulation is based on the theory of integral interplant that uses kernel function to approximate delta function. A physical property is obtained by the interpolation between a set of point inside a certain area. These points known as particles carry all the properties the fluid has, such as mass and velocity. The basic idea of this method is to approximate a function A(r) into a particle form as

$$A(\mathbf{r}_a)=\sum_{b=1}^{N}m_b\frac{A_b}{\rho_b}W(|\mathbf{r}_a-\mathbf{r}_b|,h) \tag{9}$$

Where h is the smoothing length, N is the number of neighbouring particles.

A. Density Evaluation

In SPH method, the fluid is represented by particles. All the quantities involved in the governing equations are expressed in particle approximation form. Usually, fixed particle mass are used so the mass conservative law is always satisfied. The density of particle can be calculated based on standard SPH approach which is expressed as summation interplant using a kernel function W with smoothing length h [4].

$$\rho_a=\sum_{b=1}^{N}m_bW_{ab} \tag{10}$$

Density calculated based on this form is not right near the boundaries where there are not enough neighbouring particles in the computation domain. Hence, a modified form proposed in [21] can be used instead of the standard form to improve the accuracy

$$\rho_a = \frac{\sum_{b=1}^{N} m_b W_{ab}}{\sum_{b=1}^{N} \left(\frac{m_b}{\rho_b} \right) W_{ab}} \tag{11}$$

Another most often used form for updating density [3] is

$$\frac{d\rho_a}{dt} = \sum_{b=1}^{N} m_b v_{ab\cdot} \cdot \nabla W_{ab} \tag{12}$$

The derivation of this form is based on the continuity equation but with a slight modification as shown below

$$\frac{d\rho}{dt} = -\rho \nabla \cdot \mathbf{v} = -(\nabla \cdot (\rho \mathbf{v}) - \mathbf{v} \cdot \nabla \rho) \tag{13}$$

$$-\rho \nabla \cdot \mathbf{v} = -(\nabla \cdot (\rho \mathbf{v}) - \mathbf{v} \cdot \nabla \rho) \tag{14}$$

Apply SPH approximation rule to each item on the RHS so (12) can be derived. During the process, it is assumed that the divergence of density exists for the material. However, in the interface of multi-phase flow, the density of the material is not continues. Computational instability will happen by using (12). Alternatively, another form derived based on the original continuity equation will not have this problem

$$\frac{d\rho_a}{dt} = \rho_a \sum_{b=1}^{N} \frac{m_b}{\rho_b} \mathbf{v}_{ab} \cdot \nabla W_{ab} \tag{15}$$

For a given particle, instability caused by large particle mass difference is eliminated.

B. Velocity Evaluation

According to momentum equation, pressure gradient, body force and viscous force gradient are the three factors which influence velocity. The gradient of viscous force can be expressed in SPH form [7] as

$$\left(\frac{1}{\rho} \nabla \cdot \boldsymbol{\tau} \right)_a = \sum_{b=1}^{N} \frac{4 m_b \mu_a \mu_b}{(\rho_a + \rho_b)^2 (\mu_a + \mu_b)} \frac{\mathbf{r}_{ab} \cdot (\mathbf{v}_a - \mathbf{v}_b) \nabla W}{|\mathbf{r}_{ab}|^2 + \eta^2} \tag{16}$$

The standard SPH form of pressure gradient [3] is

$$\frac{1}{\rho_a} \nabla P_a = \sum_{b=1}^{N} \left(\frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2} \right) \nabla W_{ab} m_b \tag{17}$$

This expression conserves angular momentum and linear momentum because of the symmetric form. However, same as the derivation of (12), the derivation of (17) shown below

$$\frac{1}{\rho} \nabla P = \nabla \left(\frac{P}{\rho} \right) + \frac{P}{\rho^2} (\nabla \rho) \tag{18}$$

is also based on the assumption of continuous density of the material. For multi-phase flow with large density difference, using this form will introduce unphysical instability on the interface [9]. Hence, another form is preferred

$$\frac{1}{\rho_a} \nabla P_a = \frac{1}{\rho_a} \sum_{b=1}^{N} (P_a + P_b) \nabla W_{ab} \frac{m_b}{\rho_b} \tag{19}$$

The effect from density difference is eliminated in this form. For air, pressure is calculated by the equation of state (8). For liquid, the pressure is calculated based on the Poisson's equation (7) which can be expressed in particle form as

$$\sum_{b=1}^{N} m_b \frac{P_{ab}^{n+1} r_{ab}^{n}}{|r_{ab}^2| + \eta^2} \nabla W_{ab} = -\frac{\rho}{2\Delta t} \sum_{b=1}^{N} m_b \mathbf{v}_{ab}^{n+\frac{1}{2}} \cdot \nabla W_{ab} \tag{20}$$

It can be solved by BI-CGSTAB algorithm. The pressure values obtained will be used to calculate the velocities and positions in the next time step. After that, the density and pressure values of air phase can be calculated by using (15) and (8) respectively for based on the new velocity and position.

C. Interface

For free surface flow, the pressure on the free surface is usually assumed to be zero. In the case of multi-phase flow, the pressure on the interface of liquid is assigned to be the same values as the interacting air pressure. No other special treatment is required for the interface.

D. Neighbouring particle searching algorithm

A modified cell-linked list approach is used to search for the neighbouring particles. First, the whole space domain is divided into cells of size kh, then the particles are stored according to the cell they belong to [22]. For a given particle in a particular cell, only the adjacent cells (up to 9 for two dimensional cases) will be involved for the neighbouring particle searching. In the conventional cell-linked list approach, the list of the adjoining cells is recorded. According to the symmetric distribution of the cells, only the cells with lower index plus the particular cell itself are considered in the searching algorithm. Hence, only half of the adjoining cells are searched so that the computation time is reduced. However, it is more convenient if the neighbouring particle list is stored instead of the adjacent cell list. Therefore, a modified searching algorithm is used in this paper.

Since the smoothing length is the same, the list relation for particles is symmetric and this implies that only the particles with smaller index need to be checked. The particle list will be recorded for both particles. In another word, for a pair of interacting particles, only one of them is checked to identify the neighbourhood so in total only half number of particles needs to be check. This algorithm can be combined with the conventional cell-linked list approach. Further study about computation time cost will be done in the future.

III. COURTANT NUMBER CONDITION

To maintain numerical stability a Courant-Friedrichs-Lewy (CFL) time step restriction is employed.

$$\Delta t \leq 0.1\frac{h}{\mathbf{v}_{\max}} \tag{21}$$

where h is the initial particle spacing and $\mathbf{v}_{\max}$ is the maximum particle velocity in the computation. The factor 0.1 ensures that the particle moves only a fraction (in this case 0.1) of the particle spacing per time step. Another constraint is based on the viscous term [6]

$$\Delta t \leq 0.125\frac{h^2}{\mu_{eff}/\rho} \tag{22}$$

where μ_{eff} is the effective viscosity. The allowable time-step should satisfy both of the above criteria.

IV. EXAMPLES

A dam-breaking case is studied to test the proposed method. Both single phase with water and multi-phase with air and water are simulated and the results are compared. For single phase simulation, incompressible SPH is applied and velocity is calculated based on (5) and (6). Equation (16) is used to compute the effect from viscous force and (20) is used to obtain the values of pressure. These values are substituted into (19). So far, all the items in (5) and (6) are known hence the velocity can be updated and then the position can be updated directly afterward. Water patterns at different time instances are shown in Fig. 1 to Fig. 3.

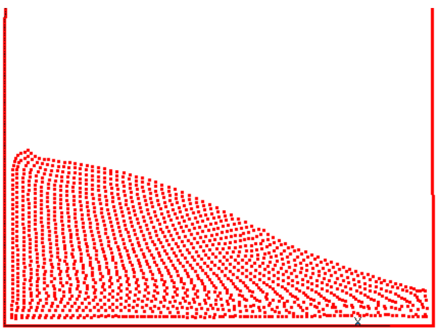

Figure 1. Single phase dam-breaking at time 0.13s

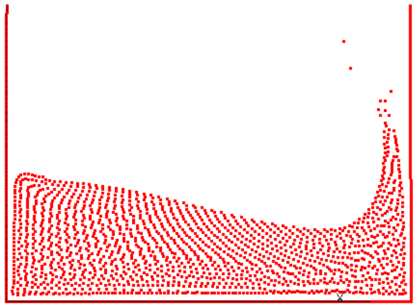

Figure 2. Single phase dam-breaking at time 0.2s

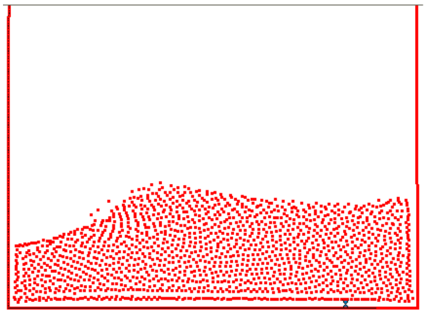

Figure 3. Single phase dam-breaking at time 0.5s

To simulate multi-phase flow, as proposed in this paper, water will be solved with incompressible SPH and air will be solved by weakly compressible SPH. The parameters in equation (8) are decided according to the compressibility required, as suggested in [9] γ =1.4 is used in this paper.

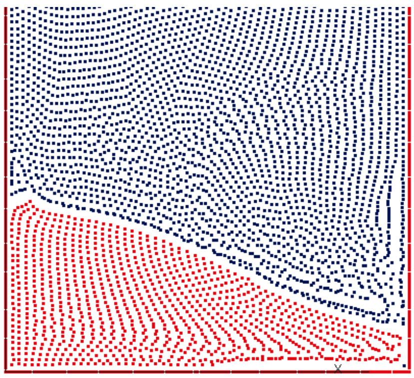

Figure 4. Multi-phase dam-breaking at time 0.13s

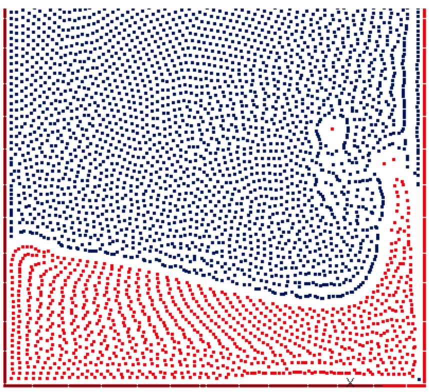

Figure 5. Multi-phase dam-breaking at time 0.2s

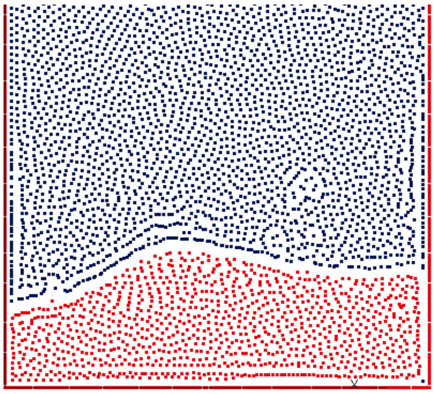

Figure 6. Multi-phase dam-breaking at time 0.5s

From the pictures shown above, wave patterns at different time with multi-phase simulation and single phase simulation are similar. There is no obvious difference found by considering the air phase above the water.

The positions of the leading edge with time being obtained from multi-phase flow are compared with experimental data [23] as well as the results obtained based on WSPH method with different kernel functions as shown in Fig. 7.

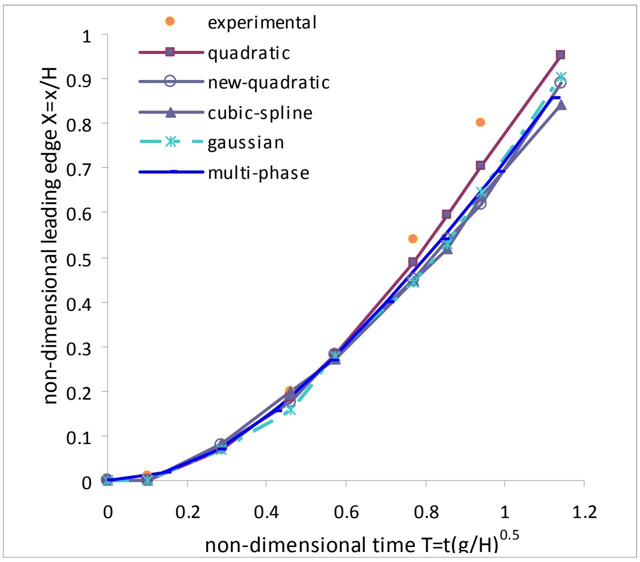

Figure 7. Positions of the leading edge with time going based on different methods

Result obtained with quadratic kernel function is slightly better than the others. Multi-phase consideration does not change the results significantly. However, these data are colleted before the impacting. Hence, it is supposed to be similar for multi-phase and single phase simulation.

According to practical experiences, increasing the pressure of the air should slow down the moment of the water. Since the pressure of the compressible fluid is calculated with the equation of state, and the parameter of this equation is proportional to the density, increasing the initial density of the compressible fluid will increase the initial pressure.

1). Assign air density =10(kg/m^2), t=0.13s

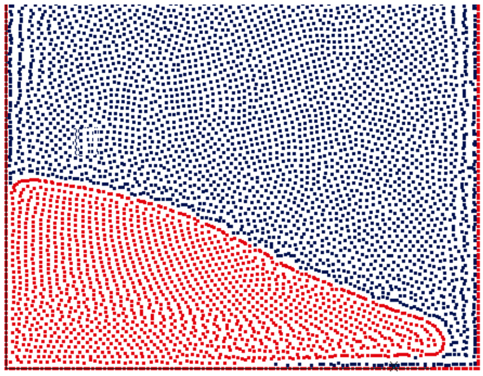

Figure 8. Multi-phase dam-breaking with air density =10(kg/m^2) at time 0.13s

2). Assign air density=100(kg/m^2), t=0.13s

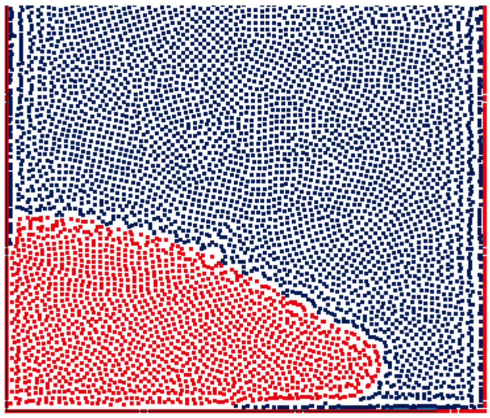

Figure 9. Multi-phase dam-breaking with air density =100(kg/m^2) at time 0.13s

3). Assign air density =1000(kg/m^2).0, t=0.13s;

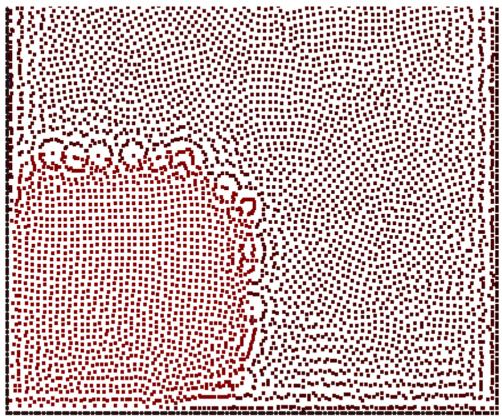

Figure 10. Multi-phase dam-breaking with air density =1000(kg/m^2) at time 0.13s

Comparing Fig. 8 to Fig. 10 with Fig. 4, it is obvious that the flowing movement of water is influenced by the density of the air seriously. Increasing air density will slow down the water movement. In the case when air and water have the same density, water can hardly flow down. But because of the compressibility of air, the water column is deformed under the pressure. This agrees with our practical experiences, which gives a qualitative validation of the proposed new approach.

V. CONCLUSION

A combined algorithm is proposed for solving air-water two phase flows. Incompressible SPH is used for the liquid and weakly compressible SPH is used for air. In other words, the pressure of liquid is calculated by solving a Poisson's equation but for air the equation of state is used. It is clearly explained that the standard SPH form of density and pressure gradient derived based on the assumption of continuous density of the material can not be used for multi-phase flow especially for the cases when the density difference is large. An alternative SPH form without the assumption of continuous density can be used.

A dam-breaking case is studied as a demonstration of the performance of the proposed method. The results obtained based on single phase flow and multi-phase flow simulations are compared. It is shown that a two phase flow consideration does not change the water movement significantly which is reasonable. The simulations for the cases with assigned air pressures / densities provide a qualitative validation of the proposed method. Further investigation of impacting pressure should be carried out in the future.

REFERENCES

[1] L.B. Lucy, "Numerical Approach to testing the fisson hypothesis," Astronomical Journal, 1977,vol. 82, pp. 1013-1024.

[2] R.A. Gingold and J.J. Monaghan, "Smoothed Particle Hydrodynamics: Theory and Application to Non-Spherical stars," Monthly Notices of the Royal Astronomical Society, 1977, vol. 181, pp. 375-389.

[3] J.J. Monaghan, "Simulating free surface flows with SPH," J. Comput. Phys., 1994, vlo. 110, pp. 399-406.

[4] J.P. Morris, P.J. Fox and Y. Zhu, "Modeling low reynolds number incompressible flows using SPH," J. Comput. Phys., 1997, vol. 136, pp. 214-226.

[5] X.Y. Hu and N.A. Adams, "A multi-phase SPH method for macroscopic and mesoscopic flows," Journal of Computational Physics, 2006, vol 213, pp. 844-861.

[6] S.J. Cummins and M. Rudman, "An SPH projection method," J. Comput. Phys., 1999, vol. 152, pp. 584-607.

[7] S. Shao and E.Y.M. Lo, "Incompressible SPH method for simulating Newtonian and non-Newtonian flows with a free surface," Advances in Water Resources, 2003, vol. 26(7), pp. 787-800.

[8] J. Pozorski and A. Wawrenczuk, "SPH computation of incompressible viscous flows," Journal of Theoretical Applied Mechanics, 2002, vol. 40, pp. 917.

[9] A. Colagrossi and M. Landrini, "Numerical simulation of interfacial flows by smoothed particle hydrodynamics," Journal of Computational Physics, 2003, vol.191(2), pp. 448-475.

[10] N. Grenier and D.L. Touze, "An improved SPH method for multi-phase simulations," Proceedings of the 8nd international conference on hydrodynamics, 2008, vol. 11, pp. 2a-01.

[11] R. Scardovelli and S. Zaleski, "Direct numerical simulation of free-surface and interfacial flow," Ann. Rev. Fluid Mech., 1999, vol. 31, pp. 567-603.

[12] J.A. Sethian and P. Smereka, "Level-set methods for fluid interfaces," Ann. Rev. Fluid Mech., 2003, vol. 35, pp. 341-372.

[13] S. Adami, X.Y. Hu, and N.A. Adams, "A new surface-tension formulation for multi-phase SPH using a reproducing divergence approximation," Journal of Computational Physics, 2010, vol. 229, pp. 5011-5021.

[14] J.J. Monaghan, R.A.F. Cas, A.M. Kos and M. Hallworth, "Gravity currents descending a ramp in a stratified tank," Journal of Fluid Mechanics, 1999, vol. 379, pp. 39-69.

[15] J.J. Monaghan and A. Kocharyan, "SPH simulation of multi-phase flow," Computer Physics Communications, 1995, vol. 87(1-2), pp. 225-235.

[16] B.W. Ritchie and P.A. Thomas, "Multiphase smoothed-particle hydrodynamics," Mon.Not.R.Astron.Soc, 2001, vol. 323, pp. 743-756.

[17] N. Grenier, M. Antuono, A. Colagrossi, A.Le Touze and B. Alessandrini, "An hamiltonian interface SPH formulation for multi-fluid and free surface flows," Journal of Computational Physics, 2009, vol. 228, pp. 8380-8393.

[18] X.Y. Hu and N.A. Adams, "An incompressible multi-phase SPH method," J. Comput. Phys., 2007, vol. 227(1), pp. 264-278.

[19] F. Sun, M. Tan and T.J. "Xing, Investigations of boundary treatments in incompressible smoothed particle hydrodynamics for fluid-structure interactions," Proceedings of the 2nd international conference on fluid mechanics and heat and mass transfer 2011, pp. 92-97.

[20] E.S. Lee, D. Violeau, and R. Issa, "Application of weakly compressible and truly incompressible SPH to 3-d water collapse in waterworks," Journal of Hydraulic Research, 2010. vol. 48, pp. 50-60.

[21] M. Rodriguez-Paz and J. Bonet, "A corrected smoothed particle hydrodynamics formulation of the shallow-water equations," Computers & Structures, 2005. vol. 83, pp. 1396-1410.

[22] J.M. Dominguez, A.J.C. Crespo, M. Gomex-Gesteira and J.C. marongiu, "Neighbour lists in smoothed particle hydrodynamics," International Journal for Numerical Methods in Fluids, 2011. vol. 67, pp. 2026-2042.

[23] J. Martin and W. Moyce, "An experimental study of the collapse of liquid columns on a rigid horizontal plane," 1952.