Ventilated cloud cavitating flow around a blunt body near the free surface

Yiwei Wang, Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Chinese Academy of Sciences, Beijing, China
Chenguang Huang, Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Chinese Academy of Sciences, Beijing, China
Xiaocui Wu, Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Chinese Academy of Sciences, Beijing, China

Long Abstract

Introduction

The interaction between the cavitation region and the free surface is an important issue for the fast cruise of surface vehicles. Ventilated cavitation occurs as an extremely complicated problem if the free surface is close to the vehicle or the cavity boundary [1]. The gas above the free surface can inject into the cavity, which may change the dynamic behaviors of the cavity significantly. The effect of wave elevation is also coupled with the distribution of cavitation region.

Relevant research works on the interactions between the free surface and the cavitating flow is very limited in literature. Most works were focused on the conditions that the free surface is separated from the cavity. Theoretical and numerical methods were established, and the influences of the free surface on the cavitating flow around hydrofoils and axisymmetric bodies are obtained[1-4]. On the other side, neglecting the influence of the free surface, the dynamic characteristics of the cloud cavitating flow are often linked with the motion of vortexes. Based on the experimental and numerical approaches, the evolution of cavities including growth, re-entry, shedding and collapse were gained [5-8].

The cavitating flow around a blunt axisymmetric body very near the free surface is investigated in the present paper. The typical experiment is carried out in a launching system based on an SHPB device, and the numerical scheme is established on the basis of large eddy simulation (LES) and volume of fraction (VOF) methods. The distinctions of evolution features between the cavities on the up and down sides are presented. The effects of the free surface and the gas injection on the cavity shape are analyzed.

1. Models

The experiment was performed by using a launching system on the basis of the SHPB technology[8], which can transiently accelerate the projectile with slight disturbance on the water. The projectile used in this study is a blunt axisymmetric body. The total length is 150 mm, and the diameter is 37 mm. As shown in Fig. 1, photographs of typical cavitation can be obtained using a high-speed camera with 12000 fps. The distance between the free surface and the up side of the projectile is 5mm, and the analysis of obtained images indicates that the speed is approximately uniform at 19.1 m/s. The cavitation number calculated as 0.537.

To simulate the motions of fluids including the phase change, a single fluid and multiphase flow equations are adopted. The governing equations are solved by a LES approach based on Smagorinsky-Lilly model. Unsteady numerical simulations are performed on the basis of finite volume method with coupled scheme by using the commercial CFD software ANSYS-FLUENT.

2. Results and discussion

From the experimental and numerical results, the quasi-periodic development of cavity shape is obtained. Photographs on typical moments of different stages are as shown in Fig.1, and the volume fraction of liquid water is 0.9 on the iso-surface. The contours of water volume fraction on the symmetric plane and wall surface are also given, in which the red color represents that the volume fraction is 1 while the blue color represents zero.

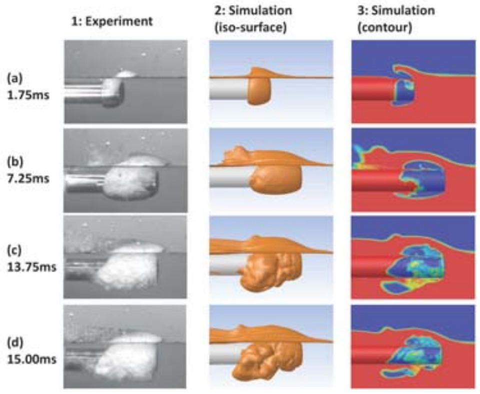

Figure 2. Time evolution of cavitation patterns

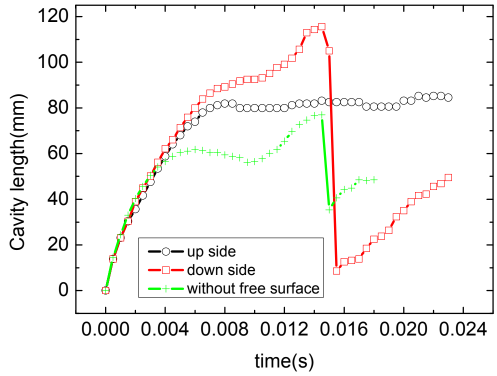

Figure 3. Cavity lengths in experiment

The variations of cavity lengths on the up and down sides are as shown in Fig.2. Refer to the evolutions aforementioned, we can see the cavity on the upper side stays approximately stable, and its length varies in a very small range after the growth stage. This may because the water layer flowing around the upper side is very thin when the vehicle moves closely to the free surface. In this condition, the restriction of water layer to the flow is weak, which cannot form high pressure at the cavity closure. The re-entry jet does not also have enough strength and thickness to cut off the main cavity, so notable fluctuation of cavity shape cannot be generated.

The length variation on the lower side is similar with the case under the same cavitation number without the free surface but has much larger amplitude. On the one hand, the influence of the free surface on the cavity on the lower side is relatively small, so powerful re-entry jet can be generated, and unsteady evolutions including breaking and shedding are still exist. The evolution of cavity on the lower side is similar with the case without the free surface. On the other hand, the air injection in the initial stage increases the inner pressure and air quantity, so the inner pressure is actually remarkably higher than the saturation pressure in the natural cavitation bubbles, which can be considered as that the cavitation number is decreased.

Acknowledgement

The authors are grateful to the national natural science foundation of china (NSFC) through grant numbers 11202215&11332011.

References

[1] O. M. Faltinsen and Y. A.Semenov. The effect of gravity and cavitation on a hydrofoil near the free surface. *Journal of Fluid Mechanics*, 597:371-394, 2008.

[2] S. Bal and S. A. Kinnas. A BEM for the prediction of free surface effects on cavitating hydrofoils. *Computational Mechanics*, 28:260-274, 2002.

[3] S. Bal. High-speed submerged and surface piercing cavitating hydrofoils, including tandem case. *Ocean Engineering*, 34:1935-1946, 2007

[4] E.Amromin. Analysis of body supercavitation in shallow water. *Ocean Engineering,* 34:1602-1606, 2007.

[5] B. Stutz and S. Legoupil. 2003, X-ray measurements within unsteady cavitation. *Experiments in Fluids*, 35:130-138, 2003

[6] B. Ji, X. W. Luo, R. E. A. Arndt, X. Peng, and Y. Wu. Large Eddy Simulation and theoretical investigations of the transient cavitating vortical flow structure around a NACA66 hydrofoil. *International Journal of Multiphase Flow*, 68:121-134, 2015.

[7] E. Roohi, A. P. Zahiri and M. Passandideh-Fard. Numerical simulation of cavitation around a two-dimensional hydrofoil using VOF method and LES turbulence model. *Applied Mathematical Modelling*, 37:6469-6488, 2013.

[8] X. Yu, C. Huang, T. Du, L. Liao, X. Wu, Z. Zheng, and Y. Wang. Study of Characteristics of Cloud Cavity Around Axisymmetric Projectile by Large Eddy Simulation. *Journal of Fluids Engineering*, 136:051303, 2014.