

Available online at www.sciencedirect.com



Journal of Hydrodynamics 2016,28(4):709-712 DOI: 10.1016/S1001-6058(16)60674-1



Numerical investigation of unsteady cavitating turbulent flows around twisted hydrofoil from the Lagrangian viewpoint^{*}

Huai-yu CHENG (程怀玉)^{1,2,3}, Xin-ping LONG (龙新平)^{1,2}, Bin JI (季斌)^{1,2,3}, Ye ZHU (祝叶)¹, Jia-jian ZHOU (周加建)³

1. Wuhan University, Wuhan 430072, China

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,

E-mail: chengiu@whu.edu.cn

(Received August 1, 2016, Revised August 8, 2016)

Abstract: Unsteady cavitating turbulent flow around twisted hydrofoil is simulated with Zwart cavitation model combined with the filter-based density correction model (FBDCM). Numerical results simulated the entire process of the 3-D cavitation shedding including the re-entrant jet and side-entrant jet dynamics and were compared with the available experimental data. The distribution of finite-time Lyapunov exponent (FTLE) was used to analyze the 3-D behavior of the re-entrant jet from the Lagrangian viewpoint, which shows that it can significantly influence the particle trackers in the attached cavity. Further analysis indicates that the different flow behavior on the suction side with different attack angle can be identified with Lagrangian coherent structures (LCS). For the area with a large attack angle, the primary shedding modifies the flow pattern on the suction side. With the decrease in attack angle, the attached cavity tends to be steady, and LCS A is close to the upper wall. A further decrease in attack angle eliminates LCS A in the boundary layer. The FTLE distribution also indicates that the decreasing attack angle induces a thinner boundary layer along the foil surface on the suction side.

Key words: cavitating flow, twisted hydrofoil, LCS, CFD, cavitation

Introduction

Much attention has been focused on cavitating flow because of its complex flow pattern and usually undesirable influence on hydraulic machinery^[1-3]. Numerous related experiments have been reported in recent decades and significantly improved knowledge about cavitating flow, such as the relationship between the re-entrant jet and the steady of cavitating flow^[4]. At the same time, numerical simulation technology is attracting increasing interest with its notable success in predicting cavitating flows. Huang et al.^[5] used filter-based density correction model (FBDCM) with the Zwart cavitation model to investigate unsteady sheet/cloud cavitating flows. Comparisons with experimental results showed that the method captured many details of the cavitating flows, such as the formation, breakup, shedding, and collapse of cavities. Ji et al.^[6] numerically investigated complex cavitating flows around a NACA66 hydrofoil with LES. The cavity shedding was theoretically related to the pressure fluctuations by a simplified one dimensional model and 3-D numerical results agreed well with the experimental results.

To provide an accurate experimental database for the validation of computational methods and contribute to the development guidelines for propeller design, Foeth^[7] conducted a series of experiments on the cavitating flow around a twisted hydrofoil, namely, the Delft Twist-11 hydrofoil. Experiments indicated that the re-entrant flow direction was largely dictated by the cavity topology, and the side-entrant jet had a noticeable influence on the behavior of the shedding cycle^[8-10]. The unsteady cavitation patterns and their evolution around the Delft twisted hydrofoil were simulated by Ji et al.^[11] with PANS model. The numerical results reproduced the 3-D cavity structure well,

^{*} Project supported by the National Natural Science Foundation of China (Grant Nos. 51576143, 11472197) and Science and Technology on Water Jet Propulsion Laboratory (Grant No. 61422230101162223002).

Biography: Huai-yu CHENG (1993-), Male, Ph. D. Candidate Corresponding author: Bin JI, E-mail: jibin@whu.edu.cn

and the frequency of the cavity shedding agreed well with the experimental observation. Zhao et al.^[12] simulated the cavitating flow around a 2-D Clark-Y hydrofoil, and a relatively new technology, the Lagrangian investigations, including Lagrangian Coherent Structures (LCS) and particle trajectory, was applied to describe the 2-D flow patterns and capture substantial circumferential motion. Their work showed that the behaviors of vortex structure in different cavitation developing stages could be described with the distributions of LCSs. The findings demonstrated that LCS was a promising method in the study of cavitating flow also shown by Tseng et al.^[3]. However, because of the 3-D characteristic of cavitating flow, the 2-D simulation cannot reproduce the cavitating flow well. As a result, some details may be lost in the analysis of LCS with 2-D numerical data.

Inspired by previous research, this paper numerically investigates the 3-D cavitating flow around the twisted hydrofoil using the Zwart cavitation model combined with FBDCM turbulence model. Lagrangian investigations, including LCS and particle trajectory, were adopted to study the 3-D behavior of cavitating flow.

Lagrangian method has been widely used in flow visualization and its comprehensive introduction can be found in Ref.[13]. The geometric structure of the Delft Twist-11 hydrofoil was introduced in detail by Foeth^[7]. In the present paper, the attract angle of the entire foil was set to -2° and the chord *C* was 0.15 m in accordance with the experiment by Foeth^[7].

Only half of the Delft Twist-11 hydrofoil was considered because of its symmetry. The hydrofoil was located in a channel with a length of 7C, a width of C, and a height of 2C. The inlet was at 2C upstream of the hydrofoil, and the inlet velocity was set to $u_{-\infty} = 6.97 \text{ m/s}$. The pressure at the outlet 5C downstream of the origin was determined by the cavitation number $\sigma = (p_{out} - p_v)/(0.5\rho_1 U_{-\infty}^2) = 1.07$. The top, bottom, and side of the channel were set as free slip walls, while the foil surface was set as a non-slip wall. The Zwart cavitation model combined with the FBDCM turbulence model^[14] was used in the present simulation. A series of mesh studies was conducted, and findings indicate that the mesh used in the simulation provided a good balance between computational efficiency and accuracy. The cavitation simulation was initialized from steady state results with fully wetted model. Then, the cavitation model combined with FBDCM and the unsteady solver was turned on for the cavitation flow simulation.

The predicted cavity shedding frequency was 27.7 Hz, which was slightly underestimated compared with the measured frequency of 32.2 Hz^[7]. Even though some differences exist between the predicted and

measured cavitation shedding cycles, the numerical results still predict the cavitation shedding dynamics well.



Fig.1 Comparison of the predicted top views of the iso-surfaces of $\alpha_v = 0.1$ (left) and the experimental pictures (right)^[7]

To show the shedding behavior of cavitation in detail, the time evolution of the predicted cavitating flow is shown by seven snapshots of typical instants in a cycle shown in Fig.1, with the experimental top-view pictures^[7] at each instant given for comparison. The numerical cavity shape is depicted by the iso-surface of vapor volume fraction α_{v} with a value of 0.1. As shown in Fig.1(a), the attached cavity reached its maximum length with a convex closure, here considered as fully developed. The flow on the sides of the attached cavity is forced into the cavity because of the pressure gradient resulting side-entrant jets. At the closure region, the 3-D shape of the hydrofoil induces the re-entrant jet to radially diverge upstream from the closure at mid-plane. The side-entrant and re-entrant jets both reach the leading edge and collide with the main stream. As a result, the attached cavity is cut off at the leading edge, thereby introducing primary shedding, as shown in Fig.1(b). The shedding cavity is advected downstream with the main flow and finally collapses, thereby inducing a U-shaped vortex at the rear of the foil and leaving a concave closure line, as shown in Figs.1(c)-1(e). During this process, the radial divergence of the re-entrant jet is further enhanced because of the concave closure line. The side-entrant and re-entrant jets finally converge, thereby causing secondary shedding, which agrees well with the experimental observation as shown in Fig.1(f). The secondary shedding is much weaker but modifies the closure line topology to a near-convex shape as Fig.1(g)

Download English Version:

https://daneshyari.com/en/article/1721828

Download Persian Version:

https://daneshyari.com/article/1721828

Daneshyari.com