Fluid-structure interaction Simulation for Hydro-elastic Performance of marine Propeller at Full-Scale

Zhipeng Qian¹, a, Nan Ji², b*, Guang Yang³, c, Linzhao Shu⁴, d

¹School of Shipping and Naval Architecture, Chongqing Jiao Tong University, Chongqing 400074, China
²School of Shipping and Naval Architecture, Chongqing Jiao Tong University, Chongqing 400074, China
³School of Shipping and Naval Architecture, Chongqing Jiao Tong University, Chongqing 400074, China
⁴School of Shipping and Naval Architecture, Chongqing Jiao Tong University, Chongqing 400074, China

a872268573@qq.com
b*Corresponding author: 2554617008@qq.com
c815293283@qq.com
d478903658@qq.com

Abstract—In this study, CFD-FEM bi-directional fluid structure interaction (FSI) methodology using STAR-CCM+ is adapted for the hydro-elastic interaction simulation of the propeller at full-scale. To validate the FSI simulation reliabilities, the open-water curve of the rigid and NAB propellers VP1304 at model scale are computed. Then, the propeller KP505 is employed for the open-water test to study scale effect and hydro-elastic performance. The change of thrust coefficient, torque coefficient, efficiency, stress distribution, deformation are presented. From the results, the hydrodynamic difference due to scale effect and hydro-elastic performance are observed. Analysis of structural response of NAB propeller shows that scale effect plays a role.

1. INTRODUCTION

The hydrodynamic characteristics of propellers are usually studied at model scale and model open-water tests are regarded as an effective method to predict the performance of marine propellers at full-scale. For the limitations of the test condition, the rotation speed of the model propeller cannot be large enough, which inevitably lead to the Reynolds number of the model propeller far smaller than of the full-scale propeller. With the development of computer performance, full-scale CFD simulation receive more attention. Jung-Kyu Choi [1] computed model and full-scale propeller under the open-water condition using different wall treatments. Results shows that the wall function calculation as an efficient calculation is recommended for the full-scale calculation due to the large Reynolds number. Sun [2] performed DES calculations of the model and full-scale propellers VP1304 to analyze scale effect on the wake dynamics. All of this suggest that more efforts to directly estimate the performance...
of a full-scale propeller have been made, rather than to indirectly estimate by a physical similarity, such as model tests.

On the other hand, under the effect of fluid pressure, the structure of blade undergoes deformation progress, which cause the change of longitudinal and skew angle. Propeller designers couldn’t handle this situation by traditional theory and empirical methods because they all are based on a rigid blade. FSI methodology has been used to study the hydro-elastic performance. Young [3] studied hydro-elastic performance of a composite propeller by a BEM-FEM method and model test. P. J. Maljaars et al. [4] carry out the non-uniform flow hydro-elastic analysis of the flexible propellers in cavitating and non-cavitating conditions, based on BEM-FEM method. Conventional boundary element method has a reasonable accuracy and computational efficiency, however BEM ignore effect of viscous flow, which has a limitation of prediction of blade boundary layer and wake region. Moreover, the effect of scale effect on propeller couldn’t be calculated quantitatively. Cao [5] computed different materials of propeller VP1304 under the open-water condition using CFD-FEM method.

In this paper, Bi-directional CFD-FEM fluid structure interaction method is used for hydro-elastic simulation. STAR-CCM+ is used as a solver for FSI analysis. The results of simulation of model propeller VP1304 under the open-water test condition are compared with the experiment and FSI simulation result of Cao [5]. The rigid and NAB propeller KP505 at model and full scale is analyzed using Bi-directional CFD-FEM method.

2. NUMERICAL METHODS

In this paper, CFD simulation of rigid ones and FSI simulation of NAB ones under open-water condition using STAR-CCM+ solver. Sliding mesh and mesh morphing technique is used to evaluate the hydrodynamic performance and structural deformation respectively.

2.1 Fluid Analysis Method

The fluid is assumed to be an incompressible fluid, which follows the generally accepted assumption about fluid around a marine propeller. Therefore, the governing equations are solved by the Navier-Stokes equations (1) and (2) of unsteady turbulent as follows:

\[ \frac{\partial U_i}{\partial x_i} = 0 \]  

\[ \frac{\partial P}{\partial x_i} + \rho \frac{\partial U_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu \frac{\partial U_i}{\partial x_i} - \rho U_i U_i \right) \]  

where \( U_i = (U, V, W) \) is the velocity component in the \( X_i = (x, y, z) \) direction; \( P \) is static pressure; \( \mu \) is the dynamic viscosity of fluid; \( \rho \) is fluid density; \( \rho U_i U_i \) expresses the Reynolds stress, and requires a turbulence model to close. In this study, K-\( \omega \) SST model is applied for turbulence model and commercial CFD program.

The calculation domain is divided into two parts—the stationary domain and the rotating domain, and the calculation domain of propeller VP1304 is shown in Figure 1. The Stationary domain should be large enough, which has the length of 10D, with 7D cylindrical diameter, to ensure that the out boundary will not influence the analysis of the near-propeller fluid flow. The rotating domain has the length of 1.2D, with 1.6D cylindrical diameter. The internal interface is established to transmit the data. The velocity of inlet boundary is set at advance speed \( V_a \). On the outlet boundary, the static pressure outlet is imposed, with 0 Pa reference pressure. Fluid solver time step is set at 1° of propeller rotation per step.
2.2 Solid Analysis Method

The equations (3) and (4) of motion for the structural deformation of the propeller blade is as follows:

\[ M_s \ddot{d} + C_s \dot{d} + K_s d = F_{sy} \]  

(3)

\[ F_{sy} = F_{hp} + F_{cor} + F_{cort} + F_{fs} = F_s + F_{fs} \]  

(4)

where \( M_s \) is the mass matrix; \( C_s \) is the damping matrix; \( K_s \) is the stiffness matrix, representing the material mass, damping and stiffness of propeller blade; \( \ddot{d}, \dot{d}, d \) are acceleration, velocity, and displacement of the structure element; \( F_{sy} \) represents the all loads acting on the structure. The structural analysis of structural response is conduct with STAR-CCM+ solver, which also has FEA solver module.

In this study, the blades of propeller VP1304 and propeller KP505 material is set at Ni-Al Bronze (NAB), which properties are \( \rho = 7400 \text{ kg/m}^3, E = 124 \text{ GPa}, \nu = 0.33 \). Solid solver time step is also set at 1° of propeller rotation per step.

2.3 Coupling Method

In this study, FSI analysis is performed by bi-directional coupling method, and the coupling time step is the same as the time step of fluid and solid solver. The analysis progress as follows [6]:

1) CFD analysis to calculate pressure distributions at the blade surface.
2) FEM analysis to calculate deformation is performed with pressure as load condition.
3) Morphing CFD mesh based on deformation data.
4) Return to step 1) and perform CFD calculation for deformed propeller.

Figure 1. Calculation domain for propeller VP1304.

Figure 2. Bi-directional coupling method.
3. Verification and Validation
To verify the validation of mesh convergence and the FSI method, propeller VP1304 open-water curve has been computed, which compared with the experimental reports on SMP’11 workshop. The advance coefficient \( J \), advance speed \( V_A \), thrust coefficient \( K_T \), torque coefficient \( K_Q \) and efficiency \( \eta \) can be calculated through equations (5) and (6) as follows:

\[
K_T = \frac{T}{\rho n^2 D^4}, \quad K_Q = \frac{Q}{\rho n^2 D^5}
\]

(5)

\[
J = \frac{V_A}{n D}, \quad \eta = \frac{J}{2\pi} \frac{K_T}{K_Q}
\]

(6)

While the calculation domain for propeller VP1304 has been explained in section II, polyhedral mesh and trimmed cell mesh were adopted for rotating domain and stationary domain respectively. Different grid refinement was used for the propeller leading edge, trailing edge, and wake region. For the solid analysis, tetrahedral mesh was generated.

Propeller rotational speed \( n \) is 15 rps, with the scale ratio \( \lambda \)=12. Eight cases with eight different advance coefficients were run, and Figure 3 shows the open water curves, both the experimental results and the computed at model scale using CFD and CFD-FEM(FSI).

![Figure 3. Open-water performance of propeller VP1304.](image)

Comparing the CFD results with the experimental results, the open-water characteristics of rigid propeller shows consistent tendency and maximum errors of \( K_T \) and \( K_Q \) occur at \( J=0.4006 \) and \( J=1.3308 \) respectively. Also, comparing the FSI results with the CFD and experimental results, the open-water characteristics of traditional NAB propeller are slightly smaller than rigid propeller, and more consistent with experiment results at \( J \) variation.

4. Analysis and Discussion
In this section, scale effect and hydro-elastic performance influence on propeller hydrodynamic characteristics and structural response will be discussed. To minimize the grid size effect, the grid topologies for the model and full-scale propellers were the same, except for the absolute near-wall spacing. As explain by Jung-Kyu Choi [1], full-scale calculation using the high \( \gamma^+ \) wall treatment nearly make no difference due to large Reynolds number, so \( \gamma^+ \) is close to 80 for full-scale propeller calculation. Also, the calculation domain, boundary conditions, material selection and other settings are the same, which have been mentioned (section II, III).

4.1 Research Object
Propeller KP505 was employed for the open-water simulation to study scale effect and hydro-elastic performance, as a representative large container vessel propeller. Model-scale (\( \lambda \)=31.6) and full-scale...
($\lambda=1$) propellers were simulated, with corresponding rotational speed $n=14$ rps and $n=2.49$ rps respectively.

4.2 Hydrodynamic Characteristics Analysis
Figures 4 and 5 show the open-water performance of rigid and NAB propeller under the model and full-scale condition. Figure 4 compares CFD simulation results with experiment and ITTC scale correction results respectively. It can be observed that $K_T$ at full-scale increases, while $K_Q$ decreases, compared with $K_T$ and $K_Q$ at model-scale. The reason can be explained in airfoil lift test of Fiddes[7]: the increase of Reynolds number will lead to the increase of lift force and decrease of drag force on blade section.

![Figure 4. Open-water performance of rigid propeller($\lambda=31.6$, 1).](image)

(a) $\lambda=31.6$
Figure 5. Open-water performance of NAB propeller and rigid propeller ($\lambda = 31.6$, 1).

The relative differences between rigid and NAB propeller values of thrust and torque coefficients are calculated as follows:

$$\Delta K_{T,\lambda} = \frac{K_{T,\text{deformed}} - K_{T,\text{undeformed}}}{K_{T,\text{undeformed}}} \times 100\% \quad (7)$$

$$\Delta K_{Q,\lambda} = \frac{K_{Q,\text{deformed}} - K_{Q,\text{undeformed}}}{K_{Q,\text{undeformed}}} \times 100\% \quad (8)$$

Figure 5 shows that the variation trend of $\Delta K_{T,\lambda}$ and $\Delta K_{Q,\lambda}$ of rigid and NAB propellers under model and full-scale condition is not consistent, with the increase of the advance coefficient $J$. $\Delta K_{T,1}$ and $\Delta K_{Q,1}$ increase with the increase of the advance coefficient $J$, while $\Delta K_{T,31.6}$ and $\Delta K_{Q,31.6}$ increase with the increase of the advance coefficient $J$.

(a) Friction components characteristics
The thrust coefficient and torque coefficient respectively include the pressure components $K_T^P$ and $K_Q^P$ and the friction components $K_T^F$ and $K_Q^F$. The distribution of each component is shown in Figure 6. It is observed that scale effect have greater effect on friction component of torque coefficient at $J$ variation, though friction component is relatively small. Comparing NAB propeller with rigid propeller at full-scale, pressure components decreases, especially under heavy load condition, while friction component increase slightly.

4.3 Structural Response Analysis

Figure 7 shows deformation of the estimated by FSI analysis under model and full-scale conditions($J=0.8$). The deformation distribution of model and full-scale propeller is consistent along the radial direction, which increases from the root of the blade to the tip of the blade. However, The chordal deformation distribution of the two sizes is different, and the deformation at the leading edge of the full-scale propeller tends to increase.

To analyze the deformation distribution of model and full-scale propeller, dimensionless relative deformation $\Delta X_r$ is defined as $\Delta X_r=\Delta X/D\times100\%$, where D represents diameter. The relative deformation distribution at the centerline of blade pressure side($J=0.8$) is shown in Table 1. For the full-scale propeller, relative deformation is much high than that of model-scale.
TABLE I. RELATIVE DEFORMATION

| r/R   | 0.5     | 0.7     | 0.9     | 0.995   |
|-------|---------|---------|---------|---------|
| $\lambda=31.6$ | 0.000536 | 0.001772 | 0.003988 | 0.00696 |
| $\lambda=1$    | 0.0168  | 0.0506  | 0.1457  | 0.2147  |

Figure 8 shows the change in pitch ratio of full-scale propeller under deformed and undeformed condition ($J=0.8$). It is observed that $P/D$ of NAB propeller near the blade tip decreases, which lead to low propeller load, explaining the change in Figures 7 and 8.

Figure 8. The radial distribution of pitch ratio ($\lambda=1$).

Figure 9 shows stress distribution under model and full-scale conditions ($J=0.8$). The stress distribution of the same material is basically the same. The low stress areas locate in the tip, middle of blade pressure side, near the leading and trailing edge of blade suction side, while high stress areas locate in the trailing edge and root.

(a) $\lambda=31.6$
5. CONCLUSION

Full-scale CFD simulation avoids the scale effect on the traditional model test and FSI simulation have accurate ability to predict hydro-elastic performance by coupling fluid and structure. In this study, to estimate the scale effect and hydro-elastic performance influence on propeller hydrodynamic characteristics and structural response, CFD-FEM methodology using STAR-CCM+ is adapted. Two kinds of marine propeller, VP1304 and KP505 are employed for open-water calculation. Analysis shows that scale effect and hydro-elastic performance affect hydrodynamic characteristics and structural response interactively, such as: (1) the thrust coefficient and torque coefficient of the full-scale NAB propeller decrease more than that of the model-scale, compared with rigid propeller; (2) moreover, scale effect and hydro-elastic performance have different influence on the friction coefficient and pressure coefficient respectively; (3) The structural response of NAB propeller have some difference in model and full-scale. the relative deformation of full-scale propeller is much high than of model-scale.

In a future work, we will try to apply the FSI methodology to the propeller KP505 behind the container vessel KCS, and conduct self-propulsion simulation.

ACKNOWLEDGMENT

Thanks are given to the research projects for their infrastructural and financial support. Sincere appreciates are delivered to scholars of the references for their outstanding contributions, and to the editors and reviewers for their helpful suggestions.

This work is supported by: (1) the National Key R & D Program of China (Grant No. 2018YFB1600400), (2) Basic and Frontier Research Project of Chongqing Science and Technology Commission (Grant No. cstc2015jcyjA70009).

REFERENCES

[1] J. K. Choi, H. T. Kim, “An investigation on the effect of the wall treatments in RANS simulations of model and full-scale marine propeller flows,” Int. J. Nav. Archit. Ocean Eng, vol. 12, 2020, pp. 967-987.
[2] S. Sun, C. Wang, C. Y. Guo, Y. Zhang, C. Sun, P. F. Liu, “Numerical study of scale effect on the wake dynamics of a propeller,” Ocean Eng, vol. 196, 2020:106810.
[3] Young, Y. L., “Fluid-structure interaction analysis of flexible composite marine propellers,” J. Fluid Struct., vol. 24, 2008, pp. 799-818.
[4] P.J. Maljaars, N. Grasso, J.H. den Besten, M.L. Kaminski, “BEM–FEM coupling for the analysis of flexible propellers in non-uniform flows and validation with full-scale measurements,” J. Fluid Struct., vol. 95, 2020:102946.
[5] F. Cao, W. Li, Y. X. You, “Numerical simulation of fluid-structure interaction characteristics for a pitch propeller with five blades in unsteady states,” Chinese Journal of Hydrodynamics, vol. 28, issue 5, 2013, pp. 575-584.

Figure 9. Predicted stress contours ($\lambda=31.6$, 1).
[6] N. Suyama, S. Fujita, K. Kimura, K. Shiraishi, Y. Sawada, C. Kawakita, “Study on method to predict performance of composite propeller using FSI analysis,” Sixth International Symposium on Marine Propulsors(smp’19), Rome, Italy, 2019.

[7] Fiddes, S., Kirby, D., Woodward, D., Peckham, D., “Investigations into the effects of scale and compressibility on lift and drag in the RAE 5m pressurised low-speed wind tunnel,” Aeronaut. J, vol. 89, 1985, pp. 93-108.