Numerical research of the effects of fouling on the performance of marine propellers

Tao Zhang, Sheng-xue Yu
Naval Research Academy, Beijing, China
zthjgc@qq.com

Abstract. As it is known, propeller has been most widely-used ship thruster, propeller blade surfaces are of polished metal and have no antifouling provision, which makes them vulnerable to fouling. Under the effects of fouling, the propulsion efficiency of propellers decreases and fuel consumption of ships increases, thus greenhouse gas emissions increase, although the effects are rarely studied. The present work aims at quantifying the effects and providing a deep insight into the physical mechanism by means of Computational Fluid Dynamics (CFD) and the surface panel method. The simulation employs the SST k-ω turbulence model and is carried on NSRDC-NACA-66-mod blade section which is widely applied to marine propeller. Barnacles are selected as the study subject from the fouling community and are directly modeled at geometry level. With Propeller 4383 being studied, the induced velocity field under several advance coefficients through the surface panel method, hence the incoming velocity and effective attack angle of the blade sections at each radius are obtained. Then the lift and drag force of each blade section under fouling conditions are got through the CFD numerical method, and then the propeller open water performance parameters under fouling conditions are obtained through blade elements theory, such as the thrust coefficient $TK$, torque coefficient $QK$ and open water efficiency $\eta_0$. The results shows that the fouling has a serious adverse effect on the propeller performance parameters, and the serious calcareous fouling leads $TK$ goes down by 36.1%, $QK$ increases by 25.7% and $\eta_0$ reduces by 49.2%.

1. introduction

The settlement and subsequent growth of flora and fauna on surfaces exposed in aquatic environments is termed fouling\(^1\). Fouling begins to occur immediately after a ship is immersed in water and will continue to accumulate throughout a ship at sea until a cleaning and repainting process is performed. It is well established that fouling on ships increases the surface roughness of the hull which, in turn, caused increased frictional resistance and fuel consumption and decreased top speed and range\(^2\).

According to Taylan\(^3\), the increase of ship resistance due to microscopic fouling is around 1-2%, whereas an accumulation of macroscopic fouling, such as barnacles, can cause an increase of ship resistance as high as 40%. Schultz et al.\(^4\) investigated the effect of fouling on the required shaft power for a frigate by the means of fouling rating of US Naval Ships’ Technical Manual\(^5\) and equivalent sand roughness. He predicted roughly that when the ship at a speed of 15 knots, heavy slime required a 21% increase in shaft power, compared to the hydrodynamic smooth condition, whereas heavy calcareous fouling led to an 86% increase. However, their study doesn’t include the effect of propeller fouling because the influence of propeller fouling on ship powering is not as well established as for generic roughness. According to US Naval Ships’ Technical Manual, fouling on the propeller can account for as much as 50 percent of the increased energy demand associated with a light to moderately fouled hull.
Therefore, light to moderately fouling of the propeller can result in 10.5% to 43% energy loss of a ship. Generally, propeller blade surfaces are of polished metal and have no antifouling provision, which makes them vulnerable to fouling. The 25th International Towing Tank Conference (ITTC) pointed out that in its report by far for propeller roughness the biggest cause is fouling and a small roughness increase of the propeller can cause large increases in the required power [6].

The aim of the present study is to investigate the impact of fouling on the hydrodynamic performance of propellers by utilizing the CFD method, the surface panel method and blade elements theory, with the study object of Propeller 4383 under several fouling conditions.

2. numerical approach

2.1. Calculation Model

The calculation is carried on 4383 Propeller, which is a 5-blade high-skewed propeller, and adopts the NSRDC-NACA-66-mod blade section.

Barnacles are dominant organism of fouling community on propellers, which present conical shapes, the upper diameter is smaller than the base and the angle between the wall and the base is less than 90 degrees. Therefore, barnacles are modelled as small conical-cylinder shapes in the research, which are shown in Fig.1. As the barnacles is modelling on the geometric-scale method, and the number of computing grid is very large, the two-dimensional model is adopted in the numerical simulation, which is shown in Fig.2. With these thoughts in mind, the computational fouling cases can be set up in Table I. Different fouling cases of Fouling_1mm, Fouling_3mm and Fouling_6mm are studied, and the Smooth case is the presentation of the control case.

| Case          | Model barnacle height h (mm) |
|---------------|-------------------------------|
| Smooth        | 0                             |
| Fouling_1mm   | 1                             |
| Fouling_3mm   | 3                             |
| Fouling_6mm   | 6                             |

2.2. SST k-w Turbulence Model

Menter[7] simulated the flow field distribution around foils by using k-e, k-w, BSL k-w and SST k-w turbulence model respectively, and found that the results of SST k-w model agreed with experiment data best. Hence the present study employs SST k-w model in commercial software FLUENT to investigate the flow field of rough 2-D blade sections and effect of fouling on the performance. The SST k-w model is described as follows:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho ku)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \Gamma_i \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k
\]  

(1)
\[
\frac{\partial (\rho w)}{\partial t} + \frac{\partial (\rho w u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \Gamma_w \frac{\partial w}{\partial x_j} \right) + G_w - Y_w + S_w D_w \tag{2}
\]

Where \( k \) is the turbulent kinetic energy, \( w \) is the specific dissipation rate. In these equations, \( G_k \) represents the generation of turbulence kinetic energy due to mean velocity gradients. \( G_w \) represents the generation of \( w \). \( \Gamma_k \) and \( \Gamma_w \) represent the effective diffusivity of \( k \) and \( w \), respectively. \( Y_k \) and \( Y_w \) represent the dissipation of \( k \) and \( w \) due to turbulence. \( S_k \) and \( S_w \) are user-defined source terms.

2.3. The Surface Panel Method

The surface panel method is a discretization method for solving elliptic equations and it is also a boundary element method. As the method directly satisfies the boundary conditions on the object surface, and does not make any assumptions about the geometry, thus more accurately characterizing complex geometry shapes\(^8\). Therefore, the surface panel method is an accurate method for calculating the speed field of propellers, and is adopted to solve the induced velocity field.

2.4. Blade Elements Theory

The velocity polygon of the dr blade section at the propeller radius \( r \) is shown in Fig 3. The complex motion on the blade can be attributed to the flow to blade section at velocity \( V_R \) and angle of attack \( \alpha_k \), and the lift \( dL \) and drag \( dD \) is generated on the blade section. Though decomposing the lift into axial component and the component forces of rotation direction of propellers, the thrust \( dT \) and rotation resistance \( dF \) of the blade section can be expressed as:

\[
dT = dL \cos \beta_i - dD \sin \beta_i \tag{3}
\]
\[
dF = dL \sin \beta_i + dD \cos \beta_i \tag{4}
\]

Fig. 3. The velocity polygon of the blade section and its force

2.5. Grid and Meshing

Considering the geometry irregularity of the blade section surface under fouling conditions, hybrid grid method is adopted: unstructured grid is used in the inner C-region near the blade section and structured grid is used in the outer C-region far away from the blade section. Near-wall grid is arranged clustering to the surface of the blade section to capture the boundary layer field accurately, especially the viscous sub-layer. Besides, in order to preserve the high resolution near the trailing edge, different C-grid has been created for each angle of attack instead of changing the angles of the incoming flow. The whole computational zone and mesh distribution for NSRDC-NACA-66-mod blade section is displayed in Fig.4.
3. Results and discussions

3.1. Grid Dependence Test

It is important to arrange the mean distance of the first grid point from the wall (y+) properly as a result of large gradient of the streamwise velocity near the wall. This distance is characterized by y+ value. Here y+ is defined as $y^+ = \left( \frac{\rho \, y_p \, u^\tau}{\mu} \right)$ where $u^\tau$ is the friction velocity and defined as $u^\tau = \left( \frac{\tau_w}{\rho} \right)^{1/2}$ with $\tau_w$ being wall shear stress. Fig.5 shows the change of CL/CD with the variation of y+, no significant change can be observed after y+ is reduced to 2, and the value of yp is about $2 \times 10^{-4}c$ (m) where c is the chord length. Therefore, the grid resolution with y+ value of 2 is chosen and used throughout all cases.

3.2. Effects of Fouling on Performance of the Blade Sections

The effects of fouling on lift coefficient CL and drag coefficient CD of NSRDC-NACA-66mod blade sections in Fig.6 and Fig.7. It can be seen that the influence of fouling on the lift and drag is large, for instance, under 5 degrees of angle of attack, compared to the Smooth case, CL is reduced by about 11% and CD increases by about 65% under Fouling_1mm case, CL is reduced by about 23% and CD increases by nearly 2.8 times under Fouling_6mm case. With the increase of fouling height, the effects on CL and CD increase, however, the results of Fouling_6mm case is nearly the same as Fouling_3mm case although the height of the former is almost twice that of the latter. Therefore, it can be inferred that once the height exceeds a certain limit, the increase of the height has little effect on CL and CD. Yet the critical value of fouling height needs to further research to determine.
3.3. Effects of Fouling on Performance of the Propellers

Axial and circumferential induced velocity is computed through the surface panel method at several advance coefficient J of 4383 propeller, thus inflow velocity VR and effective angle of attack αk at different radius locations such as 0.3R, 0.4R, …, 0.9R. The geometry shapes of blade sections at each radius can be obtained after the propeller diameter is set as 1 meter. Then through the numerical simulation method of flow field around the blade sections under fouling conditions, the lift dL and drag dD is gotten in three fouling cases of Fouling_1mm, Fouling_3mm and Fouling_6mm. Considering six advance coefficient, seven blade sections and three fouling conditions, one hundred twenty-six numerical calculations are needed in total.

The thrust coefficient \(K_T\), torque coefficient \(K_Q\) and open water efficient \(\eta_0\) are computed under three fouling cases and each advance coefficient, and the results are compared with the experimental data of the open-water propeller under smooth conditions, as shown in Fig. 8, Fig. 9 and Fig. 10. As can be seen from the figures, fouling has negative effect on hydrodynamic parameters of the propeller, the influence of fouling on the propeller performance increases with increasing of fouling height for the same advance coefficient, the influence of fouling on the propeller performance increases with increase of the advance coefficient for the same fouling condition. For example, under the cases of Fouling_1mm, Fouling_3mm and Fouling_6mm in the condition of \(J=1.0\), \(K_T\) reduces by 12.46%, 22.29% and 36.1% respectively, \(K_Q\) increases by 11.86%, 17.65% and 25.71% respectively, and \(\eta_0\) reduces by 21.74%, 33.95% and 49.17%. Near the advance coefficient (\(J=1\)) of the highest efficiency of the propeller, the hydrodynamic performance is most affected by fouling, e.g. \(\eta_0\) reduces by nearly 50% under Fouling_6mm case.

Fig. 8. The comparison of the open water performance of Propeller 4383 under 1mm fouling condition with the experimental data under smooth condition

Fig. 9. The comparison of the open water performance of Propeller 4383 under 3mm fouling condition with the experimental data under smooth condition
3.4. Conclusion
With the study object of Propeller 4383, the influence of fouling on the propeller hydrodynamic performance is evaluated through numerical calculation. The results show that fouling has adverse impact on the performance of propellers, and in the most severe case, the efficient of propellers goes down by nearly 50%. Although the structure and distributions of fouling in this paper are somewhat different from the cases of ship propellers, the impact of fouling should be paid enough attention and research efforts.

References
[1] M. P. Schultz, Frictional resistance of antifouling coating systems. J.Fluids Eng, 2004, 126(6), pp. 1039-1048.
[2] M. P. Schultz, J. A. Bendickb, E. R. Holmb, et al., Economic impact of biofouling on a naval surface ship. Biofouling 2011, 27, pp. 87-98.
[3] M. Taylan, An overview: effect of roughness and coatings on ship resistance. In: Proceedings of the International Conference on Ship Drag Reduction (Smooth-Ships). 2010, Istanbul, Turkey.
[4] M. P. Schultz, Effects of coating roughness and biofouling on ship resistance and powering. Biofouling, 2007, 23(5), pp. 331-341.
[5] Naval Ships’ Technical Manual Chapter 81, Waterborne underwater hull cleaning of naval ships. Publication S9086-CQ-STM-010/CH-081-R5. Naval Sea System Command, 2006.
[6] ITTC, ITTC - Recommended Procedures and Guidelines: Testing and Extrapolation Methods, Propulsion, Performance, Predicting Powering Margins. ITTC, 2008.
[7] F.R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 1994, 32, 1598-605.
[8] Cai Hao-peng, Research on the design method of marine propeller based on the theory of surface panel method. Harbin: Harbin Engineering University, 2011.