Study on the experiment and numerical simulation of cavitation flow mechanisms at different flow rates in water-jet propulsion pumps

Zheng Kaixuan¹, Long Yun¹* Zhu Rongsheng¹, Xu Changjian², Chen Jianping², Wang Dezhong³

¹ National Research Center of Pumps, Jiangsu University, Zhenjiang 212013
² Marine Design and Research Institute of China, NO 1688 South Xizang Road, Shanghai 200011, China
³ School of Mechanical Engineering, Shanghai Jiao Tong University, Shanghai 200240

* Corresponding author: Long Yun, Email: longyun@ujs.edu.cn

Abstract. Cavitation is the main noise source that causes the vibration of the water-jet propulsion pump. High-performance ships require the water-jet propulsion pump to be efficient and have low noise at high speed. At present, the understanding of the cavitation flow phenomenon and the interference mechanism of the vortex structure of the water-jet propulsion pump at different flow rates is not clear enough. This study takes the cavitation flow characteristics of the jet pump at different flow rates as the research object, uses high-speed photography (HSP) to obtain the flow structure of the cavitation vortex at different flow rates, studies the influence of the cavitation vortex structure on the performance of the water-jet pump, uses CFD simulation to reveal the cavitation flow structure and its evolution mechanism of the water-jet pump at different flow rates, and establishes the correlation between the cavitation flow structure and the cavitation performance. The research work provides help to grasp the influence of cavitation vortex structure on the performance of water-jet propulsion pump, captures the evolution mechanism of cavitation flow structure in the water-jet pump under different flow rates, and provides a reference for improving the cavitation performance in the water-jet pump.

Keywords: Water-jet pump, different flow, cavitation evolution

1. Introduction

Waterjet propulsion is a propulsion method that uses the reaction force of the water sprayed by the propulsion pump to propel the ship forward [1]. This technology has been widely used in various fields due to its strong anti-cavitation performance, high propulsion efficiency, and low vibration and noise levels. Application and development [2-4]. The flow rate and head of the mixed-flow water jet propulsion pump are moderate, and the advantages of axial flow pumps and centrifugal pumps are taken into account in terms of structure and performance [5]. When cavitation occurs in the water jet propulsion pump, it will not only reduce the thrust, but also cause a decrease in efficiency and increase in noise [6], which is not conducive to the safe and stable operation of the pump. The cavitation of the water jet propulsion system will lead to accelerated erosion of the blade surface, which will further lead to the degradation of the performance of the water jet propulsion pump. Serious surface erosion of hydraulic components requires suspension for repair or component replacement, which not only reduces the overall speed of the ship, but also causes high maintenance costs. Degraded performance of the water jet propulsion pump can also affect the maneuverability and maneuverability of the ship.
Prior to this, the team built a water jet propulsion pump comprehensive performance test and test platform. Through the cavitation performance test, the model pumps under 1.000Q, 0.957Q, 0.913Q, 0.870Q and 0.804Q (0.46 m$^3$/s, 0.44 m$^3$/s, 0.42 m$^3$/s, 0.40 m$^3$/s and 0.37 m$^3$/s) head with the change curve of the NPSH$_r$ [7]. In addition, combined with numerical simulation and experimental results, the cavitation flow structure, evolution law and influence on pump performance of the water jet propulsion pump under the critical NPSH at the design flow rate (0.46 m$^3$/s) were described and analyzed [8]. In order to establish the correlation between cavitation flow structure and cavitation performance, the cavitation vortex flow characteristics of cavitation generation and development are obtained to provide more scientific support for cavitation performance prediction [9-10].

By extracting the 24m/s water velocity isosurface and analyzing the water surface velocity on the isovalue surface, the team revealed the flow characteristics of the high-velocity fluid zone under different cavitation stages. Through the analysis of the equivalent surface vortex structure, the main factors that affect the development of the vortex structure in the high-velocity fluid area are summarized [11]. Then the team simulated the cavitation characteristics of the impeller tip clearance leakage flow and tip leakage vortex under different cavitation conditions. The flow mechanism of the impeller tip leakage flow and the separation vortex induced by the cavitation zone under different cavitation conditions were revealed. The main factors affecting the development of cavitation wake structure were summarized [12].

In order to further explore the flow structure of the cavitation vortex in the pump under different flow rates and the influence of the cavitation vortex on the performance of the water jet propulsion pump under different flow rates, this paper takes the high-performance water jet propulsion pump model developed by the team as the research object, and uses CFD simulation to reveal the cavitation flow structure and evolution mechanism of the jet pump under five different flow rates of 0.46 m$^3$/s, 0.44 m$^3$/s, 0.42 m$^3$/s, 0.40 m$^3$/s, and 0.37 m$^3$/s, to establish the cavitation flow structure and cavitation performance correlation between. Combined with high-speed photography (HSP) to obtain the flow structure of cavitation vortex under different flow rates, the influence of cavitation vortex structure on the performance of water jet propulsion pump is studied.

2. Numerical simulation

2.1. Research object

The research object is the water jet propulsion pump numbered WJP001. The three-dimensional model of the impeller and rear guide vane mixed-flow hydraulic model is shown in Figure 1, including the inlet suction pipe, deflector cap, impeller, guide vane body, perspective window, etc. Its physical map is shown in Figure 2. See Reference 7 for the design parameters of the waterjet propulsion test pump.

![Figure 1 3D model of Water-jet Pump](image-url)
The fluid domain of the hydraulic components of a waterjet propulsion pump, including inlet pipes, impellers, guide vanes and outlet pipes. The specific water body domain is shown in Figure 3.

2.2. **Grid division**

The number and quality of meshes have a great influence on numerical simulations. In order to ensure calculation accuracy and calculation efficiency, ICEM CFD is used to divide the calculation domain of the water jet pump inlet pipe and outlet pipe, BladeGen is used to construct the geometric model of the blade and flow channel, and Turbogrid is used to perform structural meshing on the impeller and guide vane divided. The specific division results are shown in Figure 4. Under the premise of meeting the calculation requirements, the total number of grids is about 4.13 million. Mesh independence checks have been done in previous studies.
2.3. Numerical calculation method and boundary condition setting

The complex three-dimensional cavitation flow in the water-jet pump is calculated with the commercial software Ansys CFX. The liquid phase is water at 25 °C, the density is 997 kg/m³ and the kinematic viscosity is 8.899×10⁻⁴ kg m⁻¹ s⁻¹. SST k-ω turbulence model is adopted. When the cavitation occurs, the vapor phase selects 25 °C water vapor, the density is 0.02308 kg/m³, and the dynamic viscosity is 9.8626×10⁻⁶ kg m⁻¹ s⁻¹. When the RMS residual is less than 10⁻⁵, the calculation stops. Monitor hydraulic head and efficiency until their values are constant. The boundary condition of the inlet is set as pressure inlet. The boundary condition of the outlet is set as mass outflow. Wall boundary conditions uses the non-slip wall. The impeller calculation domain is set to operate at 1450 r/min. The blade and hub are arranged to rotate, the shroud wall speed is set to Counter rotating wall. The Diffuser, Suction and Outpipe are set to rest. The interface between the rotating part and the stationary part is set to the Frozen Rotor Interface.

2.4. Control equation and cavitation model

It is assumed that the solution of cavitation flow is uniform flow, that is, multi-phase flow composed of liquid and steam is taken as a medium, which means that they have the same velocity and pressure. The continuity equation and momentum equation of the RANS equation are as follows:

\[
\frac{\partial \rho_m}{\partial t} + \frac{\partial}{\partial x_j} (\rho_m u_j) = 0
\]

(1)

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

(2)

Where, \( p \) is the mixing pressure, Pa; \( u_i \) is the velocity in the \( i \) direction, m/s; \( \mu \) is the laminar dynamic viscosity, kg/(m·s); \( \mu_t \) is the turbulent eddy dynamic viscosity, kg/(m·s). The definition of mixed density \( \rho_m \) is as follows:

\[
\rho_m = \alpha_v \rho_v + (1 - \alpha_v) \rho_l
\]

(3)

Where, the subscripts \( v \) and \( l \) represent the vapor phase and the liquid phase, respectively. \( \rho_v \) stands for vapor density, kg/m³; \( \rho_l \) represents liquid phase density, kg/m³; \( \alpha_v \) represents the vapor volume fraction. The cavitation model is evolved from the mass transfer equation to achieve the purpose of converging the volume fraction of the vapor, and its formula is as follows:

\[
\frac{\partial \rho_t \alpha}{\partial t} + \nabla (\rho_t \alpha \mathbf{u}) = m^+ - m^-
\]

(4)

The source term includes an evaporation term (for example \( m^+ \)) and a condensation term (for example \( m^- \)), Zwart[13] gives the following definition.
In the formula, \( F_{\text{vap}} \) and \( F_{\text{cond}} \) are empirical coefficients of the mass transfer process and are recommended to be 50 and 0.01\(^{[13]}\); \( r_{\text{nuc}} \) is the volume fraction of gas core of \( 5 \times 10^{-4} \); \( R_{\text{nuc}} \) is a bubble radius of \( 1 \times 10^{-6} \) m. \( p_{\text{v}} \) is the gasification pressure, Pa. According to the experimental data\(^{[13]}\), these parameters of the cavitation model were discussed and verified. It is successfully applied to the numerical simulation of cavitation flow in pump.\(^{[14]}\)

3. Results and analysis

3.1. Cavitation performance curves in different flow rates

In this study, in order to comprehensively study the cavitation performance of the water jet propulsion pump, the CFX numerical simulation method was used to solve the problem, and the curves of the head of the water jet propulsion pump with the NPSH\(_{a}\) under different flow rates was fitted. As shown in Figure 6. Obtain the critical cavitation condition point corresponding to a head drop of 3\% under different flow rates. At this time, the NPSH\(_{a}\) is equal to the NPSH\(_{c}\) of the pump. The position of the critical operating point is marked in red in the figure. In general, the head curves corresponding to different flow rates increase as the flow rate decreases. The head curves at different flow rates decrease with the decrease of NPSH\(_{a}\), and the decrease trend of the head curves near the critical NPSH becomes gentler with the decrease of flow rate. This is due to the change of flow rate which causes the change of liquid flow angle, which leads to the change of angle of attack between the blades. This will further lead to deflow on the pressure surface or suction surface of the blade, thereby intensifying the development of cavitation.

Figure 7 is the fitting curves of the head of the water jet propulsion pump with NPSH\(_{a}\) at different flow rates according to the test data\(^{[7]}\). Because the initial value at which the initial inlet pressure begins to drop is different between the numerical simulation and the test, there are some differences in the range of the abscissa. Comparing the cavitation performance curves of the two at different flow rates, it is not difficult to find from the figure that the overall change trends of the two figures are almost the same, and the head curves increase with the decrease of the flow rate. And all of them are under high flow rate, and the downward trend of the head curve near the critical NPSH is steeper. The error is not very large from the point where cavitation does not occur to the point where cavitation begins to critical cavitation. After calculation, the relative error between the numerically calculated head and the experimental head under the same NPSH\(_{a}\) is about 10\%. With the development of cavitation, the relative error between the numerically calculated head value and the experimental value also increases.
The numerical calculation results and experimental comparison of the critical NPSH curves of the water jet propulsion pump at different flow rates are shown in Figure 8. The position of the critical cavitation condition point is different under different flow rates. It can be seen from Fig. 8 that whether it is the NPSH$_c$ curve obtained by numerical calculation or experiment, as the flow rate decreases from 0.46m$^3$/s to 0.37m$^3$/s, the critical NPSH first decreases and then increases. And it reaches the minimum value near 0.4m$^3$/s, the minimum value obtained by numerical simulation is about 6.31m, and the minimum value obtained by experiment is 7.106m. The error between the numerical calculation value and the experimental value under the flow rate of 0.46m$^3$/s is very small, only 0.69%. As the flow rate decreases, the error between the two tends to increase.
3.2 Efficiency variation curves with $NPSH_a$ in different flow rates

Fig. 9 is the performance curves of efficiency decreasing with $NPSH_a$ at different flow rates obtained through numerical simulation. Fig. 10 is a performance curves obtained through experiments, in which efficiency decreases with $NPSH_a$ at different flow rates. It can be seen from the two figures that in the stage from the point where cavitation does not occur to the point where cavitation is born, the efficiency corresponding to the overall flow rate of $0.44\text{m}^3/\text{s}$ is the highest, followed by $0.46\text{m}^3/\text{s}$, and then with When the flow rate decreases from $0.44\text{m}^3/\text{s}$, the efficiency performance curves of the jet pump also gradually decreases. As the flow rate decreases, the decay rate becomes slower after the point of critical cavitation. When the flow rate is $0.37\text{m}^3/\text{s}$, the efficiency obtained by numerical simulation at the point of critical cavitation is $79.77\%$, and the efficiency obtained by experiment is about $78.13\%$. When the flow rate is $0.4\text{m}^3/\text{s}$, the efficiency obtained by numerical simulation at the point of critical cavitation is $81.95\%$, and the efficiency obtained by experiment is about $80.27\%$. When the flow rate is $0.42\text{m}^3/\text{s}$, the efficiency at the critical cavitation point obtained by numerical simulation is $82.88\%$. The efficiency obtained through the test is about $80.63\%$. When the flow rate is $0.44\text{m}^3/\text{s}$, the efficiency obtained by numerical simulation at the point of critical cavitation is $83.41\%$, and the efficiency obtained by experiment is about $81.98\%$. When the flow rate is $0.46\text{m}^3/\text{s}$, the efficiency obtained by numerical simulation at the point of critical cavitation is $83.5\%$, and the efficiency obtained by experiment is about $81.35\%$. Numerical simulation shows that the efficiency of the critical cavitation point at each flow rate shows a decreasing trend as the flow rate decreases. Numerical simulation shows that the efficiency of the critical cavitation point at each flow rate also shows a decreasing trend as the flow rate decreases. When the flow rate decreases from $0.46\text{m}^3/\text{s}$ to $0.44\text{m}^3/\text{s}$, the efficiency of the critical cavitation point There will be a slight upward trend and reach a maximum around $0.44\text{m}^3/\text{s}$.
3.3 The distribution of cavitation region of the impeller suction surface

Through the numerical calculation and post-processing of the cavitation model, the cavitation area distribution diagram of the impeller blade suction surface corresponding to the critical cavitation state under five different flow rates is obtained, as shown in Figure 9. It can be seen from Figure 9 that as the flow rate decreases, the cavitation area on the suction surface of the impeller blade under the corresponding critical cavitation condition gradually expands. When \( NPSH_c = 7.44 \text{ m} \); \( Q = 0.46 \text{ m}^3/\text{s} \), the cavitation area on the suction surface of the impeller blade is the smallest, and when \( NPSH_c = 6.40 \text{ m} \); \( Q = 0.37 \text{ m}^3/\text{s} \), the cavitation area on the suction surface of the impeller blade is the largest. And as the flow rate decreases, the cavitation area on the suction surface of the impeller blade under the corresponding critical cavitation condition is getting closer to the position of the hub and the leading
edge of the blade. In the previous research, it was found that under the working condition of 0.46m$^3$/s, with the decrease of NPSH$_c$, the cavitation area on the suction surface of the impeller blade gradually increased. When the critical cavitation condition point is exceeded, the cavitation volume fraction is very high near the trailing edge of the suction surface of the impeller blade. Because as the degree of cavitation develops, a strong vortex is formed at the trailing edge of the blade, causing the cavitation to fall off from the suction surface [12].

![Figure 9](image_url)  
(a) NPSH$_c$ = 7.44m; Q = 0.46m$^3$/s  
(b) NPSH$_c$ = 6.88m; Q = 0.44m$^3$/s  
(c) NPSH$_c$ = 6.46m; Q = 0.42m$^3$/s  
(d) NPSH$_c$ = 6.31m; Q = 0.40m$^3$/s  
(e) NPSH$_c$ = 6.40m; Q = 0.37m$^3$/s

**Figure 9** The distribution of cavitation region of the impeller suction surface at critical cavitation condition in different flow rates

3. Conclusions
In this paper, the cavitation vortex structure of the water jet propulsion pump and its influence on the performance of the jet pump are studied through experiments and numerical simulations. The main work and conclusions are as follows:

(1) We studied the cavitation performance of the model pump at different flow rates, and captured the critical cavitation point where the head drops by 3% through numerical simulation. The numerical simulation results are compared with the experimental test results. By comparing the cavitation performance curves between the two, it is found that the overall trend is consistent. As the flow rate decreases, the cavitation performance curve decreases more gently after the critical cavitation point. And the reason of head curve drop is analyzed. It is found that the numerical minimum value and the experimental minimum value of the critical NPSH of the water jet propulsion pump are obtained at Q = 0.4m$^3$/s under different flow rates. As the flow rate decreases, the errors of the two tend to increase.

(2) We studied the changing law of the efficiency with the development of cavitation degree under different flow rates. Similarly, the numerical calculation results are compared with the
experimental results. It is found that the efficiency is the highest when the flow rate is 0.44 m$^3$/s, followed by the flow rate when the flow rate is 0.46 m$^3$/s. As the flow rate decreases from 0.44 m$^3$/s, the efficiency also decreases. As the flow rate decreases, the decay rate of the efficiency curve becomes slower after the critical cavitation condition occurs.

(3) We studied the distribution of the cavitation area on the impeller suction surface under critical cavitation conditions at different flow rates. It is found that as the flow rate decreases, the distribution area of the cavitation zone on the suction surface of the impeller blade under the corresponding critical cavitation condition tends to increase. As the flow rate decreases, the cavitation area on the suction surface of the impeller blade under the corresponding critical cavitation condition is getting closer to the hub and the leading edge of the blade.

References