Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2017,29(6):979-986
DOI: 10.1016/S1001-6058(16)60812-0
www.sciencedirect.com/
science/journal/l0016058

The effect of free surface on cloud cavitating flow around a blunt body *

Chang Xu（徐畅）[1,2], Yi-wei Wang（王一伟）[1,2], Chen-guang Huang（黄晨光）[1,2], Jian Huang（黄荐）[1,2], Chao Yu（余超）[1,2],

1. *Key Laboratory for Mechanics in Fluid Solid Coupling Systems, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China*
2. *School of Engineering Science, University of Chinese Academy of Sciences, Beijing 100049, China,*
E-mail: xuchang@imech.ac.cn

(Received August 24, 2017, Revised September 6, 2017)

Abstract: In this study, the effect of the free surface on the cloud cavitating flow around a blunt body is investigated based on the water tank experiment and the CFD method. Numerical results are in good agreement with experimental data, and the mesh independence of the methods is verified. The cavity evolution process includes the cavity growth, the re-entrant jet, the cavity shedding, and the collapse, which can all be observed from the water tank experiment. The effects of the free surface on the cavity length, the thickness, and the cavity evolution period are analyzed by comparing the difference between the cavitating flows on the upper and lower sides of the body. This study also examines the effect of the distance between the free surface and the model through a series of water tank experiments and numerical simulations. The cavity stability and asymmetry, as well as the thickness and the velocity of the re-entrant jet inside the cavity, which varies with the submerged depth, are discussed with consideration of the effect of the free surface. The effect of the free surface on the cavitating flow around the blunt body is enhanced with the decrease of the submerged depth.

Key words: Cavitation, multiphase flow, water tank experiment, CFD, free surface

Introduction

The cavitation is one of the classic problems in high-speed hydrodynamics when underwater vehicles move in great speed[1-4]. The induced instable phenomena can cause serious consequences, such as noises, erosion, and vibrations of the structure. The problem becomes complicated when the interaction between the free surface and the cloud cavitating flow on the model is considered. The water tunnel[5] and water tank[6,7] tests are usually performed to analyze the problem. In recent years, the CFD method becomes one of the main research methods used for the cavitation flow, including the potential flow theory[8,9], the boundary element method (BEM)[10-12], the large eddy simulation (LES)[13-16], and other approaches[17-19] with commercial software, such as the CFX, the FLUENT[20,21] and other open source software, such as the OpenFOAM[22-24].

The interaction between the free surface and the cavitating flow is a very complex and interesting problem. Based on the numerical and experimental methods mentioned above, the flow characteristics and the mechanism of unsteady cavities were studied. The mechanism of main control parameters, such as the submerged depth, the cavitation number, the Froude number, and the gravity, which affect the cavitating flow, were analyzed based on experiments and numerical simulations[1,9,25]. Wang[13,26] studied the cloud cavitating flow around an axisymmetric projectile near the free surface, including the effect of the free surface on the cavity shape, the cavity evolution process, the re-entrant jet inside the cavity, and the vortex structure. The CFD simulations were conducted, and the results were found to be consistent with the water tank experiment data. Moreover, the atmospheric ventilation flow around a blunt body near the free surface was discussed[27]. Ventilated cavitation occurs when the model is sufficiently close to the free surface. The entrainment of a strong air into the ca-

---

* Project supported by the National Natural Science Foundation of China (Grant Nos. 11772340, 11672315), the Youth Innovation Promotion Association of CAS (Grant No. 2015015).

Biography: Chang Xu (1992-), Female, Master
Corresponding author: Yi-wei Wang,
E-mail: wangyw@imech.ac.cn

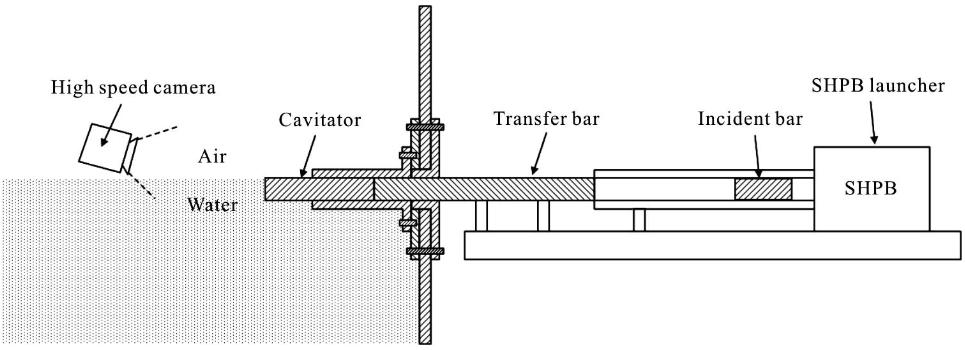

Fig.1 Water tank test facilities

vity on the upper side of the blunt body induces a large and stable cavity. The effects of other boundary conditions, such as that of the near-wall on the cloud cavitating flow around vehicles were also discussed in the recent studies[28-30].

In this work, the water tank experiments and the numerical simulations are performed to analyze the effect of the free surface on the cloud cavitating flow around a blunt body in various submerged depths. The accuracy of the numerical method and the mesh independence are verified. The cavity evolution processes, including the cavity growth, the re-entrant jet, the cavity shedding, and the collapse, can be observed through the experimental data. We first discuss the effects of the free surface on the cavity length, the thickness, and the cavity evolution period. Then, the effects of the free surface on the cavity stability, the asymmetry, and the thickness and the velocity of the re-entrant jet inside the cavity of various submerged depths are examined under a series of working conditions.

1. Water tank experiment

The water tank test facilities are shown in Fig.1. The tested model in the experiment is a slender, polished stainless-steel cylinder of 37 mm in diameter. The launching process is based on the Split-Hopkinson pressure bar technology[6], which could accelerate the launched model to a speed of 18.5 m/s in less than 50 μs. The entire cavity evolution process could be recorded by a high-speed camera with 25 000 frames per second. The water temperature is approximately 20°C. In the following sections, the cavity evolution will be mainly discussed based on experimental pictures and numerical results. The cavity shape changes with the submerged depth at the launch time, the cavitation phenomenon can be classified by the shape development into the cloud cavitation[13], the natural ventilation[27] and the supercavitation as the submerged depth decreases. We mainly focus on the cloud cavitating flow in this paper with the submerged depth varying from 15 mm to 40 mm. There will be no free surface effect on the cloud cavi- tating flow around the projectile when the distance between the upper side of the projectile and the free surface exceeds 40 mm.

2. Numerical methods

2.1 *Governing equations*

The multiphase flow equations are extensively used for solving the water-liquid/water-vapor two-phase flow problems. The governing and momentum equations are expressed as:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u_j)}{\partial x_j}=0 \tag{1}$$

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_i u_j)}{\partial x_j}=-\frac{\partial p}{\partial x_j}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial u_j}{\partial x_j}\right) \tag{2}$$

where u_i is the velocity component in the i direction, ρ is the mixture density, p is the pressure, and μ is the laminar viscosity, which can be defined as

$$\mu=(1-\alpha_v)\mu_l+\alpha_v\mu_v \tag{3}$$

where α is the volume fraction of the different phases, and l and v represent the liquid water and the water vapor, respectively. The mixture density ρ is defined as

$$\rho=(1-\alpha_v)\rho_l+\alpha_v\rho_v \tag{4}$$

The transport equation of the water vapor volume fraction is

$$\frac{\partial(\alpha_v\rho_v)}{\partial t}+\frac{\partial(\alpha_v\rho_v u_j)}{\partial x_j}=\dot{m}^+-\dot{m}^- \tag{5}$$