Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2016,28(3):431-441
DOI: 10.1016/S1001-6058(16)60646-7

www.sciencedirect.com/
science/journal/10016058

Numerical investigation of the effect of rotation on cavitating flows over axisymmetric cavitators*

Ye-jun GONG (龚也君), Jie-min ZHAN (詹杰民), Tian-zeng LI (李天赠)
Department of Applied Mechanics and Engineering, Sun Yat-sen University, Guangzhou 510275, China,
E-mail: yejungong@126.com

(Received February 5, 2015, Revised May 21, 2015)

Abstract: The rotating axisymmetric cavitator is widely applied in underwater vehicles, and its rotational motion affects the cavitating flow over the cavitator. This study focuses on the effect of rotation on the flow structure in the cavity bubble. Unsteady 2-D/3-D numerical simulations of cavitating flows over axisymmetric cavitators are performed using the volume of fraction (VOF) method and the Sauer-Schnerr cavitation model. Firstly, the 2-D simulation of cavitating flow over a circular disk or a cone cavitator is carried out at various cavitation numbers (0.15, 0.175, 0.2, 0.225 and 0.25). The simulated cavity lengths and drag coefficients are compared with the experimental data, the theoretical estimations and the published numerical results. Then the 3-D simulations of cavitating flows over the same axisymmetric cavitators with different rotating speeds are performed using the sliding mesh model (SMM). The effect of rotation on the cavity shape and the internal flow structure is analyzed.

Key words: rotation cavitator, volume of fraction (VOF), sliding mesh model

Introduction

The supercavitation technology is to use the cavitation effect for a high-speed underwater vehicle to create a gas bubble, which engulfs the whole body of the vehicle and greatly reduces its skin friction, and then the underwater vehicle can travel at a higher speed. The main applications of the supercavitation technology are in high-speed underwater vehicles, e.g., the supercavitating propeller (Brave-class, Vosper, Portsmouth, England, 1958), the supercavitating projectile (RAMICS, U.S. Navy Forces, 1994) and the supercavitating torpedo (Barracuda, Diehl BGT Defence, Berlin, German, 2004). In recent years, the supercavitation technology attracts more and more interest[1,2].

The supercavitation occurs when the cavitation number σ is lower than 0.1. The cavitation number, $\sigma = 2(p_\infty - p_v)/(\rho U^2)$, depends on the vapor pressure p_v, the ambient pressure p_∞, the fluid density ρ and the free stream velocity U. The supercavitation condition, $\sigma < 0.1$, can be reached through: (1) increasing U, (2) decreasing p_∞ of a closed-circuit cavitation tunnel, or (3) increasing p_v through ventilating non-condensable gases[1]. The third way is called the ventilated supercavitation as in contrast to the natural supercavitation.

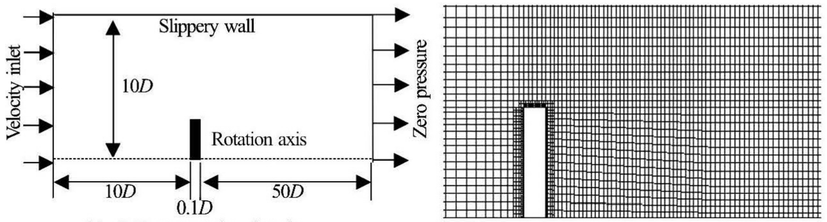
In practical applications, the leading edge of the submerged body, i.e., the "cavitator", is designed to enhance the supercavitation. The shape of the cavitator is one of the main factors affecting the cavity shape[3]. Additionally, the rotation of the cavitator can also influence the cavity shape[4]. The rotating cavitator is commonly seen in industrial and military fields, and has attracted the attention of scientists for decades. The aim of this study is to investigate the effect of rotation on the cavitating flow structure inside the cavity bubble.

The supercavitation technology has been investigated by means of experimental measurements, semi-empirical models and numerical simulations[5,6]. With the development of computational fluid dynamics

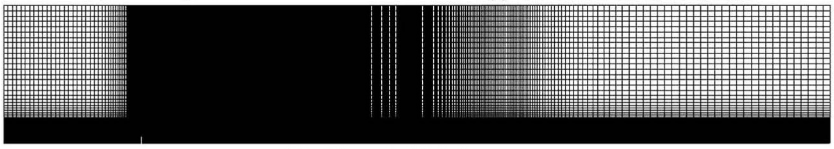
---

* Project Supported by the Sepcial Research Program of Public Welfare and Capacity Building in Guangdong Province (Grant No. 2015A020216008).

Biography: Ye-jun GONG (1980-), Female, Ph. D., Research Associate Professor
Corresponding author: Jie-min ZHAN,
E-mail: cejmzhan@vip.163.com



(a) 2-D computation domain (b) 2-D mesh local view near disk



(c) 2-D mesh global view

Fig.1 Computational domain and mesh in 2-D disk case

(CFD) and computer technologies in recent years, the numerical method is used to visualize the internal flow structure inside the vapor cavity. The re-entrant jet in the cavity closure region forces the gas convection in the cavity bubble and induces the vortex flow, which is hard to be measured in laboratory. Obviously, when the cavitator is rotating with a high speed, the internal flow structure in the gas bubble is even more complex under the action of inertial forces. In this study, 3-D cavitating flows over a circular disk and a cone cavitator are numerically simulated to see how the rotational movement of the cavitator influences the flow structure of the cavitating flow.

In the early numerical investigations of the cavitating flow, empirical correlations are widely used[5], and the flow was assumed to be inviscid[7]. However, the viscous effect on the shape of the cavity was observed experimentally. Furthermore, the linearized theory can not predict the vortex structure in the cavitating flow. In the Street's research on the rotating cavitation, the vorticity is assumed to be constant, and the stream function of the rotational flow is resolved by solving a simplified Poission's equation[4].

To further study the complex cavitating flow structure, it is necessary to develop new cavitation models. Based on the boundary iteration methodology, a boundary method was developed for the study of the steady sheet cavitation and it can also be used for the propeller cavitation[8]. The Rayleigh-Plesset equation describing the bubble dynamics was used to simulate the cavitation in a rotational supercavitating evaporator[9,10]. For the unsteady turbulent cavitating flows, several multiphase CFD methods were developed, and the homogeneous equilibrium model (HEM) was widely employed[11]. One type of the HEM method is based on the barotropic law of state[12], which is widely used but can not capture the vorticity production in a closure region due to the cavity collapse. Another HEM method is the transport-equation based cavitation model (TEM), which solves the transport equation for the vapor volume fraction with a source term describing the mass transfer due to the cavitation process. The source term is calculated using the mass transfer cavitation model[13,14]. In the past decade, the TEM was applied extensively in the numerical simulation of the cavitation phenomena[15].

In this study, the volume of fluid (VOF, Hirt, 1981) method is applied to capture the boundary of the cavity bubble under the TEM framework. The VOF technique can capture the small scale vortex flow structure in the gas bubble, therefore, it was applied widely in the simulation of the natural super-cavitation, the ventilated super-cavitation and the rotating cavitation[1,16]. The TEM+VOF method can capture the abundant vortex structures in the cavity bubble behind the rotating cavitator, where the rotational movement of the cavitator is accounted for by using the sliding mesh model (SMM) in the ANSYS Fluent. The present numerical method will be verified by the experimental measurements of cavitating flows over a circular disk and a cone cavitator. Then the validated method is applied to the 3-D simulations of cavitating flows over the same cavitators with different rotating speeds. The cavity bubble shape, the drag coefficient of the cavitator, and the flow structure in the cavity bubble will be compared and analyzed.

1. Mathematical modeling

1.1 *Governing equation*

The governing equations of the two-phase cavitating flow are as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{u}) = 0 \tag{1}$$

$$\frac{\partial \rho \boldsymbol{u}}{\partial t} + \nabla \cdot (\rho \boldsymbol{u}\boldsymbol{u}) = -\nabla \cdot p + \nabla \cdot (\mu \nabla \boldsymbol{u}) + \boldsymbol{F}_{st} + \rho \boldsymbol{g} \tag{2}$$