Numerical Simulation of Internal Flow Field of Self-Designed Centrifugal Pump

Xin Wang 1, Jun Zhang 1* and Zongshun Li 2

1 Department of Marine Equipment and Mechanical Engineering, Jimei University, Xiamen, China, 2 Fujian Sea Knight Pump Co., Ltd., Zhangzhou, China.
E-mail: bull0202@sina.com.

Abstract. The two self-designed of centrifugal pump with small vane and centrifugal pump without small vane were simulated numerically to select a centrifugal pump with higher efficiency. The internal flow characteristics of the centrifugal pump was simulated by using Reynolds time-averaged N-S equation and RNG turbulence model to obtain pressure and velocity distribution and the cavitation characteristics were simulated by using SST turbulence model and Schnerr-Sauer cavitation model to obtain the gas volume fraction distribution. The results show that at the same flow rate, the change of velocity in the pump cavity of centrifugal pump with vane is smoother and the gas volume is less, but the back-flow is aggravated near the small vane, especially when interacting with the tongue, and a large amount of gas is generated at the suction surface of the small vane. In addition, the efficiency of centrifugal pump without vane is higher than that of centrifugal pump with vane, which provides a basis for the structural optimization of centrifugal pump.

Keywords. Centrifugal pump, numerical simulation, internal flow field, cavitation.

1. Introduction
Centrifugal pump, as a widely used fluid machinery, plays a role in agriculture, water conservancy, industry and other fields. In addition to meeting the basic requirements of flow rate and head, the most important thing in the design of a pump is the high efficiency, which is closely related to the energy consumption and affects the production cost and product competitiveness [1]. The impeller is the core component of the centrifugal pump, and its performance affects the efficiency of the whole pump. Centrifugal pumps not only have a complex internal flow structure, with the computational domain divided into two sub-regions: the stator and the rotor, but also the flow is of a three-dimensional, viscous, non-constant form [1, 2]. The complex operating environment requires increasingly high performance of centrifugal pumps, and the limitations of traditional empirical methods make it particularly important to apply numerical simulations to predict the performance of centrifugal pumps [3, 4]. Siddiqi [5] used a multi-objective genetic algorithm (MOGA) to optimize a seven-vane impeller, and the results showed that the optimized impeller resulted in a more uniform pressure and velocity distribution on the meridional plane of the impeller and higher efficiency. Young-Joon An [6] used CFX to numerically simulate the three-dimensional incompressible flow of a centrifugal pump, and the impeller with a smooth curve along the front cover showed good performance. The inlet and outlet pressure difference decreases with increasing flow rate under the influence of flow variation; high head can be obtained at low flow rates. For hydraulic machinery, in addition to the basic performance parameters of head and efficiency, cavitation instability is also a crucial issue and is quite

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

Published under licence by IOP Publishing Ltd
harmful to its operational safety [7]. Yang [8] studied the effect of cavitation on centrifugal pumps by changing the inlet pressure, and the results showed that when the inlet pressure is less than the saturated vapor pressure of the transported liquid, the liquid vaporizes and gas is generated, and efficiency decreases due to cavitation in the pump. With the increase of cavitation, the efficiency decreases obviously. Wei [9] based on the SST turbulence model, the pressure pulsation characteristics of centrifugal pumps at four typical flow rates were Numerical simulations were performed based on the SST turbulence model. The results showed that uneven flow distribution at the impeller inlet would produce fluctuations at the leading edge of the impeller and large size return vortices in the flow channel, leading to low pressure areas at the impeller and the spacer tongue, which may produce vaporization and thus affect cavitation. Deng [10] used the SST turbulence model and the Zwart-Gerber-Belamri cavitation model to numerically simulate the full flow channel of a centrifugal pump and obtained that the gas first appears in the suction surface at the leading edge of the vanes, as well as cavitation exacerbates the instability of the flow and affects the centrifugal pump's ability to do work.

Impeller and volute of the water impact will produce vortex, is one of the reasons for the decline in pump efficiency, but there is less research on how to change the impeller structure affects the reflux zone. In order to reduce this loss, this paper designed two kinds of impellers with and without small vane. The large vane of the two impellers adopts the forward vane type, and the first impeller adds the small vane of the same curvature on this structure, the purpose is to compare and analyze the influence of the two vane forms on the pressure and velocity distribution of the centrifugal pump, which can provide some reference basis for improving the efficiency of the centrifugal pump.

2. Computational Model

2.1. Physical Model

According to the given structural parameters, two different three-dimensional models of the impeller were designed as shown in figure 1. In order to balance part of the axial thrust on the impeller, the impeller was designed with back vanes. The physical model of the centrifugal pump basin was obtained by flow channel extraction using SCDM software. Due to the complex structure of the centrifugal pump model, the mesh was divided with unstructured mesh in ICEM as shown in figure 2. The number of grid cells of centrifugal pump A impeller is 296527, the number of grid cells of centrifugal pump B impeller is 295601, the number of grid cells of worm housing runner is 579347, the total number of grid cells of centrifugal pump A is 1275914, and the total number of grid cells of centrifugal pump B is 1274948.

![Figure 1. Impeller three-dimensional model.](image-url)
2.2. Numerical Simulation

2.2.1. Control Equations. The MRF model is applied to the internal flow of this centrifugal pump to convert the internal flow field non-constant problem into a constant problem to solve. The equations of motion in the rotating coordinate system can be expressed as [1]:

\[
\frac{\partial \vec{v}}{\partial t} + \vec{v} \cdot \nabla \vec{v} = -\frac{1}{\rho} \nabla p + \frac{1}{\rho} \nabla \cdot \tau + \vec{f}
\]

where: \(\vec{v}\) -relative velocity vector, \(\vec{\Omega}\) -the rotational angular velocity of the rotating coordinate system, \(\vec{r}\) -the position vector of the point in the rotating coordinate system, \(\tau\) -viscous stress tensor, \(\rho\) -fluid density, \(\vec{f}\) -force per unit mass, \(2\vec{\Omega} \times \vec{v}\) -Gothic acceleration, \(\vec{\Omega} \times \vec{\Omega} \times \vec{r}\) -centrifugal acceleration.

For this centrifugal pump the full flow channel three-dimensional constant numerical calculation, the medium is water and the N-S equation for the time homogenization can be expressed as [11]:

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = f_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial u_i}{\partial x_j} - u_i' u_j' \right)
\]

where: \(u_i', u_j'\) -the pulsation value of the instantaneous velocity

For constant, incompressible flow, neglecting the bulk force, the above equation can be rewritten as:

\[
\frac{\partial u_i}{\partial t} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial u_i}{\partial x_j} - u_i' u_j' \right)
\]

2.2.2. Turbulence Model. Rotational effects are considered in the model, so the RNG turbulence model is used to improve the accuracy of the rotational flow calculation. The RNG turbulence model expression is [12]:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) - G_k + \rho \varepsilon
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_e \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\varepsilon}{k} \left( C_{1e} G_k - C_{2e} \rho \varepsilon \right)
\]
which: \( \mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \), \( \mu_{eff} = \mu + \mu_t \), \( C_{te} = C_{te} - \eta (1 - \eta / \eta_0) \), \( \eta = \sqrt{2E_y / \varepsilon} \), \( E_y = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \).

where: \( \alpha_k = \alpha_e = 1.39 \), \( \mu_{eff} \) - viscosity, \( N \times s / m^2 \), \( C_\mu = 0.0845 \), \( E_y \) - time-averaged strain rate, \( C_{te} \) - empirical constant correction term, \( \eta_0 = 4.377 \), \( C_{te} = 1.42 \), \( C_{te} = 1.68 \), \( \beta = 0.012 \).

2.2.3. Cavitation Model. In this paper, we use the Schnerr-Sauer cavitation model, whose basic equation can be expressed as [13]:

\[
\dot{m} = \begin{cases} 
\dot{m}^+ = C_{eva} \frac{3\alpha_e (1-\alpha_e) \rho_i \rho_v}{R_B \rho_m} \left( \frac{2}{3} P_v \right) \frac{P-P_v}{\rho_1} & P < P_v \\
\dot{m}^- = C_{cond} \frac{3\alpha_e (1-\alpha_e) \rho_i \rho_v}{R_B \rho_m} \left( \frac{2}{3} P-P_v \right) \frac{P-P_v}{\rho_1} & P \geq P_v
\end{cases}
\]  

(6)

where: \( \dot{m} \) - mass transfer source term, when \( \dot{m} > 0 \), indicates evaporation process, \( \dot{m} < 0 \), indicates the condensation process, \( \rho_m \), \( \rho_i \) and \( \rho_v \) - mixed phase density, liquid phase density and gas phase density, \( \alpha_e \) - volume fraction of gas phase, \( R_B \) - radius of bubble, \( P_B \) - surface pressure of bubbles, \( P \) - environmental pressure.

2.3. Boundary Conditions and Numerical Simulation Methods

The fluid inside the centrifugal pump has a complex flow form, and three pairs of cross interfaces are set at the junction of the inlet section and the impeller, the impeller and the worm gear flow path, and the worm gear outlet and the outlet section, which are used to exchange fluid parameters and ensure the continuity of the cross interfaces. The design flow rate of this centrifugal pump is known, so the inlet boundary condition of this centrifugal pump is set as velocity inlet, and the outlet boundary condition is free outflow. The medium is water, and gravity needs to be considered. The wall boundary condition is set to wall without slip condition, and the flow near the solid wall is determined by the standard wall function. The solver uses SIMPLE separated solver. The viscous term automatically adopts the second-order accuracy discrete format, and the discrete format of both turbulent kinetic energy and turbulent dissipation rate equations adopt the first-order windward format. If the residuals increase, the default values of the relaxation factors for pressure, momentum, turbulent kinetic energy and turbulent dissipation rate can be adjusted downward.

2.4. Fluid Physical Parameters and Pump Design Working Conditions Parameters

The transfer medium is water, the pump design working parameters and fluid physical parameters are shown in table 1.

**Table 1.** Pump design working parameters and fluid physical parameters.

| Design Parameters | Numerical value |
|-------------------|----------------|
| Flow rate \( Q / (m^3 \cdot h^{-1}) \) | 130,140,150 |
| Head \( H / m \) | 20 |
| Media density \( \rho / (kg \cdot m^{-3}) \) | 1000 |
| Viscosity coefficient | 1.005 \times 10^{-3} |
| kg / (m \cdot s) | |
3. Simulation Results and Analysis

3.1. Pressure Distribution Cloud

The cloud diagram of pressure distribution at the impeller inlet section is shown in figure 3. From figure 3 (a) and (b), it can be seen that the fluid pressure rises gradually in a step-like manner from the blade inlet to the outlet, and the pressure reaches the maximum at the outlet. It can be seen that the simulation results of both meet the performance requirements of pressurizing the fluid by impeller work. Low pressure zone appears in the blade leading edge of the suction surface, the pressure is negative, according to the conditions of cavitation occurrence, here easy to produce gas, and the actual situation.

3.2. Velocity Vector Distribution Cloud

The cloud diagram of the velocity vector distribution of the center section of the impeller of the centrifugal pump A worm casing is shown in figure 4(a). As can be seen from the figure, the velocity of the vane at the inlet is higher and the flow is uniform, which indicates that the impeller structure is reasonably designed. From the blade inlet to the outlet, the speed gradually decreases, while the pressure distribution cloud chart can be seen, the pressure is gradually increasing, so to meet the law of conservation of energy. Fluid in the impeller rotation driven by work to obtain energy, high speed left the outer edge of the impeller into the worm gear flow channel, the role of the small blade worm gear flow channel velocity vector is roughly uniform state distribution. By figure 4 (b) 1, 2, 3 can be seen in the worm gear flow channel there are a few tiny reflux vortex; by figure 4 (b) 1, 3, 4 can be seen in the small blade suction surface near all produced a tiny reflux vortex, which is due to the different curvature of the blade and rotational movement caused; by figure 4 (b) 3 to see the speed distribution near the spacer tongue disorder and accompanied by reflux, which is because of the resistance of the spacer tongue and impeller driven by high speed rotation of fluid [2], and when the small blade rotation to the septum area reflux situation aggravated, in the worm shell reflux vortex and small blade reflux vortex between another obvious reflux vortex, this is the septum and small blade structure interaction results, the fault type small blade to increase the pressure drop, speed more turbulent, resulting in energy loss. That small blade structure design rationality needs to be improved.

![Figure 3. Impeller inlet section pressure cloud distribution.](image-url)
The cloud diagram of velocity vector distribution in the center section of the worm gear impeller of centrifugal pump B is shown in figure 5(a). From figure 5(b) 1, 2 and 4, it can be seen that there are several continuous tiny backflow vortices in the worm gear flow channel, and the velocity vector of the worm gear flow channel is not uniform between 0° and 135° cross-section; the velocity near the spacer tongue is turbulent and accompanied by tiny backflow, and from figure 5(b) 3, it can be seen that tiny backflow vortices are generated at the outlet of this blade. Overall, the internal flow field of this centrifugal pump B is in good condition, and there is no obvious backflow vortex.

3.3. Cavitation Analysis

Cavitation is closely related to the pressure instability of centrifugal pumps, which can reduce pump performance, generate vibration and noise, and even corrode the pump flow components [14]. To eradicate cavitation, it is necessary to prevent the generation of gas at the impeller inlet, which is the pressure at the inlet is too low. In this paper, cavitation is simulated using Mixture multiphase flow model, turbulence model, Schneer-Sauer model for cavitation, Coupled solver, and PRESTO for pressure discretization format [15]. The inlet and outlet boundary conditions of the centrifugal pump are set to be pressure boundaries, and the inlet pressure is taken to be the inflection point value of 3% head drop, the reference pressure is 0 Pa, the saturated vapor pressure is 3540 Pa under the current operating conditions, and other boundary conditions are kept as default.
The cloud diagram of the gas volume fraction distribution of the impeller of centrifugal pump A is shown in figure 6. As can be seen from the figure, the cavitation of the impeller is manifested as the attached gas developing up on both sides of the blade leading edge surface, and the gas volume fraction is not uniformly distributed on each blade surface, and gradually separates from the blade surface with the growth of the gas volume fraction and develops toward the center of the blade flow path.

![Figure 6](image6.png)  
**Figure 6.** Impeller A gas volume fraction distribution.

The cloud plot of the gas volume fraction distribution in the inlet section of the impeller of the centrifugal pump A is shown in figure 7. As can be seen from the figure, there is also a small amount of adherent gas at the exit of some of the blades and it does not develop into the central flow channel [15]. From the velocity vector diagram, it can be seen that the flow in the septal tongue region is turbulent and the flow impact is large, which causes energy loss and the existence of low pressure area; in addition, when the small blade rotates to the vicinity of the septal tongue, the resulting multiple backflow vortices cause the pressure drop to become large, and the small blade suction surface also produces gas, which is more damaging to the small blade. On the whole, the low pressure area at the impeller inlet, the low pressure area near the spacer tongue and the suction surface of the small blade are easy to produce gas, and the cavitation performance needs to be improved.

Centrifugal pump B impeller and its inlet section gas volume fraction distribution cloud diagram as shown in figures 8 and 9. It can be seen that centrifugal pump B has similar cavitation characteristics with centrifugal pump A, but it generates more gas volume than centrifugal pump A, which is more likely to produce cavitation in the high-pressure area of the pump cavity, resulting in a decrease in pump efficiency.

![Figure 8](image8.png)  
**Figure 8.** Impeller B gas volume fraction distribution.

![Figure 9](image9.png)  
**Figure 9.** Impeller B inlet section gas volume fraction distribution.
4. Efficiency Comparison

With the FLUENT solution setup, parameters such as pressure, torque, and flow rate of the internal flow of the centrifugal pump can be obtained, which in turn leads to performance characteristics such as head, shaft power, and efficiency.

According to the definition of centrifugal pump head $H$, its equation expression is [16]:

$$H = \frac{P_{\text{out}} - P_{\text{in}}}{\rho g}$$  \hspace{1cm} (7)

where: $P_{\text{out}}, P_{\text{in}}$ - total pressure of inlet and outlet, $\rho$ - media density, $m^3$, $g$ - gravitational acceleration, $m/s^2$.

Axis power: $P = M \omega$, $\omega = \frac{2\pi n}{60}$

Centrifugal pump efficiency:

$$\eta = \frac{\rho g Q H}{3600 M \omega}$$  \hspace{1cm} (8)

where: $Q$ - Volume flow rate of centrifugal pumps, $m^3/h$.

The comparison of the efficiency of centrifugal pumps A, B at different design flow rates is shown in figure 10. As can be seen from the figure, the efficiency of centrifugal pump A is lower than that of centrifugal pump B at the same flow rate.

![Figure 10. Comparison of efficiency of centrifugal pumps A, B at different design flow rates.](image)

5. Conclusion

Based on CFD technology and MRF model, the three-dimensional turbulent flow inside the centrifugal pump is simulated numerically using Reynolds time-averaged N-S equation with RNG and SST turbulence models and Schnerr-Sauer cavitation model, and the pressure distribution, velocity vector distribution and cavitation characteristics of the internal flow field of the centrifugal pump are obtained. The results show that:

1. The small vane can play a role in preventing the expansion of the counterflow area, reduce the convective impact of the fluid, so that the water flow evenly and the gas volume is reduced, but the small vane makes the pressure gradient in the partition tongue area increase, the speed is more turbulent, reflux aggravation, resulting in energy loss, reducing the efficiency of the centrifugal pump.
A. The structure design rationality needs to be improved.

(2) Small vane can increase the head of the centrifugal pump, but it will increase the shaft power and reduce the efficiency.

Acknowledgements
This study is supported by Fujian Province Science and Technology Plan Funding Project (2017H0024); Fujian Province Science and Technology Commissioner Project (Min Ke Nong "2019" No. 11)

References
[1] Tang H and He F 2002 Numerical simulation of the flow field inside a centrifugal pump Pump Technology (03) 3-8+14.
[2] Wang Z J, Tong L and Zheng J S 2012 Numerical simulation and performance prediction of internal three-dimensional flow of centrifugal pumps based on CFD Fluid Machinery 40(06) 14-18.
[3] Song D M, Liao G L and Liu X Y 2017 Performance prediction of double suction centrifugal pumps based on turbulence numerical simulation Machinery 44(08) 18-21.
[4] Song S F and Liu Y T 2020 Numerical simulation of internal flow field of centrifugal pump based on CFD Energy and Energy Conservation (03) 99-100+156.
[5] Siddiqi M A, Boppa P and Strongin M P 2013 Multi-Parameters Pump Impeller Optimization American Society of Mechanical Engineers Digital Collection 387–394.
[6] Young-Joon An, Shin B R and Wahid M A 2010 Numerical Simulation of Centrifugal Pump with Double-Suction Impeller Kuala Lumpur (Malaysia) 456–464.
[7] Luo X W, Ji B and Yoshinobu Tsujimoto 2016 A review of cavitation in hydraulic machinery Journal of Hydrodynamics 28(03) 335-358.
[8] Yang Z J, Wang K T and Nie Y L 2020 Experimental study of the effect of cavitation on the performance of centrifugal pumps China Water Power & Electrification (02) 50-54.
[9] Wei Z, Yang W and Xiao R 2019 Pressure Fluctuation and Flow Characteristics in a Two-Stage Double-Suction Centrifugal Pump Symmetry 11(1) 65.
[10] Deng Z Q, Jiang J R and Liu X B 2021 Analysis of non-constant cavitation in the internal flow field of a centrifugal pump Water Resources and Power 39(06) 166-170.
[11] Chen W 2007 Simulation of internal flow and structural optimization of low specific rpm centrifugal pumps Zhejiang University.
[12] Yao Z F, Wang F J and Xiao R F 2012 Experimental investigation of pressure instabilities affected by cavitation for a double-suction centrifugal pump Earth and Environmental Science 15(6) 062040.
[13] Hu J, Hou X Y and Yu Y 2021 A Modified Schnerr-Sauer Cavitation Model with Local Flow Properties Transactions of Beijing Institute of Technology 41(01) 9-15.
[14] Yang S S, Kong F Y and Wang W T 2010 Numerical calculation and analysis of cavitation performance of centrifugal pumps Journal of Huazhong University of Science and Technology (Natural Science Edition) 38(10) 93-95.
[15] Hu K, Hu T T and Ma H F 2019 Ansys Fluent Example details Beijing: China Machine Press.
[16] Huang S and Sang D K 2010 Numerical simulation and performance prediction of 3D flow field of multi-stage centrifugal pump Mechanical Science and Technology for Aerospace Engineering 29(06) 705-708.