Ocean Engineering 140 (2017) 377–387

Contents lists available at ScienceDirect

Ocean Engineering

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

Experimental and numerical investigation on cloud cavitating flow around an axisymmetric projectile near the wall with emphasis on the analysis of local cavity shedding

Chao Yu[a,b], Yiwei Wang[a,b,*], Chenguang Huang[a,b], Tezhuan Du[a], Chang Xu[a], Jian Huang[a]

[a] *Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Chinese Academy of Sciences, No.15 Beisihuanxi Road, Beijing 100190, China*

[b] *School of Engineering Science, University of Chinese Academy of Sciences, China*

ARTICLE INFO

Keywords:
Unsteady cloud cavitation
Effect of wall
Large eddy simulation
Local cavity shedding

ABSTRACT

In this paper, the mechanisms of cavitating flow around an axisymmetric projectile near a wall with local cloud cavity shedding are analyzed using experimental and numerical methods. Several experiments are designed to observe the evolution of cavity around an axisymmetric projectile near a wall underwater with cavitation number $\sigma = 0.45$. Numerical simulations using the large eddy simulation (LES) approach, Kunz cavitation model, and volume of fluid (VOF) method are established in an open-source code OpenFOAM framework to present more information on the flow structure. The shape and length of the cavity are in good agreement with the experimental observation, which guarantees the accuracy of the numerical methods. The characteristics of cavitation around the axisymmetric projectile near the wall are reported, and the wall effect is briefly analyzed according to the experimental observations and the details obtained from the numerical results. Local cavity shedding, which increases the instability of cavity periodic shedding, is observed to be induced by the combination of the vortex structure, jet at the cavity closure, and special cavity shape on the projectile near the wall.

1. Introduction

Cavitation is a classic issue in the hydrodynamic field that has been a subject of study in the past decades. It can be defined as the breakdown of a liquid medium under a low pressure (Franc and Michel, 2004) and widely appears on the surface of underwater vehicles, such as propellers, hydrofoils, and high-speed torpedoes. Cavitation can induce abnormal dynamic behavior, noise, and erosion that can seriously affect cloud cavitation (Soyama et al., 1992; Seo et al., 2008). Thus, the mechanisms of unsteady cavitation have been investigated from various aspects using experimental and numerical methods in the recent years.

Experiment is the main research approach for unsteady cloud cavity. For example, Kubota et al. (1989) provided a detailed description of the flow structure around unsteady cloud cavitation on a stationary two-dimensional hydrofoil using experimental methods. They showed that shed cloud is a large-scale vortex structure containing various small cavitation bubbles. Kawanami et al. (1997) investigated the generation mechanism of cloud cavitation in details. Callenaere et al. (2001) investigated the instability of a partial cavity induced by the development of a re-entrant jet using experiments. In the investigation of Ganesh (2015), a propagating condensation shock wave was the dominant mechanism of periodically shedding cavity. Chen et al. (2015) investigated the cavitation evolution in a convergent–divergent channel with pressure fluctuation through a tunnel experiment and numerical method. Wang et al. (2015a, 2015b) investigated the characteristics of cavity on an axisymmetric projectile near the free surface using a launching experiment and numerical method.

Numerical simulation method presents more details effectively for the clear analysis of unsteady cavitation mechanisms. Developed numerical simulation methods are based on Reynolds-averaged Navier–Stokes (RANS) equations. For example, Watanabe et al. (2003) simulated the unsteady cavitation on a propeller based on a RANS turbulence model and the Singhal cavitation model with the use of the commercial software FLUENT. The cavity shape and pressure

* Corresponding author at: Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Chinese Academy of Sciences, No.15 Beisihuanxi Road, Beijing 100190, China.

E-mail addresses: yuchaohust@163.com (C. Yu), wangyw@imech.ac.cn (Y. Wang), huangcg@imech.ac.cn (C. Huang), dutezhuan@imech.ac.cn (T. Du), xuchang@imech.ac.cn (C. Xu), huangjian@imech.ac.cn (J. Huang).

http://dx.doi.org/10.1016/j.oceaneng.2017.06.007
Received 26 November 2016; Received in revised form 10 April 2017; Accepted 4 June 2017
0029-8018/

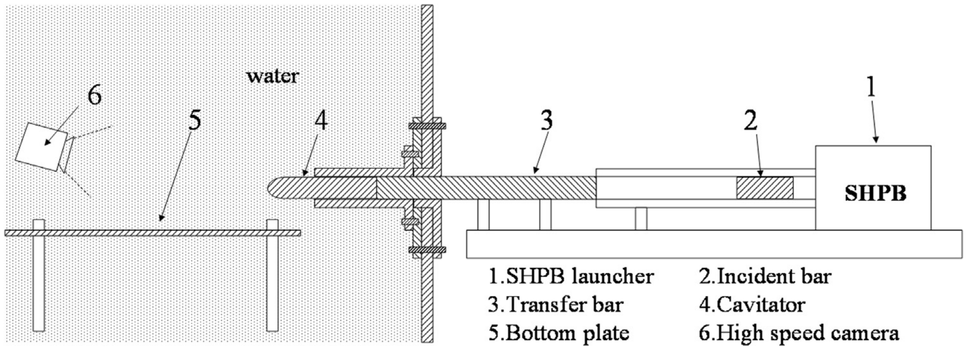

Fig. 1. Underwater launch system. (The height of bottom plate is adjustable.).

fluctuations predicted on the blade surfaces were fairly consistent with the obtained measurements. Zhou and Wang (2008) used the standard renormalization group (RNG) k-ε turbulence model for the stable cavities and the modified RNG k-ε model for the unstable cavity shedding. The relation between the numerical and experimental results were presented. Furthermore, RANS turbulence models have been widely used in the numerical simulations of cavitation flows around other underwater vehicles (Hasuike et al., 2009; Ji et al., 2011; Ji et al., 2012; Ying and Lu, 2008; Decaix and Goncalves, 2013; Goncalves, 2011; Wang et al., 2014).

Although RANS turbulence models are able to provide information on turbulent movement, they still have limitations when simulating the effect of transient cavitation pulsation; whereas large-eddy simulation (LES) methods have performed better in this case. Recently, many cavitating studies have adopted LES methods. For example, Bensow and Bark (2010) simulated unsteady cavitating flows around an INSEAN E779A propeller using implicit LES methods. They proved the validity of the method, pointed out that the LES of cavitation requires further development and exploration, and predicted some important cavitation mechanisms, which were useful in assessing cavitation erosion. Lu et al. (2014) simulated the cavitating flow around two highly skewed propellers operating in open water and mounted on an inclined shaft using and approach based on LES methods. Yu et al. (2014) simulated the cavitating flow around an axisymmetric projectile with cavitation number $\sigma = 0.58$ using LES methods. Their numerical results were in good agreement with the experiment, and presented various cavitation details and mechanisms. Moreover, some favorable results have also been published (Dittakavi et al., 2010; Huang et al., 2014; Ji et al., 2013, 2015; Roohi et al., 2013; Wang and Ostoja-Starzewski, 2007; Wang et al., 2016a, 2016b). Besides, some new approaches are developed to simulate the cavitating flow and get favorable results (Ma et al., 2010; Ma et al., 2017).

Boundary condition, including wall effect, is an important influencing factor for cavity evolution. Studies on the effect of walls in cavitating flow are limited. For example, Xin et al. (2008) used a numerical approach to study the wall effect on ventilated cavity shape and hydrodynamics. They found that the cavity size and the drag coefficient of the ventilated cavitating flow increased with the decrease in diameter of the water tunnel, and the cavity size can be different for the same ventilation rates. In the investigation of Zhou et al. (2010) the blocking effect of water tunnel affected the ventilated super cavity shape seriously. The length of the super cavity increased with the extent of the blocking effect, while its diameter decreased. He et al. (2014) simulated the flow around a hydrofoil with various distances and showed it has significant three-dimensional characteristics because of the side wall. However, in previous literature the effect of the wall were investigated with ventilated cavity rather than natural cavitation. What's more, those investigations of the blocking effect in water tunnel mainly focused on the pressure coefficient of vehicles or volume and cycle of cavity as a whole, but rarely focused on the partial blocking effects (such as local shedding).

In this present paper, new characteristics on the cavity around the projectile near the wall are observed and analyzed in the experiment for a better understanding of the internal mechanisms of unsteady cavitation. Numerical simulations using LES approach, Kunz cavitation model (Kunz et al., 2000), and VOF method are adopted to present more details and mechanisms at a typical condition where the wall is close to the projectile. The evolution process and characteristics of cavitation around the axisymmetric projectile near the wall presented using numerical simulation are in good agreement with the experimental results. Finally, the mechanisms of the wall effect, especially on local cavity shedding, are analyzed according to the experimental observations and the details obtained from the simulations.

2. Experimental setup

2.1. Experimental device

A split Hopkinson pressure bar (SHPB) launching system is established as an experimental method to investigate the characteristics of cavitation around an axisymmetric projectile near the wall (see Fig. 1). The projectile (4) is transiently accelerated by the SHPB launching system (1, 2, and 3) with slight disturbance on the water. The distance between the projectile and the wall are changed by adjusting the height of the bottom plate (5). The high-speed camera (Phantom® v2512) (6) is used to capture the trajectory and cavitation.

2.2. Projectile model and experiment condition

In this study, the test model is an axisymmetric projectile with a cone head (see Fig. 2). The diameter of the projectile is $D = 37$ mm and the length is 200 mm. The distances between the projectile and the bottom plate (d, as shown in Fig. 2) are 5 mm to 40 mm in increments of 5 mm. A contrast experiment without the bottom plate is also set up to analyze the characteristics of the cavitation around the projectile near the wall. The launch velocity is controlled by the pressure in the SHPB launching system, which is 1.2 MPa. The time-averaged velocity of the projectile is about 21 m/s over the first 20 ms after launching (interferences of the velocity instability are introduced in the Appendix section). The density of water is $998.0\ \mathrm{kg/m^3}$, the kinetic viscosity coefficient of water is $\nu = 9 \times 10^{-7}\ \mathrm{m^2/s}$, and the saturated vapor pressure is $P_{sat} = 3160 P_a$ at 25 °C. Therefore, the Reynolds number is $\mathrm{Re} = \frac{UL}{\nu} = 7.77 \times 10^5$(where the characteristic length is defined as the diameter of the projectile, $L = D$, U is the time-averaged velocity of the projectile), and the cavitation number is $\sigma = \frac{P_\infty - P_{sat}}{0.5 \rho_l U^2} = 0.45$ (where P_∞ is the operating pressure which is 1 atm at here, ρ_l is the density of water, U is the time-averaged velocity of the projectile).

2.3. Observation in experiments

The cavity length and the position of the projectile are obtained by measuring the pixels in the pictures. For example, the projectile with a