Comparative Analysis of Hydrodynamic Performance of Propeller Under Different Turbulence Models

Xiaoming Zhou 1, *, Huachen Pan 1, Xiaoqing Tian 1, Zefei Zhu 2

1 School of Mechanical Engineering, Hangzhou, China
2 Hangzhou Dianzi University, Hangzhou, China

*Corresponding author e-mail: xiaomingzhou1226@qq.com

Abstract: In order to study the applicability of different turbulence models in the prediction of propeller hydrodynamic performance, computational fluid dynamics software CFX was used to simulate the thrust coefficient, torque coefficient and efficiency of VP1304 propeller under specific viscous flow field conditions by using $\varepsilon - k$ models, $\omega - k$ models, SST models and RSM models respectively. The prediction results from the four turbulence models are compared with the experimental data of controlled pitch propeller VP1304 published by SVA Potsdam. It is concluded that the simulation prediction of performance due to the RSM model is the most accurate, Merits and demerits of RSM model is analyzed.

1. Introduction
In order to access the accuracy of numerical simulation for propeller with different turbulence models in CFD commercial software CFX and the practical reliability verification in industrial field [1], The k-epsilon model, k-omega model, Shear Stress Transport (SST) model and Reynolds Stress Model (RSM) model are used. The thrust coefficient, torque coefficient and efficiency value of VP1304 propeller under the condition of specific viscous flow field are simulated. The calculated results are compared with the experimental data of the controllable pitch propeller VP1304 published by SVA Potsdam Company, and the turbulence model with a better performance is selected.

2. Governing equation and turbulence model
2.1. Governing equation
In this paper, the viscous flow field of blade with a certain speed in uniform flow is simulated. According to the principle of relative motion, propeller model can be regarded as stationary in axial direction, water flows at a constant velocity $V$ relative to the propeller. It rotates in circumferential direction, while the propeller keeps a relative rotation velocity of 0. In general, water is considered to be incompressible, and the uniform flow can be regarded as a steady flow. The continuity equation is [2]:

$$\frac{\partial \overline{u_i}}{\partial t} = 0$$

Momentum equations are:
\[
\rho \frac{\partial \bar{u}_i}{\partial t} + \rho u_j \frac{\partial \bar{u}_i}{\partial x_j} = - \frac{\partial \bar{p}}{\partial x_i} + \mu \frac{\partial^2 \bar{u}_i}{\partial x_i \partial x_j} - \rho \frac{\partial \bar{u}_i u_j}{\partial x_i} \rho \bar{j}_i 
\]  

(2)

In the formula: \(\bar{u}_i\) is Reynolds average velocity, \(u_j\) is pulsating velocity, \(\rho \bar{u}_i u_j\) is Reynolds stress.

2.2. Turbulence model

2.2.1. \(k - \varepsilon\) model. As one of the most prominent turbulence models, the \(k - \varepsilon\) model has been implemented in most common CFD codes and is considered to be an industry-standard model. Facts have proved that it has perfect prediction ability. For general simulation, the model provides a good tradeoff between accuracy and stability. In CFX, when the near wall mesh is very fine, the turbulence model uses an extensible wall function method to improve the accuracy and stability. The extensible wall function can be implemented on any fine near-wall grid, which is a significant improvement on the standard wall function.

The model is a semi-empirical formula based on the two equations of turbulent kinetic energy and turbulent dissipation.

The differential transport equations are:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu_t + \mu_r \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon + P_{kb} 
\]

(3)

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho U_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_r \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} \left( C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon + C_{\varepsilon 3} P_{eb} \right) 
\]

(4)

where \(C_{\varepsilon 1}, C_{\varepsilon 2}, \sigma_k\) and \(\sigma_{\varepsilon}\) are constants. Check the CFX theoretical guide [1]. we know the values are 1.44, 1.92, 1.0 and 1.3, respectively. Turbulent viscosity coefficient \(C_{\mu} = \rho C_{\mu} k^2 \varepsilon\), \(C_{\mu} = 0.09\). \(P_{kb}\) and \(P_{eb}\) represent the influence of the buoyancy forces, which are described below. \(P_k\) is turbulence production caused by viscous force, and its formula is:

\[
P_k = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - \frac{2}{3} \frac{\partial U_i}{\partial x_i} \left( \frac{3}{2} \frac{\partial U_k}{\partial x_k} + P_k \right) 
\]

(5)

For incompressible flow, \(\partial U_i / \partial x_i\) is small, the second term to the right of equation has no significant effect on turbulence production. For compressible flow, the value is larger only in the region with a high velocity divergence.

2.2.2. \(k - \omega\) model. For \(k - \omega\) model the Two transport equations are expressed, as shown in the formula (6), (7).

The \(k\) equation is:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu_t + \mu_r \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta' \rho k \omega + P_{kb} 
\]

(6)

The \(\omega\) equation is:
\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j \omega) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\sigma_{\omega}} \right) \frac{\partial \omega}{\partial x_j} \right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega \gamma + P_{ab} \tag{7}
\]

In addition to independent variables, density \( \rho \) and velocity vectors \( U \) are considered to be known quantities from the Navier-Stokes equations. \( \beta' = 0.09 \), \( \alpha = 5/9 \), \( \beta = 0.075 \), \( \sigma_k = 2 \), \( \sigma_\varepsilon = 2 \). \( P_i \) is the turbulence generation rate, the calculation method is as shown in (1-5). An obvious advantage of \( k - \omega \) formula lies in the near wall treatment of low Reynolds number calculation. The model does not involve the complex nonlinear damping function required by the \( k - \varepsilon \) model, so it is more accurate and robust. It allows smooth conversion from a low Reynolds number form to a wall function formula. This is the advantage of \( k - \omega \) model.

2.2.3. SST Shear stress model. The SST turbulence model based on \( k - \omega \) is similar to the \( k - \omega \) model. With the following modifications:

The SST model combines the cross diffusion from the \( \omega \) equation, and the turbulent viscosity synthesizes the wave propagation of the turbulent shear stress, and the model constants are different. These changes make the SST model more accurate and reliable than the \( k - \omega \) model in a wide range of hydrodynamic fields. The main purpose of the design based on shear stress transport model is to predict the beginning of flow separation and flow separation with high accuracy. By including transport effects in vortex-viscous formulations, onset and quantity of flow separation under adverse pressure gradient are predicted with a high accuracy. This has led to significant improvements in flow separation forecasting. The superior performance of the model has been confirmed in a large number of verification studies [3]. It is suggested that the SST model be used in the high precision boundary layer simulation experiment. For free shear flow, the SST model is the same as the \( k - \omega \) model.

2.2.4. RSM Reynolds stress model. The Reynolds averaged momentum equations for the mean velocity are:

\[
\frac{\partial \rho U_i}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial x_j} - \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] = - \frac{\partial p''}{\partial x_i} - \frac{\partial}{\partial x_j} \left( \rho \bar{u}_i \bar{u}_j \right) + S_{M_i} \tag{8}
\]

\( p'' \) is a modified pressure, \( S_{M_i} \) is the sum of body forces, \( \rho \bar{u}_i \bar{u}_j \) is the sum of fluctuating Reynolds stresses. Unlike the vortex-viscosity model, the modified pressure has no turbulence contribution.

This relates to static pressure:

\[ p'' = p + \frac{2}{3} \mu \frac{\partial U_k}{\partial x_k} \tag{9} \]

In the differential stress model, \( \rho \bar{u}_i \bar{u}_j \) satisfies a transport equation. For the six specific Reynolds stress components of \( \rho \bar{u}_i \bar{u}_j \), it is necessary to solve each of the transport equations separately.

The Reynolds stress transport differential equation is:

\[
\frac{\partial \rho \bar{u}_i \bar{u}_j}{\partial t} + \frac{\partial}{\partial x_k} \left( U_k \rho \bar{u}_i \bar{u}_j \right) - \frac{\partial}{\partial x_k} \left( \delta_{ik} \mu + \rho C_s \frac{k}{\varepsilon} \frac{\bar{u}_i \bar{u}_j}{\bar{x}_j} \right) = P_{ij} - \frac{2}{3} \delta_{ij} \rho \varepsilon + \Phi_{ij} + P_{ij,b} \tag{10}
\]

\( P_{ij} \) and \( P_{ij,b} \) are the shear and buoyant turbulence generation terms of Reynolds stress, respectively. \( \Phi_{ij} \) is the pressure-strain tensor, and \( C \) is a constant. The buoyancy turbulence term \( P_{ij,b} \) also takes into
account the buoyancy contribution in the pressure-strain term and is controlled in the same manner as the \( k - \omega \) and \( k - \varepsilon \) models.

3. Establishment of a computational model

3.1. Geometry model and calculation domain establishment

The experimental propeller compared with the CFX simulation is the five-leaf VP1304 type. The specific parameters are shown in Table 1, and the model is shown in Figure 1.

The propeller is a controllable pitch propeller. The working area is a rectangular trough with a width of 9 meters, a depth of 4.5 meters and a length of 30 meters.

The propeller was placed at a height of 375mm from the surface of the water, 10 m from the uniform water inlet and 20 m from the static outlet.

Import the propeller's solid model igs file into the grid software Pointwise. Firstly, Select one of the blade geometry of one-fifth of the blade, and perform a dense structure meshing on the pressure and suction surfaces of the blade [2]. This part of the density is the key to the simulation results. Since the root of the blade is mounted on the surface of the hub, it is necessary to draw a mesh portion which is an intermediate hollow quadrilateral to form a non-structural surface. And use T-Rex technology to generate a certain thickness of the boundary layer, such a setting is conducive to improve the grid accuracy at the hub. Then, the surface of the blade and the hub of the non-structural body mesh where the blade is located also grows a boundary layer of a certain thickness. The thickness of the first layer of the boundary layer is set to 0.05, the growth factor is 1.2, and the step size is 10 to improve the calculation accuracy of the blade area. The generated mesh cut surface is shown in Fig. 3.

The simulated calculation basin is set to a 1/5 cylinder with a diameter of 6 times the diameter of 1400 mm, and the 1/5 blade is located 600 mm from the fluid inlet, as shown in Figure 4. The total number of grids is 1.36 million. Here, only the flow of one blade is designed and calculated to greatly reduce the amount of computer calculation and improve efficiency.

| parameter name          | symbol | unit     | Numerical value |
|-------------------------|--------|----------|-----------------|
| Propeller diameter      | D      | [mm]     | 250.0000        |
| Pitch r/R=0.75          | P0.75  | [mm]     | 407.3804        |
| Average pitch           | Mean   | [mm]     | 391.8812        |
| Chord length r/R=0.70   | C0.70  | [mm]     | 104.167         |
| Blade thickness r/R=0.75| t0.75  | [mm]     | 3.7916          |
| Side bevel              | -      | [°]      | 18.800          |
| Hub diameter ratio      | DH/D   | [-]      | 0.150           |
| Number of blades        | z      | [-]      | 5               |

Figure 1 Solid model of VP1304 propeller  
Figure 2 Meshing of propellers and hubs
3.2. Boundary conditions and solution settings
The inlet boundary field is set to a constant speed inlet condition, the outlet boundary type is set to the opening type, and the relative pressure is zero. The blade, the hub and the circumferential surface are arranged as a solid wall surface. The two cut surfaces p1, p2 of the cylindrical surface are set to interface[4], and the whole water flow is rotated around the X axis at a rotation speed of n=900r/min, so that the fluid forms a periodic flow. Simply multiply the result value by 5 after obtaining the calculation result to obtain the analog value of the entire five-blade paddle, which greatly improves the efficiency. CFX uses the solver to control the set solver [5], using the finite volume method. The process of solving is actually an iterative solution process of an algebraic equation.

In the numerical simulation calculation, the propeller inlet speed coefficients are set to 0.6, 0.8, 1.0, 1.2, and 1.4, respectively, and the rated speed is maintained at 900 r/min. According to the formula $J = \frac{V}{nD}$, the water inflow velocity under each speed coefficient is 2.25m/s, 3.0m/s, 3.75m/s, 4.5m/s, 5.25m/s.

The hydrodynamic performance of VP1304 propeller under the above viscous flow field conditions was simulated by four turbulence models, $k - \varepsilon$ model, $k - \omega$ model, SST model and RSM model.

4. Analysis and comparison of results

4.1. Result data comparison
See Table 2 for the experimental data of the controllable pitch type VP1304 propeller water released by SVA Potsdam. $J$ is the speed coefficient, $K_T$ is the thrust coefficient, $10K_Q$ is 10 times the torque coefficient, and $H$ is the water immersion efficiency.

| $J$   | $K_T$   | $10K_Q$ | $H$   |
|-------|---------|---------|-------|
| 0.60  | 0.6288  | 1.3964  | 0.4300|
| 0.80  | 0.5100  | 1.1780  | 0.5512|
| 1.00  | 0.3994  | 0.9749  | 0.6520|
| 1.20  | 0.2949  | 0.7760  | 0.7258|
| 1.40  | 0.1878  | 0.5588  | 0.7487|
In the post-CFX processing POST, the thrust and torque coefficients of the blades under different turbulence models can be solved under different advance coefficients. Thus, the thrust coefficient, the torque coefficient, and the hydrophobic efficiency are calculated. The calculation formula is as follows [6]:

\[ J = \frac{V}{nD} \quad (11) \]

\[ K_T = \frac{T}{\ell n^2 D^4} \quad (12) \]

\[ K_Q = \frac{Q}{\ell n^2 D^4} \quad (13) \]

\[ H = \frac{K_T}{K_Q} \times \frac{J}{2\pi} \quad (14) \]

Where \( V \) is the ship speed, \( n \) is the propeller speed, \( D \) is the propeller diameter, and \( \ell \) is the fluid density. From the results, the propeller thrust and torque under different speed coefficient conditions \( J \) are extracted, and the thrust coefficient \( K_T \), the torque coefficient \( K_Q \) and the hydrophobic efficiency \( H \) of the propeller are obtained. The comparison of the data results of different turbulence models at different speed coefficients with the experimental data is shown in Table 3 - Table 6, Figure 7 - Figure 11.

Table 3 Result of \( k - \varepsilon \) turbulent model

| \( J \) | \( K_T \) | \( K_T \) | \( K_T \) | \( 10K_Q \) | \( 10K_Q \) | \( 10K_Q \) | \( 10K_Q \) | \( H \) | \( H \) | \( H \) | \( H \) |
|-------|--------|--------|--------|---------|---------|---------|---------|------|------|------|------|
| 0.60  | 0.6288 | 0.6047 | -0.0241 | -3.8327 | 1.3964 | 1.3564 | -0.04 | -2.8645 | 0.4300 | 0.4257 | -0.0043 | -1  |
| 0.80  | 0.5100 | 0.4915 | -0.0185 | -3.6275 | 1.1780 | 1.164  | -0.014 | -1.1885 | 0.5512 | 0.5376 | -0.0136 | -2.4673 |
| 1.00  | 0.3994 | 0.3671 | -0.0323 | -8.0871 | 0.9749 | 0.937  | -0.0379 | -3.8876 | 0.6520 | 0.6234 | -0.0286 | -4.3865 |
| 1.20  | 0.2949 | 0.2536 | -0.0413 | -14.005 | 0.7760 | 0.7253 | -0.0507 | -6.5335 | 0.7258 | 0.668  | -0.0578 | -7.9636 |
| 1.40  | 0.1878 | 0.1514 | -0.0364 | -19.382 | 0.5588 | 0.521  | -0.0378 | -6.7645 | 0.7487 | 0.6475 | -0.1012 | -13.517 |

Table 4 Result of \( k - \omega \) turbulent model

| \( J \) | \( K_T \) | \( K_T \) | \( K_T \) | \( 10K_Q \) | \( 10K_Q \) | \( 10K_Q \) | \( 10K_Q \) | \( H \) | \( H \) | \( H \) | \( H \) |
|-------|--------|--------|--------|---------|---------|---------|---------|------|------|------|------|
| 0.60  | 0.6288 | 0.5819 | -0.0469 | -3.8327 | 1.3964 | 1.3564 | -0.04 | -2.8645 | 0.4300 | 0.4257 | -0.0043 | -1  |
| 0.80  | 0.5100 | 0.4915 | -0.0185 | -3.6275 | 1.1780 | 1.164  | -0.014 | -1.1885 | 0.5512 | 0.5376 | -0.0136 | -2.4673 |
| 1.00  | 0.3994 | 0.3671 | -0.0323 | -8.0871 | 0.9749 | 0.937  | -0.0379 | -3.8876 | 0.6520 | 0.6234 | -0.0286 | -4.3865 |
| 1.20  | 0.2949 | 0.2536 | -0.0413 | -14.005 | 0.7760 | 0.7253 | -0.0507 | -6.5335 | 0.7258 | 0.668  | -0.0578 | -7.9636 |
| 1.40  | 0.1878 | 0.1514 | -0.0364 | -19.382 | 0.5588 | 0.521  | -0.0378 | -6.7645 | 0.7487 | 0.6475 | -0.1012 | -13.517 |
**Figure. 5** Simulated values (CFX) the $k - \varepsilon$ model

**Figure. 6** Simulated values (CFX) the $k - \omega$ model

**Table.5** Result of SST turbulent model

| $J$ | $k_T$ | $K_T$ | $10K_Q$ | $10K_O$ | $H$ | $H$ | $H$ | $H$ |
|-----|-------|-------|--------|--------|-----|-----|-----|-----|
| Test value | Compute value | Absolute error | Relative error(%) | Test value | Compute value | Absolute error | Relative error(%) | Test value | Compute value | Absolute error | Relative error(%) |
| 0.60 | 0.6288 | 0.5845 | -0.0443 | -7.0452 | 1.3964 | 1.31984 | -0.0766 | -5.4855 | 0.4300 | 0.4229 | -0.0071 | -1.6512 |
| 0.80 | 0.5100 | 0.4766 | -0.0334 | -6.549 | 1.1780 | 1.147 | -0.031 | -2.6316 | 0.5512 | 0.5291 | -0.0221 | -4.0094 |
| 1.00 | 0.3994 | 0.3764 | -0.023 | -5.7586 | 0.9749 | 0.971 | -0.0039 | -0.4 | 0.6520 | 0.617 | -0.035 | -5.3681 |
| 1.20 | 0.2949 | 0.2661 | -0.0288 | -9.766 | 0.7760 | 0.7653 | -0.0107 | -1.3789 | 0.7258 | 0.664 | -0.0618 | -8.5147 |
| 1.40 | 0.1878 | 0.1516 | -0.0362 | 19.2758 | 0.5588 | 0.5337 | -0.0251 | -4.4918 | 0.7487 | 0.6331 | -0.1156 | 15.4401 |

**Table.6** Result of RSM turbulent model

| $J$ | $k_T$ | $K_T$ | $10K_Q$ | $10K_O$ | $H$ | $H$ | $H$ | $H$ |
|-----|-------|-------|--------|--------|-----|-----|-----|-----|
| Test value | Compute value | Absolute error | Relative error(%) | Test value | Compute value | Absolute error | Relative error(%) | Test value | Compute value | Absolute error | Relative error(%) |
| 0.60 | 0.6288 | 0.5951 | -0.0337 | -5.3594 | 1.3964 | 1.339 | -0.0574 | -4.1106 | 0.4300 | 0.4243 | -0.0057 | -1.3256 |
| 0.80 | 0.5100 | 0.4967 | -0.0133 | -2.6078 | 1.1780 | 1.16 | -0.018 | -1.528 | 0.5512 | 0.5452 | -0.006 | -1.0885 |
| 1.00 | 0.3994 | 0.3867 | -0.0127 | -3.1798 | 0.9749 | 0.961 | -0.0139 | -1.4258 | 0.6520 | 0.6404 | -0.0116 | -1.7791 |
| 1.20 | 0.2949 | 0.2743 | -0.0206 | -6.9854 | 0.7760 | 0.7457 | -0.0303 | -3.9046 | 0.7258 | 0.7025 | -0.0233 | -3.2103 |
| 1.40 | 0.1878 | 0.1587 | -0.0291 | -15.495 | 0.5588 | 0.507 | -0.0518 | -9.2699 | 0.7487 | 0.6975 | -0.0512 | -6.8385 |
4.2. Comparative analysis of calculation results
From the above chart, it can be concluded that the simulation results of the four turbulence models are similar. In the range of the velocity coefficient of 0.6-1.4, the linear distribution of the thrust coefficient $K_T$ is 10% lower than the experimental value, and there is no crossover; The 10 times torque coefficient $K_Q$ is in good agreement with the experimental data, and the SST and $k-\omega$ models intersect. The four models have uniformity in the performance of the hydrophobicity $H$. When the coefficient of advancement is small, the deviation is small and as the coefficient of advancement increases, the difference becomes an expanding trend. After the coefficient exceeds 1.2, the deviation becomes larger and larger and shows a downward trend.

From the detailed analysis of each turbulence chart, specific differences can be obtained. From Fig. 6, it can be seen that the torque coefficient $K_Q$ calculated under the $k-\omega$ model has a good crossover with the experimental values, which is much smaller than the error values of the other three models; From Figure 8, it can be seen that the efficiency value of the RSM model is the closest to the
experimental value; It can be seen from Fig. 9 that the solution conditions are the same, so under the influence of the common error, hydrophobic efficiency value $H$ under the four model simulations is different from the experimental value, and the deviation is larger after the coefficient exceeds 1.2. It is on a downward trend. However, it is not difficult to find that the simulated value of the turbulence model RSM is less than 3% of the calculated values of other models. This is not achievable by other turbulence models. The $k - \omega$ model has a large deviation, so it should not be used here; Based on the above, it can be concluded that the RSM model is more consistent with the performance of the test data. For efficiency values only, the simulation effect is RSM Model > $k - \varepsilon$ Model > SST Model > $k - \omega$ Model.

Combined with other analyses, although the standard two-equation model, as mentioned above, the A model and the S model can provide good predictions for many engineering processes, but some applications may not be suitable for these models. These include: 1. Flow with boundary layer separation. 2. The flow rate changes with a sudden change in the average strain rate. 3. Rotate the flow in the fluid. 4. Flow through the curved surface.

Standard two-equation turbulence models often fail to predict the onset and number of flow separations under adverse pressure gradient conditions. In general, the turbulence model based on the $\varepsilon$ equation predicts that the separation begins earlier and underestimates the subsequent separation. This is problematic because this behavior gives an overly optimistic performance characteristic. From an engineering perspective, the forecast is not conservative. The SST model developed to solve this problem has been shown to predict separation more accurately in many test cases and industrial applications. It has also been well verified in this simulation.

The Reynolds Stress Model (RSM) is more suitable for flow rates with sudden changes in strain rate or rotational flow. The SST model may be more suitable for separating traffic. Although the two-equation turbulence model provides good predictions of the characteristics and physics of most industrial-related flows. However, in turbulent flow or other fluid motions where non-equilibrium effects are important, the eddy viscosity assumption is no longer valid and the results of the eddy viscosity model are not accurate. Compared with the turbulence model with similar eddy viscosity, the Reynolds stress model includes the effects of streamline curvature, sudden change in strain rate, secondary flow and buoyancy. The Reynolds stress model is based on the transport coefficients of the various components of the Reynolds stress tensor and the dissipation rate. These models are characterized by a higher degree of universality. The drawback of this flexibility is the high complexity of the resulting mathematical system. An increase in the number of transmission equations will result in a decrease in numerical stability and an increase in the amount of calculation.

For the error existing in the optimal model, one thing to consider is to calculate the quality of the domain mesh. High-quality meshing is a necessary condition for the success of numerical simulation[7], A grid that is too thin or too dense will greatly affect the calculation results. Here we have adopted a total of 730,000, 1.01 million, 1.36 million and 1.71 million grid systems in a single blade basin grid for numerical simulation calculations. It is concluded that the grid with a single-blade grid number of 1.36 million is a relatively high-precision and high-efficiency calculation condition, but it is uncertain that this grid number is an optimal condition, but it is relatively optimal.

5. Conclusion and discussion
In this paper, the hydrodynamic performance of VP1304 propellers under different turbulence models is calculated and compared with the test results. The reason for the error is the following:

1. In terms of overall effect, RSM model > $k - \varepsilon$ model > SST model > $k - \omega$ model. The difference between the efficiency values of the latter three and the experimental values indicates that it is not suitable for numerical simulation of propeller hydrodynamic performance.

2. Although the accuracy of the RSM model is the best, the standard wall function method is used in this paper. The wall function method does not solve the laminar flow bottom layer and the mixed region. The semi-empirical formula is used to solve the region between the laminar flow bottom layer and the complete turbulence. In the meantime, the amount of calculation is greatly reduced,
accompanied by a decrease in calculation accuracy. On the other hand, the quality of the grid is one of the factors that influence it. Therefore, there is a certain error between the simulation results and the experimental data. Moreover, the universality brought by the flexibility of the RSM model is at the expense of a large amount of computation. The increase in the number of transmission equations leads to a decrease in numerical stability, which requires more time cost and computation to maintain stability.

(3) Consider using the Reynolds stress model in the following complex flow fields[8]:
1. A free shear flow with strong anisotropy, such as a strong swirl component, which includes flow in a rotating fluid.
2. The flow rate changes with a sudden change in the average strain rate.
3. Strain the complex fluid and reproduce the directionality of the turbulent flow itself.
4. Flow with strong streamline curvature.
5. Secondary flow.
6. Buoyancy flow.

In these cases, the Reynolds stress model shows better predictive performance.

(4) The simulation results of the hydrodynamic performance of the propeller are more accurate by using the CFX simulation software, which can intuitively simulate the characteristics of the flow field around the blade. It is helpful to select which turbulence model to select for similar propellers in the simulation of fixed watersheds in the future.

References
[1] GAO Fei, LI Xin. The ANSYS CFX 14.0 super-learning manual[M]. People's Post and Telecommunications Press, 2013.
[2] Bardina, J.E., Huang, P.G. and Coakley, T.J.Turbulence Modeling Validation Testing and Development[J], NASA Technical Memorandum 110446, 1997.
[3] Wang Huan, Li Wenying, Yang Fei. Propeller design and hydrodynamic analysis based on FLUENT [J]. Journal of Chinese Agricultural Mechanization, 2015, 36 (1): 63-66.
[4] WANG Chao, Huang Sheng, Xie Xue Shen. Prediction of propeller hydrodynamic performance based on CFD method [J]. Journal of the University of Naval Engineering, 2008, 20 (4): 107-112.
[5] WANG Guibiao, Xie Yonghe, Xu Songjie, et al. Analysis and comparison of different turbulence models in prediction of propeller hydrodynamic performance [J]. Journal of Zhejiang Ocean University (Natural Science Edition), 2014, 33 (6): 526-530.
[6] SHENG Zhenbang, LIU Yingzhong. Principle of ship [M]. Shanghai Jiao Tong University Press, 2004.
[7] Zhao W, Wan D. Numerical simulation of hydrodynamic performance of propeller by MRF Simple Foam solver[J]. International Society of Offshore & Polar Engineers, 2014, 588:3397-402.
[8] Huang S, Zhu X Y, Guo C Y, et al. CFD simulation of propeller and rudder performance when using additional thrust fins [J]. Journal of Marine Science & Application, 2007, 6(4):27-31.