NUMERICAL MODELING OF A SINGLE CAVITATION BUBBLE NEAR THE SOLID WALL WITH A COUPLED LEVEL SET AND VOLUME OF FLUID METHOD

JIANYONG YIN[1], YONGXUE ZHANG[2], YUNING ZHANG[3*]

[1] *China University of Petroleum-Beijing, China, jianyongyin@foxmail.com*

[2] *China University of Petroleum-Beijing, China, zhyx@cup.edu.cn*

[3] *China University of Petroleum-Beijing, China, zynlzyn.100@163.com*

** Corresponding Author*

Keywords: *cavitation bubbles; volume of fluid; level set; compressible flows*

For the simulation of the growth and collapse of the cavitation bubble near the solid wall, the open source package OpenFOAM [1] is used and the source code of two-phase solver compressibleInterFoam is modified for inclusion of equation of state for the gas and the liquid [2]. A coupled level set and volume of fluid (CLSVOF) [3, 4] method is established to track the movement of the gas-liquid interface, to improve the accuracy of the simulation of interface curvature and surface tension force. All the numerical results are well consistent with the experimental data, which demonstrates the correctness and reliability of the model.

Figure 1 shows that the comparison of the evolution of the bubble radius of the CLSVOF and VOF with the experiment data. In experiment, A high-power laser pulse (wavelength532nm, duration 6ns) of the Q-switch Nd:YAG laser is focused into a plexiglass container filled with deionized water in order to generate a single cavitation bubble. The stand-off distance (γ, where $\gamma = L / R_{max}$, L is the diatance between the solid wall and the bubble centre, R_{max} is the maximum bubble radius) is equal to 2.0. In numerical simulation, both the initial pressure inside the bubble and the initial bubble radius are almost impossible to be exactly specified, and therefore are assumed to be 9×10^7Pa and 0.072 mm, respectively. The distance between the initial bubble centre and the solid wall is 1.58mm, which is same as the experiment. It's worth noting that the numerical interfaces are captured by both compressible VOF and CLSVOF. When the cavitation bubble reaches the maximum radius, there is a relative error of radius of 1.53% between the numerical results by the method of VOF and experiment data. What's more, the relative error of radius is 1.24% between the numerical results by the method of CLSVOF and experiment data. Meanwhile, compared with the results by using the VOF method, the CLSVOF has a smaller bubble radius during the bubble collapse. As shown in Figure 1, the CLSVOF results are better consistent with the experimental data.

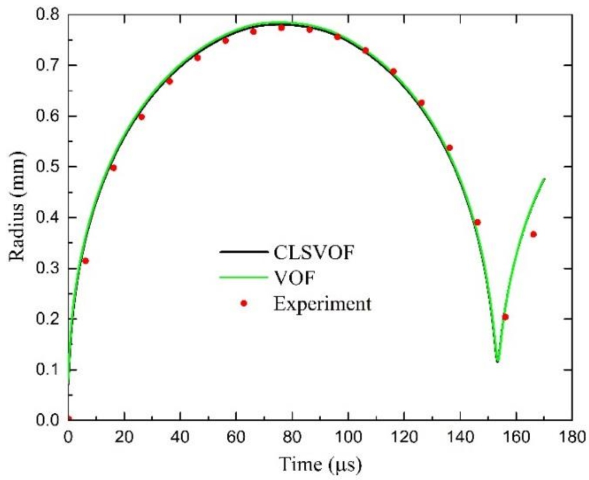

Fig. 1. The comparison of the numerical results with the experimental data with γ = 2.0

Acknowledgements

The authors would like to acknowledge the supports of the National Natural Science Foundation of China (Project No.: 51606221), the Science Foundation of China University of Petroleum, Beijing (No. 2462016YJRC003).

References

[1] Greenshields, C.J., Openfoam user guide. OpenFOAM Foundation Ltd, version, 2015. **3**(1).

[2] Koch, M., et al., Numerical modeling of laser generated cavitation bubbles with the finite volume and volume of fluid method, using OpenFOAM. Computers & Fluids, 2016. **126**: p. 71-90.

[3] Albadawi, A., et al., 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, 2013. **53**: p. 11-28.

[4] Jadidi, M., et al. A coupled level set and volume of fluid method with application to compressible two-phase flow. in Proceedings of the 22nd Annual Conference of the CFD Society of Canada, Toronto, ON, Canada. 2014.