Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2017,29(1):27-39
DOI: 10.1016/S1001-6058(16)60715-1
www.sciencedirect.com/
science/journal/l0016058

Large eddy simulation of turbulent attached cavitating flow with special emphasis on large scale structures of the hydrofoil wake and turbulence-cavitation interactions*

Bin Ji (季斌)[1,2,3], Yun Long (龙云)[1,2], Xin-ping Long (龙新平)[1,2], Zhong-dong Qian (钱忠东)[1],
Jia-jian Zhou (周加建)[3]
1. *State Key Laboratory of Water Resources and Hydropower Engineering Science, School of Power and Mechanical Engineering, Wuhan University, Wuhan 430072, China, E-mail: jibin@whu.edu.cn*
2. *Hubei Key Laboratory of Waterjet Theory and New Technology, Wuhan University, Wuhan 430072, China*
3. *Science and Technology on Water Jet Propulsion Laboratory, Shanghai 200011, China*

(Received June 9, 2016, Revised December 6, 2016)

Abstract: In this paper, the turbulent attached cavitating flow around a Clark-Y hydrofoil is investigated by the large eddy simulation (LES) method coupled with a homogeneous cavitation model. The predicted lift coefficient and the cavity volume show a distinctly quasi-periodic process with cavitation shedding and the results agree fairly well with the available experimental data. The present simulation accurately captures the main features of the unsteady cavitation transient behavior including the attached cavity growth, the sheet/cloud cavitation transition and the cloud cavitation collapse. The vortex shedding structure from a hydrofoil cavitating wake is identified by the Q-criterion, which implies that the large scale structures might slide and roll down along the suction side of the hydrofoil while being further developed at the downstream. Further analysis demonstrates that the turbulence level of the flow is clearly related to the cavitation and the turbulence velocity fluctuation is much influenced by the cavity shedding.

Key words: Cavitation, large eddy simulation (LES), vortex structure, turbulence-cavitation interactions

Introduction

The cavitation involves a very complex two-phase flow. It can have negative impacts on the hydraulic machinery such as through vibrations, erosions, noises and performance break-down[1]. Therefore, it is necessary to investigate its physical mechanism for controlling cavitation in engineering applications.

The fundamental physics of the cavitation phenomenon was widely studied. Much attention was paid on the evolution of both sheet and cloud cavitations on the hydrofoil. The typical quasi-periodic phenomenon was a focus, including the sheet cavitation inception, growth and shedding[2-4]. Li et al.[5] reproduced the quasi-periodic phenomenon by using the large eddy simulation (LES) method and the VOF model. The VOF technique is widely used for the simulation of the free surface flow[6]. The cavity interface can be predicted accurately[7]. Gnanaskandan and Mahesh[8] studied the sheet to cloud cavitation transition over a wedge. It is found that the frequency of the shedding process obtained by the LES is in good agreement with that of the experiments. These studies show that the recent methods can accurately capture the detailed features of the cavitation. Studies about the sheet to cloud cavitation show that the re-entrant jet is a very important source factor for the cavity shedding. Le et al.[9] studied partial cavities. They observed that the cavitation periodic shedding was closely associated with the re-entrant jet. Pham et al.[10] confirmed the conclusions of Le et al.[9] by analyzing the frequencies of the re-entrant jet surges and the cloud shedding. Recent studies confirm that the re-entrant jet is the main cause for the development and the shedding of the attached

* Project supported by the National Natural Science Foundation of China (Grant Nos. 51576143, 11472197).
Biography: Bin Ji (1982-), Male, Ph. D., Associate Professor
Corresponding author: Xin-ping Long,
E-mail: xplong@whu.edu.cn

cavity[2]. The studies of the cylindrical objects[11], the head-form body[12] and others[13-15] demonstrate that the re-entrant jets are responsible for the cavity shedding.

The unsteady cavity shedding often involves a complex cavity vortex structure. Therefore, the cavitation-vortex interactions are important research issues. Gopalan and Katz[16] demonstrated by means of the PIV and the high speed photography that the collapse of the vapor cavity in the downstream region was mainly responsible for the generation of the vortex. The LES method is a useful tool to reproduce the unsteady cavitating flow, with great advantages in accurately capturing the complex vortex structures. It was firstly proposed by Smagorinsky[17], and now it is widely applied for the simulations[18,19]. Roohi et al.[7] combined the 2-D LES with the VOF and the cavitation model and showed that it was useful to simulate the shape of the cloud cavity and its dynamics. Ji et al.[13] and Luo et al.[20] studied the 3-D cavitation and vortex structures based on the LES around a twisted hydrofoil. Ji et al.[21] indicated that the mass transfer along the cavity surface induces an increase in the vortex dilatation. They found that the baroclinic torque term is important along the liquid-vapor interface but has a negligible effect inside the attached cavity region. Dreyer et al.[22] studied the tip leakage vortex on a NACA0009 hydrofoil using the stereo-PIV. The results revealed that it is prone to generate the cavitation when the vortex length is the maximum. All these studies have enhanced the understanding of the cavitation-vortex interactions, but the large scale vortex structures remain a difficult issue. More recently, Peng et al.[23] performed a series of experimental observations around hydrofoils in the cavitation tunnel. They observed that the U-type flow structures were common in cloud cavities. The side reentrant jets were identified as the main cause of the sheet cavity shedding and the formation of the U-type vortex structure was controlled in the cloud cavitation, which used Q - criterion[24] to visualize the vortex structure evolution in numerical results. This helps us gain a deeper understanding of the cavitation shedding dynamics and cavitation-vortex interactions.

Besides these fundamental studies of the dynamics of the sheet and cloud cavitation and the cavitation-vortex interactions, various kinds of cavitations were also studied, among others, the cavitating flows around a semi-circular leading-edge flat plate[25], the axisymmetric projectile[26], the 3-D hemispherical head-form body[12], and behind a 3-D disk[27] by numerical and experimental methods. Wang et al.[28] studied the effect of free surface on the unsteady behavior of the cloud cavity. They found that the cavity evolution was remarkably different when the axisymmetric projectile was near the free surface. Park and Rhee[29] performed a comparative study of incompressible and compressible flow solvers for the cavitating flow. They concluded that the compressibility effects can enhance the reproduction of the cavitation. Various features of the cavitation at different cavitators are thus revealed, and the understanding and the ability to control and utilize the cavitation are enhanced.

The pressure and velocity fluctuations were also attracted some attention. Chen et al.[30] found that the pressure fluctuations are closely related to the transient cavitation behaviors, and they are excited by the acceleration due to the changes in the cavity volume. Ji et al.[31] illustrated the relationship between the cavitation evolution and the pressure fluctuations by analyzing the cavity volume fluctuation. They came to a similar conclusion that the cavity volumetric acceleration is the main source of the cavitation excited pressure. Huang et al.[32] introduced a filter-based corrected model. They found that the transient cavitation behaviors significantly enhanced the turbulence velocity fluctuations to induce a thicker turbulence boundary layer. More researches show that the streamwise velocity fluctuations generally are dominant inside the cavity[33], and the streamwise and spanwise fluctuations are equally important at the cavity closure and in the wake[8]. The turbulence-cavitation interactions continue to attract many research interests, but their mechanisms remain not well understood[34,35].

Although a great progress has been made on the cavitating flow and the cavity vortex structures by both numerical and experimental methods, the large scale structures in the hydrofoil wake are still a worthwhile topic. Furthermore, the turbulence-cavitation interactions remain elusive due to the unsteadiness and the complex flow structures. This paper studies the cavitating flow around a Clark-Y hydrofoil by investigating the turbulent attached cavitating flow with a special emphasis on the large scale structures in the hydrofoil wake and the turbulence-cavitation interactions with the LES approach.

1. Cavitation model and numerical methods

The LES method is employed in this paper to solve the unsteady Navier-Stokes equations coupled with a mass transfer cavitation model. The main features of the solver are as follows.

1.1 *Physical cavitation model*

Schnerr and Sauer[36] developed the cavitation model adopted in this paper to describe the mass transfer through the interface of the vapor and the liquid. The cavitation model is described by the follow equation