Level Set and Volume of Fluid Coupled Method for Violent Two-phase Flows

Songtao Chen, Weiwen Zhao and Decheng Wan*

Computational Marine Hydrodynamics Lab (CMHL), School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China

* Corresponding author: dcwan@sjtu.edu.cn

ABSTRACT

In this paper, a fully coupled level set and volume of fluid (CLSVOF) method for unstructured meshes is implemented in OpenFOAM. The interface is reconstructed by the piecewise-linear interface calculation (PLIC) method, where the interface normal is given by the level set method. A reinitialization algorithm suitable for unstructured meshes is employed to maintain the signed distance property of the level set function. Besides, the surface tension is instead calculated by the continuous level set function. The CLSVOF method is validated on two-dimensional and three-dimensional dam break benchmark cases and achieves good agreements with experiments. Furthermore, the comparison with the algebraic VOF method demonstrates that the CLSVOF method can obtain a sharper interface and significantly reduce spurious velocities, both of which are helpful for simulating violent two-phase flows.

Keywords: CLSVOF; PLIC; level set; unstructured meshes; two-phase flows; OpenFOAM.

1. INTRODUCTION

The interface-tracking/interface-capturing method is of great importance for two-phase flow simulations. Among them, the most common ones in marine hydrodynamics are the level set method and the volume of fluid (VOF) method. The level set method usually uses the continuous signed distance function for advection, where the zero iso-surface represents the interface. It can accurately calculate the interface curvature as well as the surface tension. However, the level set method has the problem of mass conservation, which has always been the focus of research. The VOF method is conservative, so it has been adopted by many commercial software and open-source solvers, such as STAR-CCM+ and OpenFOAM. Nevertheless, the VOF method suffers from the smearing of the interface, making it difficult to provide a sharp interface. Also, the discontinuous volume fraction function is less accurate in calculating surface tension. In this regard, many scholars have proposed their improved methods and implemented them in OpenFOAM (Vukcevic and Jasak, 2014; Ferro *et al*., 2022; Weller, 2008; Roenby *et al*., 2016; Scheufler and Roenby, 2019).

In order to combine the advantages of the level set method and the VOF method, Sussman and Puckett (2000) proposed the coupled level set and volume of fluid (CLSVOF) method. The CLSVOF method not only preserves mass conservation well, but also provides a sharp interface. Furthermore, it can give more accurate information about the interface, such as the normal vector and curvature. According to the coupling strategy, the CLSVOF method can be divided into one-way coupling and two-way coupling. For one-way coupling, Kunkelmann and Stephan (2010) used the updated volume fraction function at each time step to estimate the level set function instead of solving the transport equation. On this basis, Albadawi *et al*. (2013) proposed a simple coupled volume of fluid with level set (S-CLSVOF) method, focusing on improving the calculation of surface tension. Through the comparison with the interface compression algebraic VOF method (Weller, 2008), the S-CLSVOF method shows a significant improvement in the study of bubble dynamics, where surface tension plays an important role. However, for large-scale flows in marine hydrodynamics, the S-CLSVOF method obviously cannot achieve much improvement. For two-way coupling, both the level set function and the volume fraction function are updated by solving the transport equation and then corrected for each other. Wang *et al*. (2009) implemented the CLSVOF method in their in-house solver CFDship-Iowa v6, which is a Cartesian grid solver. The level set equation is solved using the fifth-order HJ-WENO scheme, and the volume fraction function is updated using the piecewise-linear interface calculation (PLIC) scheme and the Lagrangian interface propagation scheme. Dianat *et al.* (2017) implemented the CLSVOF method in the arbitrary polyhedral framework of OpenFOAM. For unstructured meshes, an iterative clipping and capping algorithm for interface reconstruction and a reinitialization algorithm are given.

In the present study, a two-way coupling CLSVOF method for unstructured meshes is implemented in OpenFOAM. The primary objectives are to (i) validate the accuracy and robustness of the CLSVOF method in violent two-phase flows and (ii) demonstrate the advantages of the CLSVOF method by comparison. The remainder of this paper is organized as follows. First, the numerical methods are introduced with emphasis on the CLSVOF method. Then, two dam break benchmark cases are used to validate the CLSVOF method. Finally, the main conclusions are drawn.

2. NUMERICAL METHODS

2.1 Governing equations

In the present study, the two-phase flow is described by the incompressible Navier-Stokes equations, which are as follows:

$$\nabla\cdot\mathbf{U} = 0, \tag{1}$$

$$\frac{\partial\rho\mathbf{U}}{\partial t} + \nabla\cdot(\rho\mathbf{U}\mathbf{U}) = -\nabla p_d - \mathbf{g}\cdot\mathbf{x}\nabla\rho + \nabla\cdot(\mu_{eff}\nabla\mathbf{U}) + f_\sigma, \tag{2}$$

where $\mathbf{U}$ is the velocity, ρ is the weighted average density, p_d is the dynamic pressure, $\mathbf{g}$ is the acceleration of gravity, $\mathbf{x}$ is the coordinate vector, μ_{eff} is the effective dynamic viscosity, and f_σ is the surface tension term. In this paper, the laminar model is used throughout the study.

2.2 Interface capturing methods

2.2.1 Algebraic VOF method

For the incompressible two-phase solver interFoam, OpenFOAM provides an algebraic VOF method to capture the interface, the interface compression technique. Its basic idea is to add an artificial compression term to the transport equation for the volume fraction α (Weller, 2008), which can be written as follows:

$$\frac{\partial\alpha}{\partial t} + \nabla\cdot(\alpha\mathbf{U}) + \nabla\cdot(\alpha(1-\alpha)\mathbf{U}_c) = 0, \tag{3}$$

$$\mathbf{U}_c = \min\{c_\alpha|\mathbf{U}|, \max(|\mathbf{U}|)\}\frac{\nabla\alpha}{|\nabla\alpha|}, \tag{4}$$

where $\mathbf{U}_c$ is the compression velocity, and c_α is the compression coefficient, which is 1.0 by default. In addition, the multidimensional universal limiter for explicit solution (MULES) algorithm is employed to ensure the boundedness of the volume fraction.

On the other hand, the surface tension term f_σ is calculated by the Continuum Surface Force (CSF) method:

$$f_\sigma = \sigma\kappa(\alpha)\nabla\alpha, \tag{5}$$

where σ is the surface tension coefficient (0.072 N/m for the air-water interface in the following simulations), and $\kappa(\alpha) = -\nabla\cdot(\nabla\alpha/|\nabla\alpha|)$ is the interface curvature calculated by the volume fraction.

2.2.2 CLSVOF method

To improve accuracy, we implement the CLSVOF method suitable for unstructured meshes in OpenFOAM, based on the previous work by Dianat *et al*. (2017). In our implementation, the transport equation for the volume fraction excludes the artificial compression term:

$$\frac{\partial \alpha}{\partial t} + \nabla\cdot(\alpha \mathbf{U}) = 0. \tag{6}$$

Using Gauss's theorem, the advection term in Eq. 6 can be converted to the discretized surface integral form:

$$\int_V \nabla\cdot(\alpha \mathbf{U})\, dV = \sum_f \alpha_f \mathbf{U}_f \cdot \mathbf{S}_f, \tag{7}$$

where α_f is the face-interpolated volume fraction, $\mathbf{U}_f$ is the face-interpolated velocity, and $\mathbf{S}_f$ is the face area vector. To obtain the precise value of α_f, the PLIC method is adopted, i.e., the interface within each cell is described by a plane:

$$\mathbf{n}\cdot\mathbf{x} + D = 0, \tag{8}$$

where D is a constant, and $\mathbf{n}$ is the plane normal vector. For unstructured meshes, an interface reconstruction method based on tetrahedral decomposition is used to determine D (Skarysz *et al*., 2018).

On the other hand, in order to accurately calculate $\mathbf{n}$, the level set function ϕ is introduced and solved using the transport equation as follows:

$$\frac{\partial \phi}{\partial t} + \nabla\cdot(\phi \mathbf{U}) = 0. \tag{9}$$

After solving Eq. 9, to ensure mass conservation, the level set function is first re-distanced based on the reconstructed interface in the cells containing an interface. Next, to preserve the signed distance property of the level set function, the reinitialization equation proposed by Sussman *et al*. (1994) is solved:

$$\frac{\partial \phi}{\partial \tau} = S(\phi_0)(1 - |\nabla\phi|), \tag{10}$$

$$S(\phi_0) = \frac{\phi_0}{\sqrt{\phi_0^2 + (|\nabla\phi_0|\Delta)^2}}, \tag{11}$$

where τ is the fictitious time step, $\phi_0 = \phi(\tau = 0)$ is the initial condition, $S(\phi_0)$ is the sign function, and $\Delta = \max(\Delta x, \Delta y, \Delta z)$ is the maximum dimension of each cell. In the current implementation, the gradient magnitude $|\nabla\phi|$ for unstructured meshes is calculated based on the method proposed by Dianat *et al*. (2017). Moreover, τ is set to $0.1\times\min(\Delta x, \Delta y, \Delta z)$ and 5 iterations is used to achieve the convergence of $|\nabla\phi| = 1$. Based on the above procedures, the flow chart of the current CLSVOF method is summarized in Fig. 1.

Compared with the VOF method, the surface tension term f_σ is recalculated by the continuous level set function:

$$f_\sigma = \sigma\kappa(\phi)\delta(\phi)\nabla\phi, \tag{12}$$

$$\delta(\phi) = \begin{cases} 0 & |\phi| > \varepsilon \\ \dfrac{1}{2\varepsilon}\left(1 + \cos\left(\dfrac{\pi\phi}{\varepsilon}\right)\right) & |\phi| \le \varepsilon \end{cases}, \tag{13}$$

where $\kappa(\phi) = -\nabla\cdot(\nabla\phi/|\nabla\phi|)$ is the interface curvature calculated by the level set function, $\delta(\phi)$ is the delta function that limits the effect of surface tension near the interface, and $\varepsilon = 1.5\Delta$ is the prescribed interface thickness (Sussman *et al*., 1994).

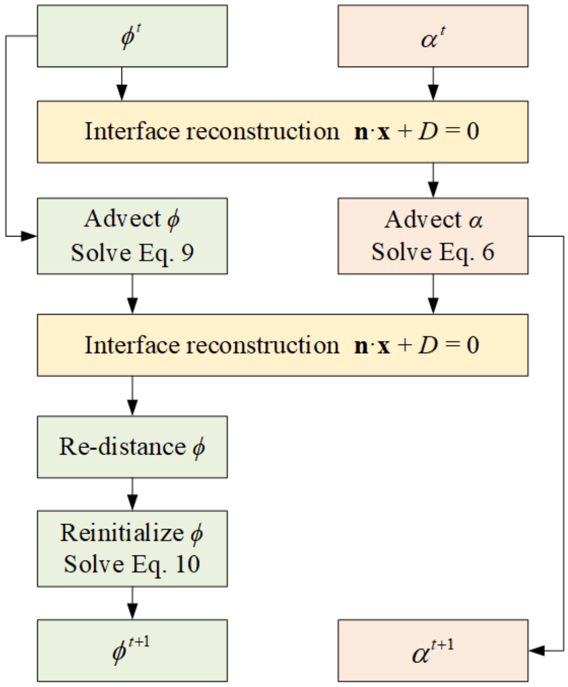


Figure 1. Flow chart of the CLSVOF method**.**

3. RESULTS AND DISCUSSION

In the present study, the current CLSVOF method is validated with two benchmark cases: two-dimensional dam break and three-dimensional dam break with an obstacle, respectively. It should be noted that all simulations are run in parallel. In the first case, the CLSVOF method is also compared with the algebraic VOF method.

3.1 Two-dimensional dam break

In this section, the dam break benchmark case carried out by Lobovský *et al*. (2014) is used to validate the CLSVOF method. The laboratory experiments are conducted in a prismatic tank of 1.61×0.6×0.15 m (length × depth × width) at the Technical University of Madrid (UPM). The dimension of the water column before release is 0.6×0.3 m (length × depth). In addition, the time histories of surface elevation at four locations of interest is extracted using the video recording technique in the experiments.

According to the above experimental setup, a two-dimensional computational domain is used in the following simulations, as shown in Fig. 2. The structured computational mesh consists of 1.61×10^4 cells with a uniform size of 10^{-3} m. The no-slip boundary condition is applied to the bottom and lateral walls, and the Neumann boundary condition is used for the atmosphere. The time step is set to 5×10^{-4} s, satisfying the maximum Courant number is less than 1. Wave probes are located at the same position as the experiments, referred to as H1~H4. As a comparison, the same case is run using the interface compression algebraic VOF method as well.

Fig. 3 first compares the surface elevation between the numerical results and the experimental measurements. The experiments provide the data for two test runs, referred to as Exp 01 and Exp 02. The results of the interface compression method are not shown here because the differences from the CLSVOF method in terms of time history are not significant. Compared with the experiments, the CLSVOF method gives convincing results at all locations except H3. This is due to the formation of a large air pocket near H3 (shown in Fig. 3c), and the *interfaceHeight* utility used to record surface elevation in OpenFOAM cannot give accurate results in the presence of multiple interfaces.

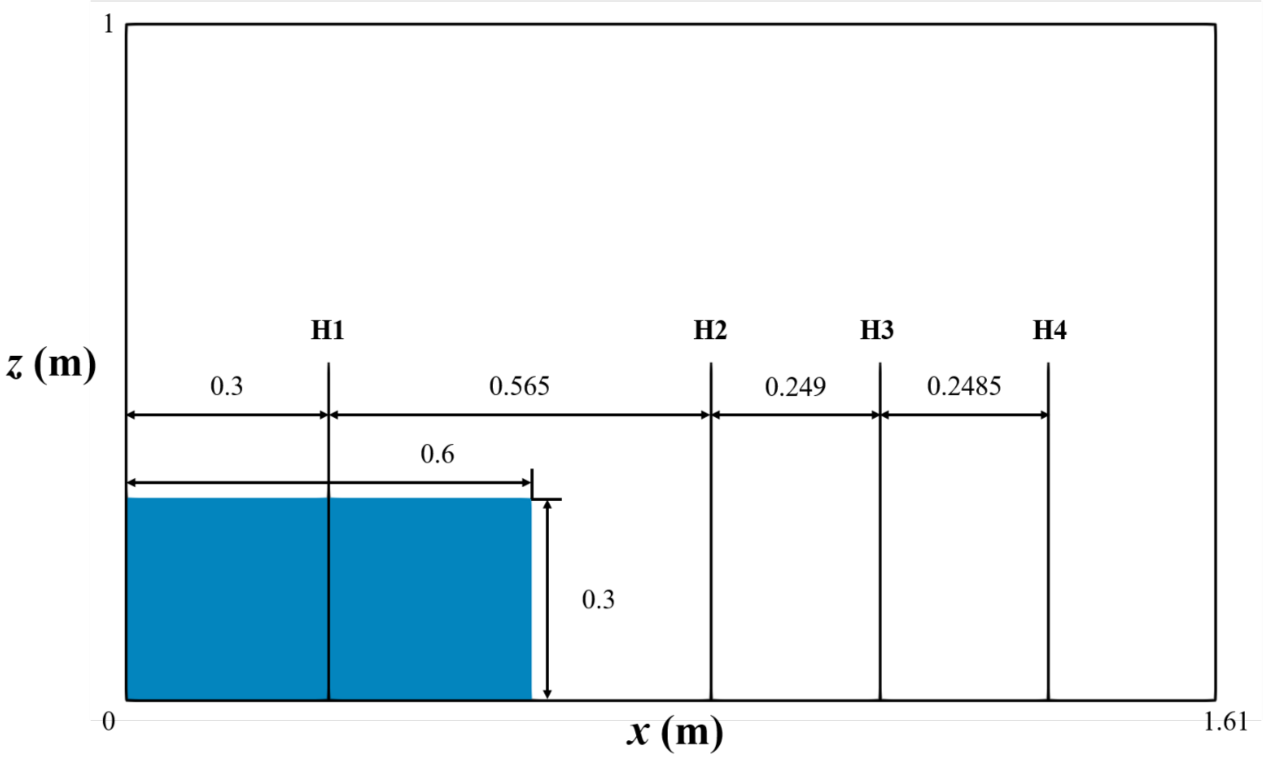
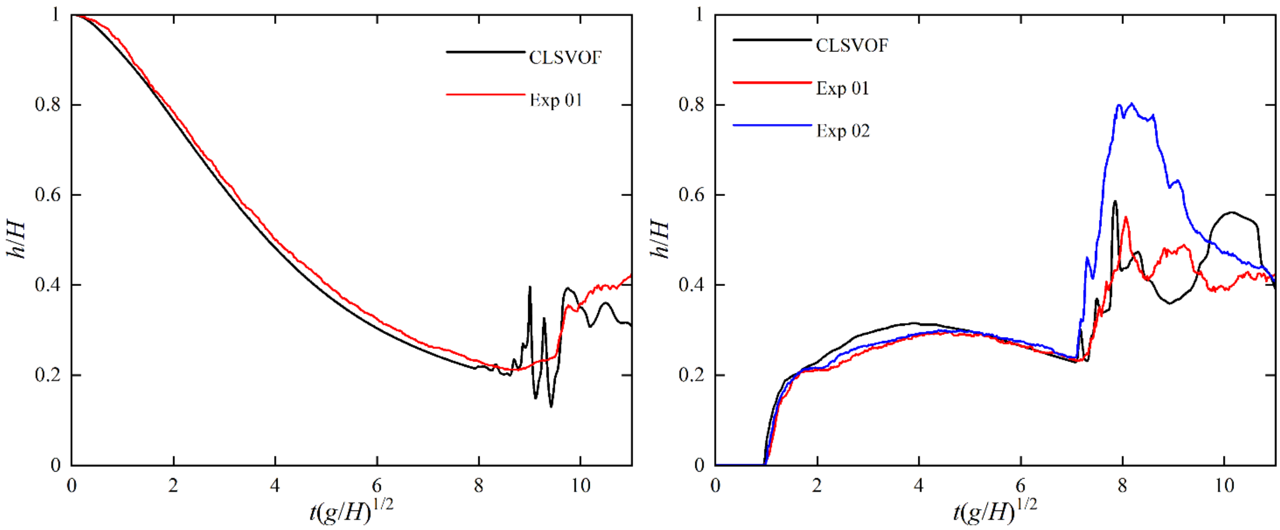
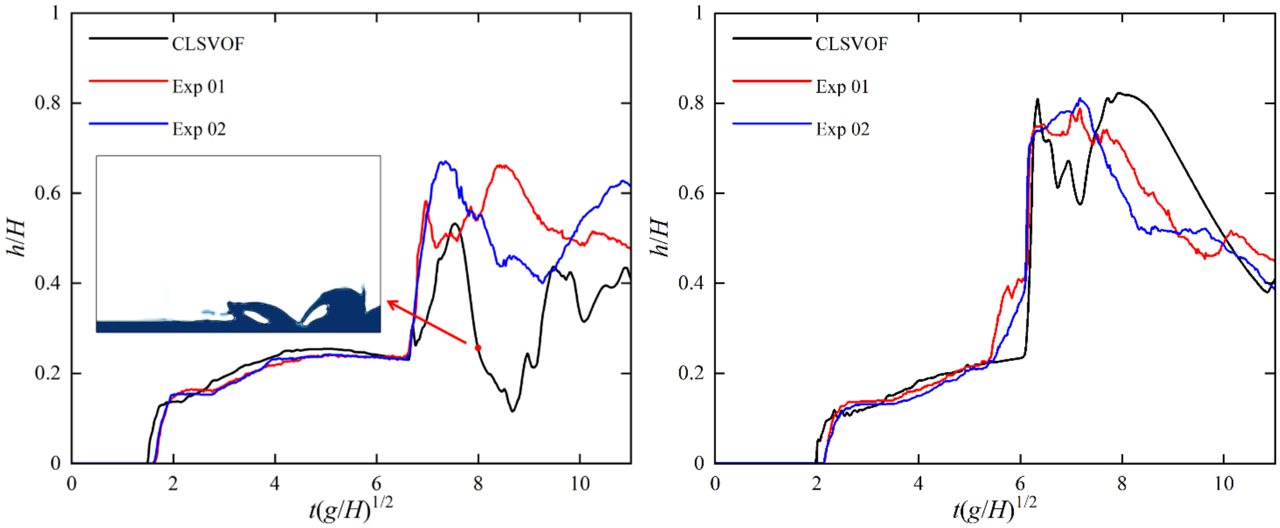


Figure 2. Computational domain and locations of wave probes.



(a) (b)

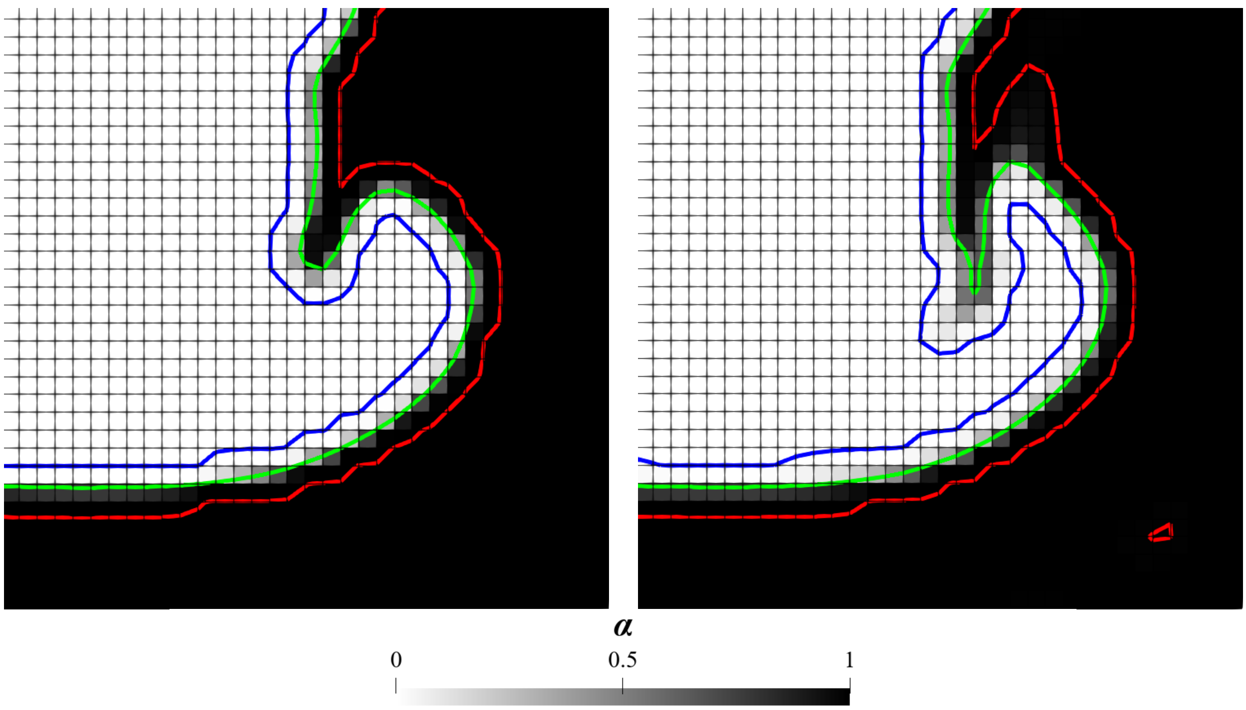


(c)

(d)

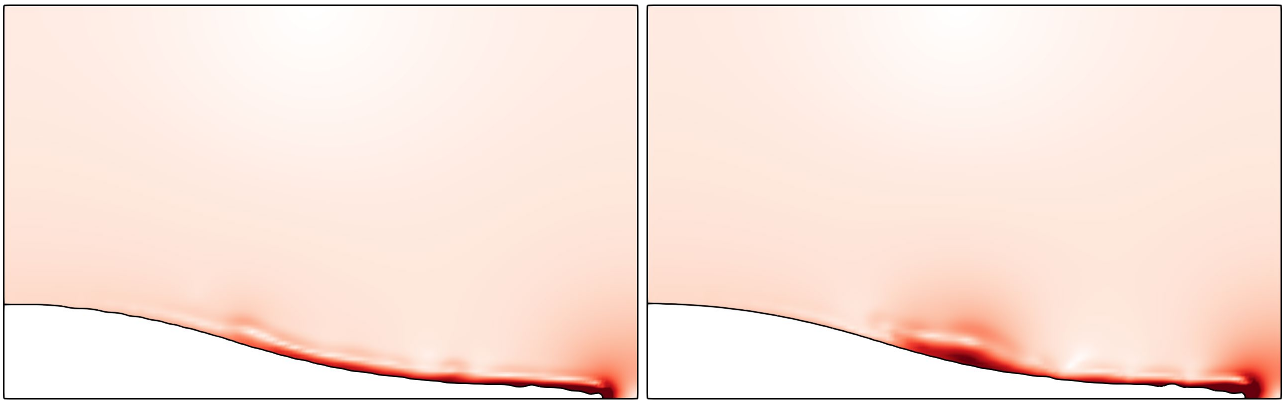
Figure 3. Comparison of surface elevation at (a) H1, (b) H2, (c) H3, and (d) H4.

To demonstrate the advantages of the CLSVOF method, a further comparison of the flow field is presented. Fig. 4 compares the contours of the volume fraction at $t = 1$ s. Among them, the blue, green and red lines represent the iso-surfaces of $\alpha = 0.01$, $\alpha = 0.5$, and $\alpha = 0.99$, respectively. Compared with the interface compression algebraic VOF method, the CLSVOF method can obviously obtain a sharper interface near the water tongue of the plunging wave breaker, i.e., a shorter distance between the blue and red lines. Fig. 5 compares the velocity magnitude of the flow field at two time instants. In the numerical simulations of large density ratio two-phase flows, the spurious velocity is a troublesome problem, especially in the light air phase. Therefore, for clarity, only the air phase is shown below. At $t = 0.4$ s, it can be seen that the interface compression algebraic VOF method suffers more from the spurious air velocities, such as near the wave front. At $t = 1$ s, this problem becomes apparent near the water tongue. At the same time, the velocity field becomes more chaotic in the upper domain. In contrast, the CLSVOF method can significantly reduce these spurious air velocities.

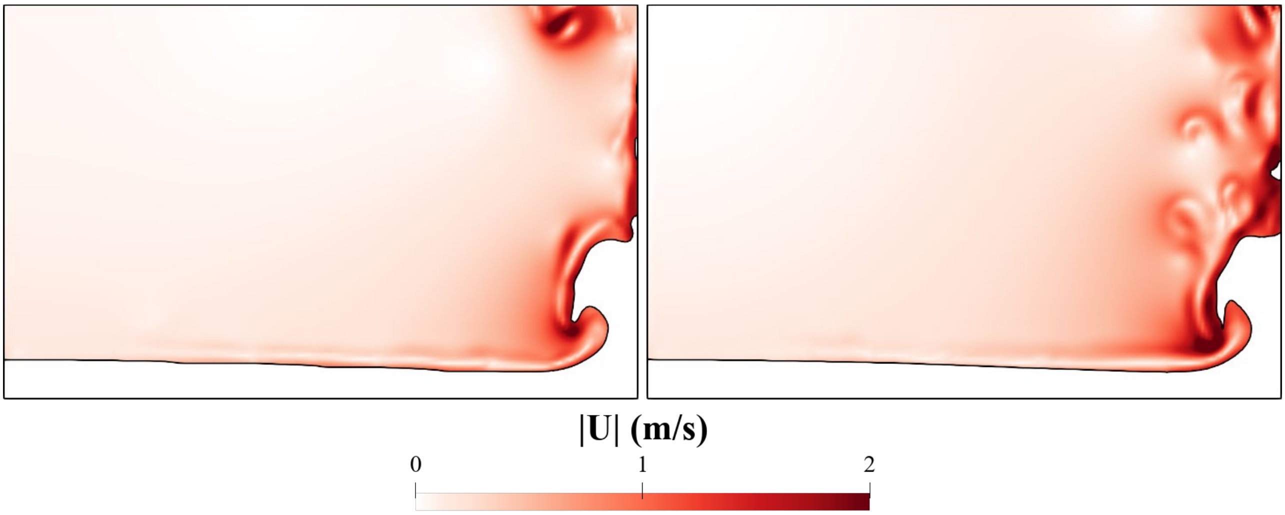


(a) (b)

Figure 4. Comparison of free surface at t = 1 s: (a) CLSVOF and (b) interface compression.



(a)



(b)

Figure 5. Comparison of velocity magnitude at (a) t = 0.4 s and (b) t = 1 s. (The left column is the results of CLSVOF and the right column is the results of interface compression.)

3.2 Three-dimensional dam break with obstacle

To further validate the CLSVOF method in three-dimensional simulations, the benchmark case performed by Kleefsman *et al*. (2005) is simulated in this section. The experiments are carried out in a tank of 3.22×1×1 m (length × depth × width) at the Maritime Research Institute Netherlands (MARIN). In this case, the dimension of the water column before release is 1.228×1×0.55 m (length × width × depth). Furthermore, a rectangular box is placed at the bottom.

The numerical setup is basically consistent with the experiments. Fig. 6 shows the computational domain, which is the same size as the tank. Fig. 7 illustrates the location of the probes, two for surface elevation and four for impact pressure on the box. The structured computational mesh consists of 3.2×10^6 cells with a uniform size of 10^{-2} m. All boundaries are treated as no-slip walls except the top, where the Neumann boundary condition is applied. The time step is set to 5×10^{-4} s, satisfying the maximum Courant number is less than 1.

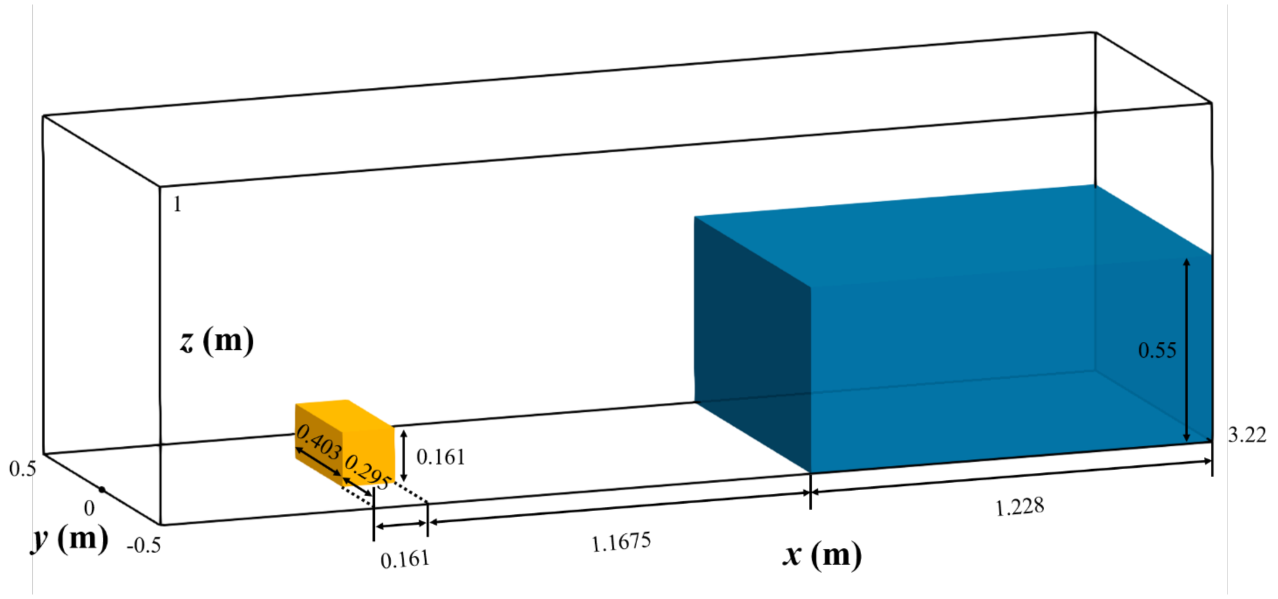


Figure 6. Computational domain.

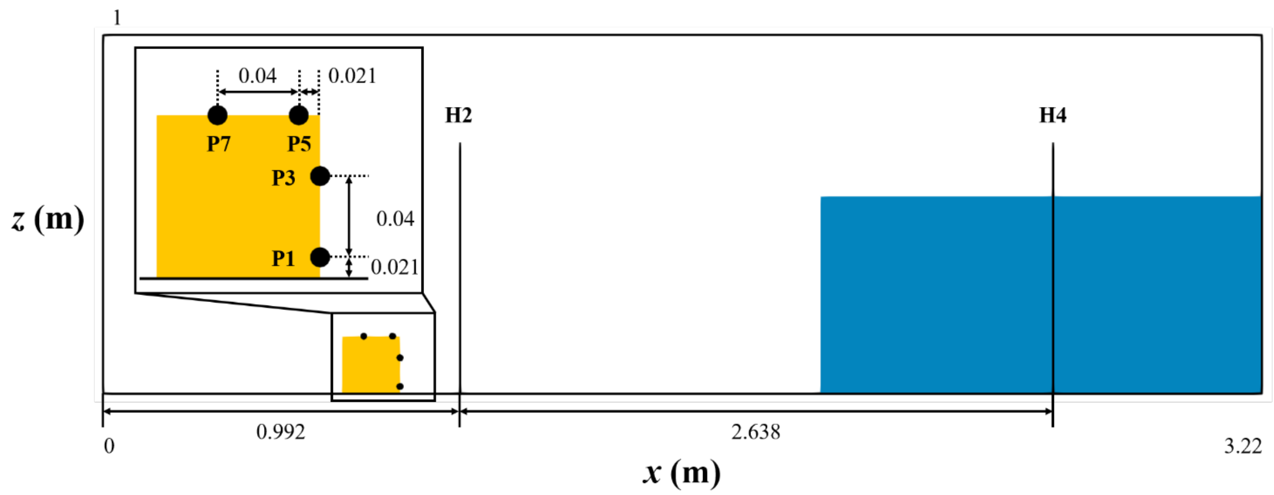
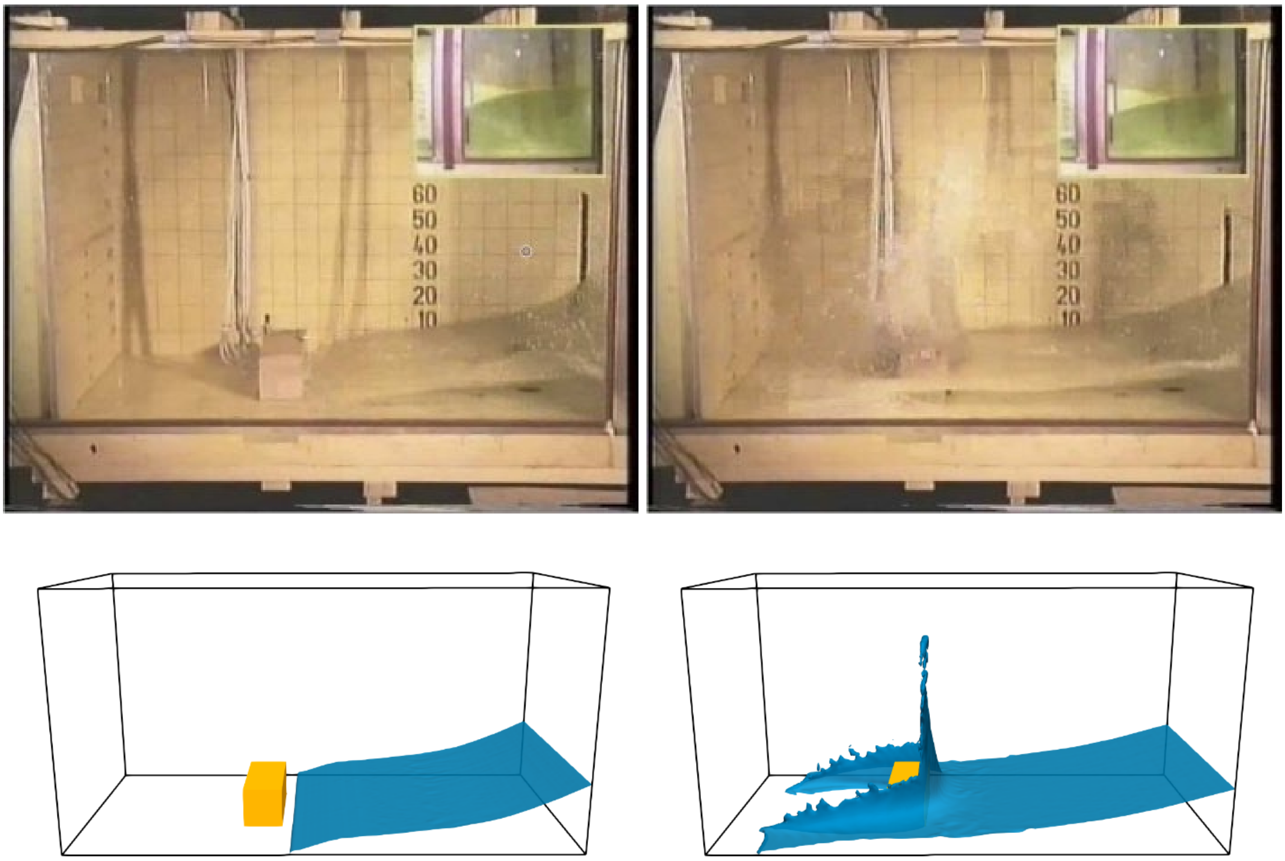


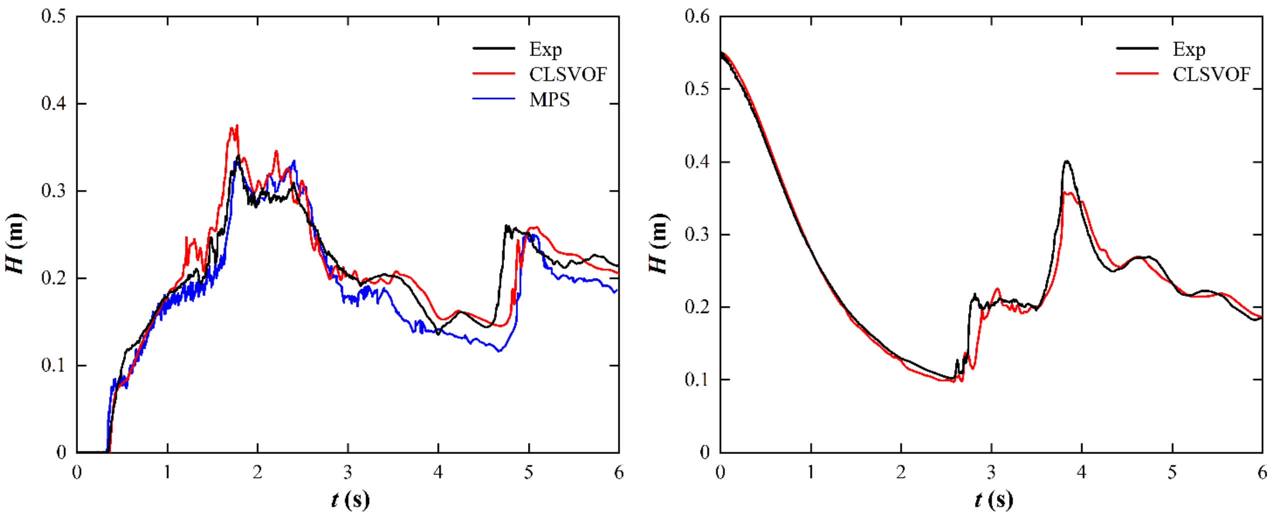
Figure 7. Locations of probes.

Fig. 8 compares the free surface at two time instants. In general, the numerical results agree well with the experiments, demonstrating the good performance of the CLSVOF method. Although the splashing in Fig. 8b is not well captured, it can be improved by increasing the mesh resolution. Figs. 9 and 10 further validate the CLSVOF method quantitatively in terms of surface elevation and impact pressure. Among them, the numerical results calculated by the in-house meshless particle solver MLParticle-SJTU (Zhang and Wan, 2014; Xie *et al*., 2021) based on the MPS method are also given. In Fig. 9, it can be observed that the numerical predictions are in good agreement with the experimental measurements and the results of the MPS method. In Fig. 10, the CLSVOF method gives reasonable predictions for impact pressure, where the overall trends are nearly identical. However, the peak is overestimated for P1 and underestimated for P3. This is because the dam break flow hits these probes on the front surface directly, resulting in high-amplitude peak pressures within a short duration. The current mesh resolution may not be sufficient to capture this phenomenon well. For the probes on the top, the strong oscillation at the initial stage is attributed to the overturning breaking waves. This phenomenon is well captured in our simulations, as shown in Figs. 10c and 10d.



(a) (b)

Figure 8. Comparison of free surface at (a) $t = 0.4$ s and (b) $t = 0.56$ s. (The upper row is the experimental photos and the lower row is the numerical results.)



(a) (b)

Figure 9. Comparison of surface elevation at (a) H2 and (b) H4.

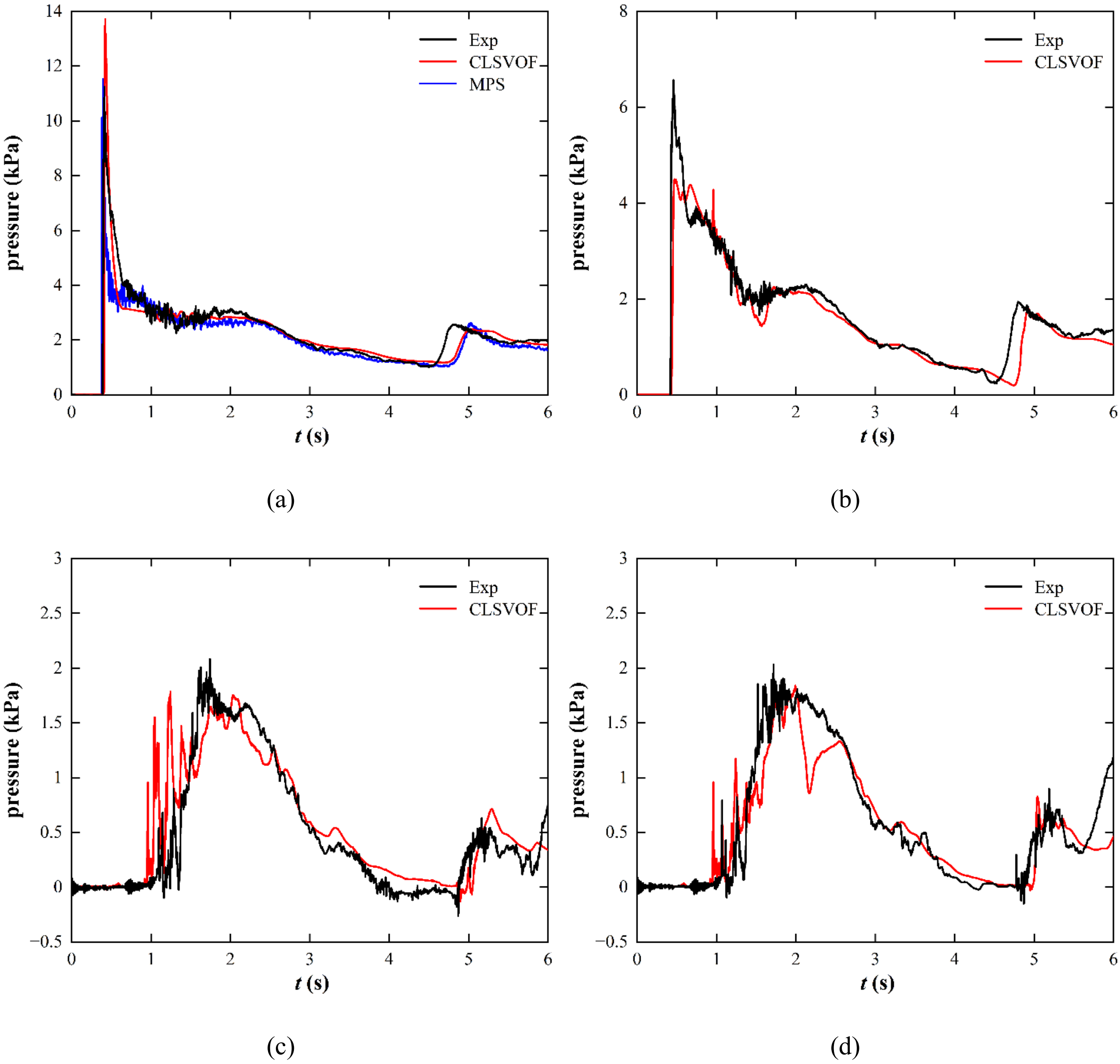


Figure 10. Comparison of pressure at (a) P1, (b) P3, (c) P5, and (d) P7.

4. CONCLUSIONS

In this paper, a two-way coupling CLSVOF method suitable for unstructured meshes is implemented in OpenFOAM, which mainly includes the interface reconstruction algorithm based on the PLIC method and the reinitialization algorithm. The accuracy and robustness of the CLSVOF method are validated on two dam break benchmark cases, where the free surface has large deformations. In addition, the CLSVOF method is compared with the interface compression algebraic VOF method. The main conclusions are as follows.

Through the comparison with the experimental measurements, it is demonstrated that the CLSVOF method can give promising results in terms of surface elevation and impact pressure. Moreover, compared with the algebraic VOF method, the CLSVOF method can obtain a sharper interface and significantly reduce the spurious air velocities. These two advantages can help to improve the accuracy and robustness of violent two-phase flow simulations in marine hydrodynamics. In the near future, we plan to combine the CLSVOF method with the ghost fluid method that has been implemented and validated in our previous work (Chen *et al.*, 2022) to achieve high-fidelity simulations of air-water two-phase flows.

ACKNOWLEDGEMENTS

This work was supported by the National Natural Science Foundation of China (52131102), and the National Key Research and Development Program of China (2019YFB1704200), to which the authors are most grateful.

REFERENCES

Albadawi, A., Donoghue, D. B., Robinson, A. J., Murray, D. B., and Delauré, Y. M. C. (2013). Influence of surface tension implementation in volume of fluid and coupled volume of fluid with level set methods for bubble growth and detachment. *International Journal of Multiphase Flow*, 53, 11-28.

Chen, S., Zhao, W., and Wan, D. (2022). On the scattering of focused wave by a finite surface-piercing circular cylinder: A numerical investigation. *Physics of Fluids*, 34(3), 035132.

Dianat, M., Skarysz, M., and Garmory, A. (2017). A coupled level set and volume of fluid method for automotive exterior water management applications. *International Journal of Multiphase Flow*, 91, 19-38.

Ferro, P., Landel, P., Pescheux, M., and Cadot, P. (2022). A level-set approach with embedded reinitialization. *Proceedings of the 18th Hydrodynamics Days*, November 22-24, Poitiers, France.

Kleefsman, K. M. T., Fekken, G., Veldman, A. E. P., Iwanowski, B., and Buchner, B. (2005). A volume-of-fluid based simulation method for wave impact problems. *Journal of computational physics*, 206(1), 363-393.

Kunkelmann, C., and Stephan, P. (2010). Modification and extension of a standard volume-of-fluid solver for simulating boiling heat transfer. *Proceedings of the 5th European Conference on Computational Fluid Dynamics ECCOMAS CFD2010*, June 14-17, Lisbon, Portugal.

Lobovský, L., Botia-Vera, E., Castellana, F., Mas-Soler, J., and Souto-Iglesias, A. (2014). Experimental investigation of dynamic pressure loads during dam break. *Journal of Fluids and Structures*, 48, 407-434.

Roenby, J., Bredmose, H., and Jasak, H. (2016). A computational method for sharp interface advection. *Royal Society open science*, 3(11), 160405.

Scheufler, H., and Roenby, J. (2019). Accurate and efficient surface reconstruction from volume fraction data on general meshes. *Journal of computational physics*, 383, 1-23.

Skarysz, M., Garmory, A., and Dianat, M. (2018). An iterative interface reconstruction method for PLIC in general convex grids as part of a Coupled Level Set Volume of Fluid solver. *Journal of Computational Physics*, 368, 254-276.

Sussman, M., Smereka, P., and Osher, S. (1994). A level set approach for computing solutions to incompressible two-phase flow. *Journal of Computational physics*, 114(1), 146-159.

Sussman, M., and Puckett, E. G. (2000). A coupled level set and volume-of-fluid method for computing 3D and axisymmetric incompressible two-phase flows. *Journal of computational physics*, 162(2), 301-337.

Vukcevic, V., and Jasak, H. (2014). A Conservative Level Set method for interface capturing in two-phase flows. *Proceedings of the 11th World Congress on computational mechanics; 5th European Conference on computational mechanics; 6th European Conference on Computational Fluid Dynamics*, July 20-25, Barcelona, Spain.

Wang, Z., Yang, J., Koo, B., and Stern, F. (2009). A coupled level set and volume-of-fluid method for sharp interface simulation of plunging breaking waves. *International Journal of Multiphase Flow*, 35(3), 227-246.

Weller, H. G. (2008). A new approach to VOF-based interface capturing methods for incompressible and compressible flow. Report TR/HGW/04, OpenCFD Ltd.

Xie, F., Zhao, W., and Wan, D. (2021). Numerical simulations of liquid-solid flows with free surface by coupling IMPS and DEM. *Applied Ocean Research*, 114, 102771.

Zhang, Y., and Wan, D. (2017). Numerical study of interactions between waves and free rolling body by IMPS method. *Computers and fluids*, 155, 124-133.