Numerical simulation of unsteady flow characteristics for cavitation around a 3-D hydrofoil

S H Ahn, Y X Xiao and Z W Wang
State Key Laboratory of Hydroscience and Engineering & Department of Thermal Engineering, Tsinghua University, Beijing 100084, China
E-mail: wzw@mail.tsinghua.edu.cn

Abstract. At present it is possible to predict more accurately by various numerical methods established for cavitation simulation around a hydrofoil. However, for the solution of the complex unsteady cavity flow, it is still marginal. In this paper, numerical method is adopted to simulate cavitation around 3-D NACA0015 hydrofoil with homogeneous two-phase flow calculation using commercial code CFX-solver with two turbulence models, the standard RNG $k$-$\varepsilon$ turbulence model and the modified RNG $k$-$\varepsilon$ turbulence model respectively. First, pressure coefficient for non-cavitating flow, time averaged values of unsteady cavity flow around a hydrofoil are verified to simulate more closely to an actual cavity flow. And then frequency analysis is performed with Fast Fourier Transform. The results show that the calculation results with modified RNG $k$-$\varepsilon$ turbulence model agree with experimental results in terms of mean cavity length and pressure drop, but the unsteady flow characteristics of oscillating cavitation still deviate slightly in terms of unsteady cavity flow.

1. Introduction
Cavitation is a phenomenon generated from expansion of the non-condensable gas when the local pressure falls below specific value in constant temperature. This phenomenon causes complex pressure fluctuation, for hydraulic machinery, the periodic change in pressure damages to some parts in the machine. Furthermore, the cavitation bubble has an impact on surrounding liquid flow, which is relevant to the mechanical efficiency. The cavitation flow can bring out erosion, noise, vibration and so on. Thus, the cavitation is the problem which should be solved.

At present it is possible to observe cavitation phenomena more clearly and to measure more accurately by the development of experimental devices in cavitation around a hydrofoil. And also various numerical methods for cavitation calculation were already established, it is possible to more accurately predict it. Coutier-Delgosha et al. [1] presented that cavitation simulation with $k$-$\omega$ turbulence model including compressibility effect can be obtained better reasonable result. Kunz et al. [2] compared numerical results with turbulence modeling of between RANS and DES. Zhou et al. [3] simulated the cavitation flow on a hydrofoil considering the influence of non-condensable gas. Ji et al. [4, 5] introduced PANS method for cavitating flow and also showed the numerical results using the actual maximum density ratio. But it is still limited to solve the complex cavitation phenomena which are compressible multiphase flow. In this paper, numerical analysis is adopted to simulate unsteady cavitating flow around 3-D NACA0015 hydrofoil. The numerical result with two turbulence models

1 Corresponding author. Tel.: +86 10 51533279.
for cavitation are compared to experimental result, first, time averaged values are compared, and then unsteady characteristics are evaluated for two turbulence models.

2. Numerical method

2.1. Governing equations
In homogeneous two phase flow, all fluids are in a flow field, share a velocity field. The homogeneous model assumes that each quantity transported for all phases are same. The homogeneous continuity and momentum equations are as:

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

(1)

\[
\frac{\partial (u_i)}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nabla^2 u - S
\]

(2)

where \(\bar{\rho}, \nu, S\) are the mixture density, viscosity and the stress tensor, which are the following:

\[
\rho = \sum_{a=1}^{N_p} \alpha_a \rho_a
\]

(3)

\[
\mu = \sum_{a=1}^{N_p} \alpha_a \mu_a
\]

(4)

\[
S = \frac{\partial}{\partial x_j} \left[ \nu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right]
\]

(5)

where \(\alpha_i\) is the volume fraction of each phase, \(N_p\) is the number of phases.

2.2. Cavitation model
The Rayleigh-Plesset model is typical cavitation model which control vaporization and condensation.

\[
R \frac{d^2R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2 = \frac{p_v - p}{\rho_l}
\]

(6)

where \(R\) represents radius of bubble, \(\bar{\rho}_l\) is water liquid density. Equation (6) is reduced by neglecting the second order terms as following:

\[
\frac{dR}{dt} = \left( \frac{2 p_v - p}{\frac{3}{\rho_l}} \right)^{1/2}
\]

(7)

The change rate of bubble volume is:

\[
\dot{V} = 4\pi R^2 \left( \frac{2 p_v - p}{\frac{3}{\rho_l}} \right)^{1/2}
\]

(8)

The vaporization bubble initially contains non-condensable gas. The sum of volume fraction of each phase is as \(\alpha_i + \alpha_v + \alpha_{ncg} = 1\) and the mixture density is as \(\bar{\rho} = \rho_i \alpha_i + \rho_v \alpha_v + \rho_{ncg} \alpha_{ncg}\). The number of bubbles per unit volume for vaporization, \(N\), can be written as:

\[
N = \frac{3 \alpha_{ncg} (1 - \alpha_v)}{4\pi R^3}
\]

(9)
and for condensation, \( N \) is following:

\[
N = \frac{3\alpha_v}{4\pi R^3}
\]  

(10)

Cavitation process is governed by kinetics of phase change; mass transfer equation between vapor and liquid is as:

\[
\frac{\partial (\rho_v \alpha_v)}{\partial t} + \nabla \cdot (\rho_v \alpha_v U) = \dot{m}_e - \dot{m}_c
\]  

(11)

where \( \alpha_v \) is vapor volume fraction. Equation (11) represents mass transfer rate of \( \alpha_v \) between evaporation and condensation.

\[
\dot{m}_e = C_e \frac{3\rho_v (1 - \alpha_v) \alpha_{ncg}}{R} \left( \frac{2\rho_v - p}{3 \rho_t} \right)^{1/2}
\]  

(12)

\[
\dot{m}_c = C_e \frac{3\rho_v \alpha_v}{R} \left( \frac{2\rho_v - p}{3 \rho_t} \right)^{1/2}
\]  

(13)

where \( C_e, C_c \) are empirical coefficient for vaporization and condensation, which are 50 and 0.01, respectively in this paper.

2.3. Numerical model and boundary conditions

Unsteady calculation of the homogeneous two-phase flow with the cavitation model is used to obtain the results of unsteady cavity flow. The cavity flow around 3-D hydrofoil has velocity component of the spanwise direction, which is not zero by the effect of the side walls [6, 7]. Therefore, 3-D hydrofoil was adopted for this calculation domain, NACA0015 based on the experimental study by Cervone et al. [8], which is symmetry shaped model between upper and lower with chord length of 115 mm and span length of 80 mm. This simulation was used structured grids of about 1.4 million nodes, the density of grids is finer near the walls, especially around hydrofoil, where o-grid method was used as figure 1. The boundary conditions of the pressure at inlet and the mass flow rate at outlet were set according to cavitation number is defined as:

\[
\sigma = \frac{p_0 - p_v}{\frac{1}{2} \rho U_0^2}
\]  

(14)

where \( p_0 \) and \( U_0 \) are the freestream pressure and velocity. The no slip condition was set at other walls including a hydrofoil. In this paper, the calculation condition is as following: angle of attack of 8 degree for leading edge, cavitation number of 1.8, temperature is constant of 25 °C. The initial condition in this domain was obtained from two-phase flow calculation of water liquid-vapor without the cavitation model to observe initial bubble change. Convergence criterion was specified as root mean square of 10^{-5}, time step was set and controlled by CFL number according to velocity change in calculation domain, and the commercial CFD code CFX-solver is used.

In this paper, the numerical results will be compared between RANS with RNG \( k-\varepsilon \) turbulence model and with the modified RNG \( k-\varepsilon \) turbulence model of which the eddy viscosity can be changed to the function of density proposed by Coutier-Delgosha et al. [1]. The standard RNG \( k-\varepsilon \) turbulence model was developed for incompressible fluid, but the cavitation is two-phase compressible flow [6]. The eddy viscosity of standard RNG \( k-\varepsilon \) model, \( C_{\mu}^{0} \), is as:

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}
\]  

(15)

where \( C_\mu \) represents turbulence model constant, \( k \) turbulence kinetic energy, \( \varepsilon \) turbulence dissipation rate. And modified one is as follows:
\[ \mu_t = f(\rho) C_\mu \frac{k^2}{\epsilon} \]  
\[ f(\rho) = \rho_v + (\alpha_t)^n(\rho_l - \rho_v) \]

where \( \alpha_t \) is the volume fraction of water liquid. And the turbulence model constant \( C_\mu \) and \( C_{\epsilon 1}, C_{\epsilon 2} \) coefficients are 0.085, 1.42 and 1.68, respectively, for RNG \( k-\epsilon \) turbulence model in this paper.

\[ \square_n \]

\( \text{Figure 1. Computational grids (angle of attack = 8 degree).} \)

3. Numerical results

The results for unsteady calculation with the standard RNG \( k-\epsilon \) turbulence model and the modified RNG \( k-\epsilon \) turbulence model were obtained. There is no significant difference between with two different turbulence models in non-cavitating condition because this flow is almost same to single phase flow. Simulation result agrees with experimental result for pressure coefficient on a hydrofoil as figure 2. It is essential to verify calculation result of non-cavitating flow for obtaining more accurate simulation result of the cavitating flow. Table 1 shows the mean cavity length and the time averaged pressure drop around a hydrofoil of between numerical and experimental result with angle of attack of 8 degree and 1.8 cavitation number. As a result, the calculation result with modified RNG \( k-\epsilon \) turbulence model is better close to experimental data, which model is changed eddy viscosity in the standard model to the function of density. The cavity flow with cavitation number of 1.8 is sufficient complex in figure 3, it shows the bubble change of vaporization and condensation with contour on the mid-plane of spanwise and iso-surface of 0.1 for vapor volume fraction, the phenomena of both bubble and cloud cavitation can be seen for this cavitation number.

\[ \text{Figure 2. Pressure coefficient distribution of experiment and calculation.} \]
Table 1. Mean cavity length and pressure drop round a hydrofoil.

|       | \( l/C \) | \( \Delta p/0.5\rho U_0^2 \) |
|-------|-----------|-----------------------------|
| EXP   | 0.78      | 0.156                       |
| RNG   | 0.372     | 0.149                       |
| RNG modified | 0.782     | 0.161                       |

\( t/T = 0.2 \)

\( t/T = 0.4 \)

\( t/T = 0.6 \)

\( t/T = 0.8 \)

(a) Iso-surface

(b) Contour

Figure 3. Instantaneous vapor volume fraction (\( \sigma = 1.8 \), 8 degree).
Figure 4 shows the result of frequency analysis with Fast Fourier Transform, the amplitude on this diagram is a non-dimensional value of pressure pulsation divided by dynamic pressure. In this result, dominant frequency of the calculation with standard RNG $k$-$\varepsilon$ turbulence model, Strouhal number of 0.18, is about half of with modified one. It is still different from experimental result for oscillation cavitation and the amplitude of pressure pulsation is also larger than measurement value.

![Frequency diagram of pressure pulsation at a hydrofoil upstream](image)

**Figure 4.** Frequency diagram of pressure pulsation at a hydrofoil upstream (left: standard RNG $k$-$\varepsilon$ model, right: modified RNG $k$-$\varepsilon$ model)

4. Conclusion

The unsteady cavitation simulation around a NACA0015 hydrofoil with two turbulence models, the standard RNG $k$-$\varepsilon$ turbulence model and the modified RNG $k$-$\varepsilon$ turbulence model of which turbulence viscosity is changed by local density, was calculated to evaluate an accuracy for unsteady cavitation characteristics comparing to the experimental results by Cervone et al. [8]. In this paper, a cavitation number with complex cavity flow, which has bubble and cloud cavitation simultaneously, is adopted because unsteady characteristics is obvious for this condition. To simulate more close to actual cavity flow, the calculation results of pressure coefficient for non-cavitating condition, mean cavity length and pressure drop around a hydrofoil for cavity flow were compared, and then the frequency analysis of pressure pulsation with Fast Fourier Transform was performed. The results show that modified RNG $k$-$\varepsilon$ turbulence model can be simulated well through the comparison of mean cavity length and pressure drop, but the dominant frequency and amplitude of pressure pulsation still have difference in terms of unsteady cavity flow.

Acknowledgments

Special thanks are due to the State Key Program of National Science of China (Grant No. 51439002), National Natural Science Foundation of China under Contract (No. 51479093), State Key Laboratory of Hydrosience and Engineering (Grant No. 2014-KY-05) for supporting the present work.

References

[1] Coutier-Delgosha O, Fortes-Patella R and Reboud J L 2003 *J. Fluid Eng.-T. ASME* **125** 38-45
[2] Kunz R F, Lindau J W, Kaday T A and Peltier L J 2003 *Proc. the 5th international symposium on cavitation* (Osaka, Japan, 1-4 Nov 2003) pp 1-4
[3] Zhou L and Wang Z 2008 *J. Fluid Eng.* **130** 011302-1
[4] Ji B, Luo X, Wu Y, Peng X and Xu H 2012 *Int. J. Heat and Mass Transfer* **55** 6582-8
[5] Ji B, Luo X W, Wu Y L, Peng X X and Duan Y L 2013 *Int. J. Multiphase Flow* **51** 33-43
[6] Yang J, Zhou L J and Wang Z W 2011 *J. Fluid Eng.-T. ASME* **133** 081301-1
[7] He M, Yuan L L, Zhou L J, Yang J and Wang Z W 2014 *Eng. Computations* **31** 501-9
[8] Cervone A, Bramanti C, Rapposelli E and d'Agostino L 2006 *J. Fluid Eng.-T. ASME* **128** 326-31