Numerical Analysis of the Impact of Compensating Ducts on Propeller’s Hydrodynamic Performance

Jinpeng Kong*, Zhuangzhi Shao & Botao Wu
CSMTC, Jiangyin, Jiangsu, China

ABSTRACT: Based on FCFD method, the multiple reference system MRF model and turbulence model were adopted to conduct numerical analysis on the hydrodynamic performance of propellers with compensating ducts and analyze the impact of compensating ducts on propellers’ front and rear flow fields, thrust coefficient, torque coefficient and efficiency. It is discovered that compensating ducts can enhance the high-speed flow area behind the propeller and play the role of diversion. Under different advance coefficients, the thrust coefficient and torque coefficient of the propeller are increased. The efficiency is obviously improved, and the energy-saving effect of compensation ducts is remarkable.

Keywords: propeller; compensating duct; hydrodynamic performance; CFD

1 INTRODUCTION

Compensation conduct is one of the most widely used energy-saving devices on ships, which is a trumpet type semicircle catheter with large point diameter and small back end diameter. This device can improve the propelling efficiency, reduce the flow separation at the rear and the navigation resistance, adjust the pre-swirl degree of propeller inlet, decrease the rotational energy loss of the wake and generate additional thrust[1][2]. Besides improving ships’ rapidity, compensating ducts also have certain impact on their maneuverability[3][4], which can improve small rudder angle control and exert certain anti-rolling function in waves. It has simple structure and is easily installed, which can be used not only in new ship building, but also on already operated ships. Their energy-saving effect is remarkable; the average energy-saving effect is about 5%.

In recent years, with the constant improvement of calculation methods and computers performance, CFD calculation begins to gain the widespread attention of engineers. Due to its low cost, high speed and other advantages compared to experimental research, CFD method can simulate relatively true and reliable results. Thus, it is widely used in engineering field. In terms of propellers’ hydrodynamic performance forecast, the application of CFD method can provide many visual results, such as the propeller blade and its surrounding flow velocity distribution diagram, propeller streamline diagram and pressure distribution map, etc. This paper established the three-dimensional model of compensating ducts and propeller blades through ICEM software and forecast the impact of compensating ducts on propellers’ hydrodynamic performance using the multi-reference system MRF model and the turbulence model.

2 MATHEMATICAL MODEL

The basic governing equations of fluid dynamics include incompressible fluid continuity equation and RANS equation[6], respectively:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0
\]

(1)

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_i} \left( \rho u_i u_j \right)
\]

(2)

In the equations, \(p\) is mean value of pressure; \(\rho\) is fluid density; \(\mu\) is fluid viscosity coefficient; \(u_i\) is velocity fluctuation.

*Corresponding author: kongjinping1981@163.com
The Reynolds equation is closed using models [7]-[9]. The equations obtained from this k model are very similar to the k-ε standard model equations [10]:

\[
\rho \frac{d k}{d t} = \frac{\partial}{\partial x_i} \left[ \left( \alpha_k \mu_{eff} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_B - \rho \varepsilon - Y_M \tag{3}
\]

\[
\rho \frac{d \varepsilon}{d t} = \frac{\partial}{\partial x_i} \left[ \left( \alpha_\varepsilon \mu_{eff} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + G_\varepsilon \frac{\varepsilon}{k} + C_{2\varepsilon} \frac{\rho \varepsilon^2}{k} - R \tag{4}
\]

In the equations, \(G_k\) is the turbulent kinetic energy induced by mean velocity gradient; \(G_B\) is the turbulent kinetic energy induced by buoyancy; \(Y_M\) is the effect of compressible turbulent pulse expansion on total dissipation rate. \(\alpha_k, \alpha_\varepsilon\) is the reciprocal number of the effective Prandtl of turbulent kinetic energy \(k\) and its dissipation rate \(\varepsilon\).

As the propeller is a rotating part, the acceleration of the fluid in the rotating coordinate system equations of motion appears as an additional term. Now, FLUENT allows the absolute velocity \(\vec{u}\) or relative velocity \(\vec{u}_r\) to be independent parameters. The relationship between these two velocities is as follows:

\[
\vec{u}_r = \vec{u} - (\vec{\Omega} \times \vec{r}) \tag{5}
\]

Wherein, the angular velocity vector \(\vec{\Omega}\) (i.e. the angular velocity of the rotating coordinate system) is the position vector \(\vec{r}\) in the rotating coordinate system. The left of the momentum equation in the rotating coordinate system is as follows:

\[
\frac{\partial}{\partial t} (\rho \vec{u}) + \nabla \cdot (\rho \vec{u} \vec{u}) + \rho \left( \vec{\Omega} \times \vec{u} \right) \tag{6}
\]

According to the relationship with \(\vec{u}\) and \(\vec{u}_r\), substitute it into the above equation. No matter using the relative velocity or absolute velocity, the continuity equation in the rotating coordinate system can be expressed as follows:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}_r) = S_m \tag{7}
\]

The effect of compensating ducts on propellers’ hydrodynamic performance is mainly analyzed through the changes of propellers’ efficiency which is represented by thrust coefficient \(K_T\), torque coefficient and speed coefficient \(K_Q\):

\[
\eta_0 = \frac{K_T \cdot J}{\pi n} \tag{8}
\]

\[
K_T = \frac{T}{\rho n^2 D^4}, \quad K_Q = \frac{Q}{\rho n^2 D^5}, \quad J = \frac{V_a}{n D} \tag{9}
\]

In the equations, \(T\) is the propeller ’s axial thrust; \(Q\) is the torque generated during propeller operation; \(\rho\) is working medium density; \(D\) is propeller’s diameter; \(n\) is propeller’s working rotational speed; \(V_a\) is propeller’s advance speed.

3 NUMERICAL CALCULATION

3.1 Computing model and meshing

The ship design speed is 14 knots and the propeller’s design speed is 127r/min. The compensation ducts adopt tapered catheters, which increases the flow speed going through the propeller disks and improves the propeller’s efficiency. The duct section adopts airfoil design as shown in Figure 1. Under the diversion effect, it can automatically generate thrust and reduce the thrust reduction of ship hull.

![Figure 1. Compensating duct’s section.](image1)

The propeller’s diameter is 5.5m and the duct is 0.518m away from the interface. The duct’s internal diameter is 2.263m and the maximum outer diameter is 2.72m, as shown in Figure 2.

![Figure 2. Propeller model.](image2)

Structured mesh is conducted on the model using professional meshing software ICEM. There are two sub-domains in the grid computing domain. They are rotation domain and stationary domain. The grids of these two sub-domains are divided individually. Then, the interface is used for the whole mesh joining and the numerical calculation is carried out. The far field is a cylinder with the diameter of 33.2m and the length of 43.2m. The interface of the stationary domain and moving domain is a cylinder with the diameter of 6.624m and the length of 1.6m.

The twist angle of the propeller is large, which makes the meshing difficult. More complex topology and mesh adjustment are required to satisfy the requirements of uniform grid density, reasonable grid amount and high grid quality. The main process of meshing includes establishing the topology of different models, densifying of the wall boundary layer, key densifying of the grids around the propeller and density transition of the grids. The model grids are all structured grids. Grids sketch is shown in Figure 3. The grid number is about 1,990,000.
3.2 Boundary conditions setting

According to the model characteristics, the types of the boundary conditions adopted by FCD include velocity inlet, far field, symmetry plane, wall surface, interface, far field boundary, etc. The velocity inlet adopts the stream velocity of 14 knots (i.e. 7.106m/s). The far field is the free flowing boundary. Considering the propeller speed, the rotational domain adopts the multi-reference MRF model. The whole calculation domain is divided into two sub-domains. Both have their own motion ways. The sub-domain of the propeller is rotational and the far field’s sub-domain is stationary. Flow field governing equations are solved in each sub-domain. On the interface of sub-domains, the information exchange of flow fields between sub-domains is conducted by means of converting speed to absolute speed. The meshing on the interface is irregular, i.e., the meshing on each side of the interface is different. Then, information on the interface is transmitted by interpolation calculation.

4 HYDRODYNAMIC PERFORMANCE ANALYSIS

Analyze the front and rear flow fields of the propeller. Observe the surrounding relative velocity vector and velocity profile. In light of open water calculation conditions, analyze the front and rear flow fields of propellers with and without ducts. Analyze the front and rear sections of the propeller and the location going through the propeller shaft. There are three sections. Their locations are respectively $y=0m$, $z=92.2m$ and $z=93.5m$, as shown in Figure 4.
4.2 The impact of compensating ducts on propeller’s efficiency

Carry out statistics of the calculation data to obtain the thrust and torque data. Calculate the efficiency of the propeller by formulas. As shown in Table 1 below, the efficiency of model propellers with compensating ducts is 14.8% higher than that of model propellers without compensating ducts. The energy-saving effect of compensating ducts is remarkable.

Table 1. The efficiency of model propellers with/without compensating ducts.

<table>
<thead>
<tr>
<th>Serial No</th>
<th>Compensating ducts</th>
<th>Thrust (N)</th>
<th>Torque (Nm)</th>
<th>Efficiency</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>yes</td>
<td>225776</td>
<td>275214</td>
<td>0.444</td>
</tr>
<tr>
<td>2</td>
<td>no</td>
<td>280115</td>
<td>297424</td>
<td>0.509</td>
</tr>
</tbody>
</table>

4.3 The impact of compensating ducts on propeller’s efficiency under different speed conditions

The ship speeds are 10, 12, 14 and 16 nautical miles per hour. Thrust, torque and propeller efficiency are obtained by data statistics and processing. A shown in Table 2, the efficiency of model propellers with compensating ducts is higher than that of model propellers without ducts. Compensating ducts can play the energy-saving role very well.

From Figure 8 we can see that with the increasing of advance coefficient, the thrust coefficient and torque efficient are decreased. The thrust coefficient and torque coefficient of models with compensating ducts are bigger than the ones without compensating ducts. As shown in Figure 9, under different advance coefficients, the efficiency of propellers with ducts is higher than the efficiency of propellers without compensating ducts. The bigger the advance coefficient, the higher the efficiency improves.

Table 2. Propeller efficiency under different speeds.

<table>
<thead>
<tr>
<th>Speed (kn)</th>
<th>Compensating ducts</th>
<th>Thrust (N)</th>
<th>Torque (Nm)</th>
<th>Efficiency</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>no</td>
<td>516979.000</td>
<td>416378.460</td>
<td>0.480</td>
</tr>
<tr>
<td></td>
<td>yes</td>
<td>542974.970</td>
<td>425445.890</td>
<td>0.493</td>
</tr>
<tr>
<td>12</td>
<td>no</td>
<td>378484.530</td>
<td>350113.850</td>
<td>0.501</td>
</tr>
<tr>
<td></td>
<td>yes</td>
<td>418750.190</td>
<td>365754.080</td>
<td>0.531</td>
</tr>
<tr>
<td>14</td>
<td>no</td>
<td>225776.000</td>
<td>275214.000</td>
<td>0.444</td>
</tr>
<tr>
<td></td>
<td>yes</td>
<td>280115.000</td>
<td>297424.000</td>
<td>0.509</td>
</tr>
<tr>
<td>16</td>
<td>no</td>
<td>59302.881</td>
<td>191538.060</td>
<td>0.191</td>
</tr>
<tr>
<td></td>
<td>yes</td>
<td>126808.900</td>
<td>220086.420</td>
<td>0.356</td>
</tr>
</tbody>
</table>

5 CONCLUSION

Based on CFD method, the impact of compensating ducts on propellers’ hydrodynamic performance was analyzed referring to the multi-reference MRF system.
and turbulence model. The front and rear flow fields of propellers, thrust coefficient, torque coefficient and efficiency were analyzed. The conclusions are as follows:

(1) After adding the compensating ducts, some turbulence occurs in the front flow field of the propeller. But, the high-speed flow area behind the propeller is slightly increased. The compensating ducts played the role of diversion.

(2) Efficiency of propeller with compensating ducts models is higher than that of models without the ducts. The energy-saving effect of compensating ducts is remarkable. They can play very good energy-saving function.

(3) With the increasing of speed coefficient, the thrust coefficient and torque coefficient are decreased. The thrust coefficient and torque coefficient of propellers with compensating ducts models are higher than that of models without compensating ducts. The bigger the speed coefficient, the higher the efficiency improves.

REFERENCES


