Simulation analysis of underwater working condition of ship rudder blade based on Fluent

Haowei Zhao*a, Guifeng Liub*, Jianggui Han*c
Mechanical Engineering, Naval University of Engineering, Wuhan 430032, China
*a1130441457@qq.com, b2838739726@qq.com, *921174821@qq.com

Abstract—The rudder blade is an important power plant of the ship. Removing damage and failure is the premise of the normal operation of the ship. In this paper, the numerical calculation method is used, combined with K-ε (k-epsilon model) and S-A (spalart allmaras) turbulence model, the underwater working conditions of ship rudder blade are analyzed, and the influence of water flow impact at different angles of attack on the damage failure of rudder blade components is discussed. The simulation results show that when the ship is sailing at constant speed, with the change of water attack angle, the main distribution range of rudder blade stress deviates, and the lift and resistance of rudder blade are also gradually increasing, which is easy to damage the parts with concentrated stress. It has theoretical guiding significance for the protection and repair of rudder blade damage.

1. Introduction
During the navigation operation of the ship, the rudder blade plays an important role as a power mechanism, which can not only ensure the navigation stability and operability of the ship, but also improve the economy and rapidity of the ship through the optimization of structural functions[1]. The rudder blade system works in high humidity and high salinity waters, and bears high-power loads such as torque, impact and vibration, resulting in complex and diverse failure forms and affecting the normal operation of the ship[2]. When the ship is sent to the dock for overhaul after long-term operation, the main maintenance problems of the rudder blade are steel plate fracture, impact loss and cleaning of marine attachments, and the damaged parts of the rudder blade are mainly concentrated on the upstream side of the rudder blade and the upstream surface of the steering. Therefore, it is of great significance for rudder blade protection and repair to simulate the working condition of rudder blade and explore the relationship between its component failure and the transient load caused by water flow impact.

Common rudder blades can be divided into single plate rudder and streamlined rudder according to different section shapes. The streamlined rudder is generally selected for long-distance navigation, so the representative streamlined rudder is selected for analysis.

2. Overview of Hydrodynamic Performance of Rudder Blade
The hydrodynamic performance analysis of the rudder blade is to obtain the dimensionless working coefficient of the rudder blade during underwater steering by calculating the overall working condition of the rudder blade at different angles of attack. The equivalent loads on the rudder blade can be obtained by converting the specified formula, so as to evaluate the possible damage of each part of the rudder blade.

When a ship is sailing on the water surface, the rudder blade will be affected by the force of water
flow on it, which is hydrodynamics. As shown in Fig. 1, when the ship is working, water flows to the rudder blade at speed \( v \) and angle of attack \( \phi \), and the rudder blade is affected by the rudder pressure at this time. The rudder pressure on the rudder blade is equivalent to the resultant force \( P \), and the operating point of the resultant force \( P \) is the intersection of the resultant force \( P \) and the chord of the rudder blade, which is related to the motion direction of the hull or rudder blade. The resultant force \( P \) can be decomposed into \( P_d \) in the direction of water flow and \( P_l \) in the direction perpendicular to the direction of water flow. It can also be decomposed into tangential force \( P_t \) and normal force \( P_n \) perpendicular to the string direction according to the string direction. The torques produced by the normal force \( N \) on the leading edge of the rudder blade and the rudder rod are \( M_0 \) and \( M^3 \).

![Fig. 1 Schematic diagram of rudder blade under water force](image)

The hydrodynamic force and torque generated by the rudder blade under the action of water flow can be expressed by dimensionless coefficients. That is, the resultant force coefficient \( C_r \), the lift coefficient \( C_l \), the drag coefficient \( C_d \), the tangential force coefficient \( C_t \), the normal force coefficient \( C_n \), the moment coefficient \( C_m \) and the rudder rod moment coefficient \( C_{m0} \). The hydrodynamic performance analysis of the rudder blade is to obtain the variation of the hydrodynamic and dimensionless hydrodynamic coefficients of the rudder blade through the calculation of the rudder blade working at different angles of attack. Among them:

\[
\begin{align*}
C_n &= \frac{P_n}{\frac{1}{2} \rho v^2 A} \quad C_l = \frac{P_l}{\frac{1}{2} \rho v^2 A} \\
C_d &= \frac{P_d}{\frac{1}{2} \rho v^2 A} \quad C_a = \frac{P_a}{\frac{1}{2} \rho v^2 A} \\
C_m &= \frac{M}{\frac{1}{2} \rho v^2 Ab}
\end{align*}
\]  

(1)

In the type, \( \rho \) is the density of water, kg/m\(^3\); \( v \) is flow velocity, m/s; \( A \) is the lateral projection area of the rudder blade, m\(^2\); \( B \) is the profile width of the side of the rudder blade, m.

The correlation of the above dimensionless coefficients is as follows:

\[
\begin{align*}
C &= \sqrt{C_n^2 + C_l^2} = \sqrt{C_d^2 + C_t^2} \\
C_n &= C_l \cos \alpha + C_d \sin \alpha \\
C_l &= C_n \cos \alpha - C_d \sin \alpha \\
C_t &= C_n \cos \alpha + C_d \sin \alpha \\
C_d &= C_t \cos \alpha - C_n \sin \alpha
\end{align*}
\]  

(2)

At present, there are generally three methods for hydrodynamic performance analysis of rudder blades: hydrodynamic empirical formula, reference to the data of rudder blade model and simulation.
by using CFD (Computational Fluid Dynamics) software. The advantages and disadvantages of the three methods are shown in Tab. 1. Therefore, using CFD software to simulate the working condition of rudder blade has a high degree of simulation.

Tab. 1 Common analysis methods of hydrodynamic performance

| Method for analysis of hydrodynamic performance | advantages | disadvantages |
|-----------------------------------------------|------------|---------------|
| The hydrodynamic empirical formula is adopted  | Simple calculation | The simulation is poor, and the error is large with the actual working situation |
| Refer to rudder model data                     | The model is similar to the actual rudder blade in geometry, motion and dynamics, and the experimental results are relatively accurate | The model experiment process is long, the experiment cost is relatively high, and the experimental condition is different from the actual condition |
| Applying CFD software                          | The experimental working condition is close to the actual working condition, and the experimental cost is relatively low | There are requirements for computer computing power |

3. Calculation Process of Hydrodynamic Performance of Rudder Blade

3.1. Modeling and meshing

In this study, the hydrodynamic performance of streamlined rudder blades was analyzed, and the model of the rudder blade was semi-suspended. The NACA0015 airfoil profile was selected as the experimental object, and the hydrodynamic performance of the rudder blade under different water attack angles was calculated respectively, so as to make theoretical preparation for the underwater fault analysis of the rudder blade.

NACA0015 airfoil profile, chord length is 1.5m, aspect ratio $\lambda=1$, the modeling environment is set as Reynolds number $Re=0.81\times10^6$, the rudder blade water attack Angle is $0^\circ$, $10^\circ$, $20^\circ$, $30^\circ$ and $40^\circ$ respectively, and the water velocity is set as 10m/s.

The boundary conditions of simulated flow field in the moving environment of rudder blade should be combined with the governing equation to make the simulation close to the actual working situation. Environmental boundaries include inlet, outlet, wall and far field boundary condition. The inlet was given the initial velocity of the flow field, and the underwater operation of the rudder blade was simulated. The outlet was set as the pressure outlet, the wall was set as the wall with no slip boundary, and the far field boundary of the fluid was set as the flow field was stable and undisturbed.

![Fig. 2 Setting rudder blade calculation domain](image)
The setting of rudder blade calculation domain is as follows: domain surface ADHE is set as inlet surface, and external calculation domain surface BCGF is set as outlet surface; the external domain surfaces ABCD, EFGH, ABFE and DCGH are set as two symmetrical planes. The surface along the -X direction and the surface along the X direction in the inner calculation domain are the boundary interfaces for rudder blade hydrodynamic calculation. The rudder blade modeling is shown in Fig.3 and Fig.4.

3.2. Calculation results and analysis of hydrodynamic performance of rudder blade
In this chapter, five flow directions are used in the numerical simulation (0°, 10°, 20°, 30°, 40°). The hydrodynamic performance of the rudder blade is simulated and calculated, and the pressure distribution cloud diagram of the rudder blade and the lift and drag coefficients of the rudder blade are obtained, and compared with the theoretical calculation results of the rudder blade, the influence of the angle of attack on the working performance of the rudder blade is analyzed.

3.2.1. Pressure distribution cloud diagram
The pressure distribution clouds of 0° and 20° are shown in Fig.5. In the diagram of each rudder blade, the right side is the upflow side, and the left side is the upflow side in the diagram of the back of the blade.
According to the above pressure cloud diagram of the rudder blade, it can be concluded that under the condition of flow attack Angle, the average velocity of water flow on the rudder blade surface is smaller than that on the back of the rudder blade, so it is easy to produce pressure difference, namely suction.

### 3.2.2. Curve of rudder blade lift and drag coefficient

The lift force and drag coefficient curves of the semi-suspended streamlined rudder with section NACA0015 at different Angle of attack are as follows:

In the simulation, it can be seen that the stress on the upstream side of the rudder blade is large when the rudder blade is working, and the stress on the upstream surface of the rudder blade is large when steering, which is easy to impact and damage the rudder blade surface.

### 4. Conclusion

In this paper, s-A model and standard k-ε model were used to analyze the viscous flow field of the rudder blade under different working conditions, to study the stress distribution, lift and drag curves of the rudder blade under different flow attack angles, and to explore the relationship between the failure of the rudder blade component and the transient load caused by water impact. The main conclusions are as follows:

1. Under the transient impact of water flow, the stress of rudder blade is concentrated during navigation, which is easy to cause damage or failure of rudder blade.

2. When the ship is running, the main stress of the rudder blade is distributed on the inflow side and inflow surface, which is consistent with the fault problems encountered in practical maintenance. It can be verified that the transient impact of water flow is an important factor of the mechanical failure of the rudder blade.

In this experiment, the underwater working condition of rudder blade is simulated and analyzed, which is verified with the actual maintenance working condition, and the ship damage is analyzed in
principle, which provides a guiding idea for ship maintenance and fault protection.

References
[1] Feng Song, Donghua Jiang, Tianling Dai. (2019) Rudder Blade Selection and Adaptive Parameter Optimization. J. Ship Standardization Engineer, 52(04) : 76-81.
[2] Deming Wu, Shouren Li, Shan Wang. (1995) Reliability Design of Ship Product Reliability Engineering. M. Harbin: Harbin Engineering University Press.
[3] Qinsheng Mei. (1981) Rudder for ship. People's Communications Press. Beijing.
[4] Jinqi Shi, Qingeng Zhang, Liwen Wang. (2016) Dynamic Simulation of Torpedo's Electric Servo Actuator. J. Torpedo Technology, 24(06): 463-468.
[5] Fei Peng, Yong Huang, Zhun Li. (2020) Finite Element Analysis of Strength of Vibrating Screen Vibrator Shaft Based on ANSYS Workbench. J. Coal Mine Machinery, 41(3): 173-176.
[6] Xiongke Zhang, Zhong Wang, Hui Fan, et. (2021) Simulation analysis of impact load on actuating mechanism of double degree of freedom. Ship Science and Technology, 43(3): 19-23.
[7] Yang Yong, Wang Zaigao, Huang Chao. (2021) Analysis of Hydrodynamic Characteristics of Balanced Rudder Based on Fluent. J. Ship Materials & Market, 29(05): 27-31.
[8] Yuan Pei-yin, Zhao Yu, Lei Lin, Qin Huai-tao, Zhou Chuang, Li Dong-ying. (2021) Simulation analysis of dynamic characteristics of collision between ship and dock based on LS-DYNA. Ship Science and Technology, 43(3): 29-32.