Prediction of hydrodynamic characteristics of combined propellers based on CFD method

Shichuan Su\textsuperscript{1} *, Shengsen Wang\textsuperscript{1}, Jiabin Cao\textsuperscript{1}, Desheng Feng\textsuperscript{2}

\textsuperscript{1}School of Energy and Power Engineering, Jiangsu University of Science and Technology, Jiangsu Zhenjiang, China
\textsuperscript{2}Nantong Office of Jiangsu branch of China Classification Society, Jiangsu Nantong, China

*Corresponding author e-mail: jstussc1@163.com

Abstract. In recent years, with the concept of "green ship" put forward, it has been a major task for ship workers to improve ship propulsion efficiency and save fuel consumption. Combined propellers show higher propulsion efficiency and better energy saving effect, so they are favored by ship workers. This article using the computational fluid dynamics (CFD) software to the single propeller, tandem propellers and contra-rotating propellers was simulated in different speed coefficient, and forecast the propeller hydrodynamic performance, comparing single propeller, tandem propellers and contra-rotating propellers open water efficiency shows: the contra-rotating propellers and tandem propellers rear blade absorbing front blade axial velocity and trailing vortex energy, increase propeller thrust, to improve the efficiency of the propeller propulsion, energy conservation and emission reduction has important significance, also conforms to the concept of "green design" of the ship.

1. Introduction
With the rapid development of ship industry, the problems of propeller cavitation, vibration and noise restrict the development of propeller, and the bottleneck of improving the efficiency of single propeller also appears. For this reason, scholars at home and abroad transfer the target to the design and research of combined propeller [1]. The emergence of combined propeller not only improves the propulsion efficiency, but also effectively improves the occurrence of cavitation and vibration. Therefore, more and more scholars are working on the combination of propellers [2-3], which includes tandem propeller, counter-rotating propeller and so on. The counter-rotating propeller is mounted on two concentric axes with opposite rotating direction, while the tandem propeller is mounted on the same axle with two propellers with the same rotating direction. The combined propeller can increase the blade area and reduce the load per unit area, and the rear propeller can effectively recover the vortex flow energy of the front propeller, thereby improving efficiency, avoiding or reducing the occurrence of cavitation and vibration.

The research methods of traditional propellers mainly rely on model tests and theoretical calculations, so the experimental period is long and the cost is high. The numerical simulation of the propeller by CFD method not only saves the manpower and material resources of the experiment, but also simulates...
the various states of the propeller in actual operation, and can obtain complete data. It has become one
of the most important methods to study ship hydrodynamic performance at this stage [4-5].

In this paper, based on the CFD method, the hydrodynamic performance of the propeller is
numerically simulated. Firstly, the influence of the turbulence model on the hydrodynamic performance
of the propeller is studied to obtain a better turbulence model. Based on this, the numerical calculation
of each propeller is verified. Secondly, according to the open water performance curve of the propeller
under different inlet speed coefficients, the problems of open water performance of single propeller,
tandem propeller and counter-rotating propeller are discussed, which lays a theoretical foundation for
the subsequent engineering design.

2. Control Equation and Turbulence Model

2.1. Control Equation.
Taking the marine propeller as the research object, the surrounding fluid can be regarded as an
incompressible fluid, that is, the density $\rho$ does not change with time. The three-dimensional steady
flow of the incompressible fluid in the propeller is described based on the RANS equation. The
continuity equation and the momentum equation are:

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

$$\rho \frac{\partial (u_i u_j)}{\partial x_i} = - \frac{\partial P}{\partial x_i} + \rho \frac{\partial}{\partial x_i} \left( \mu \left( \frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_j} \right) - u_i' u_j' \right)$$ (2)

In the formula: $\rho$ is the density of water; $u_i, u_j$ is the time-averaged velocity component; $P$ is the
pressure; $\mu$ is the turbulent viscosity; $-\rho u_i' u_j'$ is Reynolds stress term.

2.2. Turbulence Model.
In order to predict the open water characteristics of propeller more accurately, four turbulence models
including standard $k-\varepsilon$, RNG $k-\varepsilon$, RSM and SST $k-\omega$ were adopted in this paper, are used to
simulate the thrust coefficient, torque coefficient and open water efficiency of propeller at different
velocity coefficients, and the best model is selected.

Considering the complex surface space, flow field and flow rotation characteristics of propeller
model, Reynolds stress average turbulence model (RSM) is selected in this paper, because RSM model
can truly reflect the actual situation when considering streamline bending and flow rotation, and its
numerical accuracy is high[6-7].

2.3. Propeller Calculating Formula.
According to the calculation results of propeller, the following formulas are used to calculate the open
water performance of propeller. The expressions of thrust coefficient, torque coefficient and open water
efficiency are as follows [8-9]:

Speed Coefficient:

$$J = \frac{V_A}{nD}$$ (3)

Thrust coefficient:

$$K_T = \frac{T}{\rho n^2 D^4}$$ (4)

Torque Coefficient:

$$K_Q = \frac{Q}{\rho n^2 D^5}$$ (5)

Open water efficiency:

$$\eta = \frac{K_T}{K_Q} \cdot \frac{J}{2\pi}$$ (6)
Combined propeller:

\[ K_T = K_{Tf} + K_{Ta} \quad K_Q = K_{Qf} + K_{Qa} \quad \eta = \frac{K_T}{K_Q} \frac{1}{2\pi} \quad (7) \]

In the formula: \( V_A \) is the fluid velocity, \( D \) is the diameter of the propeller, \( n \) is the speed. \( K_{Tf} \) and \( K_{Qf} \) are the thrust coefficient and torque coefficients of the front propeller, \( K_{Ta} \) and \( K_{Qa} \) are the thrust coefficient and torque coefficients of the rear propeller.

3. Calculation models and meshing

3.1. Model.
Taking a marine propeller as an example, the main design parameters are rotation \( N=450r/min \), propeller diameter \( D=300mm \), pitch ratio \( P/D=1.3 \) at 0.7R and blade number \( Z=4 \). In order to effectively distinguish the various performances of the combined propeller and the single propeller, the counter-rotating propeller and the tandem propeller are combined by a single propeller, and the counter-rotating propeller is left-handed and the rear-propeller is right-handed. The front and rear propellers of tandem propellers rotate in the same direction, both of which are left-handed. Fig. 1 shows the overall model of the single propeller, counter-rotating and tandem propellers.

![Fig. 1 Models of single propeller, counter-rotating propeller and tandem propeller](image)

3.2. Grid and Computing Domain.
In this paper, the whole flow field is divided into static domain and rotating domain. The small cylinder with propeller is a rotating domain with a diameter of 1.1D and a length of 3D. The rest are static domains with a length of 10D and a diameter of 5D. Because the shape of propeller is complex and the surface curvature of propeller blade is large, the hybrid mesh method is adopted to divide the whole rotating domain into unstructured hexahedron meshes, the static domain into structured meshes, and the three-dimensional surface of propeller blade and hub is refined, and the boundary layer meshes are divided on the wall.

The computational domain setting and meshing process of the combined propeller are the same as that of the single propeller, but there are two rotational domains in the combined propeller. Moreover, the front and rear rotating domains of the combined propeller are connected by Interior, and the interface between the rotating and stationary domains is connected by interface. The calculation is based on a full-scale model, and the overall computational domain meshing is shown in Fig 2.

![Fig. 2 Meshing of computational domain and boundary condition](image)
4. Numerical calculation and analysis

4.1. Analysis of open water performance of single propeller.
In this paper, four turbulence models are selected to calculate the model, namely Standard k-ε, RNGk-ε, RSM and SSTk-ω. The rotation speed of the propeller is 450 r/min, and the numerical results calculated in the range of the speed coefficient 0.1~1.0 are compared with the experimental results, and the water performance curves of the different turbulence models are plotted, as shown in Fig. 3:

(1). It can be seen from Fig. 3(a) (b) that the thrust coefficient KT of each turbulence model has a small error with the text. When J is too small or too large, the error will obviously increase. At the same time, with J increasing, the error between the calculated torque coefficient and the text will become larger and larger. As a whole, the RSM simulation effect is consistent with the test, the Standard k-ε error is the largest.

(2). It can be seen from Fig. 3(c) that the open water efficiencies obtained by the four turbulence models are smaller than the text, and the open water efficiency of the RSM model under the various J is slightly better than the text, and the accuracy is higher. followed by the SSTk-ω model.

From the analysis of the overall open water performance curve, the open water efficiency of the RSM model is in the best agreement with the experimental value, followed by the SSTk-ω model, so the RSM turbulence model is a better turbulence model, and it shows that the method is feasible and accurate to calculate the open water performance of the propeller.

![Fig. 3 Hydrodynamic characteristics of different turbulence models](image)

(a) Thrust Coefficient (b) Torque Coefficient (c) Open Water Efficiency

4.2. Counter-rotating propeller numerical verification.
To verify the open-water performance of the counter-rotating propeller, a turbulence model of RSM was selected. The torques of the front and rear propellers of the counter-rotating propeller were equal, and the rotational speeds were all 450 r/min, and the speed coefficient is between 0.4 and 0.9, based on the RSM model. Numerical simulation of the counter-rotating propeller and plotting the open-water performance curves of the numerical simulation and test data, as shown in Fig 4: The KT and KQ of the front and rear propellers are in good agreement with the experimental data, and 10KQ decreases with the increase of J. At the same speed coefficient J, the numerical error of the rear propeller KQ is slightly larger than that of the front propeller KQ, which indicates that the wake of the front propeller has a certain effect on the rear propeller. Fig. 4 (c) shows that the thrust coefficient KT of contra-rotating propeller has good overall simulation effect, but with the increase of J, the error of 10KQ becomes larger and larger, the overall data are larger than the text, and is smaller than the text. On the whole, the numerical results of open water performance of counter-rotating propeller based on this method meet the accuracy requirements and are in good agreement with the text.
4.3. Numerical verification of tandem propellers.
The two propellers have the same direction of rotation front and rear the tandem propeller, and the rotational speed is 450 r/min. The RSM turbulence model is used for numerical simulation in the range of the speed coefficient $J$ of 0.4~0.9. $K_T$, $K_Q$ and $\eta$ are plotted and compared with the text. As shown in Fig. 5:

From Fig. 5, it can be seen that the calculated and tested of $K_T$, $K_Q$ and $\eta$ curves of tandem propellers are basically in agreement with each other under different $J$. Among them, the calculated values of $K_T$ and $\eta$ are less than the text, and $K_Q$ is larger than the text. and with the $J$ increases, $K_T$, $K_Q$ and $\eta$ and test error increase, but the overall numerical results agree well with the text, thus The feasibility and accuracy of the numerical simulation scheme of the hydrodynamic performance of the tandem propeller given in this paper are verified. And meet the needs of the project.

4.4. Open Water Performance Comparison of Combination Propellers.
We selected the ordinary propeller, tandem propeller and counter-propeller and Comparing the open water performance of three kinds of propellers by numerical calculation. For the convenience of comparison, the parameters selected by the three propellers are consistent. Open water performance is studied under the condition that the power coefficients of single propeller and combined propeller are equal according to formula $B_P = 33.07K_Q^{1/2}/V_A^{5/2}$. The open water performance results of the three types of propellers are shown in Fig. 6 Under the same conditions, the efficiency of counter-rotating propellers and tandem propellers is significantly improved compared with ordinary propellers, and its hydrodynamic performance has obvious advantages. Among them, the efficiency of the contra-rotating propeller is the highest, followed by the tandem propellers. In the range of $B_P=6$~255, the efficiency of the contra-rotating propeller is 7.28%~15.07% higher than that of the single propeller, and the efficiency of the tandem propeller is 2.68%~9.79% higher than that of the single propeller.
4.5. **Wake Analysis.**

The flow field of propeller can be simulated by CFD. The J is 0.7, analysis of single propeller and composite propeller wake field. As shown in Fig. 7, the wake outer diameter of combined propeller is smaller than that of propeller, but the wake outer diameter of single propeller is opposite.

The reason is that the wake outer diameter of single propeller increases with the increase of the J at high J. The combined propeller can absorb the wake energy of front propeller due to the mutual interference between the front and rear propeller discs. The wake diameter is smaller than the propeller diameter, which improves the energy efficiency and propeller performance.

![Fig. 6 Hydrodynamic Characteristic Curve of Power Coefficients](image)

(a) Single Propeller  (b) Counter-rotating Propeller  (c) Tandem Propeller

**Fig. 7 The trailing vortex of propeller while J=0.7**

5. **Conclusion**

1) Numerical verification of single propeller and combined propeller based on different turbulence models shows that the numerical accuracy of RSM model is high, and the study of open water performance of propeller using this method is accurate, feasible and reliable.

2) The numerical calculations compare the hydrodynamic performance of single propeller and combined propeller. The results show that the open water efficiency of combined propeller is significantly higher than that of single propeller under the same power factor, and its open water performance is obviously superior.

3) By analyzing and comparing the open water efficiency of single propeller and combined propeller, it can be seen that the combined propeller rear propeller can absorb the axial velocity and wake energy of the front propeller, reduce the influence of tangential velocity on the fluid disturbance, increase the fluid momentum flowing through the blade, reduce the wake diameter of the front propeller, increase the propeller's thrust and improve the propulsive performance of the counter-rotating propeller, which has engineering practicability.

**Reference**

[1] Wang Guoqiang, Dong Shitang. Theory and Application of Ship Propeller [M]. Harbin: Harbin Engineering University Press, 2005: 383-396. (in Chinese)

[2] Pan Guang, Xiao Zhiyong, Huang Mingming et al. Design parameter optimization of counter-rotating propeller [J]. Torpedo technology, 2010, 18(1): 6-8. (in Chinese)

[3] Sun Qin, Gu Yunde. A Simple Calculation Method for the Rotary Propeller[J]. Journal of Shanghai Institute of Shipping and Transportation, Ministry of Communications, 1981, 8(2): 198-215. (in...
Chinese)

[4] Wang Zhanzhi, Xiong Ying, Qi Wanjiang. Open water numerical prediction of counter-rotating propeller [J]. Journal of Huazhong University of Science and Technology (Natural Science Edition), 2012, 40 (11): 77-80, 88. (in Chinese)

[5] ZHANCN G Tao, YANUG Chengjun, SONU Baowei, etal. CFD simulation of the unsteady performance of contra-rotating propellers [J]. Journal of Ship Mechanics, 2011(6):605-615.

[6] WANG Fujun. Principle and Application of CFD Software for Computational Fluid Dynamics [M]. Beijing; Tsinghua University Press, 2001. (in Chinese)

[7] VERSTEEG H K, MALALASEKERA W. An Introduction to Computational Fluid Dynamics; The Finite Volume Method [M]. New York; Wiley, 1995.

[8] Hassan Ghassemi, Mohsen Taherinasab. Numerical Calculations of the Hydrodynamic Performance of the Contra-rotating Propeller (CRP) for High Speed Vehicle [J]. Polish Maritime Research, 2013, 20(2): 13-20.

[9] WANG Zhanzhi, XIONG Ying, QI Wanjiang. Numerical prediction of contra-rotating propellers open water performance [J]. Journal of Huazhong University of Science and Technology (Natural Science Edition), 2012, 40(11):7-80, 88. (in Chinese)