International Journal of Multiphase Flow 98 (2018) 79–95

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Review

Experimental and numerical investigation of ventilated cavitating flow structures with special emphasis on vortex shedding dynamics

Zhiying Wang, Biao Huang*, Mindi Zhang, Guoyu Wang, Xing'an Zhao

School of Mechanical and Vechicular Engineering, Beijing Institute of Technology, 5 South Zhongguancun Street, Beijing 100081, China

ARTICLE INFO

Article history:
Received 22 August 2016
Revised 10 April 2017
Accepted 26 August 2017
Available online 31 August 2017

Keywords:
Ventilated cavitating flow
PIV
Proper orthogonal decomposition (POD)
Vortex shedding
Vorticity transport equation

ABSTRACT

The objective of this paper is to investigate ventilated cavitating flow structures with special emphasis on vortex shedding dynamics via combining experimental and numerical methods. In the experiments, the high-speed video and time-resolved particle image velocimetry (TR-PIV) technique are used to observe ventilated cavitating patterns, and to measure the flow velocity and vorticity fields. The numerical simulation is performed by CFX with large eddy simulation (LES) model to capture the unsteady cavity shedding process, and the corresponding velocity and vorticity dynamics. The results show that the flow patterns can be classified into two principally different categories: structures mainly with vortex shedding (namely Bénard–Kármán vortex street; Bénard–Kármán vortex street with vortex filaments and Aligned vortices) and relatively stable structures (namely Aligned vortices with Re-entrant jet; Re-entrant jet and Stable supercavity). For the structures mainly with vortex shedding, the Strouhal number *St* corresponding to vortex shedding frequency and ratio h/λ corresponding to vortex streets are significantly different in variable ventilated cavitating regimes: *St* and ratio h/λ increase with enhancement of gas entrainment coefficient Q_v for the Bénard-Kármán vortex street, and then *St* declines gradually for Bénard-Kármán vortex street with vortex filaments and Aligned vortices, but ratio h/λ declines dramatically for the above both patterns. In addition, the influences of Q_v on the velocity and vorticity distributions have also been investigated. The proper orthogonal decomposition (POD) analysis of PIV measurements is used to characterize the coherent large-scale flow unsteadiness of velocity fields. It demonstrates that the ventilated cavitation plays an important role in the first mode pairs to mainly affect the vortex shedding in the wake. Moreover, the vorticity transport equation is applied to illustrate the influence of ventilated cavitation on the vorticity distribution. It can be found that the associated vortex dilatation term and baroclinic torque term are important mechanisms for the complicated change of vortices.

1. Introduction

Ventilated cavitation is obtained by injecting gas into the low pressure regions of liquid flows (Franc and Michel, 2005). There is a growing body of literature dedicated to the study of ventilated cavitation due to its practical importance in engineering and scientific relevance in fluid mechanics (Amromin et al., 2011; Ceccio, 2010; Arndt et al., 2009). Although ventilated cavitation has been studied considerably in the literature, many researchers focus their attention on the stable ventilated supercavitation at relatively larger gas entrainment coefficient (Wosnik et al., 2003; Guo et al., 2010). However, in different gas entrainment coefficient, ventilated cavitation often results in the formation of vortex shedding showing an unsteady behavior. And the phenomenon of ventilated cavitation developing in the shed vortices has been ignored for a long time. Ventilated cavitation vortex shedding can occur in the turbulent wake when gas entrainment coefficient is not large enough to maintain continuous ventilated supercavity. The various kinds of vortices which form, develop and interact with each other in the wake will induce transient loads and lead to further hydrodynamic instabilities. Hence, it is necessary to investigate ventilated cavitation vortex shedding and therefore is a topic worthy of further study (Zou et al., 2013; Ausoni et al., 2007; Gnanaskandan and Mahesh, 2016a; Liu et al., 2017).

Ventilated cavitating flows are inherently complex, involving multiple flow scales, boundary layer separation, jet sprays, and rapidly fluctuating pressure fields and so on (Harwood et al., 2016; Wang et al., 2017). To improve the understanding of the complex structures of ventilated cavitating flows, various experimental

* Corresponding author.
E-mail address: huangbiao@bit.edu.cn (B. Huang).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2017.08.014
0301-9322/

studies have been conducted (Lyer and Ceccio., 2002; Brandner et al., 2010; Kravtsova et al., 2014; Gavaises et at., 2015; Skidmore et al., 2016; Karn et al., 2016; Wang et al., 2017). Although many interesting studies have been reported on ventilated cavitation, most studies concentrated on ventilated cavitation focusing on the shape of the cavitation, the flow velocity, the pressure distributions and hydrodynamic coefficients of the models with cavitation. Kuklinski et al. (2001) performed a series of experiments to examine the stability of ventilated cavities. They found that the dominant cavity frequency was correlated with cavity length and towing speed. Kawakami and Arndt (2011) investigated the ventilated supercavity formed behind a sharp-edged disk utilizing several different configurations. Results regarding cavity shape, cavity closure and ventilation requirements versus cavitation number and Froude number were presented. Wang et al. (2012) investigated the unsteady characteristics of the ventilated partial cavitation around an axisymmetric projectile. They found that the cavity breaks off by the interaction between the gas injection and the re-entry jet at the middle location of the projectile. Wosnik and Arndt (2013) carried out experiments in high-speed water tunnel to investigate the interaction between a ventilated supercavity and its turbulent bubbly wake by the Particle Image Velocimetry (PIV). They found that the bubbly turbulent wake was resulted from the collapse of a ventilated supercavitation. Karn et al. (2016) employed high-speed and high-resolution photography and pressure transducers to understand the physical mechanisms determining closure formation and transition between different closure modes. Closure maps were constructed to depict the flow regimes of each closure mode as a function of Fr and C_{Qs} for different blockage ratios. They also found that once a supercavity was formed, the ventilation rate could be decreased to a much lower value with no change in cavitation number while still maintaining a supercavity. It was just accompanied by a change in closure modes which generally goes from twin vortex, to quad vortex, and then to re-entrant jet flow. Previous researches have mainly focused on stable ventilated cavitation, especially supercavitation, however, only a few studies exist that shed light on the ventilated cavitation vortex shedding in the turbulent wake. In this paper, unsteady characteristics of the ventilated cavitation behind a bluff body are investigated.

As ventilated cavitation often involves complex interactions between turbulent flow structures and multiphase dynamics with large variations in fluid density and pressure fluctuations, the various limitations of measurement techniques have resulted in noticeable efforts to use numerical simulations in recent years. Xiang et al. (2012) applied an Eulerian–Eulerian two-fluid model which was coupled with the balance approach based on MUltiple-SIze-Group (MUSIG) model to simulate the size evolution of the sheared off microbubbles and its complex interactions with the two-phase flow structure in the wake region. Ji and Luo (2010) proposed a three component cavitation model based on the Reynolds-Averaged Navier–Stokes (RANS) equations and mass transfer model to simulate the ventilated cavitating flow as well as natural cavitation. They found that the vapor cavity was suppressed by the gas cavity remarkably with the increase of the gas ventilation. Kinzel et al. (2009) used multiphase computational simulations based on the Navier-Stokes equations to examine the internal gaseous flows of artificially ventilated supercavities. The results indicated that air shear layers that develop on the air-liquid interface surrounding the cavity were an important mechanism of air entrainment. In the numerical modeling of cavitating flows, the selection of turbulence models is also important to predict the unsteady behavior of cavitating flows (Coutier-Delgosha et al., 2007; Goncalvès, 2011; Ji et al., 2013; Hu et al., 2014; Huang et al., 2014; Chen et al., 2015; Wu et al., 2015; Zhang et al., 2015; Chen et al., 2016). In order to better capture the transient turbulence structures, large eddy simulation (LES) using various subgrid

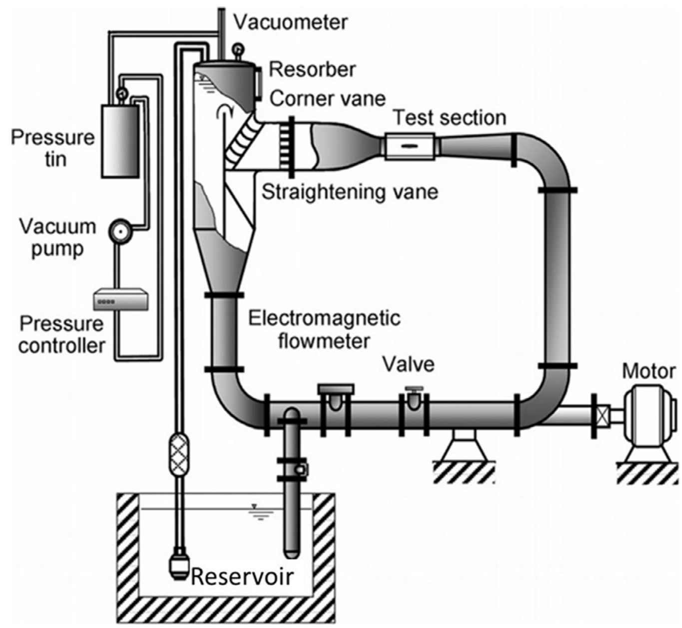

Fig. 1. Schematic of the cavitation tunnel.

models was used to simulate cavitating flows (Nicoud and Ducros, 1999; Kravchenko and Moin, 2000; Hickel et al., 2006; Lysenko et al., 2012; Hickel et al., 2014; Egerer et al., 2014; Ji et al., 2015; Peng et al., 2016). Egerer et al. (2016) develop an efficient large-eddy simulation to simulate the compressible liquid flows with cavitation based on an implicit subgrid-scale model. The method not only can improve computational efficiency by retaining the four-cell stencil of the baseline scheme, but also has better robustness for the simulation of cavitating flows. Gnanaskandan and Mahesh (2016b) applied a LES model to quantitatively investigate various aspects of the transition from sheet to cloud cavitation over a wedge. The present work applies a LES Wall-Adapting Local Eddy-Viscosity (WALE) model to concentrate on the structure of the ventilated cavitating flow and its shedding dynamics.

Although considerable researches have been focused on ventilated cavitating flow for decades, they focused their attention on the relatively stable ventilated supercavity. Little attention has been given to ventilated cavitation vortex shedding dynamics. In this study, the comprehensive understanding of the topology and formation of ventilated cavitation, the transition mechanism of ventilated vortex shedding patterns to the relatively stable ventilated supercavity and the ventilated cavitation vortex shedding dynamics behind a bluff body are investigated combing experimental and numerical method. The paper is organized as follows. Details of experimental setup are described in Section 2. The numerical setup and description are showed in Section 3. Detailed results and discussions including observations of different flow patterns and velocity and vorticity distributions are given in Section 4. Finally, the findings drawn from this investigation are summarized in Section 5.

2. Experimental setup

2.1. Cavitation tunnel

Experimental studies are conducted in a closed-loop cavitation tunnel, which is shown in Fig. 1. The test section is 0.7 m long and has a rectangular section with width of 0.07 m and height of 0.19 m. The axial flow pump is located 5 m below the test section to drive the flow into the tunnel. To separate the undesired free