

PAPER • OPEN ACCESS

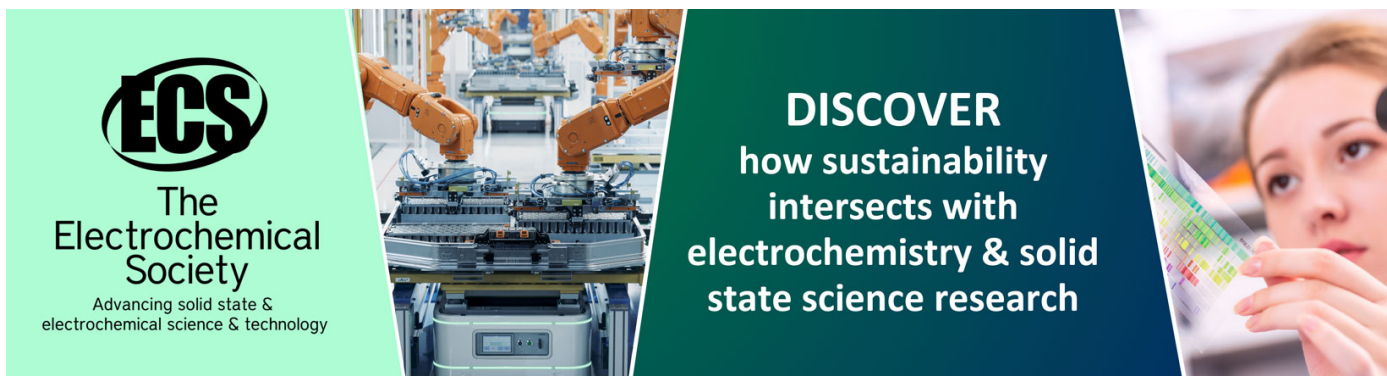
## Numerical Simulation of Cavitation Characteristics for Pump-jet Propeller


To cite this article: Yao Shi *et al* 2015 *J. Phys.: Conf. Ser.* **640** 012035

View the [article online](#) for updates and enhancements.

You may also like

- [Tip-leakage cavitation in the clearance of a 2D hydrofoil with fillets: high-speed visualization and PIV/PTV measurements](#)  
Ivan I Zapryagaev, Mikhail V Timoshevskiy and Konstantin S Pervunin
- [Hydrodynamic cavitation in microfluidic devices with roughened surfaces](#)  
Morteza Ghorbani, Abdolali K Sadaghiani, L Guillermo Villanueva *et al.*
- [Numerical simulation on the cavitation of waterjet propulsion pump](#)  
C Z Xia, L Cheng, Y N Shang *et al.*





The  
Electrochemical  
Society

Advancing solid state &  
electrochemical science & technology

DISCOVER  
how sustainability  
intersects with  
electrochemistry & solid  
state science research

# Numerical Simulation of Cavitation Characteristics for Pump-jet Propeller

**Yao Shi, Guang Pan, Qiaogao Huang, and Xiaoxu Du**

School of Marine Science and Technology, Northwestern Polytechnical University,  
China

E-mail: sy880408@163.com

**Abstract.** With  $k-\varepsilon$  turbulent model, non-cavitating performance of a pump-jet propeller was obtained by calculating RANS equations. The comparison between calculation results and experiment data shown that the numerical model and method was reliable. The cavitating hydrodynamic performance of it was calculated and analyzed with mixture homogeneous flow cavitation model based on Rayleigh-Plesset equations and sliding mesh. The effects of different inlet velocity ratio, cavitation number and flow velocity on cavitation characteristics of pump-jet were studied. When the cavitation occurred on the blades, the propeller thrust and torque decreased significantly, thereby causing open water efficiency reduced 15%. For the same cavitation number, as the inlet velocity ratio decreased, the pump-jet propeller blade cavitation phenomenon was more obvious. While for the same ratio, the smaller the number of cavitation, cavitation phenomenon was more remarkable. The more significant was that while the cavitation number was greater than a certain value, the blade cavitation phenomenon disappeared.

## 1. Introduction

Pump-jet propeller is a new type of underwater propulsion system, its main characteristic is to use a single rotor to boost and the application of deceleration ducted which could make the propeller working in a low velocity environment and improve the propeller cavitation performance. However, with the development of the submarine body to the direction of large-scale and high speed, it makes the increased probability of cavitation of jet propulsion pump. Cavitation would not only reduce the propulsion performance, produce cavitation erosion, cause the vibration of underwater navigation body, but also produce the enough cavitation noise to expose itself. Therefore, understanding and mastering the accurate forecasting technology of pump-jet cavitation becomes more and more important.

At present, the study of cavitating flows has two main methods-experiment and numerical analysis. Experimental research can accurately reflect the features of flow field variation, but it needs a lot of manpower, financial resource, and some complex conditions can't be studied; however, numerical analysis has advantages of full information, the unlimited objective condition, the cavitating flows analysis plays an increasingly important role.

Up till now, there are less public reports about the pump-jet cavitating flows, mainly concentrated in the aspect of propeller and hydraulic machinery. Liu Zhanyi<sup>[1]</sup> forecast the hydrodynamic performance of spray pump propeller based on CFD technology, the simulation results were similar with the test results, but it didn't involve the phenomenon of cavitating flows. Shi WeiDong<sup>[2]</sup> studied the



cavitation characteristics of axial flow pump under different tip clearance by using CFX software and the results agreed well with the test results. Yang Qiongfang<sup>[3]</sup> obtained E779A propeller cavitation by using Sauer cavitation model and modified shear stress transport turbulence model. Ji Bin<sup>[4-5]</sup> simulated the unsteady cavitation of propeller flow of three dimensional airfoils and propeller by using equilibrium flow cavitation model and turbulence modification model based on filter, the length and velocity field of cavitating flows and pressure coefficient agreed well with the experimental values.

Due to the cavitating flows involved quality transmission, pump-jet flow field which had the characteristic of unsteady and high Reynolds number, the numerical simulation of pump-jet cavitating flow field is difficult. Based on theory of viscous flow and multiphase flow, this thesis solved the pump-jet three-dimensional full channel cavitating steady flow field by using homogeneous multiphase model, turbulent model and sliding grid technique; studied the pump-jet performance under different speed ratio, cavitation number and inflow speed, the relevant experimental results matched well with the paper's results, and what's more, it provides a certain technical support for research and design of pump-jet in the future.

## 2. Numerical Computation Method

### 2.1. Flow Filed Basic Control Equation

Mixing multiphase flow model to solve the navier-stokes equation which based on viscous flow theory value was the major method. Gas-liquid mixture two-phase flow model had considered the interaction and alternate-slip velocity of flow field in the process of phase change, and lead into the cavitation model by phase change rate. Assuming that the density of gas and liquid two-phase flow of flow field were constant, treated the gas- liquid two-phase flow as unified mixed flow with a changeable density, its density is a function of a gas phase volume fraction. On the one hand, the navier-stokes equation was used to solve the physical quantities of unified mixed flow, the physical quantities could be expressed in average volume of gas phase and liquid phase. On the other hand, the introduction of gas phase transmission equation could solve the gas phase volume fraction. Looked the mixed flow as a uniform fluid, the control equation was<sup>[7]</sup>:

(1) Mixed phase continuity equation:

$$\begin{cases} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \\ \frac{\partial}{\partial t}(\rho f) + \nabla \cdot (\rho f \mathbf{u}) = m_{fg} \\ \frac{1}{\rho} = \frac{f}{\rho_p} + \frac{1-f}{\rho_q} \end{cases} \quad (1)$$

(2) Momentum equation of mixed phase flow:

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \cdot \mathbf{u}) = -\nabla p + \frac{1}{3} \nabla [(\mu + \mu_t) \nabla \cdot \mathbf{u}] + \nabla \cdot [(\mu + \mu_t) \nabla \mathbf{u}]$$

In equation:  $\rho$  is density of mixed phase flow,  $\rho_p$  is density of cavity phase,  $\rho_q$  is density of flow phase,  $f$  is mass component of cavity phase;  $\mathbf{u}$  is velocity vector of mixed phase,  $m_{fg}$  is interphase transport quality when cavity is born and collapsed,  $\mu_t$  is Mixed viscous coefficient.

### 2.2. Cavitation Model

The mass transfers between cavity and water lead to the generation and condensation of cavitation, cavity generation and condensation rate is controlled by the following Rayleigh-Plesset equation<sup>[8-9]</sup>:

$$R_B \frac{d^2 R_B}{dt^2} + \frac{3}{2} \left( \frac{dR_B}{dt} \right)^2 + \frac{2\sigma}{\rho_q R_B} = \frac{p_v - p}{\rho_q}$$

$$\rho = \alpha_p \rho_p + \alpha_q \rho_q$$

$$m_{fg} = 4F \rho_p \pi R_B^2 \left( \frac{2|p_v - p|}{3\rho_q} \right)^{1/2} \text{sgn}(p_v - p)$$

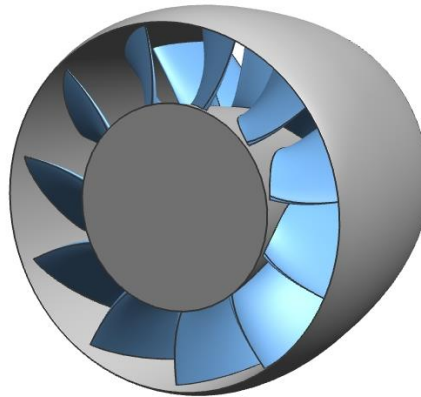
In equation:  $\alpha_p$  is volume fraction of cavity phase,  $\alpha_q$  is volume fraction of flow phase,  $R_B$  is cavity radius,  $p_v$  is pressure inside the cavity,  $p$  is pressure around the cavity,  $\sigma$  is surface tension factor,  $F$  is experience coefficient of cavity generation and condensation.

In this paper, a standard  $k-\varepsilon$  turbulence model was adopted to close the equation, the model is widely used in engineer and suitable for solving high Reynolds number flow.

### 3. Computation Model and Grid

#### 3.1 Computation Model

In this paper, the simulation model of pump-jet propeller is shown in figure 1. The propeller has 11 pieces of rotor blade, 9 pieces of stator vane, the rotors are in the former of the model, the stators are in the behind, rotors rotate clockwise (from the front looking towards back). Considering the clearance flow between the rotors and the inner surface of propeller, the minimum clearance of tip clearance flow is 1mm.



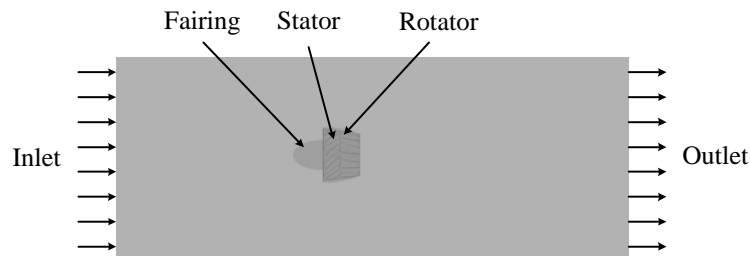
**Figure 1.** Model of pump-jet propeller

In order to better simulate the flow in the pump-jet, the front inner-axis of pump was designed as a half ellipsoid fair water cap, short axis connected smoothly with the front inner-axis of pump. At the same time, to improve the accuracy of numerical simulation, this paper adopted full channel model for numerical simulation.

According to the nature of the physical problems in this paper, the computational domains could be divided into three parts: rotor domain, stator domain and flow field calculation domain. Among them, the rotor domain was rotation calculation domain, the others were static domains. The paper simulated the interaction between rotor and static domains by the sliding grid technology.

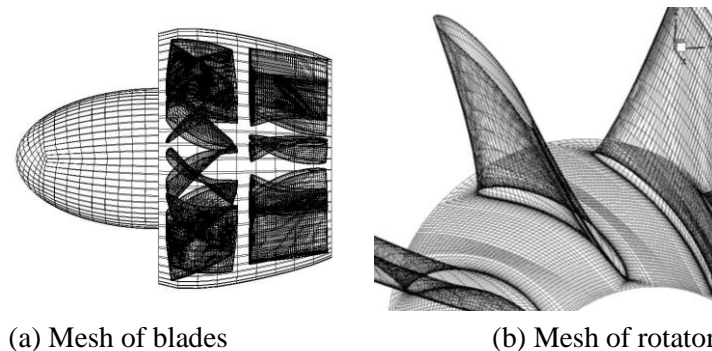
#### 3.2 Computational Domain and Mesh Generation

The performance test of pump-jet is to measure the open water characteristic, in order to make the numerical simulation in conformity with the test condition; this paper puts the pump-jet model into a cylindrical computational domain of same coaxial line with pump-jet, just shown in figure 2. Set  $D$  is the diameter of the pump-jet rotor, the distances between the pump model and the entrance boundary of the computational domain and the exit boundary are respectively  $4D$  and  $6D$ , the diameter is  $10D$ .



**Figure 2.** Computational domain and boundary

Use the blocking grid technology to generate high quality structured grid. In the surface of pump vane, use the O type mesh for encryption to accurately simulate the boundary layer. The whole calculation domain can be divided into a total of about 1.5 million cells, the rotor field has 748902 units, the stator domain has 457254 units. Figure 3 (a) and (b) respectively shows the mesh of blades and the mesh of rotator.



(a) Mesh of blades

(b) Mesh of rotator

**Figure 3.** Mesh of computational domain

### 3.3 Boundary Condition

To complete the numerical simulation of pump-jet propeller cavitation flow field, we need to set up the reasonable boundary conditions, the specific settings are as follows: the inlet boundary is speed entrance, the turbulence intensity is 5% of default parameter, the cavity volume fraction of entrance is 0, the volume fraction of liquid is 1; the reference pressure is 0Pa, the liquid temperature is 25 °C, the saturated vapor pressure is 3574pa, the average diameter of cavity is  $2 \times 10^{-6}$  m; the outlet boundary set as the export pressure, the cavitation number  $\sigma$  is controlled by pressure:

$$\sigma = (p_{out} - p_v) (0.5 \rho v_\infty^2)^{-1/2}$$

In equation:  $p_{out}$  is export pressure,  $p_v$  is saturated vapor pressure. Set pipe wall as no-slip wall, the area near wall uses the reinforced wall function model. The cylinder boundary of computational domain meet the requirement that wall can freely slide. The coupled analysis of dynamic-static effect between rotation domain and stationary domain can be realized by setting the Frozen Rotor interface implementation.

### 4. Research of Pump-jet Cavitation Characteristic

For the convenience of the analysis of calculation results, define the related physical quantities as dimensionless variables which shown in table 1.

**Table 1.** Define of dimensionless variable

Variable	Define
inlet speed ratio	$J = v(nD)^{-1/2}$
rotor thrust coefficient	$K_T = T_r (\rho n^2 D^4)^{-1/2}$

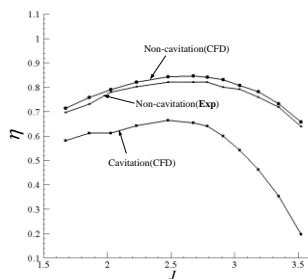
rotor torque coefficient	$K_{Q_r} = Q_r (\rho n^2 D^5)^{-1/2}$
stator duct system thrust coefficient	$K_{T_s} = T_s (\rho n^2 D^4)^{-1/2}$
stator duct system torque coefficient	$K_{Q_s} = Q_s (\rho n^2 D^5)^{-1/2}$
total thrust coefficient	$K_T = K_{T_r} + K_{T_s}$
total torque coefficient	$K_Q = K_{Q_r}$
open water efficiency	$\eta = JK_T (2\pi K_Q)^{-1/2}$

In table,  $v$  is the far field flow velocity,  $n$  is the rotor speed,  $D$  is the rotor diameter,  $\rho$  is the fluid density,  $T_r$  and  $Q_r$  are respectively the thrust and torque of rotor,  $T_s$  and  $Q_s$  are respectively the thrust and torque of stator duct system.

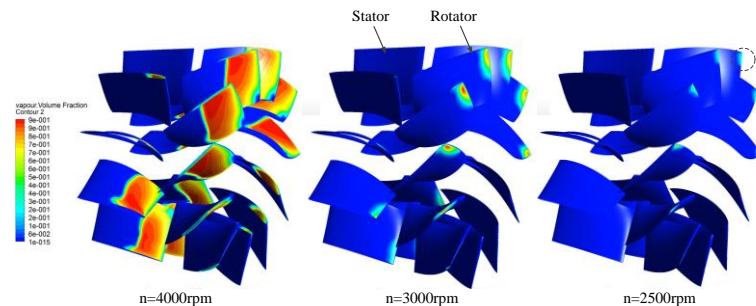
This paper calculated the open water performance of pump-jet propeller with different cavitation numbers, different inflow velocity ratio and different inflow velocity, and analyzed the influence law of open water performance on the above factors.

#### 4.1 Calculation Result Analysis with Different Inlet Velocity Ratio

From the velocity ratio definition, control it by changing the speed of pump-jet and inlet speed. Fig.4 is the graph of cavitation and non-cavitation open water efficiency at different inlet velocity ratio of 1.6730 ~ 3.5222,  $\sigma = 1.4721$  and  $v = 25.72 \text{ m/s}$ .



**Figure 4.** Efficiency at different inlet velocity ratio



**Figure 5.** Cavitation of pump-jet propeller blades at different inlet velocity ratio

As can be seen from figure 4, with the increase of inlet speed ratio, the propeller cavitation and non-cavitation open water efficiency showed a trend of increasing first and then decreasing, and reached the maximum when the velocity ratio was 2.4786. When the propeller cavitation didn't happen, the propeller efficiency was above 70% and had a good agreement with the test results<sup>[11]</sup>. The test value was slightly lower than the numerical simulation results, this might be caused that the simulation conditions were not in accordance with the test conditions. The open water efficiency of propeller cavitation is lower 15% above than the efficiency of propeller non-cavitation. Thus, improving the resistance to cavitation performance of propeller blades in the propeller design process is important. In addition, the results of numerical simulation show that the torque coefficient of stator and rotor system is almost the same, the maximum error is only 1.96%, it shows that the balance feature of pump-jet is very good.

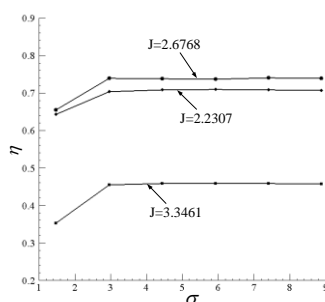
Figure 5 is the cavitation cloud of the pump-jet propeller blade under three kinds of typical working conditions. In the figure, the red area is air, the blue area is water (the same with below). Can be seen from the diagram, at low speed, the cavitation occurs only in the root of the blade tip; as the speed

increases gradually, the cavitation areas gradually expand from the blade tip root to the blade suction surface, when  $n = 4000\text{rpm}$ , almost half of the suction surface happens cavitation.

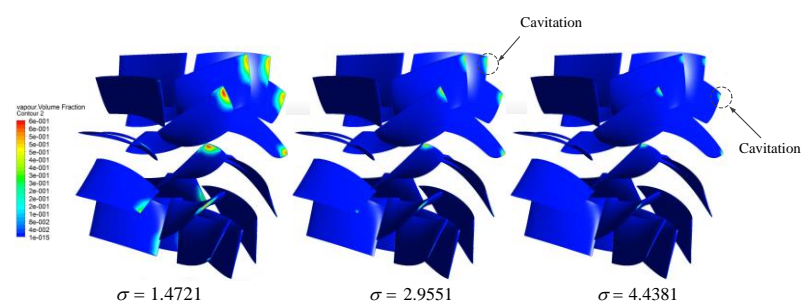
After the cavitation, the part of propeller blade worked in the gas, it made the thrust and torque reduced, so was the open water efficiency of propeller. The open water efficiency is a comprehensive evaluation of ratio, thrust and torque coefficient. Due to the extent of thrust and torque, the open water efficiency showed the trend of increasing and later reducing. It is also well explained the change laws of propeller open water efficiency in Fig.4.

#### 4.2 Calculation Result Analysis with Different Cavitation Number

From the cavitation number definition, we can change the inlet flow speed and outlet pressure to control the cavitation number. In Fig.6, there are the pump-jet propeller open water efficiency curves when the flow velocity is  $v = 25.72\text{m/s}$ , the cavitation number- $\sigma$  is  $1.4721 \sim 8.8869$ .



**Figure 6.** Efficiency at different cavitation number



**Figure 7.** Cavitation of pump-jet propeller blades at different cavitation number

As can be seen from Figure 6, with the increase of cavitation number, the propeller open water efficiency increased firstly and then kept a basic invariable feature under different inlet speed ratio. When the cavitation number was 1.4721, the pump-jet efficiency was minimum; when the cavitation number was greater than 3, with the increase of cavitation number, the propeller open water efficiency remained unchanged. Analyzed its reason, the smaller cavitation number meant the outlet pressure was low, at this point, the rotation areas of the pump-jet propulsion pruned to cavitation; the cavitation number increased, so was the outlet pressure, it made the difference between the evaporation pressure with the local fluid pressure was bigger, so the cavitation would disappear, then the pump-jet propeller open water efficiency remained unchanged. Actual calculation results show that when the cavitation number is bigger than 5.921, the propeller blade cavitation phenomenon exists not any more.

Figure 7 is the cavitation cloud of the surface of the propeller blade with three typical cavitation numbers. From the diagram, the cavitation number is bigger, the propeller cavitation area on the surface of the blade is smaller. When the cavitation number is 4.4381, the cavitation phenomenon occurs only in the tip of the blade. Therefore, with the loss of the cavitation number, the cavitation happens near the clearance area at first, and gradually increases, at the same time, it expands along the radial to the blade suction surface.

#### 5. Conclusion

In this paper, a pump-jet propeller as the research object, its performance has been studied with different inlet speed ratio, cavitation number and inflow velocity, and the grid independence has also been checked, the following conclusions are obtained by analysis:

- (1) The numerical simulation of cavitation flow field can better predict the area where the cavitation occurs, and the cavitation phenomenon occurs in the root of the blade tip at first and gradually develops to the suction side and it happens in the suction surface before the pressure surface.
- (2) Under the same cavitation number, with the increase of inlet speed ratio, the propeller open water efficiency shows a trend of increasing firstly and decreasing later, and the inlet speed ratio is smaller, the cavitation phenomenon is more obvious.



(3) Under the same inlet speed ratio, with the increase of cavitation number, the propeller open water efficiency increases firstly and then keeps a basic invariable feature. The cavitation number is smaller, the cavitation phenomenon is more obvious; when the cavitation number is greater than a specific value, the blade cavitation disappears.

(4) Under the same rotation speed, with the increase of inflow velocity, the propeller open water efficiency shows a trend of monotonically decreasing.

### Acknowledgment

This study was undertaken as part of a project supported by Ministry of education "program for New Century Excellent talents "(NCET-09-0074) , National Natural Science Foundation of China Projects(51279165), and National Defense Basic Research Project(B2720132014), China, which is gratefully acknowledged by the authors.

### 6. References:

- [1] LIU Zhanyi, SONG Baowei, HUANG Qiaogao, et al. Applying CFD Technique to Calculating Successfully Hydrodynamic Performance of Water Jet Pump. 2010 *Journal of Northwestern Polytechnical University*, 28(5):724-729.
- [2] SHI Weidong, LI Tongtong, ZHANG Desheng, et al. Effect of tip clearance on the cavitation and flow field of axial flow pumps. 2013 *Journal of Huazhong University*, 41(4):21-25.
- [3] YANG Qiongfang, WANG Yongsheng, ZHANG Zhihong. Assessment of the Improved Cavitation Model and Modified Turbulence Model for Ship Propeller Cavitation Simulation. 2012 *Journal of Mechanical Engineering*, 48(9):178-185.
- [4] JI Bin, LUO Xianwu, WU Yulin, et al. Numerical analysis of unsteady cavitating turbulent flow and shedding horse-shoe vortex structure around a twisted hydrofoil. 2013 *International Journal of Multiphase Flow* 51:33-43.
- [5] JI Bin, LUO Xianwu, PENG Xiaoxing. Unsteady Numerical Simulation of Cavitating Turbulent Flow Around a Highly Skewed Model Marine Propeller. 2011 *Journal of Fluids Engineering*, 133, 1-8.
- [6] JI Bin, LUO Xianwu, WU Yulin, et al. Partially-Averaged Navier-Stokes method with modified k- $\epsilon$  model for cavitating flow around a marine propeller in a non-uniform wake. 2012 *International Journal of Heat and Mass Transfer*, 55:6582-6588.
- [7] Wang Xianfu. Cavitating and Supercavitating Flows Theory and Applications[M]. Beijing: National Defence Industry Press, 2009.
- [8] Brennen C E. Cavitation and Bubble Dynamics[M] . Oxford: Oxford University Press, 1994: 50-79.
- [9] Johann F G. Centrifugal Pumps[M]. New York: Springer Berlin Heidelberg, 2008:257-333.
- [10] The Test Report about Hydrodynamic Characteristics of Pump-jet Propeller [R]. CSSRC, 2009. (in Chinese)