European Journal of Mechanics / B Fluids 67 (2018) 15–24

Contents lists available at ScienceDirect

European Journal of Mechanics / B Fluids

journal homepage: www.elsevier.com/locate/ejmflu

# Analysis of Near-wall effect on cloud cavitating flow that surrounds an axisymmetric projectile using large eddy simulation with Cartesian cut-cell mesh method

Chang Xu, Yiwei Wang *, Chenguang Huang, Chao Yu, Jian Huang

*Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Chinese Academy of Sciences, Beijing, 100190, China*
*School of Engineering Science, University of Chinese Academy of Sciences, Beijing, 100049, China*

ARTICLE INFO

*Article history:*
Received 18 October 2016
Received in revised form 18 April 2017
Accepted 2 July 2017
Available online 19 July 2017

*Keywords:*
Cloud cavitating flow
Near-wall effect
Large eddy simulation
Cartesian cut-cell mesh
Vortex structure

ABSTRACT

Near-wall effect is important for cavitation of flow and vortex structures. These structures are commonly investigated in cavitation of tip-vortex leakage, but are rarely discussed in cloud cavitating flow. In this study, typical experiments and numerical simulation of cloud cavitating flow were conducted near a wall that surrounds an underwater axisymmetric projectile. The experimental observations of cavity development are consistent with numerical results and validate the method's accuracy. Changes in the cavity of the distal and near wall side differ throughout the entire evolution process. The cavity grows faster on the near wall side than on the distal side, whereas the re-entry jet inside the cavity moves slowly toward the shoulder of the model. The strong vortex around the projectile is non-axisymmetric because of the collapsing cavity, which may also affect the cruising stability.



## 1. Introduction

Unsteady cavitating flow around high-speed underwater vehicles is one of the highly discussed topics in the engineering community [1,2]. Such unsteadiness causes serious consequences, e.g. noises, erosion, vibration and instability in trajectory. The near-wall effect is an important factor in the evolution of complex unsteady cavity evolution. For example, a two-dimensional cavitating flow cannot be easily generated in water tunnels because of sidewall effect. Therefore, the mechanism involved should be investigated to find solutions on controlling such effect in engineering applications.

Studying the effects of wall nearby on cavitation requires complex simulation and test equipment. Given this requirement, a limited number of research has explored this issue. Ishida and Kimoto conducted experimental analysis of the behavior of a single cavitation bubble near a wall to examine cavitation bubbles near a solid boundary using a quite complex test facility [3,4]. Zhou and Chen conducted a comparative study of ventilated supercavity around models with different shapes between the near-wall area and infinite flow [5–7] to explore near-wall effect. Wind tunnel test and CFD simulation were also conducted. He and Kida [8–10] studied near wall effect on supercavitating jet-flapped foils. Other studies focused on the interaction between free surface and cloud cavitating flow; these studies employed simulation methods, such as potential flow theory [11,12], boundary element method (BEM) [13–15] and large eddy simulation (LES) [16].

Most researchers focus on underwater cloud cavitating flow while neglecting wall effect. Experimental and numerical methods are usually used to analyze such problems. Traditional experimental methods include water tank test [17] and water tunnel test [18]. The CFD simulation method has also become increasingly popular in solving hydrodynamic problems; the tools adopted for this method include commercial [19,20], open-source [21,22] and in-house [23–25] software. Typical problems include cavitating flow of unsteady cloud around airfoil [26] and propeller models [27]. For axisymmetric projectile, an early simulation of steady and ventilated cloud cavitating flow around an underwater vehicle was demonstrated by Kunz [28]. Owis showed the cavity evolution of cavitating flow of an unsteady cloud around the same kind of vehicle [29]. Wang determined the relationship between the speed and position of a re-entry jet and adverse pressure gradient. The results of these studies can be used to predict the speed and cavity length of re-entry jets [30].

Good orthogonality and meshing quality is beneficial to the convergence of the calculation and the interface of high-precision capture, which are very important for the large eddy simulation. Fine mesh resolution is important for the LES of cloud cavitating flow, whereas cell size could substantially affect the simulation results and detailed phenomenon of cavity length [31]. For this

* Corresponding author at: Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Chinese Academy of Sciences, Beijing, 100190, China.
*E-mail address:* wangyw@imech.ac.cn (Y. Wang).

http://dx.doi.org/10.1016/j.euromechflu.2017.07.001

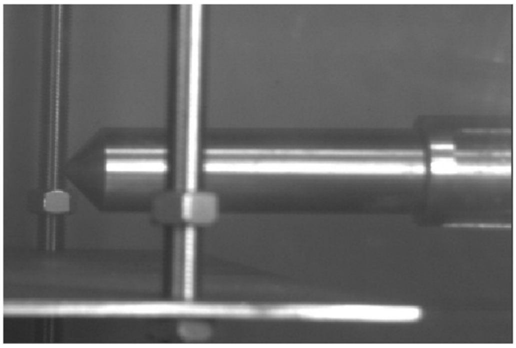

**Fig. 1.** Water tank test facility.

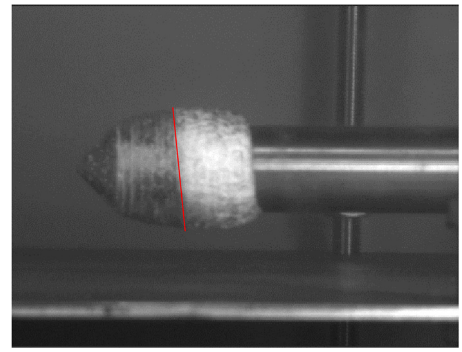

**Fig. 2.** Typical cavitation at $t = 0.006$ s. The white foam like re-entry jet inside the cavity is marked by a red line. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

reason, hexahedral mesh is usually used in the large eddy simulation. However, the structured mesh is difficult to generate for complex geometries, and the number of cells generated is quite large. Therefore, the application of structured grid will actually make the large eddy simulation more time-consuming. The Cartesian cut-cell method is an effective approach for generating unstructured mesh. This approach has underwent significant development in recent years. This approach easily achieves local refinement and perfect orthogonality and is suitable for complex geometries [32,33]. Despite this result, the application of Cartesian cut-cell meshes on unsteady cavitating flow requires further evaluation.

This study mainly involves two parts, namely, water tank test and CFD simulation. The evolution process of the cavity is recorded by a high-speed camera. Instead of using a typical structured mesh, simulation is based on LES, volume-of-fluid approach, and Cartesian cut-cell method for finite volume mesh generation. To facilitate validation, changes in cavity length are compared between that in the experiment and in CFD simulation. Mesh independence is also discussed. The key features of cavitating flow, re-entry jet, cavity shedding, cavity collapse, and the effect of the wall nearby on cavitating flow are analyzed. Consistent regularity is observed between the cavity and vortex motion.

## 2. Water tank experiment

### 2.1. Description of test facility

Fig. 1 shows the facility for water tank test used in the experiment. The model used is a steel cylinder with conical head. The longitudinal section of the cone is an isosceles right triangle. The model measures 200 mm × 37 mm × 37 mm, and the distance between the near wall side of the model and the wall nearby is 25 mm. Split Hopkinson pressure bar technology [17] is used as a launching source under typical conditions. Initially, the model is instantly accelerated to about 20 m/s and launched into a 1 m × 1 m × 2 m water tank. The temperature of the water inside the tank is about 20 °C. The entire experiment is recorded by a high-speed camera with a sampling frequency of 12 000 frames per second.

### 2.2. Typical experiment results and analysis

Fig. 2 shows a typical cavitation photograph at $t = 0.006$ s. Cavitating flow at this point developed into a stable shape. The white foam like re-entry jet inside the cavitating flow moves toward the leading edge of the model is marked by red line. As shown in the figure, the location of the near re-entry jet moves away from the front end of the model as the near wall side of the model approaches the wall nearby. The location of cavity length and re-entry jet at the distal and near wall side of the model can be measured from the figures. Cavity evolution includes cavity growth, cavity shedding, and cavity collapse. This phenomenon can be observed to validate the accuracy of the CFD simulation method in the next step.

The effect of resistance on the speed of the launched model is noticeable during experiment. The launch speed of the model at the beginning and end of the test can be derived from change of leading edge location of the model in the adjacent images, which is about 20 m/s. Cavitation number can be calculated using the following equation:

$$\sigma = \frac{p_\infty - p_v}{\frac{1}{2}\rho_l v_\infty^2} = 0.495 \tag{1}$$

where $p_\infty$ is standard atmospheric pressure, $p_v$ is saturated vapor pressure, $\rho_l$ is liquid water density, and $v_\infty$ is launch speed. The pressure inside the cavity which should be lower than the saturated pressure of water is considered as the cavitation criterion in the paper. There also exist other algorithm which may take the expression of the turbulent kinetic energy of 0.5 pk as a supplement to the saturated pressure, with reference to the Singhal model [34]. For the problem discussed in this paper, the turbulence of the incoming flow is quite low. The turbulence to the saturated pressure caused by the turbulent kinetic energy and the saturated pressure are relatively small comparing to the background pressure and the flow pressure. Small changes of the saturated pressure will not affect the cavitating flow much. The cavitation number in this paper remain constant as the model speed as well as the saturated vapor pressure of water did not change during the experiment. J. H. Kim [35] and D. R. Stinebring [36] discuss the relationship between the cavitation and the cavitation number in details. Given that the model is small and fast, the difference between the pressure exerted by gravity at the distal and near wall side of the model is nearer than flow dynamic pressure. The equation $\frac{\rho_l g d}{\frac{1}{2}\rho_l v_\infty^2} = 0.0018 \ll 1$ shows that variation of local cavitation number in $y$ direction is very small. $d = 37$ mm is the projectile diameter.

## 3. Numerical method

### 3.1. Governing equations

Multiphase flow equations are widely used to describe water-liquid/water-vapor two phases flow problems. The governing equations are,

$$\frac{\partial \rho}{\partial t} + \frac{\partial \left(\rho u_j\right)}{\partial x_j} = 0 \tag{2}$$