11th International Workshop on Ship and Marine Hydrodynamics
Hamburg, Germany, September 22-25, 2019

Numerical Investigation of Cavitation and Hydrodynamic Characteristics of Damaged Propeller

Minsheng Zhao and Decheng Wan*

State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai 200240, China

*Corresponding author: dcwan@sjtu.edu.cn

ABSTRACT

In the present study, a numerical simulation of cavitation flows around a damaged and a complete four –bladed E779A propeller has been carried out. The numerical results can be used as a reference to evaluate the working ability of a propeller in case of damage. In numerical simulation, the damaged propeller is simplified as the INSEAN E779A propeller missing one blade, hence, it's a three-bladed. The typical unsteady dynamics are predicted by the RANS method with a modified shear stress transport (SST) k-ω turbulence model. The numerical results such as cavitation shape, pressure distribution and the thrust coefficient Kt are analysed and compared with each other, to investigate the influence of propeller damage. The numerical calculations of non-cavitating flow are obtained using pimpleFoam. The results of open water characteristics are basically in accordance with experimental data, the error is about 6%, indicating the reliability of the present method. Then the cavitation flow simulations are solved by InterPhaseChangeDyMFoam in the open source CFD software platform OpenFOAM with Schnerr-Sauer cavitation model. It can be noted that the propeller with a missing blade output less thrust, and the influence of the asymmetry of the three-blade propeller on the cavitation and open water characteristics is also obvious. The disturbance of cavitation flow to pressure variation is also the reason for the reduction of propeller thrust coefficient. With the change of advance coefficient, this influence will be strengthened.

Key word: Damaged propeller; Open water characteristics; Cavitation; naoe-FOAM-SJTU solver

1 INTRODUCTION

Cavitation is a phenomenon with intense vaporization of liquid which happens when local pressure of liquid falls below the critical pressure threshold. Cavitation contains most of the complex flow in fluid dynamics and the comprehension we have is still limited. Modern ships face the challenge of high speed and high risk sea conditions. The occurrence of cavitation is unavoidable because of the speed increase, and the propeller is sometimes damaged in actual navigation. Therefore, it is necessary to study the cavitation and hydrodynamic characteristics of propeller under dangerous working conditions.

Common propeller cavitation includes tip vortex cavitation, hub vortex cavitation and blade surface cavitation. Because of the complexity of physical problems, model test technology is still the main method to solve these problems[1]. Qualitative observation of void morphology was carried out at model scale, based on experience and a large number of experimental data[2]. The main test equipment used is stroboscope, pressure sensor, hydrophone and high-speed photography. Although the model test is still a relatively intuitive method for predicting cavitation flow. However, due to the long period of model test, high cost, and scale effect, unconventional test is not easy to achieve[3], such as the influence of stern wake. The numerical simulation method arose at the moment. The numerical study of propeller cavitation was carried out based on potential flow theory in the early years. At present, numerical simulation technology based on viscous flow theory is becoming more and more important [4]. It can simulate propeller wake velocity and pressure distribution, around hydrofoil and turbine cavitation well.

Li Daqing[5] used RANS method to investigate the open water characteristics of the big skewed propeller and compared it with the experiments results, the error of hydrodynamic coefficients are less than 5%. Dular et al[6] adopted a multiphase flow model on unsteady numerical calculations around a hydrofoil. Gaggero[7] compared the difference of RANS method and Panel method in analysing the performance of propellers in unsteady flow. Krasilnikov[8] also used the RANS method to calculate the force of a propeller in unsteady flow. At first, the cavitation simulations are focused on hydrofoil and navigation body[9], the simulation about propeller are relatively rare and most of them are calculated by commercial software[10-12]. Liu [13] used Singhal complete cavitation model in commercial software Fluent to simulate E779A propeller's surface morphology. Lindau et al.[14] used a three-dimensional (3D), multiphase RANS tool, UNCLE-M, to predict the load of a designated propeller. Rhee et al.[15] used an unstructured mesh-based RANS method to simulate the cavitation characteristics of a marine propeller. A series of complete and detailed experimental studies on open water performance and cavitation characteristics of E779A propeller were carried out by INSEAN, Italian Model Pool Center[16]. The unsteady sheet cavitation of E779A propeller at model scale is analysed by Salvatore[17] with method of coupling viscous flow with potential flow. Kim [18] adopted the code of open source platform OpenFOAM to verify the simulated cavitation extension, and a more accurate result was shown. Dhinesh et al.[19] studied the viscous flow around the self-propelled hull with a RANS solver at model scale. Ahn and Kwon[20] investigated the cavitating flows around marine propulsor with a multiphase RANS flow solver based on pseudo compressibility and a mixture model using unstructured meshes. Kunz et al.[21] investigated the characteristics of the sheet cavitation on a hydrofoil with the k-ε re-normalisation group turbulence model. Wang, et al.[22] investigated the hydrodynamic performance of PPTC propeller in open water and cavitation flow using naoe-FOAM-SJTU solver developed with OpenFOAM platform.

The numerical results can be used as a reference to evaluate the working ability of a propeller in the case of damage. In numerical simulation, the damaged propeller is simplified as a three-blade INSEAN E779A propeller. All the simulation calculations of open water characteristics of propeller are obtained using pimpleFoam, and the cavitation flow simulation are solved by InterPhaseChangeDyMFoam in the open source CFD software platform OpenFOAM with Schnerr-Sauer cavitation model. The numerical results in open water of standard E779A propeller are basically in accordance with experimental data, indicating the reliability of the present method. It is found that the propeller with a missing blade output less thrust significantly, and the influence of the asymmetry of the three-blade propeller on the cavitation and open water characteristics is also obvious. The disturbance of cavitation flow to pressure variation is also the reason for the reduction of propeller thrust coefficient. With the change of advance coefficient, this influence will be strengthened.

2 NUMERICAL METHODS

2.1 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.

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial\left(\rho_m u_j\right)}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial(\rho_m u_j)}{\partial t} + \frac{\partial(\rho_m u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}[\mu(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i})] - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

$$\frac{\partial \alpha_l}{\partial t} + \frac{\partial}{\partial x_j}(\alpha_l u_j) = (\dot{m}_c + \dot{m}_v) / \rho_l \tag{3}$$

The mixture density and the viscosity are defined as follows.

$$\rho_m = \rho_l \alpha_l + \rho_v (1-\alpha_l) \tag{4}$$
$$\mu_m = \mu_l \alpha_l + \mu_v (1-\alpha_l) \tag{5}$$

In the above equations, $\boldsymbol{\rho}_l$, $\boldsymbol{\rho}_v$ are the liquid and vapor density, $\boldsymbol{\alpha}_l$, $\boldsymbol{\alpha}_v$ are the liquid fraction and the vapor fraction, $\boldsymbol{\mu}_t$ is the turbulent viscosity, representing the condensation and evaporation rates. As for LES turbulence model, the momentum equation is modified as follows.$\boldsymbol{\tau}_{ij}$ is the subgrid stress (SGS), representing the influence of small scale vortex on the momentum equation.

$$\frac{\partial(\overline{u_i})}{\partial t} + \frac{\partial(\overline{u_i} \cdot \overline{u_j})}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_j} + \gamma \frac{\partial(2\overline{S_{ij}})}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{6}$$

2.2 Modified Turbulence Model

Turbulence model plays an important role in the numerical simulation of cavitation flows. The SST k-ω turbulence model which is developed by Menter is mixed with the k-ω 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-ω models are applied following his idea.

$$\mu_t = f(\rho) C_\omega \frac{k}{\omega} \tag{7}$$
$$f(\rho) = \rho_v + \frac{(\rho_m - \rho_v)^n}{(\rho_l - \rho_v)^{n-1}}; n \gg 1 \tag{8}$$

2.3 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.

$$\alpha_v = n_0 \frac{4}{3}\pi R^3 / (n_0 \frac{4}{3}\pi R^3 + 1) \tag{9}$$
$$\dot{m}_c = C_c \frac{3\rho_v \rho_l \alpha_v (1-\alpha_v)}{\rho R} \mathrm{sgn}(P_v - P) \sqrt{\frac{2|P_v - P|}{3\rho_l}} \tag{10}$$
$$\dot{m}_v = C_v \frac{3\rho_v \rho_l \alpha_v (1-\alpha_v)}{\rho R} \mathrm{sgn}(P_v - P) \sqrt{\frac{2|P_v - P|}{3\rho_l}} \tag{11}$$

R is the average radius of gas nucleus expressed as

$$R = (\frac{\alpha_v}{1-\alpha_v} \cdot \frac{3}{4\pi n_0})^{1/3} \tag{12}$$

3 SIMULATION SETUP

The propeller type used in this paper is E779A. Because of the abundant experimental data, this propeller is one of the most widely used propellers in the research of propeller cavitation. E779A is a four-blade propeller with a diameter of 0.253m. The disk ratio, pitch ratio and other parameters are shown in Table 1. In numerical simulation, the damaged propeller is simplified as a three-blade INSEAN E779A propeller. That is used to simulate the propeller damaged by accident in real navigation. In practice, when the propeller is damaged, some blades may break, which is more complex

(a)Damaged propeller (b) Four-bladed propeller

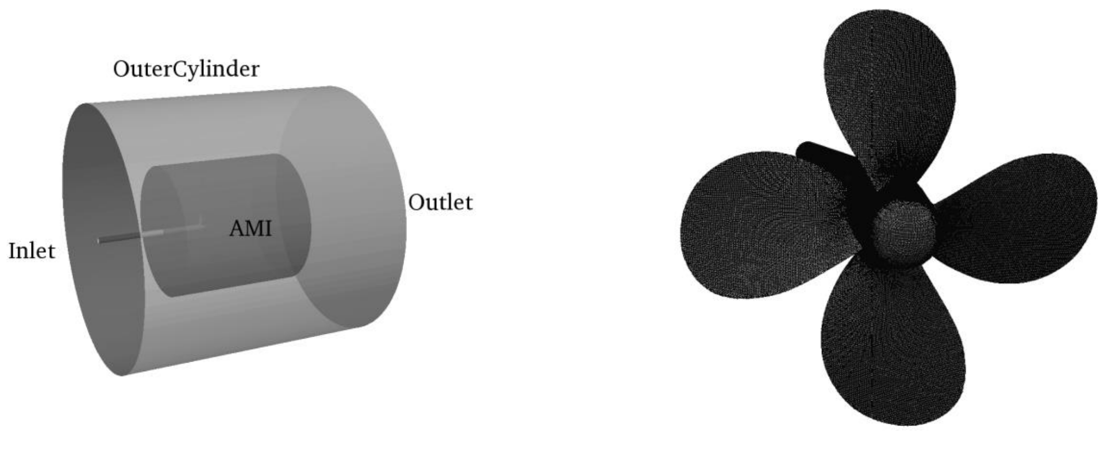
Figure 1: E779A propeller model

Table 1: E779A propeller parameters

Items	WT1	WT2
Airfoil	D	0.253
Rotor Diameter	d/D	0.200
Nacelle Diameter	$P_{0.7}/D$	1.100
Height of tower	Ae/Ao	0.689
Pitch Angle	Z	4
Tip Speed Ratio	0	4

In the present work, the interPhaseChangeDyMFoam solver in CFD open source platform OpenFOAM is used. The dynamic mesh solver is designed to solve the cavitation problem, which needs the corresponding dynamic mesh. Because of the complex geometry of propeller, structured grid is difficult to apply, so unstructured grid is adopted. The layout of propeller model, slip surface and propeller surface mesh are shown in Fig. 2. In order to ensure sufficient accuracy and minimize computational effort, the total number of grids is finally determined to be about 3.4 million.

The computational domain is $-L < X < 4\ L$, $-10\ D < Y, Z < 10\ D$, where L is the length of the cylinder and D is the diameter of the cylinder. The inlet velocity is calculated according to the advance coefficients. It should be noted that the characteristic length in this Reynolds number is based on the diameter of the cylinder, not the length. The pressure gradient and the outlet velocity gradient is zero. The simulation domain and the grid are shown in Fig. 2.



(a)Computational Domain and Slip Surface Layout (b) Propeller Surface Mesh Distribution

Figure 2: Computational domain and grid

4 RESULTS AND DISCUSSIONS,

The open water performance and cavitation characteristics of propellers are the focus of the research. Preliminary prediction results are obtained by analysing the open water performance calculation and cavitation calculation results of damaged propellers. The study of pressure distribution on propeller blade surface can find out the cause of negative influence of cavitation on propeller hydrodynamic performance.

4.1 The numerical results of open-water characteristics

Firstly, the open water performance of E779A propeller is simulated. The validity of the numerical method is verified, and then the cavitation flow simulation is carried out. In open water condition, the rotary speed of propeller is 25rps, and the advance coefficients are 0.6, 0.71, 0.77 and 0.83, which are normal forward flow. The calculated thrust coefficients under different working conditions are shown in Table 2.

From Table 2, it can be seen that the thrust coefficient is close to the experimental value as a whole. When the advance coefficient is small, the error is small. But when the advance coefficient increases, the error increases slightly, about 6%. It can be considered that the simulation results of open water characteristics are good and have certain accuracy and reliability.

Table 2: Open water characteristics simulation

J	K_T (Exp)	K_T (Num)	Error
0.6	0.293	0.281	4.09%
0.71	0.247	0.234	5.26%
0.77	0.215	0.202	6.04%
0.83	0.170	0.159	6.47%

4.2 Simulation on E779A considering cavitating flows

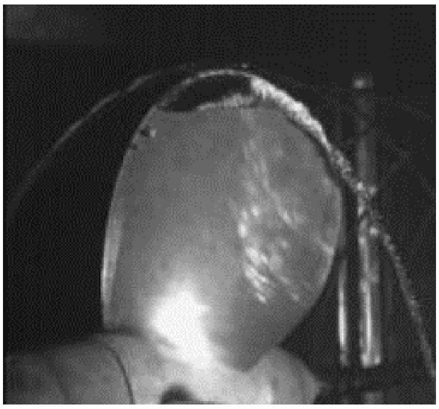
The thrust coefficients of propellers without cavitation under these three conditions are given by experimental data. When simulating cavitation, the interPhaseChangeDyMfoam solver in OpenFOAM s used. The far-field pressure can be calculated according to the propeller speed and cavitation number. The cavitation simulation parameter setting are shown in Table 3.The formula is as follows:

$$\sigma_n = \frac{p - p_v}{0.5\rho(nD)^2} \tag{13}$$

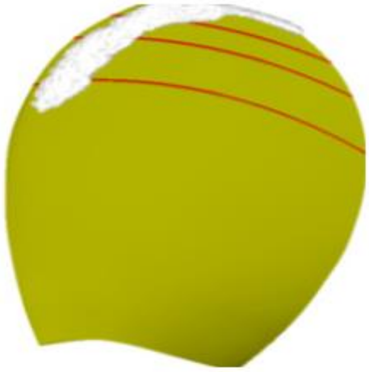
Table 3: Simulation parameter setting

Parameter	Case1	Case3	Case3
Rotating speed(rps)	24.98	24.97	25.01
Advance coefficients	0.71	0.77	0.83
Cavitation Number	1.515	1.783	2.016
Saturated vapor pressure (Pa)	2818	2818	2818
The kinematic viscosity of water (m^2/s)	9.34e-7	9.34e-7	9.34e-7
Density of water (kg/m^3)	997.44	997.44	997.44

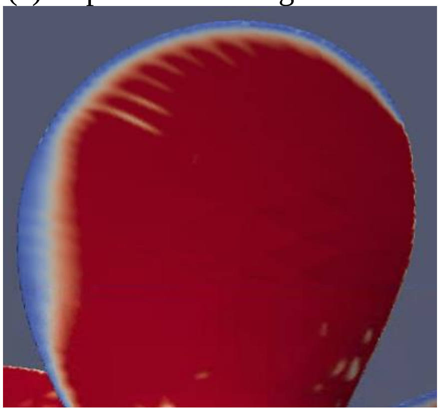
The numerical simulation of cavitation on a three-blade INSEAN E779A propeller is carried out. The cavitation calculation of propeller are simulated under three different conditions. The comparison between simulation results and experimental images are shown below. The image on the left of the first line is the image observed in the experiment. The picture on the right of the first line is the result of LIU [23]'s numerical simulation. The second line are the simulation results of the present work.



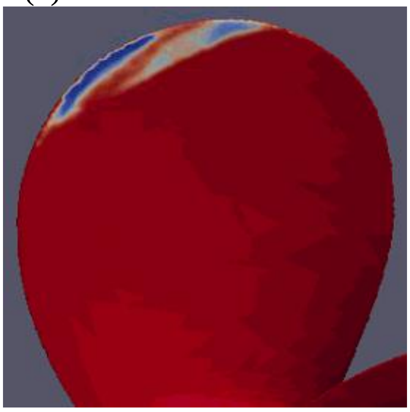
(a) Experimental images



(b) Documental results



(c) Damaged propeller



(d) Complete propeller

Figure 3: Computational domain and grid Cavitation distribution on blade surface

Figure 3 shows the surface cavitation distribution of propeller blades under the same conditions. As can be seen from the figure, for experiments and several simulation results, Cavitation mainly occurs at the tip of propeller blade and the leading edge of suction surface (shadow part). Because the experimental image is monochrome, the cavitation of the leading edge of the propeller's suction surface is not obvious. In the numerical simulation results of Liu and the research results of this paper, the cavitation phenomenon of suction surface's leading edge can be seen. It should be noticed that the blade used in Figure 3d is the second one in the clockwise direction. The results of cavitation simulation in this paper are generally close to those of experimental images and LIU[23]. From the point of view of cavitation area, the area of cavitation on a complete four-blade propeller blade is larger than that of a damaged propeller with only three blades. Cavitation affects propeller performance and erodes the surfaces of the materials. In Figure 3c, the small cavitation lines starting at the leading edge and developing into the blade surface and those patterns on lower radii near the trailing edge may be caused by or may follow certain cells with bad quality, and this cavitation pattern will be studied in the following research. Because of the asymmetry of three-blade propeller, the flow field around different blades is actually different, so the characteristics of cavitation on each blade should also be different. This difference will affect the cavitation characteristics of the propeller, and then cause additional interference to the thrust coefficient.

4.3 Comparison of Propeller Thrust Coefficients

Comparing the thrust coefficients in the case of cavitation and non-cavitation flow, it can be seen that the negative effect of cavitation on the hydrodynamic performance of propeller is great. The calculation results of thrust coefficients of E779A propeller at different propulsion coefficients are also given in open water experiments. It can be used to compare with the simulation results. Table 4 gives the relative values of simulation results and experimental results. Comparisons of the difference between cavitation and non-cavitation are also included.

Table 4: Open water characteristics simulation

	Case1	Case2	Case3
K_T(damaged)	0.201	0.173	0.135
K_T(whole)	0.234	0.202	0.159
K_T(Exp)	0.247	0.215	0.170
K_T loss(EFD)	-18.62%	-19.53%	-20.59%
K_T loss(CFD)	-14.10%	-14.36%	-15.09%

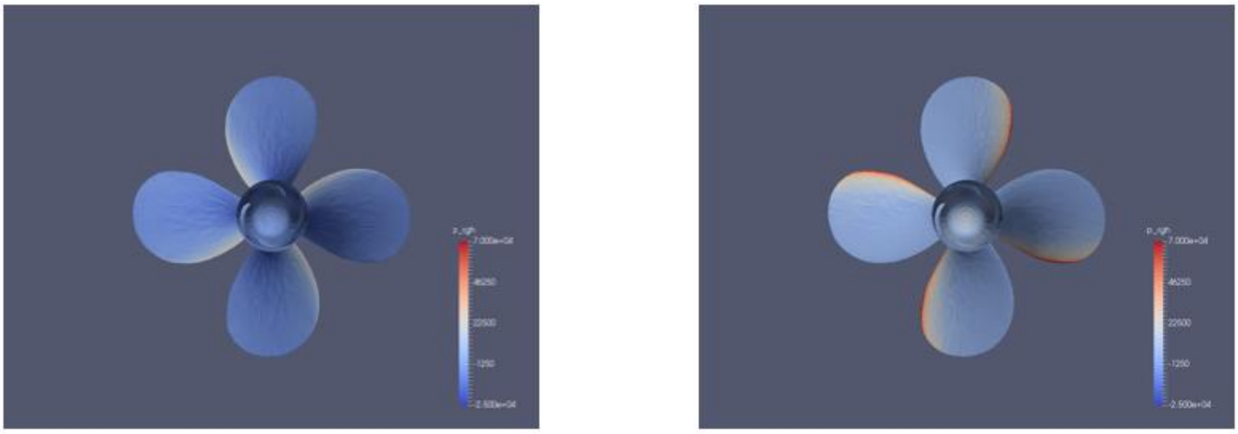
From the calculation results of propeller thrust coefficient, it is found that the thrust coefficient of the damaged propeller is obviously lower than that of the complete propeller. The thrust coefficients of damaged propellers differ from those obtained from open water tests, and the difference increases with the increase of thrust coefficients. This phenomenon is predictable, because the damaged propeller lacks a blade. In theory, the thrust coefficient of the damaged propeller should be 75% of that of the intact propeller. But the thrust coefficient of the damaged propeller is higher than 75%. In the simulation of cavitation, it can be seen that the differences of thrust coefficients between the damaged propeller and the complete propeller are smaller than in the non-cavitating flow. Therefore, the lack of one blade causes about 15% reduction of the thrust coefficient in cavitating flow. From the results of cavitation simulation, the cavitation area of the damaged propeller is smaller than that of the complete four-blade propeller. Cavitation is a factor affecting hydrodynamic performance of propeller. It can be concluded that because of the smaller cavitation area and the lower degree of cavitation, the loss of thrust coefficient of three-blade propeller is less. As for how cavitation affects propeller hydrodynamic performance will be analysed in the next part of this paper. Therefore, in actual ship navigation, a damaged propeller without one blade can also generate thrust close to 85% of the original thrust in cavitating flow, and still maintain a certain power for ship navigation.

However, it must be noted that the propeller's thrust coefficient is only an average reflection of its hydrodynamic performance. The flow field around the damaged propeller is asymmetrical. Compared with the complete four-blade propeller, the flow field around the damaged propeller must have changed. This asymmetry cannot be observed directly only by comparing thrust coefficients. Necessary analysis of the flow field will also be made in future studies to study the effects of cavitation on the damaged propeller. When the damaged propeller lacks two blades in symmetrical form, the influence on flow field and hydrodynamic performance is also worth further study.

4.4 Pressure Distribution

The pressure distribution of propeller back (suction surface) with and without cavitation is compared with that of propeller back (suction surface) as shown in Fig. 4. When the incoming flow reaches the leading edge of the propeller, a low pressure zone will be formed on the suction surface due to the separation of the flow. The pressure difference between the two sides of the propeller produces thrust. However, when the pressure in the low pressure zone is reduced to the saturated vapor pressure at that temperature, local cavitation will occur. The pressure in the cavitation zone does not decrease anymore and remains at the saturated vapor pressure. When the gas is far away from the leading edge on the suction surface, it will liquefy again. Pressure begins to return to the ambient flow field. That is to say, the existence of cavitation will inhibit the formation of low-pressure region to a certain extent. This is similar to Chen [22]'s simulation results for the cavitation flow of PPTC propeller. The reliability of the numerical method is verified. This restraint leads to the decrease of pressure difference and thrust coefficient of propeller. It can also be seen from Fig. 4 that the pressure is obviously higher than that of no cavitation when there is cavitation. It should be noted that the low pressure region of the propeller is relative. In fact, the pressure of the flow field around the propeller is higher because of the influence of the water depth of the propeller blade. Therefore, the cavitation will collapse soon after it is separated from the blade. The research on the cavitation characteristics of two-dimensional hydrofoils also verifies this point.

The analysis of the pressure distribution and size on the propeller surface shows that the occurrence of cavitation does affect the propeller thrust. But whether the negative effect of cavitation on hydrodynamic performance is the only one is debatable. Variation of flow field and complex vorticity structure will affect propeller when cavitation occurs. From the view of fluid mechanics, there are more difficulties to be analysed.



(a)Without Cavitation. (b) With Cavitation.

Fig.4 Suction Surface Pressure Distribution of E779A Propeller with and without Cavitation.

5 CONCLUSIONS

In this paper, the hydrodynamic performance and cavitation characteristics of E779A propeller are numerically simulated based on interPhaseChangeDyMFoam, a sliding grid cavitation solver in CFD platform OpenFOAM. By comparing with open water test data, the reliability of the numerical method is verified. The thrust coefficient of the propeller is close to the experimental result under the condition of lower advance coefficient, and the error is about 6%. The simulation results of the cavitation flow of the damaged E779A propeller under uniform flow condition show that the area of cavitation on the blade of the three-blade propeller is slightly smaller than that of the complete four-blade propeller due to the complexity and asymmetry of the flow field. The cavitation shapes of damaged propeller and intact propeller are close to the experimental results, and are in good agreement with the simulation results made by predecessors.

The thrust coefficient of the three-blade propeller is higher than the theoretical value. It can be considered that the area of cavitation of the three-blade propeller is slightly smaller, the negative impact of cavitation on the thrust coefficient is smaller, and the difference between the thrust coefficient and the open water test is smaller. With the increase of the thrust coefficient, the difference between the simulation results and the experiment results of the three-blade propeller thrust coefficient increases gradually. Further research shows that the cavitation of propeller blade surface will inhibit the reduction of pressure in the suction surface area of propeller, reduce the pressure difference between the two sides of propeller blade, and consequently reduce the thrust coefficient of propeller.

ACKNOWLEDGEMENTS

This work is supported by the National Natural Science Foundation of China (51879159, 51490675, 11432009, 51579145), Chang Jiang Scholars Program (T2014099), Shanghai Excellent Academic Leaders Program (17XD1402300), Program for Professor of Special Appointment (Eastern Scholar) at Shanghai Institutions of Higher Learning (2013022), Innovative Special Project of Numerical Tank of Ministry of Industry and Information Technology of China (2016-23/09) and Lloyd’s Register Foundation for doctoral student, to which the authors are most grateful.

REFERENCES

1. YANG Qiongfang, WANG Yongsheng, ZHANG Zhihong. Effects of non-uniform inflow on propeller cavitation hydrodynamics[J]. Chinese Journal of Hydrodynamics, 2011,26(5):538-550.
2. Lavis DR, Forstell BG, Purnell JG. Compact waterjets for high-speed ships. Ships Offshore Sturct. 2007, 2:115–125.

3. Pan G, Lu L, Sahoo P K . Numerical simulation of unsteady cavitating flows of pumpjet propulsor[J]. Ships and Offshore Structures, 2016, 11(1):11.
4. Kaijie Chen, Decheng Wan. Numerical simulation on cavitation flow of hydrofoil with viscosity-modified SST k-ω turbulence model[J]. Chinese Journal of Hydrodynamics, 2019, 34(2): 224-231.
5. Da-Qing L I. Validation of RANS predictions of open water performance of a highly skewed propeller with experiments[J]. Journal of Hydrodynamics. Series B, 2006, 18(3-supp-S):520-528.
6. Dular M, Bachert R, Stoffel B, et al. Experimental evaluation of numerical simulation of cavitating flow around hydrofoil[J]. European Journal of Mechanics, B/Fluids, 2005, 24(4):522-538.
7. Gaggero S, Villa D, Brizzolara S. RANS and PANEL method for unsteady flow propeller analysis[J]. Journal of Hydrodynamics Ser B, 2010, 22(5):564-569.
8. Krasilnikov V, Zhang Z, Hong F, Analysis of unsteady propeller blade forces by RANS. In: First international symposium on marine propulsors symposium on Marine Propulsors, Trondheim, Norway, 2009.
9. Bin J I, Luo X, Yulin W U, et al. Numerical analysis of unsteady cavitating turbulent flow and shedding horse-shoe vortex structure around a twisted hydrofoil[J]. International Journal of Multiphase Flow, 2013, 51(5):33-43.
10. Salvador G P , Frankel S . Numerical Modeling of Cavitation Using FLUENT: Validation and Parametric Studies[C]// Aiaa Fluid Dynamics Conference & Exhibit. 2013.
11. Bei-Bei L I, Xiu-Mei L, Zheng L, et al. Simulation and analysis for cavitation flow field in a throttle valve based on Fluent[J]. Journal of Vibration and Shock, 2015.
12. Dingxuan Z, Qian W, Miaomiao D U. Fluent-Based Numerical Simulation of the Cavitation Behavior in the Angle Nozzle[J]. Journal of Northeastern University, 2016.
13. LIU Dengcheng, HONG Fangwen, ZHAO Fen, et al. The CFD analysis of propeller sheet cavitation[C]// 8th International Conference on Hydrodynamics, Nantes, France, 2008, 2B-03: 1-6.
14. Lindau JW, Boger DA, Medvitz RB, Kunz RF. 2005. Propeller cavitation breakdown analysis. J Fluid Eng Trans ASME. 127:995–1002.
15. Rhee SH, Kawamura T, Li H. 2005. Propeller cavitation study using an unstructured grid-based Navier-Stokes solver. J Fluid Eng Trans ASME. 127:986–994.
16. Dengcheng Liu, Fangwen Hong. The Numerical Predicted of SMP11 Propeller Performance with and without Cavitation. Second International Symposium on Marine Propulsors, SMP'11. Hamburg, Germany, June 2011.
17. Salvatore F, et al. A Viscous / Inviscous Coupled Formulation for Unsteady Sheet Cavitation Modelling of Marine Propellers[C]. Fifth International Symposium on Cavitation Osaka, Japan, November 1-4, 2003.
18. Kim SE. Multiphase CFD simulation of turbulent cavitating flows in and around marine propulsors. Proceedings of Open Source CFD International Conference; 2009 November 12–13; Barcelona, Spain.
19. Dhinesh G , Murali K , Subramanian V A . Estimation of hull-propeller interaction of a self-propelling model hull using a RANSE solver[J]. Ships and Offshore Structures, 2010, 5(2):125-139.
20. Ahn S J, Kwon O J. Numerical investigation of cavitating flows for marine propulsors using an unstructured mesh technique[J]. International Journal of Heat and Fluid Flow, 2013, 43:259-267.
21. Kunz R F, Boger D A, Stinebring D R , et al. A preconditioned Navier–Stokes method for two-phase flows with application to cavitation prediction[J]. Computers & Fluids, 2000, 29(8):849-875.
22. Jianhua Wang, Lu Zou, Decheng Wan. Numerical Simulations of Zigzag Maneuver of Free Running Ship in Waves by RANS-Overset Grid Method[J]. Ocean Engineering, 2018, 162: 55-79
23. Liu D. The numerical simulation of propeller sheet cavitation with a new cavitation model[C]// International Conference on Fluid Mechanics. 2015.