Numerical analysis of the unsteady cavitation shedding flow around twisted Hydrofoil based on hybrid filter model

Citation for published version (APA):

DOI:
10.2298/TSCI1804629Z

Document status and date:
Published: 01/01/2018

Document Version:
Publisher’s PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:
• A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
• The final author version and the galley proof are versions of the publication after peer review.
• The final published version features the final layout of the paper including the volume, issue and page numbers.

Link to publication

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
• You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the “Taverne” license above, please follow below link for the End User Agreement:
www.tue.nl/taverne

Take down policy
If you believe that this document breaches copyright please contact us at:
openaccess@tue.nl
providing details and we will investigate your claim.

Download date: 05. Sep. 2020
NUMERICAL ANALYSIS OF THE UNSTEADY CAVITATION SHEDDING FLOW AROUND TWISTED HYDROFOIL BASED ON HYBRID FILTER MODEL

by

Desheng ZHANG\textsuperscript{a}, Jian CHEN\textsuperscript{a}, Lei SHI\textsuperscript{a}, Guangjian ZHANG\textsuperscript{a}, Weidong SHI\textsuperscript{a}, and Bart P. M. van ESCH\textsuperscript{b}

\textsuperscript{a} Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, Zhenjiang, China
\textsuperscript{b} Department of Mechanical Engineering, Eindhoven University of Technology, Eindhoven, the Netherlands

Original scientific paper
https://doi.org/10.2298/TSCI1804629Z

Cavitation is a common phenomenon in components of fluid machinery and it may induce material damage and vibration. A more accurate and commercial turbulence model is required to predict cavitation. In this paper, we make a combination of filter-based model (FBM) and density correction method (DCM) to propose a new DCM FBM. Firstly, the new DCM FBM and the homogeneous cavitation model are validated by comparing the simulation result with the experiment of cavitation shedding flow around the Clark-y hydrofoil and the filter size is determined as well. Then, the cavitation pattern cycle and shedding vortex structure of the twist hydrofoil experimented by Delft University of Technology were predicted using the DCM FBM. The predicted 3-D cavitation structures and development cycle of twist hydrofoil as well as the collapsing features show a good qualitative agreement with the high speed photography results. Numerical results show that the improved turbulence model could predict the cloud cavity evolution well, including the cloud cavity generation, shedding and dissipation. It is found that the re-entrant jet induced by the by adverse pressure gradient is the main reason to generate the cloud cavity shedding. The secondary shedding is also observed which is result from the combination of the radially advancing re-entrant jet and side-entrant jet simulated by the DCM FBM turbulence method.

Key words: cloud cavitation, twist hydrofoil, density correction, turbulence model, numerical simulation

Introduction

Cavitation occurs in components of fluid machinery, for instance, suction side of axial pump blade, gap between blade tip, and casing and tip of propeller. The pressure fluctuation caused by the periodic generation and collapse of cavitation bubble induce noise and vibration. Therefore, cavitation control is expected to improve the performance and reliability of the hydraulic machinery.

Turbulence models based on Reynolds averaging has been widely used [1, 2] because of the commonality and memory saving. The disadvantages of RANS model is that it over-predict the turbulent viscosity in the rear part of cavity, where compressible two-phase...
flow dominants, making the fluid too thick for the re-entrant jet to penetrate. A DCM is proposed by Coutier-Delgosha et al. [3], which is effect in predicting shedding cavitation [4-6]. To improve the cavitation prediction accuracy, large eddy simulation (LES), direct numerical simulation, and lattice Boltzmann method which can offer convincing time-dependent results are employed to model the unsteady cloud cavitation. Ji et al. [7] and Luo et al. [8] used LES model coupled with a mass transfer cavitation model to predict unsteady 3-D turbulent cavitation flow around a twist hydrofoil. However, LES requires a very fine mesh, and the computation cost is also very expensive. Furthermore, it is difficult to obtain a grid-independent solution. Until now, LES method limits the industrial application, especially in hydraulic machines.

In recent years, hybrid models provide a solution for the prediction of turbulent eddy viscosity in the cavitation flow. Taking into account the advantages of RANS and LES, Johansen et al. [9] proposed a FBM blending the RANS and LES model to calculate the wake flow of the square object, and obtained a higher prediction accuracy. Many validations [10-17] have been conducted to establish a hybrid FBM for the cavitation flows in a square cylinder, a hydrofoil and hollow-jet valve. The results show FMB can better capture the unsteady features of the cavitation flows than the normal RANS models. However, FMB is not effective in the near-wall region. It indicates the two equation model will be covered when the filter size $\Delta \gg l$. Furthermore, the filter scale in FBM and the maximum density ratio in the cavitation model have the important effects on the numerical results [14, 15].

Inspired by their work, the objective in this paper is to investigate an economical and accurate simulation method to analyze the cavitation shedding flow around a 3-D twisted hydrofoil, which employed the modified FBM turbulence model and homogeneous cavitation model. The unsteady behavior and evolution of the cavitation shedding flow was analyzed by both numerical and experimental results.

**Numerical method description and set-up**

**Governing equations**

Highly developed turbulence is one of the important features of cavitation flow in hydraulic machineries. Its interaction with cavitation and the nature of compressibility of cavity contribute to the intricacy of cavitation flow. The FBM was developed on the basis of standard $k-\varepsilon$ model by Johansen et al. [9]. The governing equations are presented:

$$
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon \tag{1}
$$

$$
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon k} \frac{\varepsilon}{k} - C_{\varepsilon \alpha} \frac{\varepsilon^2}{k} \tag{2}
$$

$$
\mu_t = C_{\mu} \frac{\rho k^2}{\varepsilon} \tag{3}
$$

in which $C_{\mu} = 0.09$, $F$ is the filter function determined by the filter size $\Delta$ and turbulence length scale $l_{\text{RANS}} = k^{3/2}/\varepsilon$:

$$
F = \min \left[ 1.0, C_3 \frac{\Delta}{l_{\text{RANS}}} \right] \tag{4}
$$
Reboud et al. [11] and Coutier-Delgosha et al. [4] proposed a DCM, which is applicable is reducing the turbulent viscosity in the near wall region. As a combination of FBM and DCM, turbulent viscosity in MFBM is presented:

\[
\mu_{i-MFBM} = C_i \rho \kappa^2 F
\]  

where

\[
F = \min \left[ f(n) C_3 \frac{\Delta}{\eta_{RANS}}, \frac{\nabla \rho}{\rho \alpha} \right], \quad f(n) = \frac{\rho \alpha + (1-\alpha)(\rho_l - \rho)}{\rho \alpha + (1-\alpha)(\rho_l - \rho)}
\]

**Geometry model**

In this paper, a 3-D twisted hydrofoil Twist 11 invented by Delft University of Technology is adopted. The experimental results published provide reference for the simulation [12, 13]. The section of Twist 11 is NACA0009, with a chord length \( c \) of 0.15 m and span \( y = 0.3 \) m.

**Simulation set-up and boundary conditions**

The real physical domain is calculated as illustrated in fig. 1, in order to save resources and time. Inlet is 2C from the leading edge and the outlet is 4C from the trailing edge of hydrofoil. The distance between the upper wall and bottom is 2C and left wall is 2C from right wall. Inlet velocity is \( V_\infty = 6.97 \) m/s and outlet pressure is defined by \( \sigma = (p_{out} - \rho V_\infty^2)/(0.5 \rho_i V_\infty^2) \). Symmetric boundary condition is set in the mid plane of span and non-slip boundary condition is employed in walls of tunnel and the surface of hydrofoil.

Convergence evaluation in every step is very important in unsteady cavitation calculation. According to Li [14] and Ji et al. [15], excessive iterations in one time step consume too much computation resources, while few iterations induce low precision. So high resolution scheme is used to discrete advection term and second order backward Euler is employed in transient term. Time-step is \( \Delta t = T_{ref}/200 = 1.076 \times 10^{-4} \), according to Coutier-Delgosha et al. [16], in which \( T_{ref} = c/V_\infty \), and \( V_\infty \) is the inlet velocity.

The O-H type structure grid is generated using ANSYS-ICEM, and mesh near wall is refined (\( 30 \leq y^+ \leq 100 \)). With the increase of element number, discrepancy caused by discretization decreases, while error accumulated inclines accordingly. Consequently, it is important to check the grid independence. Three different nodes numbers in span direction are chosen.

As shown in tab. 1, minimum and maximum pressure as well as lift \( C_l \) and drag coefficients \( C_d \) predicted by three grids are very close. Grid 2 shows properly the same results with grid 3. So considering numerical precision and resource consumption, Grid 2 is chosen for computation (fig. 2).

Figure 2. Three cases of mesh around the Delft twisted hydrofoil surface; (a) mesh case 1, (b) mesh case 2, (c) mesh case 3

Table 1. Results of the mesh independence study for twisted hydrofoil

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Nodes</th>
<th>$p_{\text{min}}$</th>
<th>$p_{\text{max}}$</th>
<th>$C_l$</th>
<th>$C_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1 (coarse)</td>
<td>1251000</td>
<td>-51941.7</td>
<td>54322.2</td>
<td>0.4643</td>
<td>0.02872</td>
</tr>
<tr>
<td>Case 2 (medium)</td>
<td>2502000</td>
<td>-51930.9</td>
<td>54309.2</td>
<td>0.4642</td>
<td>0.02871</td>
</tr>
<tr>
<td>Case 3 (fine)</td>
<td>3753000</td>
<td>-51920.6</td>
<td>54346.2</td>
<td>0.4642</td>
<td>0.02871</td>
</tr>
</tbody>
</table>

Results and discussion

Typical cavitation evolution

With the shedding of cavity on the suction surface of twisted hydrofoil, lift force. Time averaged lift coefficient calculated by MFBM is 0.464, which shows a closer result than other reports [7, 18, 19] with the experiment [17], shown as in tab. 2.

Table 2. Time-averaged lift coefficients predicted by various models

<table>
<thead>
<tr>
<th></th>
<th>Cal.</th>
<th>Exp.</th>
<th>Error [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>RANS SST $k$–-$\omega$ with correction Li et al., [19]</td>
<td>0.43</td>
<td>0.5167</td>
<td>17</td>
</tr>
<tr>
<td>RANS SA with correction Bensow, [18]</td>
<td>0.43</td>
<td></td>
<td>17</td>
</tr>
<tr>
<td>Implicit LES Bensow, [18]</td>
<td>0.45</td>
<td></td>
<td>13</td>
</tr>
<tr>
<td>PANS Ji et al., [20]</td>
<td>0.453</td>
<td></td>
<td>12</td>
</tr>
<tr>
<td>DCM FBM by present</td>
<td>0.464</td>
<td></td>
<td>10</td>
</tr>
</tbody>
</table>

Massive high speed photographs are shot in top view and presented by Foeth [17], one periods of cavitation flow predicted by numerical are compared and depicted in fig. 3, in which cavity is indicated by iso-surface of vapor volume fraction $\alpha_v = 0.1$. Instants 1–7 marked in fig. 3(a) show cavitation in the one period. Figure 3(b) is the high speed photographs provided by Foeth [17]. Instant 1 shows a convex closure of sheet cavity. As the re-entrant jet driven by adverse pressure gradient enhancing with the inclining adverse pressure gradient, re-entrant jet penetrates into the bottom of cavity and limits the advance of cavity closure downstream. Due to existence of 3-D effect, minor shedding of cavity in area A occurs, which is caused by side-entrant jet. With the re-entrant jet migrates towards the leading edge of hydrofoil, the interface between cavity and liquid start to destabilize. Meanwhile, quasi-steady sheet cavity starts to evolve into highly unsteady cloud cavitation. When re-entrant jet migrates to the leading edge of hydrofoil, main cavity breaks and detaches, as shown in Instant 2 when the total volume of cavity decreases to the minimum. With the main cavity shedding, residual cavity starts to grow and form a concave closure, as illustrated in Instant 3. From In-
stant 4 to Instant 7, shedding main cloud transport downstream and start to roll together. Cavi-
yty volume reaches maximum in Instant 5 and with the rolling and breakup of cavity, volume
of cavity drops abruptly.

Figure 3. Comparison of observed and predicted cavity volume evolution ($\sigma = 1.07$); (a) predicted
cavitation evolution in one typical cycle by numerical method, (b) experimental observation of
cavitation evolution in one typical cycle (for color image see journal web site)

It is reasonable to say that numerical method adopted here well reproduce the cavita-
tion evolution, including the generation, breakup, detachment and collapse of cavity.

**Shedding vortex cavitation structure and evolution**

In fig. 4(a), the adverse pressure is large enough to drive the re-entrant jet penetrate
into the cavity and migrate towards the head of the twisted hydrofoil. In fig. 4(b), a mount of
liquid accumulates in the vicinity of hydrofoil head, making a local high pressure region and
inducing the breakup of sheet cavity. Corresponding to this instant, cavity volume drops to the
minimum. In fig. 4(c), instability of shedding cavity increases, while transport downstream,
rolling into highly unsteady cloud cavitation, which is dominant shedding instance in cavita-
Figure 4. Cavitation patterns during one cavity shedding cycle; numerical top view (left), experimental top view (middle), numerical bird’s-eye view (right)
(for color image see journal web site)
tion flow field. Cloud cavity becomes more unsteady and transports downstream. Hereafter, generation of sheet cavity slows down and cloud cavitation starts to shrink, as shown in fig. 4(d), and collapses in the high pressure region in fig. 4(e). Figure 4(d) presents the secondary shedding of cavitation near the closure of sheet cavity. Foeth [17] believes that this is caused by the side-entrant jet and we will investigate that on the basis of numerical simulation. Velocity field near the suction side of hydrofoil is depicted in Instant 6 as illustrated in fig. 5. It is distinctive the existence of side-entrant jets on two sides of the attached cavity and the re-entrant jet proceeds radially towards the head of hydrofoil. Re-entrant jet and side-entrant jets collide while advancing and form a liquid band, as indicated in rectangle block in fig. 5. With the liquid accumulating, secondary shedding appears on two sides of sheet cavity.

Based on the previous analysis, it is plausible to say that numerical method adopted in this paper reproduces the pattern and evolution of cavitation around a twisted hydrofoil with precision and accuracy, including the main cloud shedding of main cloud and secondary shedding occurs on both sides of cavity. Specifically, the lobe like cavity and radial side-entrant jet are also predicted in this code. An over-estimation of the collapse of shedding cloud cavity is shown in fig. 4(e), which is consistent with phenomenon predicted by PANS model [20], LES [18, 21] and modified SST $k-\omega$ model [19].

Conclusions

• A modified FBM (MFBM) turbulence model is adopted to solve the over-prediction of turbulent viscosity. Combining MFBM with Zwart cavitation model, cavitation flow is simulated and validated. In a macro view, cavitation pattern and evolution show a good agreement with the experiment, which is a good evidence that numerical method employed here is accurate and suitable for cavitation flow simulation.

• Structure and evolution of cavitation on the suction side of a twisted hydrofoil is investigated. Compared with other turbulence, numerical results show that the improved turbulence model could predict the cloud cavity evolution well both in Shedding frequency of cavity and time averaged lift coefficient as well as the process of the cloud cavity generation, shedding and dissipation result from the re-entrant jet.

• The Q-criteria of the vorticity are adopted to study the development of the cloud cavity shedding driven by the high pressure adverse gradient. Both main and secondary cavity shedding are observed by the DCM FBM turbulence model proposed in this paper, and the new method shows more accurate and economical compared to PANS model, LES and modified SST $k-\omega$ model.

Acknowledgment

This work was financially supported by the National Natural Science Foundation of China (Grant No. 51776087, 51479083 and 51579118). Transformation of scientific and technological achievements Project of Jiangsu Province (Grant No. BA2016167), Primary
Research & Development Plan of Jiangsu Province (Grant No. BE2015001-3, BE2015146, and BE2016166), 333 Project of Jiangsu Province and Six talent peaks project in Jiangsu Province (Grant No. HYGC-008).

References