The Analysis of Propulsion Performance of Mainstream Ship’s Propeller in The Yangtze River

Niu Niu1*, Shuli Wu2, Yizhe Xu3, Yuewen Su4, Peixuan Xue5 and Youguang Wu6

1 School of Energy and Power Engineering, Wuhan University of Technology, Wuhan, Hubei, 430061, China
2 School of Energy and Power Engineering, Wuhan University of Technology, Wuhan, Hubei, 430061, China
3 School of Energy and Power Engineering, Wuhan University of Technology, Wuhan, Hubei, 430061, China
4 School of management, Wuhan University of Technology, Wuhan, Hubei, 430070, China
5 School of Energy and Power Engineering, Wuhan University of Technology, Wuhan, Hubei, 430061, China
6 Science and Technology on Reactor System Design Technology Laboratory, Nuclear Power Institute of China, Chengdu, 610213, China

*Corresponding author’s e-mail: 1336169863@qq.com

Abstract. In order to analyse the propulsion performance of the mainstream ship’s propeller in The Yangtze River route, this study’s propeller is designed according to a certain prototype in The Yangtze River which is also taken as the research object in this study. Its propulsion performance in uniform flow and non-uniform flow are respectively analysed by CFD method; and the cavitation effect on propeller performance is also studied. Then the propeller thrust and torque under different speed are obtained; open water efficiency and pressure distribution on blade are calculated as well. The results show that the non-uniform flow has changed the flow field near the propeller, causing the propelling performance in the non-uniform flow is higher than the calculated value in the uniform flow at the same speed of advance. The thrust and torque of a single blade will fluctuate greatly at different circumferential positions, while for overall propeller, non-uniform flow has little influence on thrust coefficient and torque coefficient. The results also illustrate that SST model is more suitable for analysing the propulsion performance under cavitation effect. And the difference between these two cavitation models is subtle. It is found that the smaller the cavitation number is, the stronger the cavitation is and the greater the influence on propulsion performance is. Cavitation happens when advance ratio is relatively higher.

1. Introduction
With the advent of the Golden Waterway in the Yangtze River which is featured for the 100 million tons’ through capacity, both the number and tonnage of vessels are dramatically increasing. Propeller, as a mechanical equipment to power vessels, transforms the power generated by diesel engine into thrust that helps to control ship motion. In this case, the propulsion performance of a propeller directly affects
the ship's speed and operating cost. Therefore, it is of great practical significance to analyze the propulsion performance of ship propellers on the Yangtze River route considering the construction of 100 million tons’ Golden Waterway.

Gao and Pan [1] utilized three Turbulence Models based on CFD theory in uniform flow to predict the propulsion performance of propellers. Then propeller open water parameters were calculated. After a comparison with the test values, it can be found that the results obtained from the Reynolds Stress Model were quite reliable. M.Liefvendahl [2] adopted LES and dynamic grid technology to study how the propeller thrust, torque and pressure distribution on a single blade fluctuated under the influence of submarine wake. However, after referring to literature review, it can be assumed that there was still a large error between test values and the simulation results calculated by integrating a vessel and its propeller when considering non-uniform flow. Feng[3] took E779A and PPTC as subjects and specifically analysed how propeller thrust and torque were affected by the cavitation with different cavitation numbers in an uniform flow. It turned out that within a certain range of speed of advance, the propulsion performance could be affected by the change of cavitation numbers. What’s more, Wang[4] depicted a collapse performance curve for the propeller thrust and then compared it with test values after calculating propeller’s forces under various cavitation numbers.

This paper is arranged as follows: at first, the specific type of propeller is selected and model is built base on this prototype. Then performance of the chosen propeller is analyzed in a non-uniform flow. And finally, the performance of the propeller is studied considering the cavitation. As has shown in the Figure 2-1.

2. Theory of Computation

2.1. Governing Equations
Mass conservation equation:
\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0 \]  

Momentum conservation equation:
\[ \frac{\partial \rho u_j}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_j}{\partial x_j} - \rho u_i u_j \right) + S_j \]  

Where p stands for static pressure; \( \rho \) stands for liquid density; \( u_i, u_j \) are velocity vector; \( \mu \) is Turbulent viscosity; \( -\rho u_i u_j \) is the Reynolds Stress.

3. The Propeller Design

3.1. Hull Resistance Calculation
The most common container ship is taken as model and the size of which is 49.9m×9.5m×3.5m. Currently, CAD drawings which include two-dimensional hull lines (mainly refers to transverse, longitudinal and waterline) are imported into the CATIA software according to their respective positions and as a result, three-dimensional model of the ship can be obtained. As has shown in Figure 1, computational domain should be established for only half of the hull. The distance between the entrance of the computational domain and the bow is set as the length over all (LOA); the distance between the exit of the computational domain and the stern is set as 3.5 times of the LOA; the distance between the top or bottom and the waterline equals to LOA. Seen from the ship beam, the distance between the boundary and mid-longitudinal section is twice the number of LOA. On grid partitioning, a hybrid grid method is adopted due to the fact that the hull surface is irregular. In this case, grids should be denser and unstructured grids are utilized in places like bow and stern where the shape changes greatly. In
comparison, structural grids can be used where the flow field changes slightly [5]. Specific settings are shown as Figure 2.

![Figure 1 3D model of the ship](image1)  ![Figure 2 Grids of the Computational Domain](image2)

### 3.2. Establishment of Geometric Model of the Selected Propeller

Considering features of those most common ships in the Yangtze River as well as some practical experience, B-Series Propeller is used here. Then an effective Horsepower Curve has been depicted and the propeller’s relevant parameters are also determined after considering different ship resistance under various speed of advance. After that, with the help of parameters like 2-D coordinates of blade sections at different radii, distribution of pitches at each radius, sectional chord length, longitudinal inclination angle (LIA) and maximum thickness, 3-D coordinates of propeller’s blades at different radius can be calculated in the Global Coordinate System. The calculation is as follows:

\[
\begin{bmatrix}
X \\
Y \\
Z
\end{bmatrix} = \begin{bmatrix}
\frac{r \sin \left( X_i - L_i \right) \cos \phi - Y_i \sin \phi}{r} \\
\frac{\left( X_i - L_i \right) \sin \phi + Y_i \cos \phi \tan \theta}{r} \\
\frac{r \cos \left( X_i - L_i \right) \cos \phi - Y_i \sin \phi}{r}
\end{bmatrix}
\]  

(3)

Where X, Y and Z are 3-D coordinates of blades; r stands for radius of blade section; X1 stands for the abscissa of the 2-D coordinates of blade sections and Y1 is the ordinate. \( \phi \) is the angle of pitch, \( \theta \) is trim angle, \( L_i \) is the distance from leading edge at radius r to the reference center line. Those basic parameters of propellor can be calculated as Table 1. The eventual 3-D model is shown as Figure 3.

#### Table 1. Basic Structural Parameters of The Propeller

| Diameter (D/m) | Trim Angle (\( \theta \)/°) | Pitch Ratio (P/D) | Area Ratio (\( A_e/A_o \)) | Number of Blades (Z) |
|----------------|----------------------------|-------------------|--------------------------|---------------------|
| 1.4            | 15                         | 0.752             | 0.40                     | 4                   |

![Table 1](image3)

![Figure 3 3D Model of the Propeller](image4)

### 4. Analysis of the Propulsion Performance in a Non-Uniform Flow
4.1. Establishment of Numerical Models

(1) Grid Partition on Propeller
In this step, basic principles are that grids are partitioned from lines to areas, then from areas to bodies so as to enhance the quality of results. Density of grids are increased in areas including propeller’s leading edge and trailing edge. The grids of propeller are shown as Figure 4.

![Figure 4 Grid Partition of Propeller](image)

a) Partition of Blade Tips    b) Partition of Blade Roots

(2) Grids Partition and Boundary Conditions Setting in the Numerical Model
Computational domains in uniform flow covers both rotating domain and stationary domain. And the diameter and length of a rotating domain are 1.2D and 1.0D respectively while the diameter and length of stationary domain are 5D and 10D respectively (Here, D stands for the diameter of a propeller). Structured hexahedron mesh is used here to mesh the stationary domain. Additionally, unstructured tetrahedron mesh is adopted to mesh the rotating domain [6]. Finally, as shown is Figure 5, there are about 1.1 million grids in the rotating domain and 0.9 million grids in the stationary domain. A Propeller-Hull Interaction model is established in non-uniform flow. In this case, the diameter of computational domain is 5 times of LOA and the width of it is 2.5 times of the beam of ship. The height of it is 5 times of the ship’s depth. And the distance between the entrance of the computational domain and bow equals LOA while the distance between the exit of computational domain and stern is 3 times of LOA. And the distance between port side or starboard side to the boundary of the domain is twice the number of a beam [7]. Still, structured hexahedron mesh is used here to mesh the front and back parts as well as the farside part. Then, unstructured tetrahedron mesh is adopted to mesh the central part including the oar. Here, boundary layer is added to the hull [8]. As a result, there are about 1.1 million grids in the rotating domain and 3.9 million grids in the stationary domain. Besides, it should be noticed that the grid partition method used in meshing propeller and its rotating domain should be in consistent with those used in open water. The overall settings of grid partition and boundary condition in the whole computational domain are depicted in Figure 6 and Figure 7.

![Figure 5 Mesh of Computational Domains of the Propeller](image)
4.2. The Propulsion Performance Study in Non-Uniform Flow

To start with, propeller’s steady-state values which are obtained from the Reliable Turbulence model and MRF model are set as the initial values. Then Sliding Mesh Model is selected to study the propulsion performance in uniform flow. Test values come from Towing Tank in Wuhan University of Technology (the revolution speed of propeller keeps at 324r/min). The comparison between test values and simulation values are shown as Figure 8.

![Figure 6 The Boundary Conditions of Computational Domains](image)

![Figure 7 Grids Partition of Propeller Model](image)

(a) Comparison of Thrust Coefficient

(b) Comparison of Torque Coefficient
Comparison of Open Water Efficiency

Figure 8 Simulation Values and Test Values of Open Water Parameters

It can be told from the comparison results that the largest deviation is no more than 5%. Such a small deviation means the simulation is quite precise.

Since the hull would create wake in navigation, the propeller actually works in a non-uniform flow. And every blade in circumferential positions, it has various thrust coefficients. So, a single blade’s propulsion performance is evaluated when it’s in different circumferential position during a complete rotating process. The speed of the vessel is 2.289m/s while the revolution speed of propeller remains at 324r/min. What’s more, 0° is in the upward position of a circumference, with the 90° in the right hand side, 180° in the downward position, 270° in the left hand side. And the calculation values of single blade and of the whole propeller are compared in Figure 9 and Figure 10.

Figure 9 Thrust Coefficient at Different Circumferential Position

Telling from Figure 10, it can be concluded that if only a single blade is taken into consideration, wake flow may have great impacts on the torque coefficient and the thrust coefficient. Then, when a blade is at different circumferential position, its torque coefficient and thrust coefficient changes dramatically. These two figures decrease from 0°, reaching a lowest number at 270°. Then they grow gradually after that. Meanwhile, the thrust coefficient fluctuates at 0.25 while the torque one fluctuates...
around 0.3. By contrast, within the whole rotation process, figures for the whole propeller changes slightly, remaining quite stable.

Thrust coefficient, torque coefficient and open water efficiency in both uniform and non-uniform flow are compared from Figure 11 to Figure 13.

![Figure 11 Comparison of Thrust Coefficients](image)

After analysing above mentioned figures, the pattern that how thrust changes with the speed of advance in non-uniform flow is basically the same with that in uniform flow. The former thrust and the calculated value of thrust coefficient are slightly higher than that of the latter ones. If the speed of advance is low, the deviation between them is small. However, with the increase of speed of advance, the deviation is growing and finally it reaches 56.9% under a given speed of advance.

![Figure 12 Comparison of Torque Coefficient](image)

After an analysis of Figure 12, it can be found that how torque coefficient changes with the speed of advance with the interaction with hull is mainly the same as without hull. The former torque and calculated torque coefficient are higher those of the latter ones. This difference is tiny under a slow speed of advance. However, this difference rises as the speed of advance climbs up. Finally, the difference reaches 32.7% under a give speed.

![Figure 13 Comparison of Open Water Efficiency](image)

After a thorough analysis, the tendency that how open water efficiency changes with the speed of advance is almost the same no matter the hull is taken into account or not. Curves can be divided into two phases. Phase I: when 0.3<j<0.4, two curves almost are exactly the same; Phase II: when J>0.4, it is rather clear that with the growth of speed, the difference of these two curves is enlarging. And finally, this number could reach 10.4%.

5. Propulsion Performance Analysis under Cavitation

5.1. Analysis of Propulsion Performance affected by Cavitation by Using Different Turbulent Models
Take Schnerr & Sauer Model as an example, under the same cavitation number ($\sigma = 3$), results obtained from Standard turbulent model, RNG model and SST model are compared in the Figure 14.

![Figure 14 Comparison of Propulsion Performance under Different Turbulent Models](Image)

After an analysis, it can be concluded that within a certain range of speed of advance, thrust coefficient and torque coefficient obtained from these models are basically the same. However, $J=0.6$ is a boundary value; when $J$ is larger than 0.6, and with the increase of the speed, the calculated values by three models are getting closer to the number under no cavitation; and when $J$ is smaller than 0.6, with the decrease of speed of advance, all the calculated values go down rapidly, far less than the number of no cavitation. For thrust coefficient, the difference can be 79.7%; for torque coefficient, this number could be 74.8%. Meanwhile, with the decrease of advance ratio, calculated values start to deviate. And according to the curses, this deviation could be larger. During calculation, it can also be found that convergence rate of SST model is larger than that of its rivals of other two models, leading to a shorter computing time. All in all, within the given range of speed of advance, the results obtained from three turbulent models vary little. However, SST model is faster to finish its task. Thus, for propeller used in this paper, SST model is a better choice to analyze the propulsion performance.

5.2. Propulsion Performance Analysis under Various Cavitation Models
SST turbulent model is selected here and cavitation number is set as 3 ($\sigma = 3$). After the Schnerr & Sauer cavitation model and Zwart-Gerber-Belamri model are built, the calculation results from these models are compared with values without cavitation. And in this chapter, the mass fraction of Non-Condensing Gas is set as $1.5 \times 10^{-7}$ [9].

The water vapor volume fraction distributions depicted by two cavitation models have been shown in Figure 15. And the advance ration is 0.7, cavitation number is 0.3 ($J=0.7, \sigma = 3$). In the figures, area in dark colors means the vapor volume is larger than 0.5, and those areas are places where cavitation happens [10]. It can also be found that areas suffering from cavitation in two pictures are almost the same in both their size and location, all in leading blade. And from blade roots to tips, the cavitation is getting stronger with the color getting darker.
Having carried out model experiments about cavitation as above mentioned, it can be obtained that both cavitation models are accurate in simulating the cavitation areas and locations. And then the propulsion performances under two cavitation models with or within cavitation are compared. All the results can be found in Figure 16.

Telling from Figure 16, the curves of both thrust coefficient and torque coefficient calculated by two cavitation models are quite similar to those in Figure 4-1. Again, at about J=0.6, curves start to deviate. When the advance ratio is larger than 0.6, with the increase of advance ratio, the results from both cavitation models are getting closer to numbers without cavitation. And when the advance ratio is lower than 0.6, with the decrease of this number, the results fall down drastically, way much lower than figures without cavitation. And the largest deviation of thrust coefficient can be as many as 78.4% while that of torque coefficient can be 68.1%. Besides, the values from Schnerr & Sauer model are likely to surpass that of Zwart-Gerber-Belamri model. To sum up, at given speeds, results from these two cavitation models are basically the same and the simulation of blade’s cavitation are of few difference.
5.3. Propulsion Performance Analysis under Different Cavitation Numbers
Choose k-ωSST turbulent model and Schnerr & Sauer cavitation model, then set cavitation numbers as 3, 4 and 5. Results, shown as Figure 17, are obtained and then compare with those without cavitation.

![Figure 17 Comparison of Propulsion Performance under Different Cavitation Numbers](image)

After a thorough study into the results in Figure 4-4, when the cavitation number is 3, critical value would be $J=0.6$. If the cavitation number is 4, the critical value would be $J=0.55$; and for cavitation number $\sigma=5$, the critical value would be $J=0.5$. If the advance value is lower than the critical values, both thrust and torque coefficients would fall as the decrease of advance ration. The deviation of thrust coefficient can reach 76.1% while the number of torque is about 64.7%. And in this case, if the advance ratio is fixed, the lower the cavitation number is, the smaller the thrust and torque coefficient are. If the advance value is higher than the critical values, with the rise of advance ratio, the thrust coefficient and torque efficient go down but basically follow the pattern of numbers without cavitation. It means that the lower the cavitation number is, cavitation is and would be stronger, having more influence on propulsion. Likewise, under the same cavitation number, the smaller the advance number is, the stronger the cavitation is and the lower the thrust and torque coefficients are. This is because the entering speed remain the same while the revolution speed of propeller climbs up.

Figure 18 to Figure 20 show the cavitation distribution of blade surface when $J$ equals 0.5 and $\sigma$ equals 3, 4 and 5 respectively. It can be concluded that under a same advance ration, the cavitation changes with $\sigma$. The smaller the $\sigma$ is, the stronger the cavitation is. To be more specifically, when $\sigma$ is 3, most of the blade back is affected by cavitation and even the blade root of the blade face is affected as well. This pattern matches with what has been mentioned above – the lower the cavitation number is, the stronger the cavitation, leading to a rapid decrease in propulsion performance.
6. Conclusion

Through an analyzing the propulsion performance of mainstream propeller in the Yangtze River, the following conclusions can be obtained:

(1) In a non-uniform flow, distribution of pressure on propeller’s blade surface would be changed, causing the thrust of single blade fluctuates when it’s at different circumferential position. However, this effect is not significant for the whole propeller. And it can be told that propeller’s thrust and torque in a non-uniform flow are larger than that in a uniform flow.

(2) SST model is a better choice in dealing with analyzing propulsion performance under cavitation.

(3) two cavitation models mentioned in the study have little influence on study the propulsion performance

(4) with the decrease of cavitation number, the cavitation is and would get stronger. If the advance ratio is high, cavitation would happen. Accordingly, the thrust and torque would fall down drastically and the advance ratio is getting larger. Finally, the influence on propulsion performance is getting stronger as well.
Acknowledgment
Project funding: National Innovation and Entrepreneurship Training Program for College Students (Item number:201810497071)

References
[1] Gao F, Pan C, Cai W. (2010) Numerical Analysis and Validation of Propeller Open-water Performance Based on CFD. J. Journal of Mechanical Engineering, 46(8): 133-139.
[2] Liefvendahl M, Troëng C. (2011) Computation of Cycle-to-Cycle Variation in Blade Load for a Submarine Propeller, using LES. J. Society of Naval Architects and Marine Engineers-Transactions, 4:77-83.
[3] Feng X, Lu C, Wu Q. (2012) Numerical Simulation of Propeller cavitation in Uniform Flow. J. Shipbuilding of China, 3:18-27.
[4] Wang G. Investigation on the Numerical Simulation of Propeller Hydrodynamics, Cavitation and Noise[D]. Huazhong University of Science and Technology, 2013.
[5] Kwangjun P, Hyunggil P, Jongsoo S. RANS simulation of cavitation and hull pressure fluctuation for marine propeller operating behind-hull condition[J]. International Journal of Naval Architecture & Ocean Engineering, 2013, 5(4):502-512.
[6] Shen H, Su Y. (2010) Use of the Sliding Mesh Technique to Analyze Interaction between Ship Hulls and Propellers. J. Journal of Harbin Engineering University, 31(1): 1-7.
[7] Negrato C. Prediction of the performance of ducted propellers with BEM and hybrid RANS-BEM methods[J]. 2015. Hasuike N, Yamasaki S, Ando J. Numerical and Experimental Investigation into Propulsion and Cavitation Performance of Marine.J. Propeller, 2:17-25.
[8] Li C, Zhou Q, Pan Y. Effects of Three Turbulence Models on the Cavitation Performance Evaluation for Ducted Propellers. J. Chinese Journal of Ship Research, 2:73-79.
[9] Project funding: National Innovation and Entrepreneurship Training Program for College Students (201810497071)