Numerical simulations of cavitation flow around a hydrofoil with consideration of thermal effects

X R Zhao¹, H Y Luo², Y X Xiao¹, Z W Wang¹, S H Ahn¹, Y Y Luo¹

¹ State Key Laboratory of Hydrosience and Engineering & Department of Thermal Engineering, Tsinghua University, Beijing 100084, China
² Water Conservancy and Civil Engineering College, Tibet Agriculture and Animal Husbandry College, Lingzhi 860000, China

lhy_xz@sina.com

Abstract. Cavitation is one of the most important and common issues in hydraulic machines. Much attention has been paid on cavitation flow prediction, however, most of the current researches ignore the thermal effects during the cavitation process and have not taken into account the thermodynamic variation of phase change. In the presented paper, the cavitation flow characteristics around a hydrofoil NACA0015 was analyzed by numerical simulations and the results were compared with the experimental data from a reference paper. The focus was put on the cavitation process with respect to thermal effects. Steady and unsteady simulations were performed by ANSYS CFX 16.0 with a modified SST $k$-$\omega$ turbulence model under three different temperature conditions (25°C, 50°C, 70°C). The energy conservation equation was considered to describe the temperature field around the hydrofoil. To ensure the accuracy of simulations, the source term of thermal effects in vapor bubble transformation process was added into the evaporation and condensation functions of cavitation. The results indicated that simulations based on thermodynamic effects could detect the cavitation characteristics in detail and agree well with the experimental results.

1. Introduction

Cavitation is an important problem in the field of hydraulic machinery. Cavitation refers to the process of forming vapor bubbles in the low pressure region of the liquid flow field. The main reason for cavitation is the decrease of pressure in the flow field. Cavitation has a significant unsteady characteristic and could cause pressure fluctuation. In the meanwhile, due to the phase transition between water and water vapor, there exists a non-uniform temperature distribution in the flow field.

Although cavitation is a common phenomenon in pumps, hydraulic turbines and pump-turbines, it would take more time and effort to study cavitation in the hydraulic machinery accurately and finely. Therefore, many researchers are focused on the flow characteristic of the hydrofoil. Yang [1] applied a modified RNG $k$-$\varepsilon$ turbulence model, which takes both the water and vapor density into account, to simulate a 3D hydrofoil NACA66. The numerical results of shedding frequency agree well with the experimental data. Ahn [2] compared the standard RNG $k$-$\varepsilon$ turbulence model and the modified RNG $k$-$\varepsilon$ turbulence model in the unsteady simulations of NACA0015, which indicated that the modified RNG $k$-$\varepsilon$ turbulence model could describe the mean cavity length and pressure drop precisely. Roohi [3] adopted the VOF (volume of fluid) method in LES simulations to explore the dynamic feature of cloud cavitation and supercavitating flow structure around a hydrofoil. Huang [4] performed LES simulations on a Clark-Y hydrofoil and found that the sheet/cloud cavitation has a strong influence on vortex and...
boundary flow around the foil. Sedlar [5] conducted experimental and numerical analysis on a NACA2412 hydrofoil and compared the advantage and disadvantage of several simulation models. In the current study on hydrofoil cavitation, only a few papers take the change of temperature field into consideration. However, due to the water-vapor phase transition process, there exists temperature change in the flow field. In the current paper, the influence of thermal effects in phase change process caused by temperature difference was simulated. Cervone [6] carried out several experiments to analyze the cavitation phenomenon around NACA0015 hydrofoil under different temperature conditions, which could provide the analysis base for simulations in this study.

2. Numerical method

2.1. Cavitation model

In cavitation simulations, Rayleigh-Plesset cavitation model is adopted to control water vapor generation and condensation, which defines the grow rate of vapor bubble as follow:

$$R_B \frac{d^2 R_B}{dt^2} + \frac{3}{2} \left( \frac{d R_B}{dt} \right)^2 + \frac{2 \sigma}{\rho_l R_B} = \frac{p_v - p}{\rho_l}$$

(1)

Where $R_B$ is the radius of vapor bubbles, $\rho_l$ is the water density and $p_v$ is the vaporization pressure. For low oscillation frequencies flow, the quadratic term could be ignored. Therefore, the growth rate of bubble radius is:

$$\frac{dR_B}{dt} = \sqrt{\frac{2p_v - p}{\frac{3}{2} \rho_l}}$$

(2)

And the change rate of vapor bubble mass is:

$$\frac{dm_B}{dt} = \rho_v \frac{d}{dt} \left( 4 \pi R_B^3 \right) = 4 \pi R_B^2 \sqrt{\frac{2p_v - p}{\frac{3}{2} \rho_l}}$$

(3)

The number of bubbles per unit volume for vaporization is:

$$N_B = \frac{3 \alpha_v}{4 \pi R_B^3}$$

(4)

Where $\alpha_v$ is the volume fraction of vapor. And the condensation and evaporation rate can be expressed as:

$$m_{\text{cond}} = C_{\text{cond}} \frac{3 \alpha_v \rho_v}{R_B} \sqrt{\frac{2p_v - p}{\frac{3}{2} \rho_l}}$$

(5)

$$m_{\text{evap}} = C_{\text{evap}} \frac{3 (1 - \alpha_v) \rho_l}{R_B} \alpha_{\text{nuc}} \sqrt{\frac{2p_v - p}{\frac{3}{2} \rho_l}}$$

(6)

Where $\alpha_{\text{nuc}}$ is the volume fraction of the nucleation cells.

In this study, the thermal effects on bubble growth process in cavitation under different water temperature are taken into consideration. According to bubble dynamic theory [7], the thermal term should be added on the condensation and evaporation rate, which illustrates the impact of the difference between real bubble temperature and reference temperature. Therefore, the equation (5), (6) can be modified into the follows.

$$m_{\text{cond}} = C_{\text{cond}} \frac{3 \alpha_v \rho_v}{R_B} \sqrt{\frac{2p_v - p}{\frac{3}{2} \rho_l}} + \frac{\rho_v L}{\rho_l T_{\text{ref}}} \max (T_{\text{ref}} - T)$$

(7)
\[ m_{\text{evap}} = C_{\text{evap}} \frac{3(1 - \alpha_v)\rho_v}{R_B} \alpha_{\text{nuc}} \sqrt{\frac{2p_v - p}{3 \rho_i}} + \frac{\rho_vL}{\rho_i T_{\text{ref}}} \max(T_{\text{ref}} - T) \]  

In the present paper, the \( C_{\text{cond}} \) is set as 0.01, \( C_{\text{evap}} \) is 50, and \( \alpha_{\text{nuc}} \) is 0.0005.

### 2.2. Turbulence model and boundary conditions

In this research, ANSYS CFX 16.0 is used to solve Reynolds-averaged Navier-Stokes (RANS) equations. The SST \( k-\omega \) turbulence model, which combines the \( k-\omega \) and \( k-\varepsilon \) models, is adopted in steady and unsteady simulations. This model has better adaptability to capture the flow structure near wall. Due to the compressibility in water-vapor mixed flow, many researchers argue that the common turbulence model could overrate the actual eddy viscosity when cavitation appears in the flow field. Coutier-Delgosha [8] suggested to apply a correction function on density to modify the RNG \( k-\varepsilon \) turbulence model. In this paper, the modification is adopted as the eddy viscosity in SST \( k-\omega \) turbulence model, as follow:

\[ f(\rho) = \rho_v + \left( \frac{\rho_v - \rho_m}{\rho_v - \rho_l} \right)^{10} (\rho_l - \rho_v) \]  

\[ \mu_c = \frac{\alpha_1 f(\rho) k}{\max(\alpha_1 \omega, SF_z)} \]  

The inlet boundary is set as normal speed, which is in accordance with the experiment, and the temperature is defined as 25°C, 50°C and 70°C for different simulating conditions. The outlet is set as static pressure according to the cavitation number \( \sigma \), which is:

\[ \sigma = \frac{p_{\infty} - p_v}{\frac{1}{\pi \rho u^2}} \]  

### 2.3. Mesh generation

Hydrofoil NACA0015 with 5° attack angles is simulated. The structured hexahedral meshes are produced by ANSYS ICEM 16.0. The calculated domain is composed with 1.68 million of mesh elements and the generated mesh is finer near the foil, of which the average dimensionless wall distance \( y^+ \) is 2.77.
3. Results and analysis

3.1. Pressure coefficient distribution

The calculating results with the modified cavitation model were obtained through steady and unsteady simulations. The pressure coefficient distribution of polyline between the Z=0 plane and the foil surface was compared with experimental data under the three temperature conditions. As shown in figure 2, the simulating results of the Rayleigh-Plesset cavitation model and modified cavitation model, and the experimental data are compared. When comparing the three temperature conditions, it is detected that the cavitation bubble length decreases when the temperature is higher. Especially in the 70°C condition, the bubble occupies only approximately 18% of the foil length. This phenomenon is also displayed in figure 3, which is the absolute pressure distribution at Z=0 plane.

According to figure 2, it is indicated that under 25°C and 50°C conditions, both calculating results agree well with the experimental data with angle of attack of 5 degrees and 1.5 cavitation number. However, there exists a significant difference between the pressure coefficient distribution of two cavitation models under 70°C condition. The result of modified model is in accordance with experimental data while the disparity of results between Rayleigh-Plesset cavitation model and experiments cannot be ignored. It can be concluded that the modified cavitation model is accurate enough to describe the cavitation flow around hydrofoil under varied temperature conditions and it fill gaps on cavitation prediction of Rayleigh-Plesset model under high temperature conditions.

(a) 25°C

(b) 50°C
Figure 2. Comparison of pressure coefficient between experimental and simulating results

(c) 70°C

Figure 3. Absolute pressure distribution at Z=0 plane

(a) 25°C
(b) 50°C
(c) 70°C

3.2. Vapor volume distribution

Figure 4 illustrates the vapor volume fraction distribution at 1.70s, 1.85s and 2.0s under 25°C, 50°C, 70°C conditions. It is demonstrated that the cavity flow with cavitation number of 1.5 does not change with time significantly and only the length of vapor bubble has an unobvious unsteady characteristic. It can be concluded that the cavitation flow is relatively stable with angle of attack of 5 degrees and 1.5 cavitation number.

When comparing the cavitation phenomena under the three temperature conditions, it is detected that the vapor distribution is not similar. Under 25°C condition, the length of bubble is comparatively longer. But under 50°C condition, the high volume fraction area (0.9~1.0) is more obvious than other conditions. Under 70°C condition, the cavitation phenomenon is quite inapparent.
Figure 4. Vapor volume distribution at several moments under 25°C, 50°C, 70°C conditions

4. Conclusions
In this paper, a modified cavitation model was proposed, which takes the thermal effects in phase transformation process of water vapor bubbles into consideration through adding source term on the condensation and evaporation rate. Steady simulations on a hydrofoil NACA0015 with 5° attack angles were performed with this modified model and Rayleigh-Plesset cavitation model under 25°C, 50°C, 70°C conditions and the applied turbulence model is also with correction.

The pressure coefficient distribution was compared with the experimental data from Cervone’s research. The results indicate that the modified model can predict cavitation phenomenon more accurately than the Rayleigh-Plesset model, especially under higher temperature condition. Mean cavity length decreases obviously when the temperature is higher and the thermal effects are more significant in this condition.

The unsteady cavitation characteristic was detected through simulations. It is illustrated that under these three temperature conditions, the cavity flow does not change with time significantly. The cavitation is stable with angle of attack of 5 degrees and 1.5 cavitation number. But the flow distribution is varied when the temperature is different.

Acknowledgments
Special thanks are due to the State Key Program of National Science of China (Grant No. 51439002), National Natural Science Foundation of China (No. 51479093), Special Funds for Marine Renewable Energy Projects (Grant No. GHME2012GC02), the National Key Research and Development Program of China (No. 2017YFC0404200), Chinese National Sci-Tech Support Plan (Funding NO. 2012BAD10B01) for supporting the present work.

References
[1] Yang J, Zhou L J, and Wang Z W. "Numerical simulation of three-dimensional cavitation around a hydrofoil." Journal of Fluids Engineering 133.8 (2011): 081301.
[2] Ahn S H, Xiao Y X, and Wang Z W. "Numerical simulation of unsteady flow characteristics for cavitation around a 3-D hydrofoil." IOP Conference Series: Materials Science and Engineering. Vol. 72. No. 2. IOP Publishing, 2015.
[3] Roohi E, Zahiri A P, and Passandideh-Fard M. "Numerical simulation of cavitation around a two-
dimensional hydrofoil using VOF method and LES turbulence model." Applied Mathematical Modelling 37.9(2013):6469-6488.

[4] Huang B, Zhao Y, and Wang G. "Large Eddy Simulation of turbulent vortex-cavitation interactions in transient sheet/cloud cavitating flows." Computers & Fluids 92.3(2014):113-124.

[5] Sedlar M, et al. "Numerical and experimental investigation of three-dimensional cavitating flow around the straight NACA2412 hydrofoil." Ocean Engineering 123(2016):357-382.

[6] Cervone A, et al. "Thermal cavitation experiments on a NACA 0015 hydrofoil." Journal of fluids engineering 128.2 (2006): 326-331.

[7] Brennen C E. "Cavitation and Bubble Dynamics." 33.4(1995):609-617.

[8] Coutier-Delgosha O, Fortes-Patella R, and Reboud J. "Evaluation of the turbulence model influence on the numerical simulations of unsteady cavitation." Journal of Fluids Engineering 125.1 (2003): 38-45.