

Experimental and numerical investigation of ventilated cavitating flow with special emphasis on gas leakage behavior and re-entrant jet dynamics



Zhiying Wang^a, Biao Huang^{a,*}, Guoyu Wang^a, Mindi Zhang^a, Fufeng Wang^{a,b}

^a School of Mechanical and Vehicular Engineering, Beijing Institute of Technology, 5 South Zhongguancun Street, Beijing 100081, China

^b School of Changing Non-Commissioned Officers, Academy of Equipment, Beijing 1002249, China

ARTICLE INFO

Article history:

Received 21 March 2015

Accepted 30 July 2015

Keywords:

Ventilated cavitating flow

High-speed camera

Gas leakage behavior

Re-entrant jet dynamics

ABSTRACT

The objective of this paper is to investigate the ventilated cavitating flow structure by combining experimental and numerical methods. A high-speed camera technique is used to record cavity evolution patterns. The numerical simulation is performed by CFX with a free surface model and a filter-based model, and the gravity effect is considered. The results show when the gas entrainment coefficient Q_g is constant, two typical mechanisms of the gas leakage exist at different Froude numbers Fr , namely toroidal vortices mode and two hollow tube vortices mode. With the increasing of Fr , the cavity would transfer from the two hollow tube vortices to the toroidal vortices. Moreover, when the Fr number keeps constant, the enlargement of the cavity causes the gravitational effect to be more significant for the case of larger value of Q_g . The detail analysis of re-entrant behaviors is also conducted. One type of re-entrant flow is unsteady with air cluster being periodically rejected at the rear of the cavity. The other type of the re-entrant flow shows that the majority of the cavity is transparent, only the region at the tail of the cavity is nontransparent, due to the re-circulation of water back into the cavity.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

Occurrence of cavitation in hydrodynamic applications such as test bodies, propellers and underwater weapon (torpedoes and projectiles) can lead to problems including pressure pulsations, sudden changes in loads, vibration, noise, and erosion. Most of these problems are related to the transient behavior of cavitation structures (Brennen, 1995; Joseph, 1995; Arndt, 2002; Ji et al., 2014). Although the cavitation may not be avoided, it is not always an undesired phenomenon in fluid dynamics. For the past decade, the researchers try their best to minimize the undesired effects of cavitation and maximize the advantage of cavitation. Especially, drag reduction by ventilated cavitation (Amromin et al., 2006, 2011) is a topic of great interest. Supercavitation (Franc and Michel, 2005; Chen et al., 2008; Wosnik and Arndt, 2013) may be maintained through vaporous cavitation sustained by a sufficiently high speed or by ventilated cavitation through the artificial injection of gas. Ventilated supercavitation (Ceccio, 2010; Arndt et al., 2009) is one of the successful technologies for skin friction reduction of underwater vehicles.

To improve the understanding of the complex structures of cavitating flows, various experimental studies have been conducted.

In experimental study, by decreasing the cavitation number, four different types of cavitation can be defined: inception cavitation, sheet cavitation, cloud cavitation, and supercavitation (Wang et al., 2001). Schauer (2003) conducted experimental study of a ventilated supercavitating vehicle. He quantified the gas injection coefficient under a variety of different cavitations and Froude numbers corresponding to the re-entrant flow regime. Wosnik et al. (2003) carried out in high-speed water tunnel to investigate some aspects of the flow physics of a supercavitating vehicle. The digital strobe photography images were taken to describe the ventilated cavity shape and wake details, and the Particle Image Velocimetry (PIV) was applied to characterize the wake of ventilated cavities. It was found the strut shape critically affected air demand through cavity-strut wake interaction. Kopriva et al. (2005) performed an experimental study of test body drag reduction by ventilated partial cavitation in steady and unsteady flows. They found that the ventilated cavitation on the OK-2003 test body was effective in reducing drag and sharply increasing the lift to drag ratio. 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. Zou et al. (2013) discussed the effect of supercavity shedding on supercavity stability combining models of gravity and angle of attack with Logvinovich model. They found that

* Corresponding author. Tel.: +86 010 68912395 607.

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

the supercavity shedding was the intrinsic property and was closely related to the dynamics parameter. Saranjam (2013) investigated on unsteady supercavitation characteristics, including formation, evolution and loss of supercavitation. To achieve the goal, the natural supercavitation was conducted through a moving body in experiment. And for the numerical simulation, the RANS equations are coupled to a six-degree-of freedom rigid body motion model.

It is known that cavitation always involves complex interactions between turbulent flow structures and multiphase dynamics with large variations in pressure fluctuations, fluid density and instability (Tseng and Shyy, 2010; Callenaere et al., 2001; Ji et al., 2013; Huang et al., 2014; Stanley et al., 2014). As the phenomenon is complex, these physical mechanisms are not well understood, and also, the present of discrete interfaces, non-equilibrium interfacial dynamics make numerical and physical modeling of multiphase flow rather challenging. Most numerical models of cavitating flow assume the fluid to be homogeneous and isothermal, and apply one vapor–liquid mixture

continuity equation (Kunz et al., 2001; Singal et al., 2002; Seif et al., 2010). The key challenging here is how to define the mixture density. Based on the assumption of a homogenous equilibrium medium, many cavitation models are proposed. Kubota et al. (1992) proposed a cavitation model where the liquid–vapor mixture is treated as a single fluid to satisfy the Navier–Stokes equations. Some researchers (Merkle et al., 1998; Kunz et al., 2000; Zhang and Khoo, 2014; Wang et al., 2014) have employed the artificial compressibility method with special attention given to the preconditioning formulation. Singhal et al. (2001) considered the effect of noncondensable gas in their “full cavitation model”. Kunz et al. (2000) pursued a three species formulation of the Navier–Stokes equations, which account for the liquid, vapor as well as an incondensable gas field for the natural and ventilated cavitating flows. Ji et al. (2010) used a three-component model to capture the cavity development for natural and ventilated cavitation. Xiang et al. (2012) handled the two-phase flow field in the ventilated partial cavity through the Eulerian–Eulerian two fluid model, coupling with the population balance approach. Celik et al. (2014) developed a new method to predict the cavity on two-dimensional (2D) and three-dimensional (3D) hydrofoils by a potential-based Boundary Element Method. In order to reduce smearing at the cavity interface and capture the sharp interface, Huang et al. (2011) used the Level Set method with a compressive

discretization scheme in both time and space to minimize the smearing of the free surface at the interface for simulation of natural and ventilated cavity.

In the numerical modeling of cavitating flows, the selection of turbulence models is important to predict the unsteady behavior of cavitating flows. The Reynolds-Averaged Navier–Stokes (RANS) turbulence models are widely used for single phase, but it is not suitable for compressible two-phase mixture flows, so some modifications (Goncalves, 2011; Hu et al., 2014) are required. In order to better capture the transient turbulence structures, a large eddy simulation (LES) model was used to simulate sheet/cloud cavitation on a NACA0015 hydrofoil (Wang and Ostojic-Starzewski 2007; Ji et al., 2015). Johansen et al. (2004) formulated a filter-based model (FBM) as a compromise between RANS and LES. Wu et al. (2005) assessed the validity of FBM turbulence model through unsteady simulations of different geometries. They found that the FBM could be better to capture the unsteady features than standard RANS models.

Although ventilated cavitation has received much attention for decades, the structure of ventilated cavitation in different conditions is still not well understood, and hence additional studies are still needed combined the experimental and numerical investigation. The aims of the present study are to (1) gain a broad and improved understanding of the physics of ventilated cavitating flow, (2) quantify the influence of Froude number Fr and gas entrainment coefficient Q_v on the global flow structures, (3) improve the understanding of the detail structures of the unsteady ventilated cavitation via physical and numerical analysis.

2. Experimental setup

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 stream bubbles in the flow, a tank with volume of 5 m³ is placed upstream of the test section, with a corner vane and a straightening vane being set to reduce the turbulence level of the flow. The vacuum pump is connected to the top of the tank controlling the pressure in the tunnel.

The cavitation patterns are observed with a high-speed digital camera (HG-LE, by Redlake) at a sampling frequency of 3000 fps to maintain desirable spatial resolutions, with two light sources illuminating the flow field from different directions. Fig. 2 shows the schematic measurement of the cavitation phenomenon.

In present study, the ventilated cavitation study is carried out using a test body with a sharp-edged disk at the nose and special ventilation ports. A schematic of the test body is shown in Fig. 3. The length of the test body is $L=0.12$ m, and the diameter is $D=0.02$ m. The position of the test body inside the test section is shown in Fig. 4. The test section is $35D$ in length, $9.5D$ in height, and $3.5D$ in width. The flow can be observed from three windows, located at the top, bottom, and side. The cavitation tunnel is capable of generating free stream velocity ranging from 2 to 15 m/s with minimum nature cavitation number of $\sigma=2(p_\infty-p_v)/(\rho_l U_\infty^2)=0.3$, where p_∞ is the reference static pressure, p_v is the saturated vapor pressure of water, ρ_l is the water density, and U_∞ is the free stream velocity. The test body is clamped to the back wall of the test section with a mechanical locking system.

In order to ensure the formation of the ventilated cavitation, a gas ventilated system is designed and used, which is shown in Fig. 5, the main components include gas supply, pressure-regulating system, flow measurement devices. In this system, the ventilation pressure and speed can be freely adjusted, and the volumetric flow rate can be accurately measured by the rotameter.

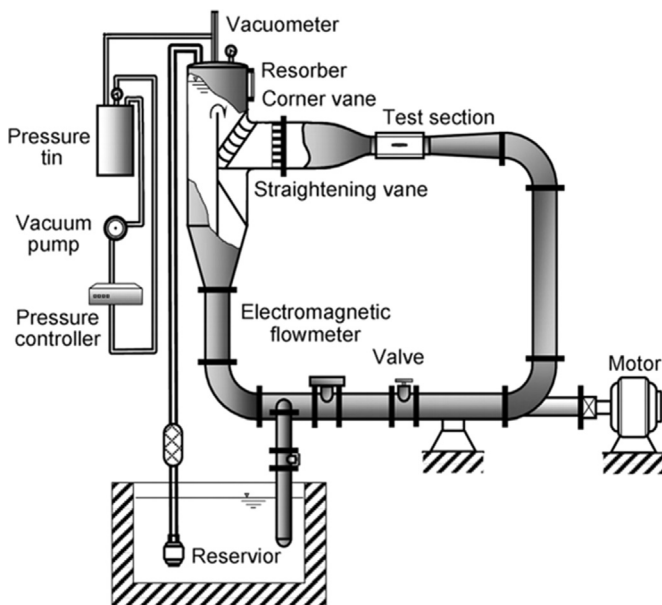


Fig. 1. Schematic of the cavitation tunnel.

Download English Version:

<https://daneshyari.com/en/article/8065412>

Download Persian Version:

<https://daneshyari.com/article/8065412>

[Daneshyari.com](https://daneshyari.com)