Simulation Analysis of Influence of the Blade Elasticity on Propeller Performance

Yuhang TAN, Weicai PENG and Lijiao SHEN

China Ship Development and Design Center, National Key Laboratory on Ship Vibration & Noise, Wuhan 430064, China
tanyuhang@163.com

Abstract. The blade elasticity can influence hydrodynamic and structural performance of the propeller. The elasticity of the propeller is considered in the hydrodynamic simulation based on the CFD/FEM fluid-structure interaction method. And the hydrodynamic results of the elastic are compared with the rigid. Which show that the accuracy of elastic hydrodynamic prediction improved significantly compared with the rigid. In addition, the hydrodynamic prediction considering the elasticity also shows the transient fluid flow considering the influence of blade vibration. The study indicates considering blade elasticity can significantly improve the accuracy of propeller hydrodynamic prediction, and the prediction of fluid pulsation and propeller response can be used to analyse the flow noise and structural vibration.

1. Introduction

The study of propeller performance is extremely important as the mainstream thruster in the ship. How to accurately predict the propeller in the flow field has very important engineering meaning. The influence factors of ship such as propeller hydrodynamic performance, cavitation, structure vibration and flow noise and so on are studied based on the accurate calculation of propeller hydrodynamics.

The early hydrodynamic prediction is mainly based on the theoretical method of lifting line, lifting surface and panel method. And then CFD technology is used to predict the hydrodynamic performance. However, the calculation model often regards the propeller as a rigid structure without considering the deformation. With the improvement of prediction requirements and the development of composite propeller research, the influence of propeller elasticity on hydrodynamic performance and structural performance has to be taken into account.

Considering the propeller elasticity, the fluid-solid coupling effect should be analyzed in hydrodynamic prediction. In 1996, Lin et al. [1] considered the effect of fluid-solid coupling and firstly used VLM/FEM fluid-solid coupling method to analyze the propeller hydrodynamic performance. Then Young et al. [2] analyzed the composite propeller based on BEM/FEM fluid-solid coupling method, considering the influence of fluid-solid coupling and non-uniform wake. With the development of CFD hydrodynamic prediction, HE et al. [3] analyzed the hydroelastic behavior of composite propellers in non-uniform flow based on CFD/FEM fluid-solid coupling analysis method. In recent years, the CFD/FEM fluid-solid coupling method has become the mainstream to analyze fluid-solid coupling problems.

Considering the influence of elastic effect on hydrodynamic and propeller’s performance, the CFD/FEM fluid-solid coupling method is adopted based on ANSYS-CFX modules. The influence of
propeller elasticity on hydrodynamic prediction and its own response characteristics are analyzed through the simulation.

2. Calculation model
The calculation model is established based on the CFD/FEM method. The two-way fluid-structure coupling calculation of elastic propeller is realized by computational fluid mechanics software CFX and structural calculation module ANSYS Mechanical in Workbench platform. The fluid-structure coupling calculation model is composed of the elastic propeller model and the external flow field model. The two-way fluid-structure coupling calculation method is used to realize the transfer of fluid pressure and structural deformation data in the coupling process.

2.1. Propeller model
A marine MAU four-blade propeller is used as the model. Design parameters of the propeller are given in Table 1. 3-D model of the propeller is shown in Figure 1(a). The propeller model is meshed by mesh module of the ANSYS Mechanical. And make the blade surface mesh denser. The propeller mesh is shown in Figure 1(b). The material of the propeller is copper alloy. The density is 8300kg/m$^3$, Yang's modulus is 110GPa, and the Poisson's ratio is 0.34.

| Table 1. Design parameters of the propeller |
|---------------------------------------------------|
| **Propeller Type** | **MAU** |
| Diameter | 4.78m |
| Number of Blades | 4 |
| Advanced Speed | 15.48kn |
| Efficiency | 0.545 |
| Pitch/Diameter Ratio | 0.682 |
| $A_e/A_0$ | 0.544 |
| Hub/Diameter Ratio | 0.18 |
| Direction of Rotation | Right |

![3-D model](image1)

(a) 3-D model

![3-D mesh](image2)

(b) 3-D mesh

**Figure 1.** 3-D model and mesh of the propeller

2.2. Flow field model
The whole flow field is divided into two cylindrical computing fields, which are a inner rotating domain surround by a outer static one. The diameter of the outer is 5D (D is propeller diameter) and long 7.5D. The upstream from the inlet to the propeller is 2.5D, and the downstream from the propeller to the outlet is 5D. The diameter of inner rotating domain is 1.2D, which is 0.63D (about 3m) long.
Mesh generation adopts the form of combination of structured grids and unstructured grids by using ICEM software, which is shown in Figure 2. And fluid-solid coupling interface of the flow domain adopts boundary layer mesh generation method.

![3-D Mesh of the fluid model](image)

Figure 2. 3-D Mesh of the fluid model

In the flow field, the Z axis is the flow direction, the flow field inlet is speed entrance and the outlet is pressure exit where pressure is zero. The outside boundary of outer domain is set as Free Slip Wall and the rotate speed of inner domain is 155r/min. Those two domains are connected by using General Connection Interface model.

The simulation of flow field model is based on CFD method, which uses incompressible unsteady RANS equation as the governing equation of the flow field. The turbulence model adopts SST $k-\omega$ turbulence model, which has good performance on wall bounded boundary layer flows [4].

2.3. Fluid-solid coupling model

The coupling simulation is solved by using the separation method, based on the CFX-ANSYS coupling module in ANSYS Workbench. CFX module is used for flow field simulation, and Transient Structural module for propeller’s transient calculation. Two modules transmit the data on the Fluid-Structure Interface (FSI) to achieve the two-way unsteady fluid-structure coupling calculation.

In the constraint of the structure part, the head face of propeller shaft is set as the fixed constraint, and the blade as the FSI. In the flow field, the blade contact surface in the rotating domain is set as the coupling surface for data transmission, and the shaft as the No Slip Wall. In the time step setting of flow field calculation, the time step is set to 0.002s, the total time of coupling calculation is 2s, and the time setting of structure transient calculation is the same as the flow field.

In the coupling simulation, Transient Rotor Stator is adopted for the interface between the rotating field and the outer field. The advance coefficient $J$ is set at the condition of 0.3, which means the inflow velocity is set at 3.7m/s. The hydrodynamic results of the steady calculation under this condition are taken as the initial value of the coupled calculation of the flow field. The Second Order Implicit Euler method is used to solve the transient problem. The solving process monitors propeller’s thrust and torque.

3. Hydrodynamic validation

First, CFX simulation software is used to verify the reliability of steady hydrodynamic simulation model of rigid. The turbulence model and the boundary conditions are the same as the coupling calculation Settings. The Frozen Rotor is selected for the interface between the rotating field and the outer field, and the propeller boundary surface in the rotating field is set as No Slip Wall.
By changing the speed of the inlet, steady hydrodynamic results of the propeller advance coefficient from 0.1 to 0.7 are calculated in CFX, including the thrust coefficient $K_t$, the torque coefficient $K_q$ and the open-water efficiency $\eta$.

The simulation results are verified by using the experimental regression formula of open-water performance of AU series propellers in Shanghai Jiao Tong University [5]. The thrust coefficient and torque coefficient regression formulas are as follows:

$$K_T = \sum_{i=0}^{n_i} \sum_{j=0}^{n_j} \sum_{k=0}^{n_k} A_{ijk} (P/D)^i (J)^j (A_E/A_p)^k$$

$10K_Q = \sum_{i=0}^{n_i} \sum_{j=0}^{n_j} \sum_{k=0}^{n_k} B_{ijk} (P/D)^i (J)^j (A_E/A_p)^k$

where $A_{ijk}$ and $B_{ijk}$ are the regression coefficients of thrust coefficient and torque coefficient respectively.

Through the experimental regression formula, curves of the thrust coefficient, torque coefficient and open water efficiency of the propeller are calculated. The open-water performance comparison of CFX results and experimental curves are shown in Figure 3. It is shown that the results of the CFX calculation are slightly less than the experimental results, but the trend is consistent. The error of the thrust coefficient is within 5%, and the error of the torque coefficient is within 10%. Which indicates that the calculation results of the CFX are reliable and can be used for the hydrodynamic simulation of the propeller.

![Figure 3. Open-water performance comparison between CFX and experiment](image)

4. Simulation analysis

4.1. Coupling hydrodynamic analysis

After the coupling calculation is completed, the monitoring data results are shown in Figure 4. The coupling calculation reaches stability after 0.2 s. The absolute average values of thrust and torque after stabilization are $7.275 \times 10^5$ N and $4.110 \times 10^5$ N·m respectively. Corresponding to the test values, the comparison results of thrust coefficient $K_t$ and torque coefficient $K_q$ of steady and coupling calculations are shown in Table 2. The coupling calculation results are 7.19% and 14.74% higher than the steady hydrodynamic results respectively, and 2.81% and 4.09% higher than the errors respectively. Compared with the rigid hydrodynamic calculation, the thrust of the coupling results decreases from 4.09% to 2.81% relative to experiment results, and the torque from 9.28% to 4.09%. Which indicate that the coupling hydrodynamic prediction is more accurate than the steady hydrodynamic prediction, and the prediction accuracy is obviously improved.
Table 2. Comparison of the open-water performance of simulations and test on $J=0.3$

|        | $K_t$   | Error (%) | $10^*K_g$ | Error (%) |
|--------|---------|-----------|-----------|-----------|
| Elasticity | 0.2088  | 2.81      | 0.2468    | 4.09      |
| Rigidity | 0.1948  | -4.09     | 0.2151    | -9.28     |
| experiment | 0.2031  |           | 0.2371    |           |

Figure 4. Monitor data of the force, and torque

The contour of elastic blade’s pressure at 2s is shown in Figure 5, and the rigid hydrodynamics in Figure 6 for comparison. Compared with the rigid hydrodynamics, the pressure distribution of the coupling simulation remains unchanged. The local pressure on the guide of the blade thrust surface is the largest. And the pressure near the tip of the suction surface is the smallest, which results in negative pressure. The extreme pressure is shown in Table 3. The maximum pressure of the elastic blade decreases by 3.5% relative to the rigid blade, but the absolute value of the minimum negative pressure increases by 3.1%.

In addition, the transient results of coupling contain the flow field pulsating information, which can be used to analyse flow noise and stern vibration induced by flow and propeller vibration.

Figure 5. Distributions of the elastic blade’s pressure on 2s
Table 3. Comparison of the blade’s extreme pressure

|        | MAX (Pa)        | MIN (Pa)       |
|--------|-----------------|----------------|
| Elasticity | 2.2773×10⁵     | -2.2220×10⁵   |
| Rigidity | 2.3393×10⁵     | -2.1544×10⁵   |

4.2. Propeller response analyse

Selecting the results of structural calculation at 2s, the distributions of Von Mises equivalent stress on blade surface are shown in Figure 7. It can be seen from the stress distributions that the maximum stress occurs in the middle of the blade root and gradually decreases along the radial direction of the blade surface. But the distributions of the stress on thrust and suction of the blade surface are different. At the same radius, the stress distributions of the thrust and the suction are opposite.

As the blade tip deformed the most, the displacement-time curve at the blade tip is shown in Figure 8. The average value of the tip displacement is 18.8mm, 0.4% of the diameter, and the amplitude is about 0.1mm, 5.3% of the maximum deformation. Which can be seen that considering the propeller elasticity, in addition to improving the accuracy of hydrodynamic prediction, the predicted structural vibration response cannot be ignored.
5. Conclusion
In this paper, we consider the elasticity of the propeller to simulate the hydrodynamic performance and response characteristics of the MAU four-blade copper alloy propeller by using CFD/FEM fluid-solid coupling method based on CFX and Transient Mechanical in Workbench platform. The main conclusions are as follows:

1) The two-way fluid-solid coupling hydrodynamic results with considering structural elasticity are closer to the test data compared with the rigid hydrodynamic results, which means the prediction accuracy increases obviously.

2) After considering the propeller elasticity, the pressure distribution of the coupling simulation remains unchanged. But the maximum predicted pressure of the blade decreases, and the absolute value of minimum negative pressure increases.

3) Considering the structural elasticity, the transient hydrodynamic simulation results can show the flow field pulsation information which considers the influence of propeller response compared with rigid transient hydrodynamics. Which can predict structural response and flow noise more accurately.

The study also provides a research approach for the prediction of propeller flow noise and structural response.

Acknowledgments
The authors are grateful for supports from the National Natural Science Foundation of China (No.51609226)

References
[1] Lin H J and Lin J J 1996 Nonlinear hydroelastic behavior of propellers using a finite-element method and lifting surface theory J. Mar. Sci. Technol. 1(2) 114–24
[2] Yong Y L 2007 Hydroelastic behavior of flexible composite propellers in wake inflow 16th Int. Conf. on Composite Materials (Kyoto)
[3] He X D, Hong Y and Wang R G 2012 Hydroelastic optimisation of a composite marine propeller in a non-uniform wake Ocean Eng. 39 14-23
[4] Li D Q 2006 Validation of RANS predictions of open water performance of a highly skewed propeller with experiments J. Hydrodyn. Ser. B 18(3) 520-28
[5] China State Shipbuilding Corporation 1998 Practical Manual for Ship Design (General Catalogue) (Beijing: National Defence Industrial Press) pp 316-17