Cloud Cavitating Flow around an Axisymmetric Projectile in the shallow water

[1,2]Chang Xu; [1,2]Yiwei Wang*; [1,2]Jian Huang; [1,2]Chenguang Huang

[1] *Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Chinese Academy of Sciences, Beijing, China;* [2] *School of Engineering Science, University of Chinese Academy of Sciences, Beijing, China*

Abstract

Through a series of water tank experiments and numerical simulations, this study examines the cloud cavitating flow that surrounds an axisymmetric projectile in shallow water. Experiment results show the free surface effect, near-wall effect, and mutual effect on the cavity evolution process. The experimental observations of cavity development are consistent with numerical results, which validate the accuracy of the methods. The stability and asymmetry of the cavity shape around the projectile under the effects are studied. The weak constraint effect of the free surface will affect the re-entrant jet and improve the stability of the cavity nearby. The low-pressure area between the lower side of the projectile and the bottom wall may also affect the re-entrant jet, thus inducing partial cavity shedding.

Keywords: cavitation, multiphase flow, water tank experiment, CFD

Introduction

Cavitation occurs when a vehicle runs fast underwater. Liquid water is subjected to rapid pressure changes that cause the formation of cavities in a low-pressure region. Cavitation phenomenon can be classified by shape development into incipient cavitation, sheet cavitation, cloud cavitation, and supercavitation with high to low cavitation number changes [1, 2]. For a typical high-speed craft underwater launching problem, the process can be considered a representative problem on the unsteady transient cavitation flow and its interaction with the free surface [3-7] and nearby wall [8, 9]. Such unsteadiness, with phenomena such as breaking, shedding, and collapsing, may have serious consequences, such as noises, erosion, vibration, and instability in trajectory. Therefore, the influence of cavitating flow on the performance of navigation vehicles cannot be neglected [10-13]. Free surface and bottom wall near the cavitating flow are significant boundary conditions that should be considered. Moreover, the mechanism involved should be investigated to determine how such effect can be controlled in engineering applications.

In this work, the cloud cavitating flow that surrounds an axisymmetric projectile is investigated using water tank experiment and CFD simulation methods, with a focus on the mutual effect of the free surface and the bottom wall near the projectile. Numerical methods with LES approach and Cartesian cut-cell mesh method are validated and verified based on the water tank experiment results. Cases with a 15 mm distance between the model and both boundary conditions are analyzed, and the numerical methods are validated and verified. The cavity evolution process and the difference between the cavity on the upper and lower sides of the projectile are observed experimentally, and the re-entrant jet inside the cavity is discussed based on the experimental and numerical results.

Water Tank Experiment

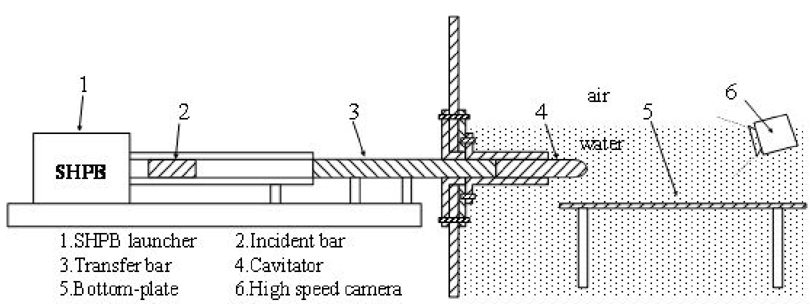

Figure 1 Schematic of the launching process.

*Corresponding Author, Yiwei Wang: wangyw@imech.ac.cn

The schematic of the launching process is shown in Figure 1. SHPB (Split-Hopkinson pressure bar) technology [14] is used in the water tank experiment, which can accelerate the projectile to 20.5 m/s in 200 μs. Little disturbance to the flow field is observed before the launching. A 200 mm × 37 mm × 37 mm steel cylinder with a conical head model is tested. The temperature of the water inside the tank is approximately 20 °C. The entire experiment is recorded by a high-speed camera with a sampling frequency of 25000 frames per second. Moreover, compared with the water tunnel, this system will save considerable energy. Importantly, the present experimental method still has some disadvantages. The launching speed of the model is set by adjusting the pressure in the air chamber with high pressure. However, the propagation of stress wave is insensitive to contact and friction conditions, and thus, the launching speed disperses under the same pressure in the air chamber. In actuality, exact speed values need to be obtained by analyzing the model motion in images.

Numerical Methods

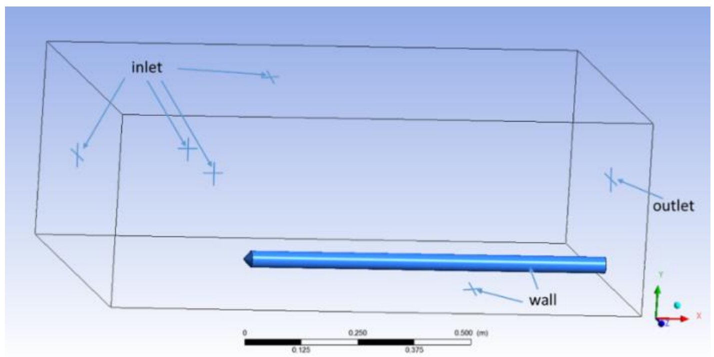

Figure 2 Calculated domain and boundary conditions.

Commercial software ANSYS Fluent is used to simulate cavitating flow near the wall of an underwater axisymmetric projectile. The computational domain and boundary conditions are shown in Figure 2. A semi-infinite projectile model is used, and the effect of the tail on the shoulder cavity is neglected. The model is fixed, with the free surface moving toward the model. For the velocity inlet boundary condition, the inlet velocity is set as 20.5 m/s with no turbulent perturbations. The bottom is also set as a non-slip wall boundary. LES and the WALE model are adopted to simulate turbulent flow with Cartesian cut-cell mesh method. he total cell number is approximately 3.42 million with good orthogonality.

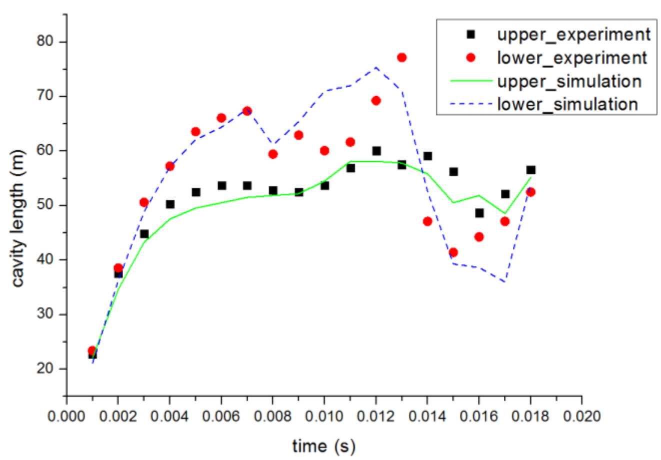

Figure 3 Comparison of the cavity length at the upper and lower sides of the projectile between the experimental and simulation results when the distance between the projectile and the free surface/bottom wall is 15 mm.

*Corresponding Author, Yiwei Wang: wangyw@imech.ac.cn

The cavity lengths at the upper and lower sides of the projectile are compared in the experimental and simulation results to validate the accuracy of the simulation method when the distance between the projectile and the free surface/bottom wall is 15 mm (Figure 3). The results show a small difference between CFD simulation and experimental data, which may be caused by the differences of the boundary conditions between the experiment and numerical methods. The comparison shows that the simulation is reasonably accurate.

Results and Discussion

Cavity evolution involves four stages, namely, cavity growth, re-entrant jet, cavity shedding, and collapsing. Figure 4 shows a comparison of cavity images between the experimental and simulation results when the distance between the projectile and the free surface/bottom wall is 15 mm. Cavity grows in the first stage, and then the length remains nearly constant on the upper side in the re-entrant jet stage. Partial cavity shedding happens at the same time on the lower side of the projectile, and the lower-side cavity is much more unstable than that in the upper. Then, as the re-entrant jet inside the cavity reaches the leading edge of the cavity, it cuts off the cavity on the upper side of the model first and the cavity begins to shed. The total cavity length experiences a secondary growth in this stage. The cavity collapses at the closure in the last stage, and the length of the cavity is drastically decreased. The growing cavity in the new cavity evolution process is then observed. This analysis shows that the re-entrant jet inside the cavitating flow exerts a negligible influence on cavity evolution.

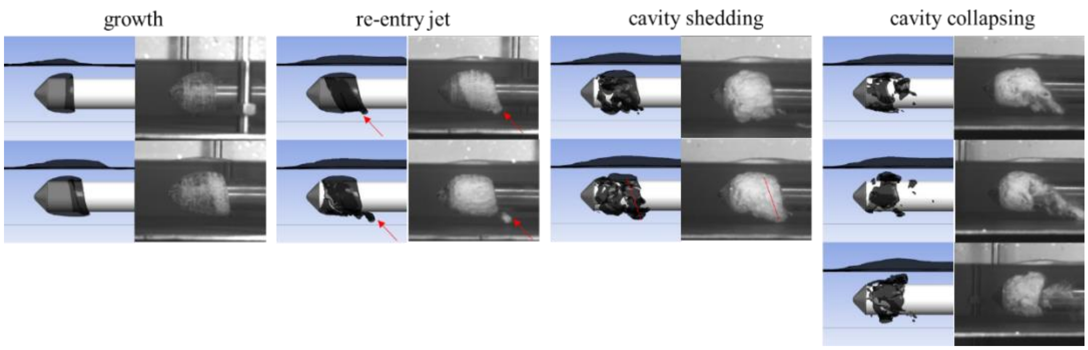

Figure 4 Comparison of the cavity patterns between experiment and simulation results.

Free Surface Effect on Cloud Cavitating Flow

To analyze the free surface effect on the cloud cavitating flow around the tested projectile in the water tank experiment, we plot the upper- and lower-side cavity lengths of the experimental results. The differences can be clearly seen from the comparison shown in Figure 5. First, the overall upper- and lower-side cavity length of the experimental case without free surface is longer than the case with free surface near the projectile. The cavity evolution process of the two cases is generally similar. However, the period of the cavity evolution process in shallow water is shorter than in the case without free surface. Partial shedding of cavity on the lower side of the model also occurs in the two cases. Another phenomenon that cannot be ignored is the much more stable cavity during the period in the case with free surface nearby than in the case without free surface. The cavity on both the upper and lower sides of the projectile becomes more stable when the projectile is close to the free surface. The free surface may have good effects on high-speed cruising in a certain depth range and operating conditions.

Near-Wall Effect on Cloud Cavitating Flow

Figure 6 shows a comparison of the upper- and lower-side cavity lengths in the water tank experimental results of the tested projectile in shallow water and near the free surface only without a bottom wall nearby. Generally, the overall cavity evolution process for the two cases is similar. The cavity on the upper side is stable during this time. However, a comparison of the plotted lines of the upper-side cavity length shows that the bottom wall could lead to a slightly more stable cavity. This finding means that the bottom wall may also have positive effects on high-speed cruising in a certain depth range and operating conditions. A longer cavity and a much longer period of the cavity evolution under the near-wall effect for the lower-side cavity are shown in the figure. The partial cavity shedding is not shown in the case without a nearby wall, which means that the phenomenon is caused by the near-wall effect.

*Corresponding Author, Yiwei Wang: wangyw@imech.ac.cn

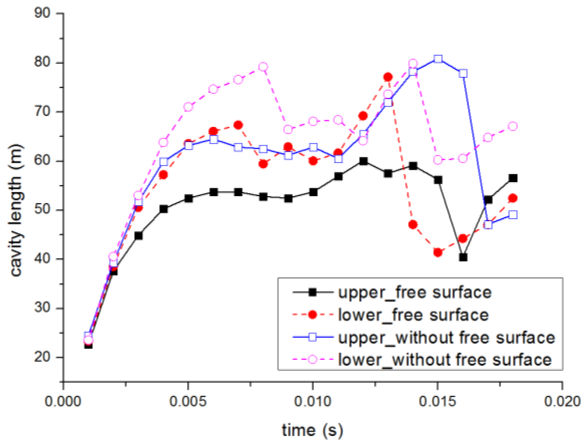

Figure 5 Comparison of the upper- and lower-side cavity length in the water tank experimental results of the tested projectile in shallow water and near the bottom wall only.

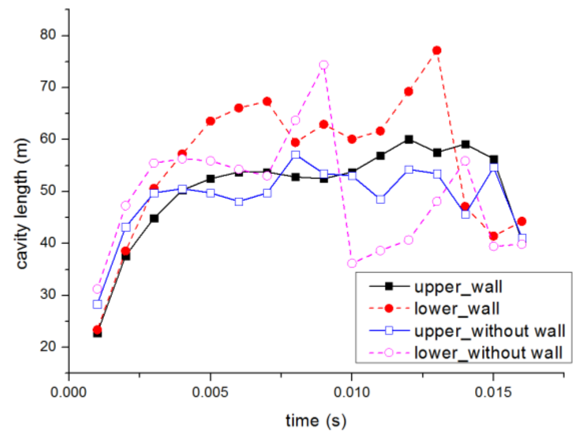

Figure 6 Comparison of the upper- and lower-side cavity lengths in the water tank experimental results of the tested projectile in shallow water and near the free surface only.

Mutual Effect of the Free Surface and Bottom Wall on Cavitating flow

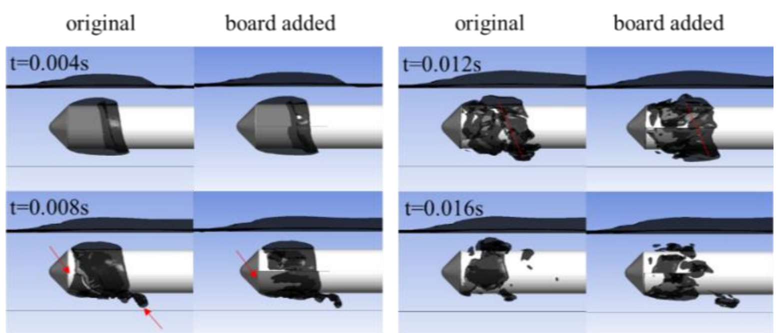

Figure 7 Comparison of the original simulation results and the simulation results of the case with a nonthick board added in the middle of the projectile on the Y–Z plane.

A board without thickness is added in the middle of the projectile on the Y–Z plane to hold back the interaction of the cavity on the upper and lower sides. Figure 7 shows a comparison of the simulation results of the original and additional board. The differences that need focus are marked by red lines and arrows in the figure. The most obvious difference occurs at t = 0.008 s when partial cavity shedding occurs. Without interaction inside the cavity, cavity at the lower side of the projectile will not shed at the shoulder of the model, that is, the period will be longer.

Conclusion

The cloud cavitating flow that surrounds an axisymmetric projectile in shallow water is analyzed in this article. Water tank experiment and CFD simulation are conducted to solve the problem and the results agree well. Mutual effects on the cavitating flow in shallow water include the free surface effect, near-wall effect, and interaction inside the cavity. Free surface could induce a more stable cavity around the projectile. A weaker free-surface constraint than infinite flow field makes the cavity on the upper side of the projectile shorter and thicker, and shortens the overall cavity evolution process. Partial cavity shedding occurs on the near-wall side of the projectile. A large low-pressure zone between the lower side of the projectile and the bottom wall leads to a longer and thicker cavity than the upper side.

Free surface and wall have complex effects on the cavity around a projectile, and such effects may change along with the distance between the model and the boundary conditions. The results of this study are limited to typical working conditions for a typically shaped model, thereby requiring further in-depth analysis.

*Corresponding Author, Yiwei Wang: wangyw@imech.ac.cn

References

[1] Franc, J. P., and Michel, J. M., 2004, *Fundamentals of cavitation*, Springer.

[2] Wang, G., Senocak, I., Shyy, W., Ikohagi, T., and Cao, S., 2001, "*Dynamics of attached turbulent cavitating flows*," Progress in Aerospace Sciences, 37(6), pp. 551-581.

[3] Young, Y., Harwood, C., Montero, F. M., Ward, J., and Ceccio, S., "*Ventilation of Lifting Bodies: Review of the Physics and Discussion of Scaling Relations*," Applied Mechanics Reviews.

[4] Harwood, C., Young, Y., and Ceccio, S., 2016, "*Ventilated cavities on a surface-piercing hydrofoil at moderate froude numbers: cavity formation, elimination, and stability*," J. Fluid Mech, 800, pp. 5-56.

[5] Wang, Y. W., Xu, C., Jian, H., and Huang, C. G., 2017, "*Study on flow characteristics and stability mechanism of unsteady cavitating flow near the free surface*," Journal of Hydrodynamics, Ser. A.

[6] Wang, Y., Xu, C., Wu, X., Huang, C., and Wu, X., 2017, "*Ventilated cloud cavitating flow around a blunt body close to the free surface*," Physical Review Fluids, 2(8), p. 084303.

[7] Wang, Y., Wu, X., Huang, C., and Wu, X., 2016, "*Unsteady characteristics of cloud cavitating flow near the free surface around an axisymmetric projectile*," International Journal of Multiphase Flow, 85, pp. 48-56.

[8] Xu, C., Yu, C., Huang, J., Wang, Y. W., and Huang, C. G., 2017, "*Experimental and numerical analysis of cloud cavitating flow that surrounds an axisymmetric projectile in shallow water*," Journal of Hydrodynamics, Ser. A.

[9] Yu, C., Wang, Y., Huang, C., Du, T., Xu, C., and Huang, J., 2017, "*Experimental and numerical investigation on cloud cavitating flow around an axisymmetric projectile near the wall with emphasis on the analysis of local cavity shedding*," Ocean Engineering, 140, pp. 377-387.

[10] Robert T. Knapp, J. W. D., Frederick G. Hammitt, 1970, *Cavitation*, McGraw-Hill.

[11] Stutz, B., and Reboud, J., 1997, "*Experiments on unsteady cavitation*," Experiments in fluids, 22(3), pp. 191-198.

[12] Stutz, B., and Reboud, J.-L., 2000, "*Measurements within unsteady cavitation*," Experiments in fluids, 29(6), pp. 545-552.

[13] Callenaere, M., Franc, J.-P., Michel, J.-M., and Riondet, M., 2001, "*The cavitation instability induced by the development of a re-entrant jet*," Journal of Fluid Mechanics, 444, pp. 223-256.

[14]Wei Y., Wang, Y., Fang, X., Huang, C., and Duan, Z., 2011, "*A Scaled Underwater Launch System Accomplished by Stress Wave Propagation Technique*," Chinese Physics Letters, 28(2), p. 024601.

*Corresponding Author, Yiwei Wang: wangyw@imech.ac.cn