The design and layout of rudders and propellers will greatly affect the handling performance of underwater vehicles, as well as the flow-induced noise characteristic which mainly produced by the propeller. In the present investigation, a redesigned ducted propeller with rear rudder inside the duct is proposed for revealing the effect of propeller-rudder coupling on the hydrodynamic and acoustic performance of underwater vehicles at low speeds. The hydrodynamic numerical simulation of the redesigned ducted propeller is discussed by comparing the different spacing between the propeller and the rudder. Furthermore, the effects of rudder angle on the hydrodynamics and acoustic characteristics of the propeller are also investigated. The Reynolds-averaged Navier-Stokes (RANS) method is employed to formulate the steady flow field of the propeller. Moreover, after the steady flow field obtained, Ffowcs-Williams-Hawking (FW-H) equation is adopted to reveal the flow-induced noise characteristic of the propeller.

Keywords: CFD, Hydrodynamic performance, Flow-induced noise, FW-H, Marine propeller

1. Introduction

Determining the open water performance and noise performance of the new propeller is an important step in initial ship design. This will ensure that ship sails at the set cruising speed with optimal hydrodynamic performance and low noise. However, it is difficult for a full size propeller to experimentally measure its open water performance and noise performance. Therefore, the rapidly development of CFD method has been widely applied to the initial design and argumentation of the propeller to reduce R&D investment and cycle.

Zhang Zhi-rong et al. investigated computational results for viscous free surface flow along S60 ship hull at different conditions, and then compared experimental data with CFD results. The two sets of data are well consistent, which demonstrated that CFD plays an irreplaceable role in computer-aided propeller and ship design. [1] Huang Sheng et al. simulated the interaction between propeller and the rudder with additional thrust fins with CFD software FLUENT. [2] M. A. Elghorab et al. studied open water characteristics of a propeller using FLUENT and compared pressure distribution on propeller face with different advance coefficient. [3] Senthil Prakash. M.N analyzed hydrodynamic performance of a four blades Wageningen B series propeller and validate CFD method is available in calculating hydrodynamic of the flow around the propeller. [4] Additionally, computational-aided method is also used to predict hydroacoustic performance of underwater propeller. Ville M. Viitanen et al. [5] investigated flow noise induced

In the present study, a redesigned ducted propeller with rear rudder inside the duct is proposed and computational fluid dynamics method is applied to reveal hydrodynamic performance and acoustic characteristic of this new propulsion at low speed. Unstructured mesh and RANS method are employed to analyze propeller using CD-adapco’s commercial CFD software package STARCCM+ version 12.06.011. Two main factors: the spacing of propeller-rudder and the rudder angle are discussed in detail in subsequent chapters.

2. Formulation

2.1 Governing equations

There are three basic governing equations for fluid dynamics in general. Energy conservation is not involved in this simulation because the calculation assumes that the fluid is incompressible. The mass conservation equation and the momentum conservation equation are described below.

Mass conservation is also called the continuity equation, which is defined as Eq.1.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0
\]  

(1)

Momentum conservation equation is defined as Eq.2.

\[
\frac{\partial}{\partial t}(\rho \mathbf{v}) + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot \left( \mathbf{\mu} \left[ (\nabla \mathbf{v} + (\nabla \mathbf{v})^T) \frac{2}{3} \nabla \cdot \mathbf{v} I \right] \right)
\]

(2)

Where: \( \mathbf{v} \) - the velocity vector
\( p \) - static pressure
\( \mu \) - molecular viscosity
\( I \) - the unit tensor

When using the Reynolds averaging approach, the Navier-Stokes equations can be written as [7]:

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

(3)

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

(4)

Where: \( \delta_{ij} \) - the Kronecke delta
\(-\rho u_i u_j\) - the Reynolds stresses

2.2 Hydrodynamic theory of the propeller

In this paper, open water characteristics are shown in the form of the advance coefficient \( J \), the thrust coefficient \( K_T \), the torque coefficient \( K_Q \) and the open water efficiency \( \eta_o \). Each of variables is given below.

The advance coefficient:

\[
J = \frac{V_A}{n \cdot D}
\]

(5)

The thrust coefficient
The torque coefficient

\[ K_T = \frac{T}{\rho n^2 D^4} \]  

(6)

The open water efficiency

\[ K_Q = \frac{Q}{\rho n^2 D^5} \]  

(7)

The open water efficiency

\[ \eta_0 = \frac{T v_A}{\omega Q} = \frac{K_T}{K_Q} \rho n^2 D^4 \frac{v_A}{\omega} = \frac{K_T}{K_Q} \cdot \frac{J}{2\pi} \]  

(8)

2.3 Propeller radiated underwater noise modelling

In STARCCM+, a built-in noise model based on the FW-H equation can be used to analyze the hydroacoustic problems. The FW-H method was originally proposed by Ffowcs-Williams and Hawkings in 1969 and then used to calculate aerodynamic noise. In recent years, this method has been used to calculate the noise radiation in water [6].

The equation can be written as:

\[ \Box^2 p = \frac{\partial}{\partial t} \left[ \rho_0 v_n \delta(f) \right] - \frac{\partial}{\partial x_i} \left[ \frac{\partial}{\partial x_j} \left( \frac{\partial}{\partial x_j} \left( H(f) \right) \right) \right] \]  

(9)

Where \( p \) is the acoustic pressure in the undisturbed medium and \( \rho_0 \) is the fluid static density. \( v_n \) is the source surface’s local normal velocity. \( \delta(f) \) and \( H(f) \) respectively. The Lighthill stress tensor is defined as \( T_y \) and D’Alembert operator can be expressed in another form \( \Box^2 = -\frac{1}{c_0^2} \frac{\partial^2}{\partial t^2} + \nabla^2 \). Physically speaking, the right side of the equation represents thickness, loading and quadrupole noise terms.

3. Propeller geometry and numerical method

The redesigned ducted propeller contains of three principal components: duct, propeller and rudder. The specific information of each component is shown in Table 1.

Table 1: The redesigned Ducted propeller characteristic

<table>
<thead>
<tr>
<th>Duct form</th>
<th>JD-75</th>
</tr>
</thead>
<tbody>
<tr>
<td>Propeller</td>
<td>Ka4-70</td>
</tr>
<tr>
<td>Rear Rudder (Section)</td>
<td>NACA0045</td>
</tr>
</tbody>
</table>

A four blades Ka series propeller is selected for current study and its basic geometric datas are collated in Table 2.

Table 2: Ka-propeller parameters

<table>
<thead>
<tr>
<th>No. of Blade</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter</td>
<td>210mm</td>
</tr>
<tr>
<td>Expanded area ratio</td>
<td>0.55</td>
</tr>
<tr>
<td>Hub ratio</td>
<td>0.2</td>
</tr>
<tr>
<td>Pitch ration</td>
<td>1.0</td>
</tr>
</tbody>
</table>
The three-dimensional geometry of the propeller and the overall diagram of the new propeller can be seen in Figure 1 and Figure 2.

![Figure 1: Propeller model](image1)

![Figure 2: The redesigned ducted propeller model](image2)

In order to simulate the operation of the propeller in water medium, the computational domain is divided into two parts: the rotating domain and the static domain.

1) The rotating domain: this area is used to simulate rotation of the propeller. The rotation of the propeller is simulated using the moving reference frame (MRF) and the shape of the rotating domain is a cylinder which contains propeller.

2) The static domain: this area is enclosed by a cylinder which contains duct, rudders and shaft. Relative to the propeller, the cylinder should be large enough to accurately simulate the actual flow. However, it has to be considered that too large static domain will result in a large number of meshes and excessive solution time [8].

The overall computational domain is shown in Figure 3.

![Figure 3: The modelling of computational field](image3)

In this simulation, the boundary conditions are set as follows. The inlet boundary and the wall surface of the static domain are selected as the velocity inlet while the outlet boundary is the pressure outlet. Due to complexity of the flow at the blades, duct and rudders, the mesh of these areas should be properly refined.
The advance coefficient is adjusted by keeping the propeller rotating speed constant (set 15rps in the simulation) while changing the forward speed of the thrust. To obtain open water characteristic of the redesigned propeller, the simulation calculations were performed for the following values of the advance coefficient \( J: 0.2, 0.4, 0.6, 0.8 \) and \( 1.0 \). The turbulence model used in this paper is the RNG (Renormalization Group) \( k-\varepsilon \) model based on Reynolds-averaged Navier-Stokes (RANS) method.

4. Results and discussion

4.1 The effect of spacing between propeller and rudder

In this section, three models with different spacings, 45mm, 55mm and 65mm, were simulated in STARCCM+. After the unstructured grid is generated, the solution settings are made to obtain the open water characteristic curve of the propeller under different working conditions.

![Numerical hydrodynamic coefficients of the propeller with different spacings](image)

Figure 6: Numerical hydrodynamic coefficients of the propeller with different spacings

Figure 6 shows that when comparing with Wang's experimental data [9], the KT curve results are in good agreement while 10KQ curve has a large difference. This means that the rear rudder has a great influence on the open water characteristics of the conventional ducted propeller. Furthermore, the propeller under the three different working conditions almost have the same highest efficiency when \( J=0.6 \). However, under the same advance coefficient, three hydrodynamic coefficients curves present some differences. This indicates that the distance between propeller and rudder is an influential factor in the open water performance of the redesigned ducted propeller. Comparing the three working conditions, as the spacing of propeller-rudder increases, the thrust coefficient, the torque coefficient and efficiency of the propeller decrease. The reason may be that the interaction between propeller and rudder is a favorable interference. When the spacing of propeller-rudder is reduced, the presence of the rudder can increase the efficiency of the propeller.
4.2 The effect of rudder angle

The influence of rudder angle on the open water performance of the propeller is analyzed by keeping the distance between the rudder and the propeller at 45mm while making the non-adjacent pair of rudders deviate by 5 degrees and 10 degrees.

![Propeller with rudder angles from rear view](image)

Figure 7: The propeller with rudder angles from rear view: (a) 5 degrees; (b) 10 degrees

By comparing the open water data of the propeller-rudder spacing at 45mm in section 4.1, Figure 8 can be obtained as shown below.

![Numerical hydrodynamic coefficients of the propeller with different rudder angles](image)

Figure 8: Numerical hydrodynamic coefficients of the propeller with different rudder angles

Figure 8 shows that the hydrodynamic coefficients of the propeller decrease with the appearance of the rudder angle. The open water coefficient decrease significantly between the advance coefficient of 0.6 and 1 with rudder angle increases. Moreover, as the advance coefficient increases, the thrust coefficients of D45-R5 and D45-R10 is lower than D45 while the torque coefficient hardly changes. The decrease in open water performance may be due to the rudder angle disturbing the propeller wake field, which in turn affects the performance of the propeller. This interference will become more and more obvious as the rudder angle increases.

Furthermore, the flow-induced noise generated by the propulsion of the underwater vehicles during operation is an important source of underwater radiation noise, this noise will also destroy the marine ecological environment. Hence, acoustic research of the rudder angle’s influence has been studied in this part.

FW-H equation is applied to analyze flow-induced noise generated by the propulsion. The sound pressure level (SPL) of propeller with and without rudder declination is analyzed when the distance between propeller and rudder is kept at 45mm. At the same time, the advance coefficient is set to 0.6. The receiving point is arranged in the far field and its position is shown in Figure 9.
Figure 10 shows that the amplitude of the sound pressure level spectrum decreases from low frequency to high frequency. At the same time, the average value of the flow noise of the rudder angle propeller is higher than that of the rudderless propeller. In general, as the rudder angle increases, the sound pressure level at the same frequency increases. The possible reason is that the appearance of rudder angle disturbs the flow field inside the duct, especially the flow field behind the propeller, which may adversely affects the acoustic performance of the propeller.

5. Conclusions

This paper mainly focuses on the open water characteristics and acoustic performance of a redesigned ducted propeller. The discussion was numerically investigated by using CD-adapco’s commercial CFD software package STARCCM+ version 12.06.011. Firstly, the effects of the spacing between propeller and rudder are studied through comparing the results from the numerical analysis of 45mm, 55mm and 65mm propeller. In addition, the effects of rudder angle on the hydrodynamics and acoustic characteristics of the propeller are also investigated. The following results have been concluded:

1) The interaction between propeller and rudder is a favorable interference. The open water performance of the propeller improves while the spacing of propeller-rudder decreases.

2) The presence of the rudder angle will decrease the open water performance of the propeller. The possible reason may be that the rudder angle affects the propeller wake field and thus affects the propeller performance.

3) Furthermore, the appearance of the rudder angle will enhance sound pressure level (at the same frequency) of the propeller. The may reason is that flow field inside the duct is disturbed by the rudder angle, which will lead to detrimental effects on the acoustic performance of the propeller.

Nomenclature

- $D$: Propeller diameter
- $N$: Rotational speed
- $V_A$: Advance velocity
- $T$: Thrust of the propeller
- $Q$: Torque at the propeller axis
- $J$: Advance coefficient
- $K_T$: Thrust coefficient
- $K_Q$: Torque coefficient
- $\rho$: Density of the water
$\eta_0$ Propeller efficiency  
D45 The distance between propeller and rudder is 45mm  
D45-R5 The distance between propeller and rudder id 45mm, rudder angle is $5^\circ$  
D45-R10 The distance between propeller and rudder id 45mm, rudder angle is $10^\circ$  
D55 The distance between propeller and rudder is 55mm  
D65 The distance between propeller and rudder is 65mm

REFERENCES


