Numerical simulation, verification and experiment validation on hydrodynamic performance and radiated noise properties of skewed propeller

Fuqiang Ma¹, Yansen Liu, Lin Mu and Bing Jia
Scientific and Technology on Underwater Test and Control Laboratory, Dalian, China

¹E-mail: ma_fu_qiang@126.com

Abstract. Numerical simulation of hydrodynamic performance and radiated noise properties of the standard propeller is carried out by computational fluid dynamics (CFD) approach and acoustic analogy equation, and verifying and validating the numerical result by experiments in cavitation tunnel. Taking Potsdam Propeller Test Case (PPTC) propeller as an example, the geometric model, grid model and numerical simulation boundary conditions are established. The comparison between numerical calculation and experimental results of open water characteristics of propeller, including thrust coefficient, torque coefficient and efficiency shows that the numerical model is reliable. Under the condition of good agreement between numerical simulation and experimental results of unsteady hydrodynamic characteristics, which contains the pressure distribution of blades, cavitation properties, the wake vortex distribution of the propeller, the numerical calculation and characteristic analysis of propeller radiated noise are carried out. The results show that the relation between the radiated noise of the propeller and the inherent structure and the motion state is correlated. The research can provide a theoretical basis for the target detection and recognition based on such characteristics.

1. Introduction
The radiated noise of propeller is one of the major noise sources of underwater vehicle, which is an important basis for detecting and identifying targets. With the continuous integration and development of hydroacoustics and CFD technology in recent years, the numerical method to study the radiated noise of propeller has become a hot issue in the world. Because of the steady improvement of Computational Fluid Dynamics (CFD) technologies, as well as computer performances, numerical simulations have become a valuable and reliable tool for design purposes [1]. In the specific case of propeller, CFD analysis can be effectively used to predict overall performance as well as to investigate the effects of particular flow phenomena such as cavitation, therefore, the verification and validation of propeller numerical simulation model has become a subject worthy of attention [2].

2. Numerical model and simulation approach

2.1. The geometric model
The propeller is an international standard propeller model, which was named Potsdam Propeller Test Case (PPTC) and was proposed on the first workshop on cavitation and propeller performance [3], the main parameters of the propeller model are shown in Table 1.
According to the transformation matrix between geometric parameters and spatial coordinates, combined with the spatial coordinate value, the profile curve of blades and hub is generated, then the three-dimensional geometric model of the vehicle and contra-rotating propellers is formed, as shown in Figure 1 [4].

| Parameters                  | Value |
|-----------------------------|-------|
| Number of blades            | 5     |
| Diameter (m)                | 0.25  |
| Chord0.7R (m)               | 0.1   |
| P/D0.7R                     | 1.6   |
| Skew at tip (°)             | 19.12°|
| AE/AO                       | 0.7   |

2.2. The grid model
In order to ensure the accuracy of calculation of unsteady hydrodynamic performance of propeller and to meet the accuracy requirements of fluctuating force in calculation of radiated noise characteristics, the method of partition and grid is adopted for the calculation region. In the near wall region of propeller, the method of mesh encryption is adopted, while in the far field region away from propeller, the method of sparse division is adopted, the total number of mesh nodes is 4832536. The density of mesh nodes is the largest in the trailing edge region, leading edge region and tip section of the propeller, as shown in the Figure 2.

2.3. The numerical setup
The calculation domain is generally divided into the rotating region around the propeller and the static region of the outflow field [5], as shown in the Figure 3. The inlet and outlet of the calculation domain are set as the velocity inlet and the pressure outlet respectively. The surfaces of the propeller, hub and shaft are all set as the condition of no sliding wall. The rotation speed of the rotation region is consistent with that of the propeller.
3. **The experiment validation on hydrodynamic performance**

3.1. *The experiment of propeller model*

According to the numerical simulation, the experiment of the propeller model was carried out. The experiment was carried out in a cavitation tunnel of the Shanghai Chiao Tung University, the width of cavitation tunnel is 1000mm, the experiment schematic is shown in Figure 4.

![Figure 4. The schematic of the experiment.](image)

3.2. *The verification of open water characteristics*

In the simulation, the propeller speed is set as \( n = 600 \text{r/min} \) and the advance velocity coefficient is determined by changing the inflow velocity. The open water performance of propeller includes thrust coefficient and torque coefficient of propeller, and the specific calculation formula of the velocity coefficient is as follows:

\[
J = \frac{V_A}{(n \cdot D)} \quad K_T = \frac{T}{\rho n^2 D^3} \quad K_q = \frac{Q}{\rho n^2 D^3}
\]

(1)

From the Figure 5, we can see the calculation results of thrust coefficient, torque coefficient and the efficiency under the condition of \( J = 0.5-1.2 \). It shows that the calculated results of propeller thrust coefficient and torque coefficient are verified, and it can be seen that the grid constructed in this simulation meets the engineering precision requirements, and the open water calculation results are reliable.

The Table 2 shows that the calculated results of propeller thrust coefficient and torque coefficient are verified, and it can be seen that the simulation meets the engineering precision requirements.
Figure 5. The verification of open water performance: (a) the thrust coefficient; (b) the torque coefficient; (c) the efficiency.

Table 2. The results of verification on hydrodynamic performance.

| Parameters | Thrust coefficient | Torque coefficient |
|------------|--------------------|--------------------|
| R_G        | 0.3125             | 0.398              |
| p_G        | 3.35               | 2.65               |
| C_G        | 2.2                | 1.5                |
| U_G        | 0.00077            | 0.00092            |

3.3. The verification of unsteady hydrodynamic properties
It can be seen that the suction side of the pressure is significantly lower than the pressure side, the pressure difference is the driving force of the propeller, which conforms to the pressure distribution law of propeller rotating in Figure 6. By analyzing the pressure distribution on the suction surface of the blade, it can be seen that the pressure decreases gradually from the trailing edge to the leading edge and reaches the minimum value at the leading edge, so the leading edge of the blade tip position is the easiest to produce cavitation position.
In the Figure 7, the blue area represents the distribution of water, and the green area represents the distribution of water vapour. It can be seen that the leading edge of the suction surface of the blade produces slightly vortex-cavitation, and the pressure surface is in a non-cavitation state, combined with the analysis of the same experimental conditions, the cavitation position and the main cavitation morphology of the numerical simulation results are similar to the experimental results.

**Figure 6.** The pressure distribution of blades: (a) suction side; (b) pressure side.

**Figure 7.** The cavitation properties distribution of blades: (a) suction side; (b) pressure side.
It can be seen from Figure 8, due to the rotation of the propeller, the wake track is in a spiral shape and the wake diameter gradually decreases with the rotation. According to the photo of the experiment in corresponding working condition, the shape of wake trace of the propeller is similar to the numerical simulation result.

Figure 8. The wake distribution: (a) result of the numerical simulation; (b) experiment result.

4. Acoustic signature simulation
Based on the analysis of unsteady hydrodynamic properties, the fluctuating force on the blade surface were extracted, and the simulation on radiated noise properties and acoustic signature analysis of the propeller were carried out under the conditions of \( v=5\text{m/s}, \ n=23\text{r/s}, \ v=3\text{m/s}, \ n=13\text{r/s} \) and \( v=1.5\text{m/s}, \ n=15\text{r/s} \) respectively by using the acoustic analogy method [6, 7]. The simulation results are shown in Figure 9, Figure 10 and Figure 11 respectively.

Figure 9. Radiated noise properties under the conditions of \( v=5\text{m/s}, \ n=23\text{r/s} \).
Figure 10. Radiated noise properties under the conditions of \( v=3\text{m/s} \), \( n=13\text{r/s} \).

Figure 11. Radiated noise properties under the conditions of \( v=1.5\text{m/s} \), \( n=15\text{r/s} \).

As can be seen from these figures, the first order blade-passing frequency feature of propeller is the most significant feature, other low frequency spectrum lines are basically the harmonics of the propeller shaft frequency and the blade-passing frequency. The low frequency line spectrum reflects the multiplicative frequency feature, and the low-frequency line spectrum characteristics of the propeller can reflect the relationship between the inherent structural properties and the motion states of the propeller.

5. Conclusions
In this paper, the numerical model on radiated noise of propeller is verified and validated through the simulation calculation and experimental. The hydrodynamic characteristics of open water and the unsteady hydrodynamic characteristics were verified and the simulation on radiated noise properties and acoustic signature analysis of the propeller were carried out. The results show that the numerical simulation results are in good agreement with the experimental results, which reflects the accuracy and reliability of the numerical model and its related calculation methods.

References
[1] Fred S, Robert W and Jun S 2005 *Int. J. Num M. Flu.* 50(11) 1335-40
[2] Aktas B, Atlar M, Turkmen S, Korkut E and Fitzsimmons P 2016 *Ocean.Eng.* 120 136-151
[3] Stefano G and Diego V 2018 *J. Marine Sci. App.* 17 1-4
[4] Bensow R E and Bark G 2010 *Implicit J Fluids Eng.* 132(4) 88
[5] Bertetta D, Brizzolara S, Gaggero S, Savio L and Viviani M 2012 *Ocean.Eng.* 53 177-195
[6] Ffowcs Williams J and Hawkings D 1969 Vol. A264 321-342
[7] Akkermans R A D, Stuermer A and Delfs J W 2016 *AIAA J.* Vol. 54 No. 4 1413-1423