Applied Mechanics and Materials
ISSN: 1662-7482, Vols. 641-642, pp 283-287
doi:10.4028/www.scientific.net/AMM.641-642.283

Submitted: 2014-07-05
Accepted: 2014-07-08
Online: 2014-09-19

Hydrocomp Design and Prediction of Hydrodynamic Performance of Propeller

Aifeng Zhang[1,a], Tianlei Zhang[1,b], Jiangming Ding[2,c], Jinyang Liu[2]

[1]Transportation Equipments and Ocean Engineering College, Dalian Maritime University, Dalian, China, 116026

[2]Transportation College, Wuhan University of Technology, Wuhan, China, 430063

[a]april98@21cn.com, [b]silvia-vampire@hotmail.com, [c]goodluckdjm@sina.com

Keywords: Propeller, Hydrodynamic Performance, Atlas Method, ANSYS

Abstract. With the continuous development of large-scale, high-speed ship, considering economy and security of the ship, propeller design and performance research are of importance. In this paper, the best propeller is obtained by using the propeller design software HydroComp PropExpert and HydroComp PropCAD. Propeller geometry model is generated by Solidworks. Applying computational fluid dynamics method (CFD), we can take numerical simulation for the flow around the propeller. Finally, with the help of the software ANSYS, hydrodynamic performance of the design propeller can be forecasted. The results from the calculation and analysis can provide some reference for engineering practical problems and theoretical study.

Introduction

Mapping design which is based on the results of open water propeller model test draws the various types of special pattern related to geometry parameter and performance to design. In fact, according to the known condition, within the range of details of propeller that can be provided in the map, we can choose the most suitable scheme [1]. In 1969, Holland ship model experiment using the regression analysis method handled open water propeller regression polynomial for the first time. Since then, propeller map design took a step into computerization. Hydrocomp computer aiding ship propulsion system design software is a professional software which is developed by Hydrocomp USA company, for solving the design of ship propulsion system. It can solve the core problem quickly and accurately in the research field of ship propulsion system. Hydrocomp series software includes three modules: NavCAD, PropExpert and PropCAD, for differently solving the practical application problems in fields of ship propulsion system analysis, design and production of propeller [2]. Propeller design changes from traditional manual drawing way into the computer analysis, which not only greatly improves the design efficiency, but also makes the design more scientific and standardized[3]. This paper uses the software to design marine propeller. On the other hand, the hydrodynamic performance of the propeller is calculated based on ANSYS ICEM CFD software, which is a powerful pre-processing software and it can provide high quality meshes for the mainstream CFD software(FLUENT CFX etc), making the result more accurately[4].

Mapping Design

Determine the Propeller Elements. Using PropExpert software sets general hull, hull parameters, main engine, energy transfer, propeller size and speed. Finally we get the main elements design propeller, as shown in table 1.

PropCAD Generate Propeller Data Modeling. Import PropCAD the data which is generated by PropExpert and carry on propeller geometry data modeling. Finally, the elements of each blade section are shown in table 2.

Propeller CFD Model

Geometric Modeling. Solidworks software is used to establish the geometry modeling of the whole propeller, as shown in figure 1. Figure 2 is the result of the whole calculation domain modeling.

Table 1 The Main Elements of the Design Propeller

Propeller Type	Diameter $D(m)$	Pitch $P(m)$	Design RPM $N(r/\min)$	Design Speed $V(kn)$	Rated Power （kw）
B 4-85	2.398	1.579	285	12.7	1000

Table 2 Propeller Each Blade Section Elements

r/R	Radius(m)	Chord(m)	Max Thickness (m)	Skew Back(m)	Pitch(m)
0.1	0.1199	0.60687	0.09784	-0.07434	1.20004
0.2	0.2398	0.69988	0.08777	-0.08168	1.29478
0.3	0.3597	0.79289	0.0777	-0.08902	1.38952
0.4	0.4796	0.86288	0.06763	-0.08902	1.48426
0.5	0.5995	0.90616	0.05755	-0.07764	1.55531
0.6	0.7194	0.9209	0.04748	-0.05598	1.579
0.7	0.8393	0.90248	0.03741	-0.02203	1.579
0.8	0.9592	0.82973	0.02734	0.03029	1.579
0.9	1.0791	0.66581	0.01726	0.09912	1.579
0.95	1.13905	0.51478	0.01223	0.14226	1.579
0.975	1.16903	0.37218	0.00971	0.16404	1.579
0.9875	1.18401	0.26724	0.00845	0.17474	1.579
0.9938	1.19157	0.18707	0.00673	0.18011	1.579
1	1.199	0.00719	0.0036	0.18539	1.579

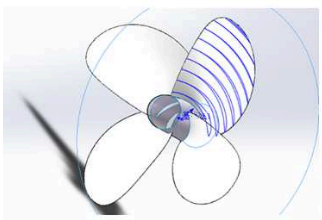


Fig. 1 The Complete Propeller Model

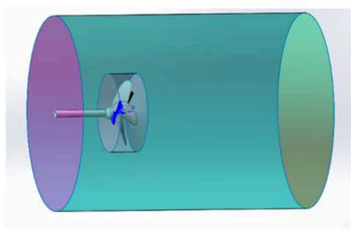


Fig. 2 The Computational Domain Modeling

Meshing. Static domain includes the out-domain inlet section, out-domain outlet section, out-domain cylindrical boundary, inner-domain inlet section, inner-domain outlet section, inner-domain cylindrical boundary, propeller shaft and diversion cover; rotation domain includes inner-domain inlet section, inner-domain outlet section, inner-domain cylindrical boundary, blade surface and hub surface. The derived results of static domain and rotating domain are input into ANSYS ICEM to do flow field grid job. Grid of static domain and rotation domain are obtained as shown in figure 3 and figure 4, respectively.

Numerical Calculation and Results

Propeller Open Water Characteristic Curve. Using CFX-Pre module to complete pre-processing, we first export the grid of static domain and rotation domain, then run the solver to calculate the torque and thrust of propeller and obtain the numerical value of advance coefficient *J*, thrust Coefficient K_T, torque coefficient K_Q, open water efficiency η_0, which are under different speed. The open water performance curve is drawn as shown in figure 5.

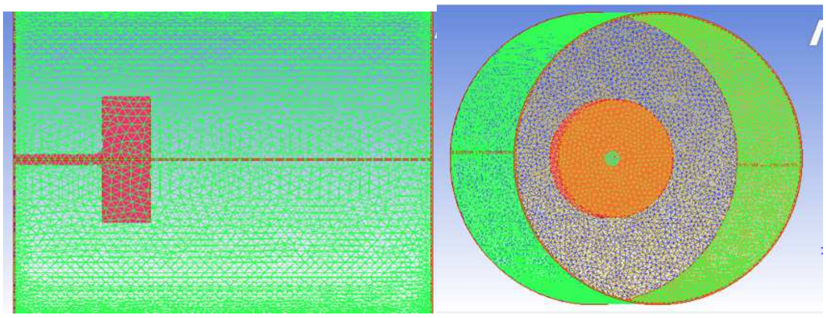


Fig.3 Static Domain Surface Mesh

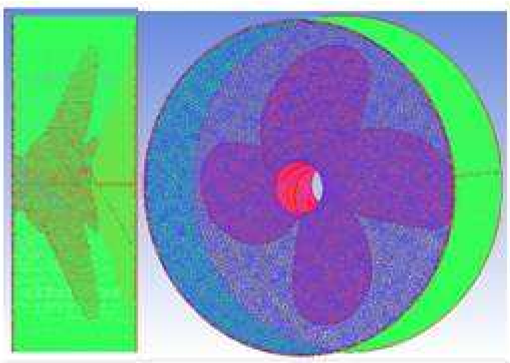


Fig.4 Rotating Domain Surface Mesh

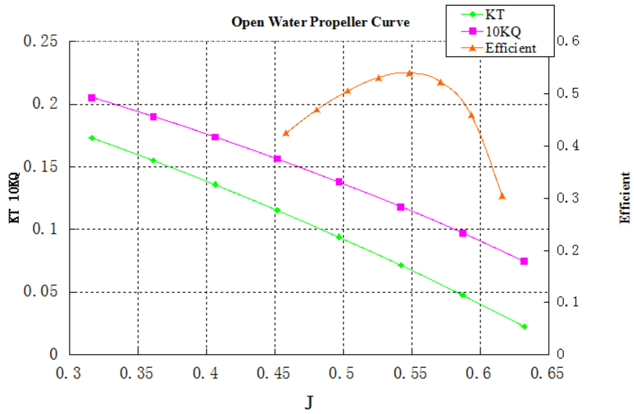


Fig.5 Open Water Performance Curve of Propeller

As can be seen from figure 5: curve obtains reasonable results and the graphic is smooth under different speed; when the revolution speed remains constant, along with the increase of the inlet velocity (the same as advance coefficient), thrust and torque decrease in correspond. For open water efficiency, along with the increase of advance coefficient, this value first increases and then decreases. We can also conclude that when the propeller is in design condition, corresponding open water efficiency is not the largest open water efficiency [5].

Analysis of Flow Field Characteristics.

Pressure Contour: In the CFD-Post, making advance coefficient of 0.452 (the corresponding speed of 10 knots) is an example to analyze pressure nephogram of propeller, as shown in figure 6 and figure7 respectively. We can get the following rules: the pressure distribution on the pressure surface continually increases from root to tip at first, thereafter decreases and reaches the minimum at the leaf tip. The pressure gradually decreases from the leading edge to the trailing edge while the suction surface near the leading edge but non leading edge shows the minimum pressure. Comparing the pressure in leaf and leaf back, we can see that under the condition of the same advance coefficient, the leaf surface pressure is bigger than leaf back surface pressure, it is the same as the forward thrust produced by revolving propeller[6]. The results are consistent with other literatures [7, 8].

The Velocity Vector Diagram. Figure 8 shows the velocity vector diagram of YZ plane and ZX on the cross section. As it can be seen, the propeller has significant pumping phenomenon for the upper reach of the stream, making the fluid outflow from outside to the inside of propeller disk; the velocity of fluid in both sides of blade increases gradually along the radial direction. The velocity vector grows from root to tip, while synthetic vector on leading edge of the leaf tip is maximum with the obvious direction[8].

Conclusion

It is a more convenient way than manual calculation for propeller design based on the software. It can eliminate the needs for steps of manual calculation, atlas query, cavitation and strength check. In addition to the boundary conditions and reasonable computational domain, it is a key to get accurate results for ensuring high grid resolution. Based on ANSYS, some important viscous flow features and phenomenon of the propeller such as the hydrodynamic performance, the pressure distribution on blade surface, flow separation are given qualitatively and quantitatively. It not only provides a theoretical basis for analyzing the noise and vibration, but also provides a strong technical support for the design of new efficient propeller.

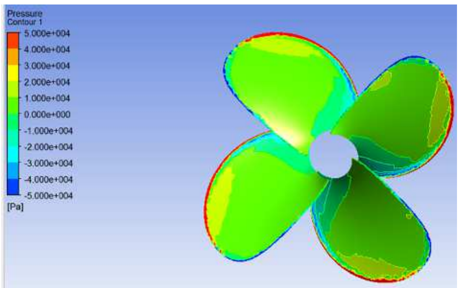


Fig.6 Pressure Surface

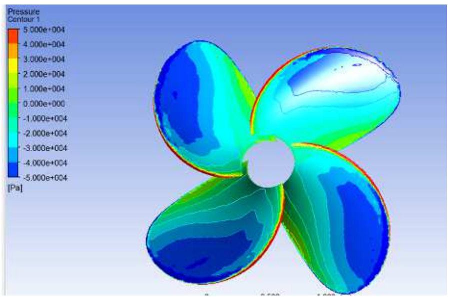
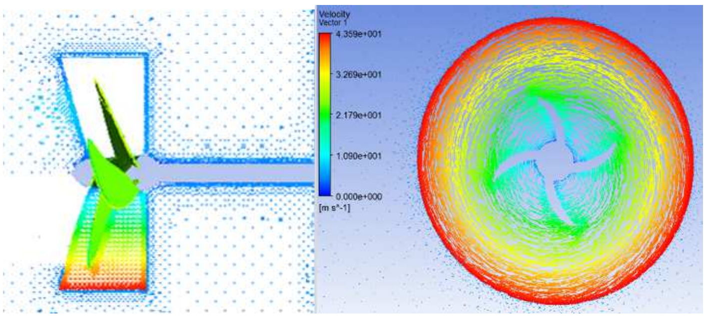


Fig.7 Suction Surface



(a)YZ plane (b)ZX plane

Fig. 8 The Velocity Vector Diagram

Acknowledgements

This work was financially supported by "the Fundamental Research Funds for the Central Universities"(3132014318).

Reference

[1] Shijun Wang: China Water Transport no.5 (2010), p. 43. (In Chinese).

[2] USA Hydrocomp company: Hydrocomp PropExpert 2004 Help [R].

[3] Guangdong Institute of Technology and Compiling group of shipbuilding system: *The Design of Marine Propeller* (China Communications Press, Beijing 1976). (In Chinese).

[4] Fujun Wang: *Computational Fluid Dynamics Analysis—The principle and Application of CFD Software* (Tsinghua University Press, Beijing 2004). (In Chinese).

[5] Jianping Ye and Guangyu Zhuang: Ship Science and Technology Vol.35 (2013), p.29. (In Chinese).

[6] Yuyue Miao and Jianglong Sun: Research on Chinese Ship Vol.6 (2011), p.63. (In Chinese).

[7] Chao Wang, Sheng Huang and Xuecan Xie: Journal of Naval University of Engineering Vol.20 (2008), p.107. (In Chinese).

[8] Fudong Gao, Cunyun Pan, Wenshan Cai and Zheng Yang: Chinese Journal of Mechanical Engineering Vol.46 (2010), p. 133. (In Chinese).