Numerical Investigation of Cloud Cavitation Flows around Clark-Y Hydrofoil

Minsheng Zhao, Decheng Wan
Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University
Shanghai, China
* Corresponding Author: dcwan@sjtu.edu.cn

ABSTRACT

The present work focuses on the shedding characteristics of cloud cavitation around Clark-Y hydrofoil. The numerical simulation is carried out using InterPhaseChangeFoam solver with Schnerr-Sauer cavitation model based on OpenFOAM. The SST k-Ω model is used for computing, and the eddy viscosity coefficient of the turbulence model is modified to restrict the viscosity of the water vapor mixing zone. The hydrodynamics performance of two-dimensional wing in cavitation flow, including lift coefficient, drag coefficient and cavitation shape, is investigated and analyzed. The numerical results are in consistent with the experimental ones, indicating the ability of numerical method to successively predict the cavitating flow. The SST k-ω model with modified eddy viscosity coefficient performs better at capturing the unsteady cavitation vortex cluster in the development process of the cavitation, and the unsteady characteristics of the cloud cavitation are changed periodically. Further studies show that the re-entrant jet is the main cause of cavitation shedding.

KEY WORDS: Cavitation; Clark-Y; modified SST k-ω model; OpenFOAM

INTRODUCTION

Cavitation is a physical phenomenon which generally happens when the local pressure in the liquid drops below the limit of vapor pressure. This phenomenon often occurs on the surface of hydraulic machines such as ship propellers, pumps and hydrofoils. Unsteady cavitation and cavity shedding generate pressure fluctuation and vibrations, which can lead to material erosion and degradation of machine performance. In recent years, with the increase of ship speed, the occurrence of cavitation is unavoidable. Therefore, an investigation of unsteady cavitation behavior is essential to accurately predict ship performance and improve propeller design. Experiments of unsteady cavitation on hydrofoils have been carried out by Wade and Acosta[1], Izumida et al[2], Le, Franc & Michel utilized mean and dynamic pressure measurements to research cavities that formed on two-dimensional hydrofoils. Kawanami et al[4,5] employed high speed video camera to observe the evaluation of cloud cavitation and measured the cavity shedding frequency. However, these experiments above can be expensive and limited by measurement techniques, which causes the development of Computational Fluid Dynamics. Three-dimensional turbulent flows around bodies of irregular shape with two-equation models have been solved by Markatos et al[6], and Abdelmeguid et al[7]. In recent years, Karabelas et al[8], solved the high Reynolds-number flow over rotating cylinder using finite volume method and modified k-ε model. Nowadays most cavitation models are based on the Rayleigh–Plesset equation. This equation describes the evaluation of bubbles after encountering the pressure disturbance. Researchers have improved the CFD cavitation models, like Singhal[9], Schnerr and Sauer[10], Reboud et al[11], predicted the cloud cavitation with an artificial reduction of the turbulent viscosity in the water-vapor mixing zone. In the present paper numerical simulations are carried out with Schnerr-Sauer cavitation model based on OpenFOAM. Two-dimensional cloud cavitation around Clark-Y hydrofoil in the uniform flow has been investigated. The hydrodynamics performance of two-dimensional wing in cavitation flow is in consistent with the experimental ones, indicating the ability of numerical method to successfully predict the cavitating flow. It is found that the SST k-ω model with modified eddy viscosity coefficient has an advantage at capturing the unsteady cavitation vortex cluster, and the unsteady characteristics of the cloud cavitation are changed periodically. Further studies show that the re-entrant jet is the main cause of cavitation shedding.
NUMERICAL METHOD

Governing Equations
The governing equations for cavitation flow are based on a single phase flow approach, regarding the mixture of fluid and vapor as a single phase whose density can change according to the pressure. The flow field is solved by the mixture continuity and momentum equations plus a volume fraction transport equation to model the cavitation dynamics. As for RANS turbulence model, the equations are presented below.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0 \quad (1)
\]

\[
\frac{\partial (\rho u_j)}{\partial t} + \frac{\partial (\rho u_j u_i)}{\partial x_j} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{\partial \tau_{ij}}{\partial x_j} \right] \quad (2)
\]

\[
\frac{\partial \alpha_i}{\partial t} + \frac{\partial (\alpha_i u_i)}{\partial x_j} = (\dot{m}_i + \dot{m}_i) / \rho_i \quad (3)
\]

The mixture density and the viscosity are defined as follows.

\[
\rho_m = \rho_l \alpha_l + \rho_v (1 - \alpha_l) \quad (4)
\]

\[
\mu_m = \mu_l \alpha_l + \mu_v (1 - \alpha_l) \quad (5)
\]

In the above equations, \( \rho_l, \rho_v \) are the liquid and vapor density, \( \alpha_l, \alpha_v \) are the liquid fraction and the vapor fraction, \( \mu_l \) is the turbulent viscosity, \( \dot{m}_i, \dot{m}_i \) represent the condensation and evaporation rates. As for LES turbulence model, the momentum equation is modified as follows. \( \tau_{ij} \) is the subgrid stress (SGS), representing the influence of small scale vortex on the momentum equation.

\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_j u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_j} + \gamma \frac{\partial (\Delta S)_{ij}}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \quad (6)
\]

Modified Turbulence Model
Turbulence model plays an important role in the numerical simulation of cavitation flows. The SST k-omega turbulence model which developed by Menter is mixed with the k-omega model in the near-wall area and the k-epsilon model in the far field. Reboud gave the suggestion that an artificial reduction of the turbulent viscosity of this model can predict a more accurate frequency of the periodical shedding of cavitation. So a serial of modified SST k-omega models are applied following his idea.

\[
\mu_i = f(\rho) C_{\mu} \frac{k}{\omega} \quad (7)
\]

\[
f(\rho) = \rho_i + \left( \frac{\rho_a - \rho_i}{\rho_a - \rho_i} \right)^n \quad (8)
\]

Mass Transfer Model of Schnerr-Sauer
The mass transfer model which also called cavitation model adopted here was developed by Schnerr and Sauer. In their papers, the vapor fraction is related to the number of gas nucleus per unit volume and the average radius of gas nucleus. The condensation and evaporation rates are defined as follows

\[
\alpha_v = n_0 \frac{4}{3} \pi R^3 / \left( n_0 \frac{4}{3} \pi R^3 + 1 \right) \quad (9)
\]
\[
\dot{m}_c = C \frac{3 \rho_c \alpha_c (1 - \alpha_c)}{\rho R} \text{sgn}(P_c - P) \sqrt{\frac{2}{3 \rho_c}} \left| \frac{P_c - P}{3 \rho_c} \right|^{\frac{2}{3}} \quad (10)
\]

\[
\dot{m}_v = C \frac{3 \rho_c \alpha_c (1 - \alpha_c)}{\rho R} \text{sgn}(P_c - P) \sqrt{\frac{2}{3 \rho_c}} \left| \frac{P_c - P}{3 \rho_c} \right|^{\frac{2}{3}} \quad (11)
\]

\( R \) is the average radius of gas nucleus expressed as

\[
R = \left( \frac{\alpha_c}{1 - \alpha_c} \cdot \frac{3}{4 \pi n_0} \right)^{\frac{1}{3}} \quad (12)
\]

The parameter \( n_0 \) is the number of gas nucleus per unit volume as an important parameter for the description of mass transfer rates between vapor and fluid. It needs to be provided as input. In this paper, it is set with a default value of 1.6e+13.

**CASE DESCRIPTION**

**2D Computational Domain and Boundary Condition.**

For the investigations in the 2D computational domain, the test geometry is an Clark-Y hydrofoil at 8 degree angle of attack with chord length \( C = 100\text{mm} \). The computational domain is shown in Fig. 1. The size of the domain is \( 500 \times 300\text{mm} \), extending one chord length ahead of the leading edge and three chord lengths behind the trailing edge.

![Computation domain](image)

The inlet velocity is 10m/s, the pressure gradient and the outlet velocity is zero. Reference pressure is obtained by the cavitation number through equation 13. The cavitation number in the present work is 0.8.

\[
\sigma = \frac{P - P_c}{\frac{1}{2} \rho U^2} \quad (13)
\]

**RESULT AND DISSCUSSIONS**

**Cavitation Flow**

The two-dimensional cloud cavitation around Clark-Y hydrofoil in the uniform flow is investigated at 8 degree angle of attack and under 0.8 cavitation number, and the simulation results are compared with experimental ones. It is shown that the numerical method presented is able to simulate the whole period of the unsteady flow. The comparison of calculated cavitation shape and vapor content with the experimental ones is shown in Fig. 2. In Fig.
2(c) the cavity reached the maximum length. Retraction occurred at the junction between the end of the cavity and the hydrofoil, indicating the existence of the re-entrant jet, which caused the occurrence of small bubble shedding at the tail of the cavity. In Fig. 2 (d)–(f), the re-entrant jet flowed to the middle of the hydrofoil, causing the instability of cloud cavitation, which detached from hydrofoil and moved downward, forming an obvious cloud drop. The calculated results basically described the break up and detachment behavior of the cavitation, showing great agreement with experiment observation.

The modified SST k-ω model reduces the turbulent viscosity, and it’s closer to the experimental cavitation image. As can be seen, the phenomenon of cavitation breakup and falling off with time is more obvious, and the captured small vortices are more abundant. The distribution of speed when cloud cavitation detached is shown in Fig. 3. Both of them showed the existence of the re-entrant.

The change of pressure with time at different place of the hydrofoil is monitored, and the pressure coefficient curves are shown in Fig. 4. The monitor points are located at x/C=0.3, x/C=0.6 and x/C=0.9. It is found that the pressure coefficient fluctuated greatly as the location of monitor point moved downward, meaning there is a relatively stable low pressure zone at the front of the Hydrofoil. On the other hand, the pressure fluctuation at the tail of hydrofoil is large, and the cavitation motion is intense. The cloud cavitation is divided into two parts because of the re-entrant. The development, break up and shedding of cavitation caused the instability of pressure.
The lift and drag coefficient curves are shown in Fig. 5. The comparison of these curves and the cavitation shape in Fig. 5 shows that the lift and drag force of the hydrofoil fluctuated periodically with the change of the cavity length on the surface of hydrofoil. The calculated timely averaged lift and drag force is compared with the experimental results and other researchers’ work, shown in Table 1. The results in present paper are in good agreement with the experimental ones.

<table>
<thead>
<tr>
<th></th>
<th>( C_l )</th>
<th>( C_d )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment[12]</td>
<td>Present work</td>
<td>Data[13]</td>
</tr>
<tr>
<td>0.8</td>
<td>0.78</td>
<td>0.66</td>
</tr>
</tbody>
</table>

**CONCLUSION AND PROSPECT**

(a) The simulations obtained based on OpenFOAM of cavitation around Clark-Y hydrofoil are in agreement with the experimental result.

(b) The SST k-\( \omega \) model with modified eddy viscosity coefficient performs better at capturing the unsteady
cavitation vortex cluster, and the unsteady characteristics of the cloud cavitation changed periodically.

(c) In the follow-up study, the simulation accuracy of the modified SST k-ω model for different hydrofoils and working conditions can be further studied.

REFERENCES