

On the flow structure of cloud cavitating flow around an axisymmetric body near the free surface

This content has been downloaded from IOPscience. Please scroll down to see the full text.

2015 J. Phys.: Conf. Ser. 656 012174

(<http://iopscience.iop.org/1742-6596/656/1/012174>)

View [the table of contents for this issue](#), or go to the [journal homepage](#) for more

Download details:

IP Address: 159.226.231.116

This content was downloaded on 07/01/2016 at 09:10

Please note that [terms and conditions apply](#).

On the flow structure of cloud cavitating flow around an axisymmetric body near the free surface

Yiwei Wang*, Xiaocui Wu, Chenguang Huang, XianXian Yu

Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Chinese Academy of Sciences

No.15 Beisihuanxi Road, Beijing, China, 100190

***EMAIL:** wangyw@imech.ac.cn

Abstract. The influence of the free surface on the cavitating flow is an important issue involved in the design of high speed surface vehicles. In the present paper, unsteady cavitating turbulent flow around an axisymmetric body near the free surface was investigated by both launching experiment and LES simulation. The vortex motion induced by cavity shedding under the effect of the free surface is emphatically analyzed by comparing with the submerged condition. The vortex shedding process around the projectile is not synchronized, while the asymmetric characteristic in collapse process is more remarkable, with the generation of multiple vortex ring structures.

1. Introduction

Characteristics of cavitating flow, especially unstable structures of cloud cavitating flow, always have close relationship with the vortex motion [1]. Therefore, the turbulence resolution method plays a fatal role on the establishment of numerical approach on unsteady cavitating flow. On the simulation for engineering applications, solving Reynolds-averaged Navier–Stokes (RANS) equations and turbulence models with physical modifications is the main way. Approaches such as modified RNG $k-\epsilon$ turbulence model [2-6] are widely used. In the recent years, large eddy simulation (LES) method is adopted for the research on cavitating flows, which can capture considerable details of large-scale eddies in the flow field with high accuracy [7-14]. For example, an implicit large eddy simulation method was adopted to simulate dynamic cavitation behavior of a propeller by Bensow et al [7]. The cavitation shedding horse-shoe structures were captured numerically by Ji et al [8, 14].

The influence of the free surface on the cavitating flow is an important issue involved in the design of high speed surface vehicles, but relative investigations are very limited. Fatingsen et al [15] established a numerical approach based on theoretical models on the supercavitating flow near the free surface. Bal et al presented boundary element methods (BEM) [16-18] for cavitating hydrofoils near a free surface. Other similar works are mainly performed concerning stable supercavitating flow structures [19, 20]. The influence of the free surface on the unsteady cloud cavitating flow is very lacking in literature.

The present paper focuses mainly on the influence of the free surface on the flow structure of the cloud cavitating flow. Typical evolutions of nonaxisymmetrical cavity shapes around a slender body near the free surface are obtained both by LES method and a launching experiment. Numerical methods are validated by comparing results with underwater launching experiments. The vortex



motion induced by cavity shedding under the effect of the free surface is emphatically analyzed by comparing with the submerged condition.

2. Experimental setup

The launching system is established on the basis of the SHPB technology [21]. It can transiently accelerate the projectile with slight disturbance on surrounding water. The projectile used in this study is a slender cylinder with a conical head. Its material is polished stainless steel. The total length is 246 mm, and the diameter is 37 mm. The conical angle is 90° . Typical photograph of cavitation can be obtained using a high-speed camera with 25000 fps (as shown in Fig.1). The cavitation number is 0.62, while the projectile speed is 17.8 m/s.

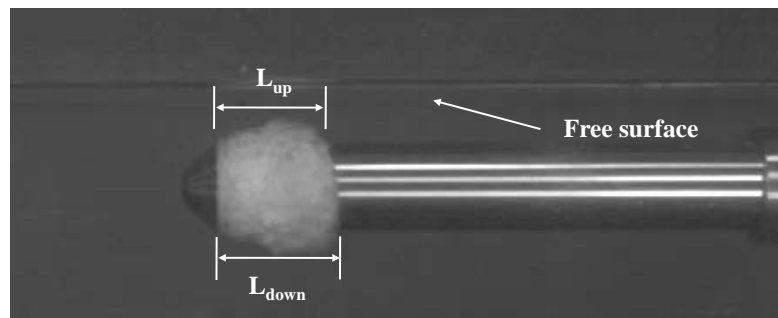


Fig. 1 Typical photograph of cavity

3. Numerical methods

Mixture/multiphase flow equations are adopted to simulate the motions of liquid water and vapor and air including the phase change. The mass transfer rate of evaporation and condensation are derived from the bubble dynamic equations of generalized Rayleigh-Plesset equation by Zwart et al.

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. The volume of fluid (VOF) method and the modified high resolution interface capturing (HRIC) scheme are used to capture the interfaces of the free surface and cavity profile. The mesh used in the present research has approximately 4 million cells, which is similar with the reference [13]. The independence analysis and validation of similar grid have been performed on simulating typical evolution of cloud cavitating flow.

4. Results and discussion

4.1. Cavity evolution

An additional case without the free surface is also performed. Typical evolution photographs in experimental and numerical results are as shown in Fig.2. Quasi-periodical variations of cavity shape are obtained. Typical evolutions of cavity includes growth, re-entry jet development, cavity shedding and collapse, which are similar for both the process considering and neglecting the effect of the free surface. The cavity shapes affected by the free surface are non-axisymmetrical. The re-entry jet in the upper part of the cavity is generated earlier and has a slightly faster velocity (as shown in Fig.2(a)). When the re-entry jets arrive at the shoulder of the model, the cavity on the lower side is a little longer than the upper side (as shown in Fig.2(b)). The cavity sheds by the interaction between main flow and re-entry jet, and the cavity on the upper side sheds earlier (as shown in Fig.2(c)). The shedding cavity on the lower side is farer away from the main cavity, which collapses more strongly (as shown in Fig.2(d)). Cavity shapes on the lower side are similar with the case without the free surface, but cavity in the latter case is longer. Because the cavity shape is integral and like a ring, the development on the lower side is hindered by the upper side.

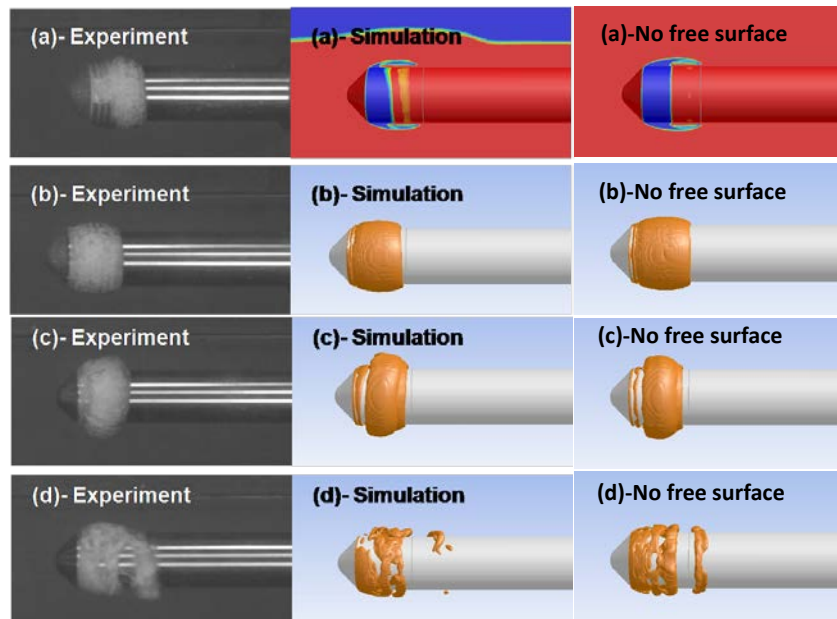


Fig. 2 Comparison of cavity shapes between experimental and numerical results

4.2. Influence on shedding vortex structures

The motion of shedding cavitation clouds has strong correlation with the vortex motion. The flow structures can be visualized based on the Q-criterion. In the present study, the iso-surface of $Q=50000$ is shown to represent the vortex core, and the colour represents the magnitude of the velocity component in the axial direction.

Vortex structures in the shedding process are as shown in Fig.3. For the case without the free surface in the right views, vortices are rolled outwards gradually, and the upstream parts begin to break apart at first under the influence of the main flow. For the case with the free surface in the left views, the shedding process around the projectile is not synchronized, and the shedding vortex is inclined. The broken vortex is thinner on the upper side under the influence of the free surface.

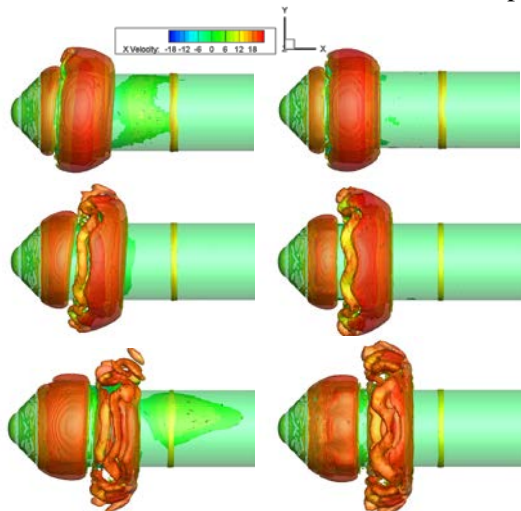


Fig. 3 Vortex structures in the cavity shedding process ($Q=50000$, left-with free surface, right-without free surface)

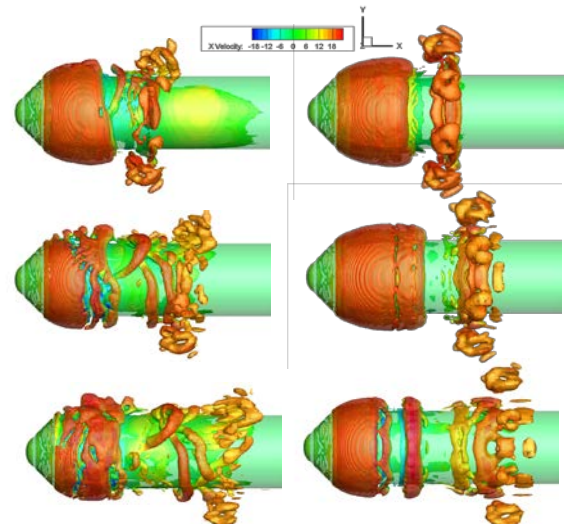


Fig. 4 Vortex structures in the cavity collapse process ($Q=50000$, left-with free surface, right-without free surface)

The shedding vortex breaks up strongly induced by the collapse of the shedding cavity (as shown in Fig.4), which exhibits more remarkable asymmetry during this period under the influence of the free surface. A multiple vortex ring structure is formed after cavity collapse, inducing significant non-uniform distributions of velocity and pressure, which may cause large lateral force and affect the stability of cruising.

5. Conclusions

Unsteady cavitating turbulent flow around an axisymmetric body near the free surface was investigated by both the launching experiment and the LES simulation. The evolution of flow structures is mainly analyzed, and several conclusions can be drawn as follows:

- (1) On the upper side near the free surface, the cavity shape and cycle time are both shorter, while cavity shapes on the lower side are similar with the case without the free surface.
- (2) The vortex shedding process around the projectile is not synchronized, and the broken vortex is thinner on the upper side under the influence of the free surface.
- (3) The vortex collapse process is strongly asymmetric, and a multiple vortex ring structure is formed after cavity collapse. The remarkable asymmetry may cause large lateral force and affect the stability of cruising.

Acknowledgements

The authors are grateful to the national natural science foundation of china through grant numbers 11202215&11332011 and the Youth Innovation Promotion Association of CAS (2015015).

References

- [1] Arndt R E A 2002 *Annu. Rev. Fluid Mech.* **34** 143-175
- [2] Coutier-Delgosha O, Stutz B, Vabre A, Legoupil S 2007 *J. Fluid Mech.* **578** 171-122
- [3] Coutier-Delgosha O, Fortes-Patella R, Reboud JL 2003 *ASME J. Fluids Eng.* **125**(1) 38-45
- [4] Coutier-Delgosha O, Deniset Fo, Astolfi JA, Leroux J-B 2007 *ASME J. Fluids Eng.* **129**(3) 279-292
- [5] Zhou L, Wang Z 2008 *ASME J. Fluids Eng.* **130**(1) 011302
- [6] Coutier-Delgosha O, Reboud J, Delannoy Y 2003. *Int. J. Numer. Meth. Fl.* **42**(5):527-548
- [7] Bensow RE, Bark G 2010 *ASME J. Fluids Eng.* **132**(4) 041302.
- [8] Ji B, Luo XW, Arndt REA, Peng X, Wu Y 2015 *Int. J. Multiphas. Flow* **68** 121-134
- [9] Wang G, Ostojca-Starzewski M 2007 *Appl. Math. Model.* **31**(3) 417-447
- [10] Huang B, Zhao Y, Wang G 2014 *Comput. Fluids* **92** 113-124
- [11] Dittakavi N, Chunekar A, Frankel S 2010 *ASME J. Fluids Eng.* **132**(12) 121301
- [12] Roohi E, Zahiri AP, Passandideh-Fard M 2013. *Appl. Math. Model.* **37**(9) 6469-6488
- [13] Yu X, Huang C, Du T, Liao L, Wu X, et al 2014 *ASME J. Fluids Eng.* **136**(5) 051303
- [14] Ji B, Luo XW, Peng X X, Wu Y L 2013 *J. Hydrodyn.* **25**(4) 510-519
- [15] Faltinsen OM, Semenov YA 2008 *J. Fluid Mech.* **597** 371-394
- [16] Bal S, Kinnas SA 2002 *Comput. Mech.* **28**(304) 260-274
- [17] Bal S 2011 *J. Mar. Sci. Tech. Japan* **16**(2) 129-142
- [18] Bal S 2007 *Ocean Eng.* **34** 1935-1946
- [19] Amromin E 2007. *Ocean Eng.* **34** 1602-1606
- [20] Chen X, Lu C J, Li J, Chen Y 2011. *J. Hydrodyn.* **23**(6) 730-736
- [21] Wei Y P, Wang Y W, Fang X, Huang C G, Duan Z P 2011 *Chin. Phys. Lett.* **28** 024601