Applied Mathematical Modelling 40 (2016) 5835–5857

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

Numerical investigation on the dynamic behavior of sheet/cloud cavitation regimes around hydrofoil

Ying Chen*, Xin Chen, Zhaoxin Gong, Jie Li, Chuanjing Lu

MOE Key Laboratory of Hydrodynamics, School of Naval Architecture, Ocean & Civil Engineering, Shanghai Jiao Tong University, A802 Mulan Building, 800 Dongchuan Road, Shanghai 200240, China

ARTICLE INFO

Article history:
Received 19 March 2015
Revised 27 November 2015
Accepted 13 January 2016
Available online 27 January 2016

Keywords:
Cavitation
Hydrofoil
Numerical simulation
Barotropic law
Vortex structure

ABSTRACT

The evolution of the behaviors between sheet cavity and cloud cavitation often has significant influence on the hydrodynamic forces acting on propellers. The present work focuses on the numerical investigation on the dynamics of the sheet/cloud cavitation regime around NACA0012 hydrofoil. A barotropic law approach with a compressibility factor is introduced to improve the numerical robustness by ensuring the system to be hyperbolic. The governing equations and the turbulence model in originally complete forms are employed in the in-house RANS solver. The stable sheet cavity and the periodic shedding of the sheet/cloud cavitation are respectively predicted at different angles of attack. The cavity length and the lift and drag coefficients are basically in accordance with experimental ones. The cyclic transition between the attached sheet cavity and the large block of vapor cloud reflects the primary features of the experimental images and principles. The time variation of the lift and drag presents periodic oscillations in correspondence with the experiments, and is found to be a superposition of different characteristic frequencies. The correlation between the evolutions of the vortex flow and the pressure distribution along suction side is analyzed. It is found that the main oscillation component has the same frequency as the shedding of vapor cloud, while the secondary oscillation with smaller amplitude and higher frequency is caused by the alternate generation and shedding of the vortexes in the period. The adverse pressure gradient near cavity closure leads to the development of the vortexes and the inverse flow.

1. Introduction

As a fundamental property of liquid, cavitation often occurs in the flows of many hydraulic devices, such as turbines, pumps, pipe systems, fuel injectors, underwater vehicles, etc. Depending on the level of cavitation number, several cavitation regimes can be observed in liquid flows: incipient cavitation, shear cavitation, sheet/cloud cavitation, and supercavitation. In marine engineering, the occurrence of sheet/cloud cavitation regime is quite common. Cloud cavitation is an undesirable phenomenon which significantly degrades the hydrodynamic performance and results in noise, vibration and erosion.

---

* Corresponding author Tel.: +86 13761302148; fax: +86 21 34204308.
E-mail address: cyofjs@sjtu.edu.cn (Y. Chen).

http://dx.doi.org/10.1016/j.apm.2016.01.031

A comprehensive understanding of the transition principle between sheet cavitation and cloud cavitation on hydrofoil is beneficial to the design of propellers.

Sheet cavitation on a stationary hydrofoil oscillates cyclically within certain ranges of cavitation number and angle of attack (AOA), even if the freestream is steady. Unsteady cavitation experiments on hydrofoils have been previously performed by Wade and Acosta [1], Izumida et al. [2], Kubota et al. [3], and Le et al. [4]. It is well known that the sheet/cloud cavitation instability is mainly caused by the re-entrant jet periodically generated at the closure of the cavity. Kawanami et al. [5,6] utilized high-speed video camera to observe the cloud cavitation around hydrofoil and measured the cavity sizes and shedding frequency. The influence of re-entrant jet on the shedding process was analyzed as well. Fujii et al. [7] explored the periodic behavior of the cavitating flows around kinds of hydrofoils. Kawakami et al. [8] as well compared the frequency characteristics of the cavitation at various flow conditions. La Torre et al. [9] experimentally studied the added mass effect of a range of steady partial cavities around the NACA0009 hydrofoil, to determine its influence on the natural frequency of the fluid-structure system. Although experimental observation can show many of the phenomena occurring, it still suffers from the limitations in measurement techniques. The computational researches on cavitating flows have been developed greatly in last tens of years.

In CFD framework of turbulent flow computation, the flow around typical geometries has been widely studied during the past few decades. For example, in the early years Markatos et al. [10] and Abdelmeguid et al. [11] solved the three-dimensional turbulent flows around bodies of irregular but basically cylindrical shape with two-equation models, using the finite difference method for parabolic flows on non-orthogonal coordinate system. In recent years, Karabelas et al. [12] solved the high Reynolds-number flow over rotating cylinder using finite volume method and modified $k-\varepsilon$ model, through which the stability of acting forces and the vortex-shedding suppression were investigated. Karabelas and Markatos [13] studied the condensation of water vapor in the highly convective flow on airfoil geometry, with the turbulence effect accounted for by Spalart–Allmaras one-equation model.

Cavitating flows are basically turbulent and thus need to be handled by specific cavitation model, coupled with the treatment of turbulence effect. The solving strategies of cavitation can be classified into several categories, among which the most popular one is the two-phase flow method, including two-fluid and one-fluid models. Spalding [14] and Markatos [15] previously proposed the two-fluid models for the solving of two-phase flows. One-fluid model is also known as the homogeneous equilibrium model (HEM), and has been greatly developed and widely used in these years. The one-fluid model treats the cavitating flow field as filled with the mixture of two fluids, and the mixture flow is governed by only one set of equations. Kubota et al. [16] proposed a bubble two-phase flow (BTF) model based on the Rayleigh–Plesset equation. This model describes cavitation by expressing the evolution of bubble radius as a function of the local pressure. Schnerr and Sauer [17] and Frobenius et al. [18] further improved the model by constructing more complex relations between the local pressure and the bubble radius distribution. Another way of the one-fluid model is well known as the transport equation model (TEM), in which a governing equation for the liquid or vapor phase fraction is solved. Merkle et al. [19], Kunz et al. [[20], 2000], Yuan et al. [21] and Singhal et al. [22] respectively proposed such models where the phase change sources are related to the amount by which the local pressure is below the vapor pressure. These models have the advantage of considering the mass transfer as a process, but sometimes produce comparatively steady cavities. Morgut et al. [23] discussed the effect of the empirical parameters in these model based on experimental benchmark and tuned the parameters to the optimal ones. Tseng and Wang [24] investigated the coefficients of the cavitation and turbulence models through simulation of the unsteady flows around cylindrical bodies, NACA66 and Clark-Y hydrofoils. Zhang et al. [25] also improved Singhal's model and verified it through simulation of the quasi-steady cavitating flow around NACA66 foil. The third way is the so called barotropic equation model first proposed by Delannoy and Kueny [26]. In this model the density of the mixture is directly linked to the local static pressure, and the phase change is assumed to be instantaneous. This model has been successfully used to simulate the unsteady cavitating flows by Song and He [27], Coutier-Delgosha et al. [28], Byeong et al. [29], and Goncalves [30].

In recent years, the cavitating flows around hydrofoils have been widely investigated using the numerical models mentioned above. Zhou and Wang [31] simulated the cavitation around NACA66 hydrofoil and modified a kind of eddy–viscosity turbulence model to obtain the unstable cavity shedding which was not well predicted in the previous works. Roohi et al. [32] simulated the cavitating flows over Clark-Y hydrofoil using LES turbulence model and VOF technique, and suitable accuracy was observed for the dynamics of both steady and transient cavities. Huang et al. [33] also performed combined numerical and experimental study on the unsteady structure of the sheet/cloud cavitation around the Clark-Y hydrofoil, using a filter-based density corrected model to regulate the turbulent viscosity in both the cavitation region and the wake. Celik et al. [34] simulated the flows around the partially cavitating 2D and 3D hydrofoils with cross section of the NACA 16 series. Timoshevskiy et al. [35] investigated the different cavitation regimes around the NACA0015 hydrofoil at several attack angles. They compared the dynamics of growth and convection of bubbly cavity at small attack angle and the transition of sheet cavity to unstable one at higher angles.

Although many important advances have been made in the numerical analysis of sheet/cloud cavitating flows in those works, some issues about the evolution of the vortex flow structure have not been resolved yet. In the present work, a kind of improved barotropic equation model is used to simulate two different types of sheet/cloud cavitation regimes around NACA0012 hydrofoil for varieties of composite flow parameters. This research will provide a better understanding of the cavitation regimes dependent on flow conditions and the physical mechanism driving cavitation shedding.