The Numerical Analysis of Hydrodynamic Characteristics of Ducted Propeller by using SST k-ω Model

Wang Yue¹,²,³, Ren Wanlong¹,²,³, Liu Gang¹,²,³, Zhang Yuanming¹,²,³ and Hao Zongrui¹,²,³

¹Institute of Oceanographic Instrumentation, Qilu University of Technology (Shandong Academy of Sciences), Qingdao, 266100, China
²Shandong Provincial Key Laboratory of Marine monitoring instrument equipment technology, Qingdao, 266100, China
³National Engineering and Technological Research Center of Marine Monitoring Equipment, Qingdao, 266100, China

Abstract. Hydrodynamic characteristics of the four-blade ducted propeller are numerically simulated by CFD method. The hydrodynamic performance of the propeller is analyzed by the SST k-ω Model, including the external characteristic parameters such as thrust and torque. The forward velocity at the propeller surface of the ducted propeller, the distribution of the induced velocity along the propeller surface, and the relationship between the propeller thrust and the surrounding velocity field are analyzed. The pressure distribution on the inner wall of the conduit is complicated. The pressure in the front half of the inner wall is small and large in the rear half. Axial, radial and tangential velocity distributions of sections at different distances from the paddle surface are calculated. The velocities on the dorsal side of the propellers are significantly higher than that on the foliar side and lower near the conduit.

1. Introduction
The ducted propeller is a commonly used propeller in underwater submersible systems. The trajectory control and navigational attitude adjustment of underwater submersible are generally accomplished by application of ducted propellers [1, 2]. In the case of large thrust and low speed, the propeller efficiency of the ducted propeller is significantly higher than that of the ordinary propeller. Therefore, for low-speed, high-thrust thrusters, ducted propellers are applicable [3]. The ducted propeller can also function as a rectification in use, reducing vibration and improving the stability of the ship sailing in the wind and waves. Also, ducted propellers are used to save energy [4].

As the application of ducted propellers is continually expanding, the computational fluid dynamics (CFD) method is introduced into the research of ducted propellers [5-7]. A lot of research and discussion have been carried out. The focus of these researches is the relationship between the flow field and thrust. Most studies use the slip grid to simulate the relative motion of the blades and the pipe in the water. By setting different boundary conditions to simulate the effects of different speeds and different flows on the thrust characteristics of the propeller. Many different models are used to do the calculations, such as standard k-ε, RNG k-ε, SST k-ω, etc. The pressure distribution, velocity distribution, and vortex shedding, cavitation are investigated. These studies are mainly on the characteristics of the propeller and...
on the surface of the propeller, however, there are few works studies the flow field at the specific plane behind the propeller disk. In this paper, the three-dimensional flow fields of the ducted propeller disk are simulated. Different planes behind the propeller disk are used to study the flow characteristics of the ducted propeller. The mechanism of complex flow region formation is discussed.

2. Computational Methods

2.1. Control Equations

2.1.1. Governing Equation of Fluid Flow. The laws of mass and momentum are applied for compressive laminar and turbulence flow. All control equations are represented by a conservation law. The coordinate system used to describe the problem is the Cartesian coordinate system in the Euler description. When the flowing fluid is defined as a continuum, the governing equations used to solve are as follows.

Continuity equation:

\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0 \]  \hspace{1cm} (1)

\( u \) is the instantaneous velocity in the \( j \) direction and \( \rho \) is the density of the fluid.

Momentum equation:

\[ \frac{\partial}{\partial t} (\rho u_j) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho f_i \]  \hspace{1cm} (2)

\( P \) is the static pressure, \( \tau_{ij} \) is the viscous stress tensor and \( f_i \) is the volume force. \( \tau_{ij} \) can be expressed as:

\[ \tau_{ij} = \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{\gamma} \mu \left( \frac{\partial u_m}{\partial x_m} \right) \delta_{ij} \]  \hspace{1cm} (3)

\( \mu \) is the hydrodynamic viscosity, \( \delta \) is the Kronecker function.

Bring (2) into (1) to get the Navier-Stokes equation:

\[ \frac{\partial}{\partial t} (\rho u_j) + \frac{\partial}{\partial x_i} (\rho u_i u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \left( \frac{\partial u_m}{\partial x_m} \right) \delta_{ij} \right) + \rho f_i \]  \hspace{1cm} (4)

2.1.2. SST \( k-\omega \) Model. SST \( k-\omega \) Model was proposed in 1993 by F.R. Menter [8]. As the Wilcox \( k-\epsilon \) model is accurate in the near wall region and the \( k-\epsilon \) model is suitable for freestream, this model combines these advantages of the Wilcox \( k-\omega \) model and the \( k-\epsilon \) model. The \( k-\epsilon \) model is transformed into a \( k-\omega \) model. The two models are weighted averaged by function \( F_1 \) and then added to obtain the BSL \( k-\omega \) model. Based on the BSL \( k-\omega \) model, the SST \( k-\omega \) model modified the definition of the eddy-viscosity for adverse pressure gradient boundary-layer flows.

Original \( k-\omega \) model and transformed \( k-\epsilon \) model are weighted and averaged to obtain the following model:

\[ \frac{\partial}{\partial t} (\rho k) = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta \rho \omega k + \frac{\partial}{\partial x_i} \left[ \left( \mu + \sigma_{k1} \mu_r \right) \frac{\partial k}{\partial x_i} \right] \]  \hspace{1cm} (5)
\[
\frac{\partial}{\partial t} (\rho \omega) = \frac{\gamma}{v_t} \tau_i \frac{\partial u_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x} \left[ \mu + \sigma_\omega \mu_t \right] \frac{\partial \omega}{\partial x_j} + 2\rho (1 - F_1) \sigma_\omega^2 \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \tag{6}
\]

Where,
\[
\mu_t = \frac{\rho a_{1k}}{\max(\alpha_1 \omega \Delta F_2)}, \quad v_t = \frac{a_{1k}}{\max(\alpha_1 \omega \Delta F_2)}, \quad F_2 = \tan \text{arg}^2, \quad a_r g_1 = \min \left[ \max \left( \frac{2 \sqrt{\beta^2 \sigma_\omega^2}}{d_2}, \frac{2 \sqrt{\beta^2 \sigma_\omega^2}}{d_2} \right), CD_{\omega} = \max \left( 2 \frac{\rho a_{1k}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} 10^{-20} \right) \right]
\]

\[
\beta^*, \beta, \gamma, \alpha_i, \sigma_{1i}, \sigma_{0i}, \text{and } a_i \text{ are empirical constants. For any constant } \phi, \text{ there is}
\]
\[
\phi = F_1 \phi_1 + (1 - F_1) \phi_2 \tag{7}
\]

The subscript 1 indicates the constant value in the inner layer, and the subscript 2 indicates the constant value in the outer layer.

2.2. Geometric Model Parameters and Meshing

The parameters of the simulated ducted propeller are listed as blow.

| Table 1. Parameters of the simulated ducted propeller. |
|-------------------------------------------------------|
| Propeller Diameter | Propeller solidity ratio | Number of blades |
| \( R_p = 120[\text{mm}] \) | \( \Phi = 0.55 \) | \( N = 4 \) |
| Inclination | Inlet diameter | Outlet diameter |
| \( \alpha = 0[^\circ] \) | \( R_i = 150[\text{mm}] \) | \( R_o = 150[\text{mm}] \) |

There are three methods for solving the problem of rotating motion commonly: dynamic mesh, slip mesh, and motion reference system settings. In this study, the method of motion reference system settings is chosen.

An unstructured tetrahedral mesh is used through the domain. The total nodes of channel 173413 and the total number of grids is 972572. The calculating grids are as shown in figure 1.

Figure 1. Duct and propeller surface meshing.
3. Results and Discussion
The pressure distribution of the duct at a speed coefficient \( J = 0.57 \) is shown in figure 2. It can be seen that the pressure distribution on the outer wall of the duct is relatively uniform. The axial force generated by the outer wall of the duct, i.e., the thrust, is substantially negligible. The pressure distribution on the inner wall of the duct is complicated. The pressure is small in the front half of the inner wall and large in the second half. There is a certain low-pressure area. Combined with the position and pressure distribution of the propeller, it is found that the low-pressure area of the inner wall of the duct is related to the position of the propeller. This is the location and source of the duct tip vortex.

![Figure 2. The pressure distribution of the duct at a speed coefficient \( J = 0.57 \).](image)

The low-pressure zone is followed by the appearance of a high-pressure zone. This indicates that the propeller load has a large pressure influence on the duct, and the gap flow pressure fluctuation generated by the tip portion is complicated. Complex turbulence creates complex stress changes on the duct.

In order to analyze the internal flow field of the ducted propeller, the axial, radial and tangential velocity distributions at the different distances \( x/R_p \) from the paddle surface in the direction of the propeller axis are selected in the case of the inlet coefficient \( J = 0.5 \). The results at \( x/R_p = 0, 0.2 \) are shown in figure 3, figure 4, and figure 5.

![Figure 3. Distribution of axial velocity.](image)

Figure 3 shows the axial velocity distribution at the \( x/R_p = 0, 0.2 \) positions after the paddle surface. The axial velocity on the dorsal side of the propeller is significantly higher than that on the foliar side, and the velocity near the duct is low, in accordance with the Bernoulli equation analysis. The speed difference suggests that the foliar side is the pressure side and the back side is the suction side. The velocity gradient is larger at \( x/R_p = 0 \) and smaller at \( x/R_p = 0.2 \). This indicates that the viscous action between the fluids tends to converge the flow field.
Figure 4. distribution of radial velocity.

Figure 4 shows the radial velocity distribution at the \( x/R_p = 0, 0.2 \) positions after the paddle surface. The radial velocity of the flow field inside the duct is relatively small, and the outer zone is almost 0. Near the inner wall of the duct, a region with a small radial velocity is observed and mainly concentrated near the propeller tip. This is due to the fact that during the rotation of the propeller, the resulting tip vortex forms a complex flow area. The tangential velocity distribution is shown in figure 5, which is similar to the radial velocity distribution.

4. Conclusion

In this study, the flow fields of the ducted propeller were simulated by CFD method. Axial velocity, radial velocity, and tangential velocity distributions were figured and discussed. It indicates that The axial velocity on the dorsal side of the propeller is significantly higher than that on the foliar side, and the velocity near the duct is low, in accordance with the Bernoulli equation analysis. Near the inner wall of the duct, a region with a small radial velocity is observed, which is due to the complex flow area formed by the rotation of the propeller.

References

[1] AVILA J P J and ADAMOWSKI J C 2011 Ocean Eng. Experimental evaluation of the hydrodynamic coefficients of a ROV through Morison's Equation, 38(17-18) 2162-70.

[2] JAULIN L 2009 Automatica Robust set-membership state estimation: application to underwater robotics, 45(1) 202-6.

[3] Gong J, Guo C, Zhao D, Wu T and Song K 2018 Ocean Eng. A comparative DES study of wake vortex evolution for ducted and non-ducted propellers, 160 78-93.
[4] Mysa R C, Tuyen L Q, Ma A and Nguyen V T 2018 Proceedings of The ASME 37th Int. Conf. Ocean, Offshore Arctic Eng. Numerical investigation of energy saving device effects on ships with propeller-hull interactions, 7B V07BT06A044.

[5] Gaggero S, Villa D, Tani G, Viviani M and Bertetta D 2017 Ocean Eng. Design of ducted propeller nozzles through a RANSE-based optimization approach, 145 444-63.

[6] Vasudev K L, Sharma R and Bhattacharyya S K 2018 Ships Offsh. Struct. Shape optimisation of an AUV with ducted propeller using GA integrated with CFD, 13(2) 194-207.

[7] Andersson J, Eslamdoost A, Vikstrom M and Bensow R E 2018 Ocean Eng. Energy balance analysis of model-scale vessel with open and ducted propeller configuration, 167 369-79.

[8] Menter F R 1993 23rd Fluid Dynamics, Plasmadynamics, Lasers Conf. Fluid Dynamics and Co-located Conf. Zonal Two Equation $k-\omega$ Turbulence Models for Aerodynamic Flows, 6 AIAA-93-2906.

Acknowledgment
This work was financially supported by the National Key R&D Program of China (2016YFC1402306), Natural Science Foundation of Shandong (ZR2017MEE046), Shandong Key Research and Development Program (2016JMRH0541), Shandong Key Research and Development Program (2018GGX103018), Qingdao Basis Research of Application Program (16-5-1-101-jch) and Qingdao Basis Research of Application Program (17-1-1-97-jch).