Energy Saving Mechanism of Propeller with Endplates at Blade Tips

Yijie Liu, 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

Ship energy-saving has always been one of the performance improvements that researchers are trying to improve. It is an effective energy-saving way to maintain a certain chord length at the tip of the propeller blade and install an arc end plate at the tip of the blade. With the development of CFD(computational fluid dynamics), many researchers combine the design and research of energy-saving devices with the CFD methods. This paper calculated the hydrodynamic of propeller with endplates based on OpenFOAM, the flow field and models are processed by sliding grid technology. By comparing the open water characteristics and thrust performance of standard MP687 propeller and improved propeller, it is found that the propeller with endplates can maintain a larger circulation strength and prevent the transverse disturbance at the end of the blade; at the same time, it can reduce the cavitation zone at the blade of the propeller and play the role of noise reduction, erosion reduction and vibration reduction; besides, it can also eliminate tip vortices, achieve the effect of energy-saving.

KEY WORDS: endplates at blade tips; energy-saving; sliding mesh; OpenFOAM

INTRODUCTION

For the high-energy-consuming shipping industry, researchers pay more and more attention to the "Green Ship" design, which can reduce energy consumption and cost. Among all the methods of energy saving and fuel consumption reduction, the improvement of propeller and propeller is the most effective. At present, most of the energy-saving propulsion technologies focus on improving the propeller inlet, improving the pressure on the blade, reducing the friction force on the blade surface[1, 2], so as to make more use of the energy that can not be used and improve the transmission shafting. Although many propulsion devices have been developed, there are still many problems affecting the efficiency of propulsion, but this shows that energy-saving propulsion devices still have great potential for development.

Early researchers mainly carried out research on energy-saving devices through ship model test or theoretical research. In recent years, the numerical simulation method for simulating fluid flow by means of computer has gradually emerged. It can greatly reduce the design time and cost, so it has attracted the attention of many scholars. Current numerical simulation methods mostly use viscous flow method. Viscous flow numerical simulation methods can be divided into three categories: DNS (direct turbulence numerical simulation), RANS (Reynolds average N-S equation) and LES (large eddy numerical simulation). DNS method is a direct discrete solution to the N-S model, which can accurately obtain the flow field information; RANS method can be solved by a smaller calculation process; LES method can save the calculation process by filtering small vortices and only calculating large vortices, and the results obtained are more accurate than RANS method. At present, many scholars [3] are continually studying and improving the viscous flow simulation methods.Chang [4] et al. simulated the cavitation flow around the propeller by RANS method, and analyzed the image results of cavitation on the suction surface of the propeller under the numerical simulation, which verified the feasibility of RANS simulation. Peters [5] et al. successfully simulated the cavitation flow around a marine propeller using a RANS-based solver and Volume Fluid (VOF) method.

Combining with the numerical simulation method, many energy-saving propellers have been designed and studied by researchers. Berger et al. [6] studied the optimal design process of hub cap fins by using CFD methods. Takafumi Kawamura et al. [7] used CFD methods to study the difference of energy-saving effect of hubcap fins on ship models and real-scale ships. Anirban et al. [8] discussed the influence of scale effect on hydrodynamic characteristics of ducted propeller based on CFD methods. Fahri [9] used commercial software to study the front catheter installed on chemical tankers. Sunho [10]et al. used commercial software STAR-CCM+ to study the wake field of the pre-rotating stator in the self-propelled sailing process of KVLCC2 in real ship scale. Based on OpenFOAM open source platform, the numerical simulation of MP687 propeller with end plates and standard MP687 propeller is carried out in open water. By comparing the force and vorticity changes of the propeller before and after adding endplates at blade tips, the energy-saving mechanism and energy-saving effect are analyzed.

NUMERICAL METHOD

Governing Equations

The governing equation of numerical simulation is incompressible fluid continuum equation. As for RANS turbulence model, the equations are presented below.

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u_i)}{\partial x_i}=0 \tag{1}$$

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_i u_t)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_i}\right)-\rho\overline{u_i' u_j'}\right) \tag{2}$$

In the above formulas, u_i and u_j are the i and j components of velocity, P is the hydrostatic pressure, μ is the hydrodynamic viscous coefficient.

$-\rho\overline{u_i' u_j'}$ is the Reynolds stress term. Reynolds stress term ensures the closure of RANS equation by turbulence model.

SST k-omega turbulence model

In SST k-omega model, k-omega model is used near the wall and k-epsilon model is used in the far field. At present, it is one of the models with high usability, which is mainly used to simulate Reynolds stress. The main equations for this model are as follows.

$$\frac{\partial\rho k}{\partial t}+\frac{\partial\rho u_i k}{\partial x_i}=\tilde{P}_k-\beta^*\rho k\omega+\frac{\partial}{x_i}\left[(\mu+\sigma_k\mu_t)\frac{\partial k}{\partial x_i}\right] \tag{3}$$

$$\frac{\partial\rho\omega}{\partial t}+\frac{\partial\rho u_i\omega}{\partial x_i}=\alpha\rho S^2 P_k-\beta\rho\omega^2+\frac{\partial}{x_i}\left[(\mu+\sigma_\omega\mu_t)\frac{\partial\omega}{x_i}\right]+ 2(1-F_1)\rho\sigma_{\omega 2}\frac{1}{\omega\partial x_i}\frac{\partial k}{\partial x_i}\frac{\partial\omega}{\partial x_i} \tag{4}$$

$$F_1=\tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega y},\frac{500\nu}{y^2\omega}\right),\frac{4\sigma_{\omega 2}k}{CD_{k\omega}y^2}\right]\right\}^4\right\} \tag{5}$$

$$CD_{k\omega} = \max\left(2\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i}, 10^{-10}\right) \tag{6}$$

The decoupling of velocity and pressure adopts PIMPLE algorithm, which combines the advantages of PISO algorithm and SIMPLE algorithm. SIMPLE algorithm is used in the time step, and PISO algorithm is used in the time step, so that the time step can be increased properly on the premise of ensuring stability.

Hydrodynamic Characteristics of Propellers

The evaluation of propeller and improved propeller can be considered from thrust coefficient, torque coefficient and propulsion efficiency. The propeller's speed coefficient is J, thrust coefficient is K_T, torque coefficient is K_Q, and propulsion efficiency is η_0.

$$K_T = \frac{T_P}{\rho n^2 D^4} \tag{7}$$

$$K_Q = \frac{Q_P}{\rho n^2 D^5} \tag{8}$$

$$\eta_0 = \frac{K_T}{K_Q}\frac{J}{2\pi} \tag{9}$$

ρ is density, n is speed of propeller, D is diameter of propeller.
Density is 998.2kg/m^3, and the kinetic viscosity coefficient of water is ν=1.106×10^{-6} m^2/s.

CASE DESCRIPTION

Computing Model

The propeller used in this paper is MP687 propeller. The improved propeller is MP687 propeller with endplates at blade tips. The specific parameters of the propeller and the improved propeller are shown in Tables 1 and 2.

Table 1. MP687 propeller model parameters

	MP687	
Diameter (m)	D	0.203
Hub ratio	r_h/R	0.180
Pitch ratio	$P_{0.7}$/D	0.750
Disk ratio	Ae/Ao	0.500
Trim angle（deg）	θ	5
Number of propeller blades	Z	5

Table 2. MP687 propeller model parameters

	endplates	
Extend (m)	a	0.09
Intermediate (deg)	α	87
Height (m)	b	0.03
Number of propeller blades	Z	5

The geometric model of MP687 propeller and MP687 propeller with endplates at blade tips is built by using commercial software CATIA according to the spatial coordinates of propeller and hub cap fin, as shown in Figure 1 and Figure 2.

Fig. 1 MP687 propeller model.

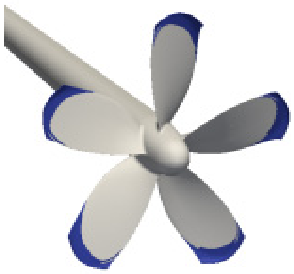

Fig. 2 MP687 propeller model with endplates at blade tips.

Mesh Generation

Arbitrary Mesh Interface (AMI) is a sliding mesh method, which can be used to solve unsteady flow problems. The computational domain is divided into two sub-regions. The propeller is completely enveloped in a cylindrical rotating sub-region, as shown in Fig. 3. Because the geometry of propeller blade is complex and has irregular surface, unstructured mesh is used in calculation. The mesh generation is shown in Fig. 4 and Fig. 5, and the total mesh is about 4 million.

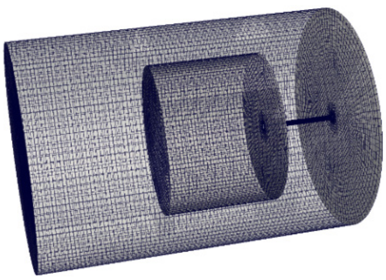

Fig. 3 Computational domain.

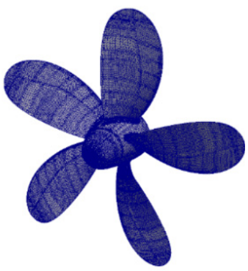

Fig. 4 Computational grid of MP687 propeller.

Fig. 5 Computational grid of MP687 propeller with endplates.

Accuracy Verification of AMI Method

In order to verify the accuracy of the AMI method in dealing with rotating grids, the open water performance of propeller is simulated and compared with the experimental data provided by NMRI. The calculation of open water propeller MP687 is performed by pimpleDyMFoam solver in OpenFOAM. During the calculation, the speed of the propeller is fixed and RPS=20 is maintained.

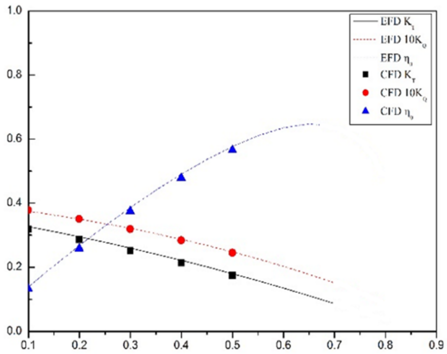

Fig. 6 Open Water Characteristic Calculating Curve of Propeller.

The numerical simulation is carried out under the condition of J=0.2, J=0.3, J=0.4, J=0.5. In Fig. 6, the error between the open water characteristics of propeller calculated by AMI method and the test results is within a small range, which verifies the accuracy and reliability of the AMI method in the calculation of rotating mesh.

RESULT AND DISSCUSIONS

Analysis and comparison of hydrodynamic characteristics

The speed of the propeller remains unchanged at 20r/s, and J is adjusted by changing the flow velocity. The hydrodynamic performance of propeller with and without endplates under J=0.2 is compared in Table 3. The thrust of the propeller itself increases significantly, the torque decreases significantly, and the propulsion efficiency of the whole system improves significantly.

Table 3. comparison of hydrodynamic characteristics (J=0.2)

	J	Thrust(N)	Torque(N*m)	KT	10KQ	η0
Without endplates	0.2	195.380	4.848	0.2881	0.3522	0.2604
With endplates	0.2	198.831	4.763	0.2932	0.3460	0.2697

Force Analysis of Propeller

The surface pressure of two kinds of propellers are analyzed and the pressure distribution maps are drawn, as shown in Fig. 7 and Fig. 8. By comparing Fig. 7 with Fig. 8, it is not difficult to find that in 7, the high pressure area at the guide edge of the blade is obviously larger than that of the standard propeller due to the addition of endplates, which means that the pressure difference between the front and rear of the propeller will increase, thus leading to the increase of the propeller thrust. The reason for this phenomenon is related to the addition of endplates, which can increase the high pressure area and improve efficiency.

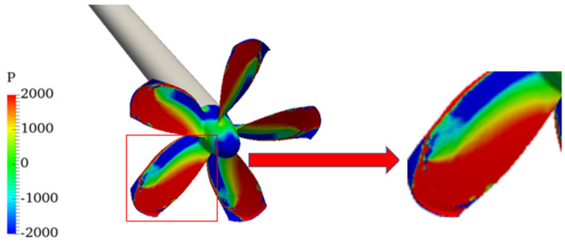

Fig. 7 Surface Pressure Diagram of MP687 Propeller with Endplates.

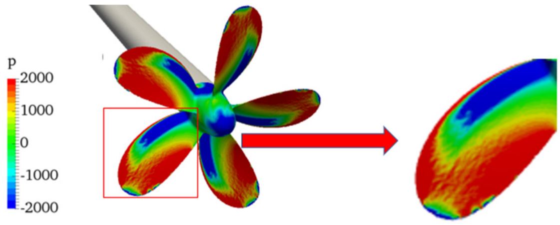

Fig. 8 Surface Pressure Diagram of MP687 Propeller.

Contrast of Vortex Ejection

The vortex spray charts of the two models are shown as fellows. It is easy to find that the improved propeller tail vorticity is more uniform, and because the end plate can eliminate the tip vorticity, thus eliminating unnecessary energy consumption, greatly improving the energy saving and propulsion capacity of the propeller.

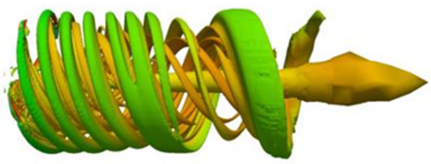

Fig. 9 Vortex Spray Chart of MP687 Propeller with Endplates.

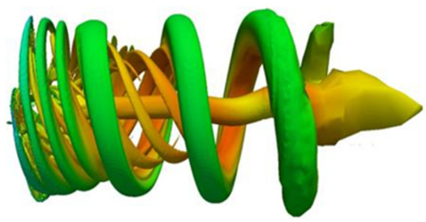

Fig. 10 Vortex Spray Chart of MP687 Propeller.

CONCLUSION AND PROSPECT

(1) Using solver under OpenFOAM platform, based on RANS equation, adding A turbulence model and

combining with the AMI meshing method is an effective numerical simulation approach for propeller, which can achieve good accuracy. An effective way to predict propeller performance and design energy-saving device is discussed for researcher to carry out numerical simulation of propeller in this paper.

(2) The improvement of propeller propulsion efficiency by endplates is mainly based on two aspects: on the one hand, endplates can increase the high pressure area and increase the pressure difference between front and rear blades. On the other hand, the endplates can reduce the tip vorticity and eliminate the tip vorticity, so as to improve the propulsion efficiency of the propeller.

(3) In the follow-up work, the influence of endplates on wake field of propeller can be further studied.

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] TAKEO N, NORIO I, HISASHI K. Energy saving technology of pbcf (propeller boss cap fins) and its evolution[C]. Okayama, Japan, 2010.

[2] OUCHI K, OGURA M, KONO Y, et al. A research and development of PBCF(propeller boss cap fins)-improvement of flow from propeller boss[J]. Journal of Society of Naval Architects of Japan. 1988, 163: 66-78.

[3] Kim S E. A numerical study of unsteady cavitation on a hydrofoil. Proceedings of the 7th international symposium on cavitation, [J]., 2009.

[4] Chang Y C , Hu C N , Tu J C , et al. Experimental investigation and numerical prediction of cavitation incurred on propeller surfaces [J]. Journal of Hydrodynamics, Ser. B, 2010, 22(5-supp-S1):764-769.

[5] Peters A , Lantermann U , Moctar O E . Numerical Prediction of Cavitation Erosion on a Ship Propeller in Model-andFull-Scale[J]. Wear, 2018.

[6] Berger S , Druckenbrod M , Pergande M , et al. A Two-Stage Optimisation Method for Full-Scale Marine Propellers Working Behind a Ship[J]. Ship Technology Research, 2014, 61(2):64-79.

[7] Kawamura T, Ouchi K, Nojiri T. Model and full scale CFD analysis of propeller boss cap fins (PBCF)[J]. Journal of Marine Science and Technology, 2012, 17(4):469-480.

[8] Bhattacharyya A,Krasilnikov V,Steen S.A CFD-based scaling approach for ducted propellers[J]. Ocean Engineering,2016,(7):116-130.

[9] FAHRI C. A numerical study for effectiveness of a preduct[J]. Ocean Engineering, 2007, 34: 2138-2145.

[10] SUNHO P, GWANGHO O, SHIN H R, et al. Full scale wake prediction of an energy saving device by using computational fluid dynamics[J]. Ocean Engineering, 2015, 101: 254-263.