+ MODEL
ARTICLE IN PRESS
+ MODEL
Available online at www.sciencedirect.com
ScienceDirect
Publishing Services by Elsevier
International Journal of Naval Architecture and Ocean Engineering xx (2016) 1—10
http://www.journals.elsevier.com/international-journal-of-naval-architecture-and-ocean-engineering/

# Drag reduction of a rapid vehicle in supercavitating flow

D. Yang [a], Y.L. Xiong [b,c,*], X.F. Guo [d]

[a] *School of Naval Architecture and Ocean Engineering, Huazhong University of Science & Technology, Wuhan, 430074, China*
[b] *School of Civil Engineering and Mechanics, Huazhong University of Science & Technology, Wuhan, 430074, China*
[c] *Hubei Key Laboratory of Engineering Structural Analysis and Safety Assessment, Luoyu Road 1037, Wuhan, 430074, China*
[d] *The Second Institute of Huaihai Industrial Group, Changzhi, 046000, China*

Received 13 November 2015; revised 13 May 2016; accepted 12 July 2016
Available online ■ ■ ■

**Abstract**

Supercavitation is one of the most attractive technologies to achieve high speed for underwater vehicles. However, the multiphase flow with high-speed around the supercavitating vehicle (SCV) is difficult to simulate accurately. In this paper, we use modified the turbulent viscosity formula in the Standard K-Epsilon (SKE) turbulent model to simulate the supercavitating flow. The numerical results of flow over several typical cavitators are in agreement with the experimental data and theoretical prediction. In the last part, a flying SCV was studied by unsteady numerical simulation. The selected computation setup corresponds to an outdoor supercavitating experiment. Only very limited experimental data was recorded due to the difficulties under the circumstance of high-speed underwater condition. However, the numerical simulation recovers the whole scenario, the results are qualitatively reasonable by comparing to the experimental observations. The drag reduction capacity of supercavitation is evaluated by comparing with a moving vehicle launching at the same speed but without supercavitation. The results show that the supercavitation reduces the drag of the vehicle dramatically.


*Keywords:* Drag reduction; Numerical simulation; Supercavitation; Underwater vehicle; Turbulent model; Multiphase flow; High-speed torpedo; Supercavitating vehicle; Computational fluid dynamics; Cavitation number

## 1. Introduction

Hydrodynamic drag is one of the greatest interests in marine hydrodynamics and aerodynamics. For a moving vehicle, drag force on one hand induces a lot of energy consumption in our daily life. On the other hand, it limits the speed of the vehicle. In addition, the drag force of a stationary object enhances structural load, which is often accompanied with high cost during the design process to maintain the structure strength. As a consequence, drag reduction is a long-standing challenge for both scientists and engineers. A series of flow control methods have been put forward for decreasing drag, such as putting tiny amount of polymer additives into water (Xiong et al., 2010; Xiong et al., 2013), using porous media (Bruneau and Mortazavi, 2008), as well as shape optimization (Bruneau et al., 2013), and so on (Beaudoin and Aider, 2008; Choi et al., 2008). One of the most promising ways to reduce drag resistance is by filling water vapour or gas to isolate the underwater vehicle, one of which is well known as supercavitating drag reduction (Arndt et al., 2005; Ceccio, 2010; Arndt, 2013). In a supercavitating flow, the cavitator contacts with water constantly. The rest parts of the vehicle mostly contact with either the saturated water vapour produced by natural phase exchange or the gas releases from an artificial ventilation vent. Since the surface of underwater vehicle contact with vapour or gas, the friction drag exerted on the vehicle is negligible. Therefore, the vehicle could achieve very high speed in water. This technology supplies us an alternative of high speed voyage in the future.

* Corresponding author. School of Civil Engineering and Mechanics, Huazhong University of Science & Technology, Wuhan, 430074, China.
*E-mail address:* xylcfd@hust.edu.cn (Y.L. Xiong).
Peer review under responsibility of Society of Naval Architects of Korea.

http://dx.doi.org/10.1016/j.ijnaoe.2016.07.003

However, it should be noted that this technique is still in the very early stage even though it has been utilized in the so-called supercavitating torpedo (Choi et al., 2005a,b; Alyanak et al., 2006). There are several challenges need to be overcome in order to achieve manipulated flight in the water: 1) the balance of the gravity of the vehicle is difficult since the Archimedes force is becoming negligible because the vehicle is enveloped by low-density medium; 2) the traditional propeller is ineffective since it is difficult to extend the turbine into water in order to get thrust by pushing water; 3) it is hard to control the moving direction of the underwater vehicle; 4) how to isolate the tremendous noise aroused by the condensation of bubbles; 5) how to brake the vehicle safely. All of the above questions are related to the fundamental problems, which are among how to calculate the unsteady pressure drag on the cavitator, as well as the shape and size of the cavity accurately, and how to optimize the shape of cavitator to achieve less drag but larger cavity.

To solve these problems, both the potential theory and Computational Fluid Dynamics (CFD) are utilized to calculate the shape of the cavity and the corresponding drag (Dieval et al., 2000; Choi et al., 2005b; Alyanak et al., 2006; Gao et al., 2012; Likhachev and Li, 2014; Pan and Zhou, 2014). Furthermore, there are also a great number of the semi-empirical theories and experimental studies have been done on the supercavitating flow (Tulin, 1998; Hrubes, 2001; Ito et al., 2002; Kulagin, 2002; Nouri and Eslamdoost, 2009; Yi et al., 2009; Cameron et al., 2011; Kim and Kim, 2015). In general, the semi-empirical theory and the potential theory could give an accurate result quickly, however they are not so versatile for the transient flow and complex geometric configuration. Meanwhile they are incapable to give detailed flow behaviour of the cavitation (Tulin, 1998; Kim and Kim, 2015). On the contrary, CFD and the experimental measurements are flexible to obtain abundant results, but they are both time-consuming and hard to implement. Especially, the cavitation model, multiphase flow method, numerical methods, and the turbulence model influence the accuracy of the result (Singhal et al., 2002; Coutier-Delgosha et al., 2007; Seif et al., 2009).

In this manuscript, we are going to examine our numerical results by experimental data, and then we are going to study the drag coefficient of different cavitators by numerical simulations. In the last part of the paper, the supercavitating drag reduction capacity is compared by using an unsteady numerical simulation. The selected case corresponds to our outdoor supercavitating experiment.

## 2. Governing equations and computational setup

A single fluid approach was used to simulate the unsteady flow of mixture phase consisting of vapour phase and water phase. The governing equations which are composed by mass and momentum conservation equations as well as a transport equation of water vapour have the following form, respectively:

$$\frac{\partial}{\partial t}(\rho_m) + \nabla\cdot\left(\rho_m \vec{v}_m\right) = \dot{m} \tag{1}$$

$$\frac{\partial}{\partial t}\left(\rho_m \vec{v}_m\right) + \nabla\cdot\left(\rho_m \vec{v}_m \vec{v}_m\right) = -\nabla p + \nabla\cdot\left[\mu_m\left(\nabla v_m + \nabla v_m^T\right)\right] \tag{2}$$

$$\frac{\partial}{\partial t}(\rho_m f) + \left(\rho_m \vec{v}_v f\right) = \nabla(\gamma\nabla f) + R_e - R_c \tag{3}$$

Here $\vec{v}_m = \sum_{k=1}^{2}\alpha_k\rho_k\vec{v}_k/\rho_m$ is the averaged velocity of mass; $\alpha_k$ denotes a volume fraction of phase-$k$, the subscript $k$ and $m$ represent the phase-$k$ and mixture phase, respectively. The mixture density and viscosity are calculated by$\rho_m = \sum_{k=1}^{2}\alpha_k\rho_k$ and $\mu_m = \sum_{k=1}^{2}\alpha_k\mu_k$. The mass fraction of water vapour $f$ can be calculated based on the density relation as $1/\rho_m = f/\rho_v + (1-f)/\rho_l$, the suffix of $m$, $l$ and $v$ denote the mixture, liquid and vapour phase, respectively. To improve the numerical stability, an artificial diffusion term in the transport equation is employed; the diffusion coefficient is reasonable small and may avoid a sharp interface which arises remarkable numerical instability in cavitating simulation. The source term $R_e$ and $R_c$ in the transport equation represent the generation and condensation of vapour, respectively. Source terms are sensitive to the local absolute static pressure and turbulent kinetic energy. Here we adopted the Singhal's cavitation model which has the following form:

$$if\ p \le p_v: \quad R_e = C_e\frac{V_{ch}}{\zeta}\rho_l\rho_v\sqrt{\frac{2(p_v-p)}{3\rho_l}}(1-f) \tag{4}$$

$$else\ if\ p > p_v: \quad R_c = C_c\frac{V_{ch}}{\zeta}\rho_l\rho_v\sqrt{\frac{2(p-p_v)}{3\rho_l}}f \tag{5}$$

where $p_v$ is the phase change threshold pressure which is calculated by $p_v = p_{sat} + 0.5p_{turb}$ and $p_{turb} = 0.39\rho_m k$, here $k$ is the turbulent kinetic energy. $V_{ch}$ is a characteristic velocity which measure the effect of local relative velocity between liquid and vapour and is estimated as$\sqrt{k}$. The empirical coefficient $C_e$ and $C_c$ are set as 0.02 and 0.01, respectively, as recommended in the literature (Singhal et al., 2002). $\zeta$ is the surface tension of liquid.

For the most circumstances, cavitating flows are also turbulent flow, which are characterized by fluctuating velocity fields. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering applications. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations which are computationally less expensive to solve. To model the influence of turbulence in the present study, the *SKE* two equations model as well as standard wall function are utilized in our simulations. The two equations are written as follows: