World Journal of Engineering Research and Technology

www.wjert.org

Impact Factor: 7.029 **Coden USA: WJERA4**

INTERNAL FLOW FIELD ANALYSIS IN A LOW SPECIFIC-SPEED PUMP AS TURBINE

Zhi-Yang Zhao*, Meng-Xi Hu, Shu-Xian Shi, Shen Chen, Ke-Xin Xie, Qi-Kai Gong

College of Mechanical Engineering, Quzhou University, Quzhou, 324000, China.

Article Received on 04/10/2025 **Article Revised on 24/10/2025** **Article Published on 01/11/2025**

***Corresponding Author**
Zhi-Yang Zhao
College of Mechanical Engineering, Quzhou University, Quzhou, 324000, China.

DOI: https://doi.org/10.5281/zenodo.17490456

How to cite this Article: Zhi-Yang Zhao*, Meng-Xi Hu, Shu-Xian Shi, Shen Chen, Ke-Xin Xie, Qi-Kai Gong. (2025). Internal Flow Field Analysis In A Low Specific-Speed Pump As Turbine. World Journal of Engineering Research and Technology, 11(11), 147–163.

ABSTRACT

Aiming at the problems of low efficiency and poor stability when low specific speed pump operates as turbine, the computational fluid dynamics software Fluent is applied, and the RNG k-ε turbulence model is adopted. With liquid water as the working medium, numerical calculations of internal flow are carried out for each structure of the low specific speed centrifugal pump in turbine mode, including the inlet section, volute, impeller and outlet section. The pressure and velocity cloud diagrams of each structure of the centrifugal pump and the external characteristics of the centrifugal pump are compared and analyzed under three different flow rate conditions. The research results show that when the fluid flows through each component, the pressure and velocity show regular changes: in terms of pressure, the inlet pressure of the turbine is the largest, and the pressure gradually decreases after the fluid enters the impeller through the volute. When the working condition is improved, the pressure of the inlet section, volute and impeller rises synchronously, which can improve the pump efficiency and suppress cavitation; in terms of velocity, the fluid flow velocity in the inlet and outlet pipes of the centrifugal pump is not high. After the fluid enters the volute, the flow velocity at the impeller edge gradually increases and forms a gradient, and the flow velocity near the water outlet section drops to the lowest. When the working condition is improved, the flow velocity of the water inlet section, volute and impeller increases significantly. Appropriately increasing the flow rate can enhance the flow stability of the impeller to

meet the actual use requirements, thereby improving the performance and reliability of the centrifugal pump.

KEYWORDS: Pump as Turbine, Pressure, Velocity, Numerical Calculation.

1. INTRODUCTION

In the context of advancing energy sustainable development with reduced carbon emissions, the centrifugal pump operating as a hydraulic turbine, an energy recovery device[1], has been increasingly widely utilized in various industrial sectors including chemical engineering, electric power, energy, water conservancy, and machinery. Initially, the pump-as-turbine (PAT) is characterized by high operability, simple structure, and convenient maintenance. It can convert the residual pressure energy[2-3] and residual kinetic energy of the fluid into mechanical energy, enabling efficient energy recovery and utilization, which is of great significance for improving energy utilization efficiency and reducing energy consumption. However, the characteristics under reverse rotation were not taken into account in the design of centrifugal pumps, leading to numerous adverse factors in practical applications. This study focuses on the low-specific-speed pump operating in reverse as a hydraulic turbine. Owing to the geometric features of structures such as its volute[4] and impeller, the low-specific-speed PAT commonly suffers from low efficiency and poor stability during actual operation. This not only impairs its energy recovery efficiency but also hinders its development and widespread application. Therefore, in-depth analysis and investigation of the internal flow field characteristics of the low-specific-speed PAT, and exploration of the inherent causes and mechanisms underlying its insufficient efficiency and stability, have become one of the key research directions in this field. This work aims to provide a reference for the further development and utilization of low-specific-speed centrifugal pumps.

In traditional research methods, the complexity of the internal flow field in fluid machinery poses numerous challenges to research, such as limitations in the precision of measurement equipment, limited consideration of directions, and flow field interference. However, with the rapid development of Computational Fluid Dynamics (CFD) technology, the numerical simulation method[5] has provided a new approach for the research on the internal flow field of pump as turbine. Nevertheless, the accuracy of numerical simulation results is related to the simplification of the fluid domain model, the quality of mesh generation, and the selection of turbulence model.[6] In this paper, by constructing an appropriate geometric model, selecting a proper turbulence model and numerical calculation method, the distribution laws of various parameters such as pressure and velocity in each structure of the pump-as-turbine flow field are obtained. Through analysis and research, a reliable and powerful theoretical basis is provided for the optimal design of centrifugal pump performance.

In this study, for the low-specific-speed centrifugal pump operating in turbine mode, a comparative analysis is conducted under three different flow rate conditions.[7] In terms of pressure characteristics, the pressure distribution and variation laws with operating conditions in regions such as the tongue, volute[8], and impeller[9] are analyzed emphatically, exploring the influence of pressure changes on the efficiency and anti-cavitation performance of the pump as turbine. In terms of velocity characteristics, the velocity changes and trends in the inlet and outlet sections, volute, and impeller regions are analyzed, as well as their influence on flow stability and energy loss.[10] Through in-depth research and analysis of the flow field characteristics[11] of the low-specific-speed pump as turbine, it not only helps to explore the internal causes of low efficiency and poor stability of the low-specific-speed pump as turbine, but also provides a meaningful reference for its structural optimization, efficiency improvement, and performance enhancement. This is of great theoretical value and significance for promoting the application and development of low-specific-speed pump as turbine in more aspects.

2. Computational Model and Method

2.1 Computational Model

The prototype pump of the centrifugal pump operating as a hydraulic turbine in reverse rotation is a low-specific-speed centrifugal pump. Its design parameters are as follows: flow rate 20 m³/h, head 34 m, rotational speed 2900 r/min. The structural parameters are detailed in Table 1.

Table 1: Main Structural Parameters.

Main Structure	Parameter	Value
Impeller	Inlet Diameter D_1/mm	124
	Outlet Diameter D_2/mm	44
	Inlet Width b_1/mm	10
	Blade Wrap Angle /(°)	135
Volute	Inlet Diameter D_s/mm	32
	Outlet Width b_0/mm	16
	Base Circle Diameter D_0/mm	140
Inlet Section	Diameter D_{in}/mm	32
	Length Lin L_{in}/mm	100
Outlet Section	Diameter D_{out}/mm	44
	Length L_{out}/mm	55.5

2.2 Computational Method

The overall computational domain mesh is shown in Figure 1, and the number of meshes in each region is shown in Table 2. The RNG k-ε turbulence model is selected for internal flow calculation. Liquid water is used as the working medium. The turbine inlet is set as a velocity inlet boundary condition, and the outlet is set as a free pressure outlet boundary.

Figure 1: Overall Model Mesh.

Table 2: Number of Meshes.

Region	Inlet Section	Volute	Impeller	Outlet Section
Number of Meshes	25200	472946	1239259	162048

3. Result Analysis

3.1 Pressure Contour of Volute

When high-pressure liquid enters the pump casing from the suction inlet of the centrifugal pump, the fluid moves in the volute cavity, generating an impact force on the impeller, which pushes the impeller to make circular motion and then drives the turbine shaft to rotate, converting the pressure energy of the fluid into mechanical energy. This efficient process is a main characteristic of the centrifugal pump, realizing continuous medium transportation and head improvement. The pressure contour distribution inside the centrifugal pump turbine volute is directly related to the incoming flow rate, and different working conditions correspond to the situation in Figure 3.

(a) Low Working Condition (b) Medium Working Condition (c) High Working Condition

Figure 3: Pressure Contours of Centrifugal Pump Turbine Volute under Different Working Conditions.

When discussing the influence of operating condition changes on the pressure distribution in the centrifugal pump volute, refer to Figure 3. The pressure contours of the volute under low, medium, and high operating conditions show clear gradient change characteristics. Under the three operating conditions, due to the continuous influx of liquid water, the pressure at the inflow of the centrifugal pump volute turbine climbs to the highest point, and the pressure gradient inside the volute gradually decreases. At the same time, the pressure of the volute on the turbine inlet side rises significantly. When the liquid medium enters the volute cavity, the pressure decreases significantly, and the pressure drop is particularly notable near the impeller. When comparing different operating conditions (low, medium, and high), the difference in this phenomenon is more obvious. As the operating condition intensity increases and the flow rate increases, the pressure of the volute at the turbine inlet rises significantly. When the operating condition improves, the pressure inside the volute continues to rise. The variable operating condition environment of the centrifugal pump enhances the flow dynamic characteristics, resulting in changes in the volute pressure distribution.

3.2 Pressure Contour of Volute Tongue

During the operation of the centrifugal pump turbine, the volute tongue plays an important role. It can reduce the internal vortex of the volute, realize stable fluid transportation, improve pump efficiency and reduce useless energy consumption. The volute tongue can also control the flow direction of the fluid, make the liquid in the volute flow passage pass more smoothly, reduce flow resistance. It has a positive impact on fluid pressure recovery, and can promote the timely recovery of pressure in the junction area of the impeller and volute, improving the operating efficiency of the pump.

Scientific planning of the profile and installation position of the volute tongue can promote uniform fluid distribution, improve volume utilization efficiency, control medium backflow and local accumulation, thereby improving the operating efficiency of the pump. The use of the volute tongue can optimize the fluid distribution pattern in the volute, reduce the performance oscillation phenomenon caused by different flow distributions, achieve continuous and stable operation of the unit and prolong its service life. In the experiment, the pressure data in the volute tongue area was collected emphatically, and the pressure field distribution of the centrifugal pump turbine volute tongue under each working condition is shown in Figure 4.

(a) Low Working Condition (b) Medium Working Condition (c) High Working Condition

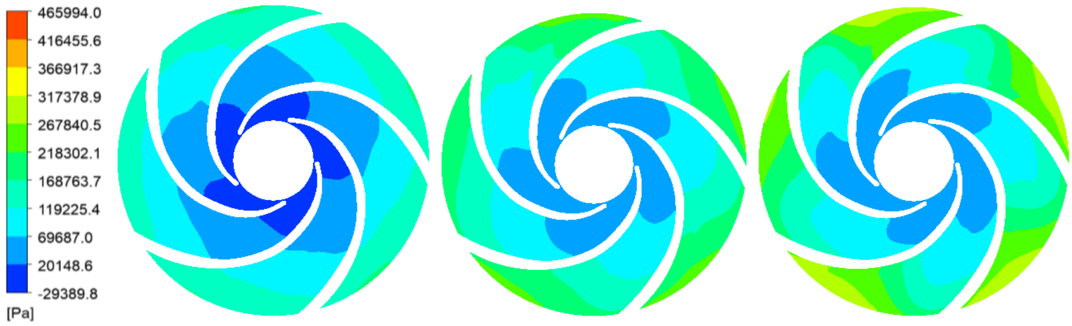
Figure 4: Pressure Contours of Centrifugal Pump Turbine Volute Tongue under Different Working Conditions.

It can be seen that by studying the pressure contours of the centrifugal pump turbine volute tongue under low, medium, and high operating conditions, it is found that the pressure at this part is mainly low pressure. As the fluid flow rate increases, the pressure in the volute tongue area rises. When the fluid movement speed is slow, the pressure value at this position is significantly low. When the flow rate continues to increase, the kinetic energy of the water phase increases significantly, resulting in a sharp rise in pressure at the volute tongue. This phenomenon is particularly prominent in high-pressure operating conditions. As the flow rate continues to increase, the pressure at the volute tongue gradually increases, reflecting a strong correlation between the volute tongue.

3.3 Pressure Contour of Impeller

Conducting visualization research on the pressure distribution of centrifugal pump impellers is of great significance and helps optimize the pump's performance. This analysis can intuitively reflect the distribution pattern of fluid pressure inside the impeller, which is used for flow characteristic analysis and design problem identification. By comparing pressure contours under different operating conditions, performance diagnosis of the impeller under various working states can be carried out, thereby timely detecting faults such as uneven pressure and cavitation.

Fluid dynamics research can obtain important experimental data through pressure contours, enhancing the understanding of complex fluid interaction principles. With the help of pressure contour analysis, the working efficiency and reliability of centrifugal pumps can be effectively improved. The pressure contours of centrifugal pump turbine impellers under different operating conditions are detailed in Figure 5.



(a) Low Working Condition (b) Medium Working Condition (c) High Working Condition

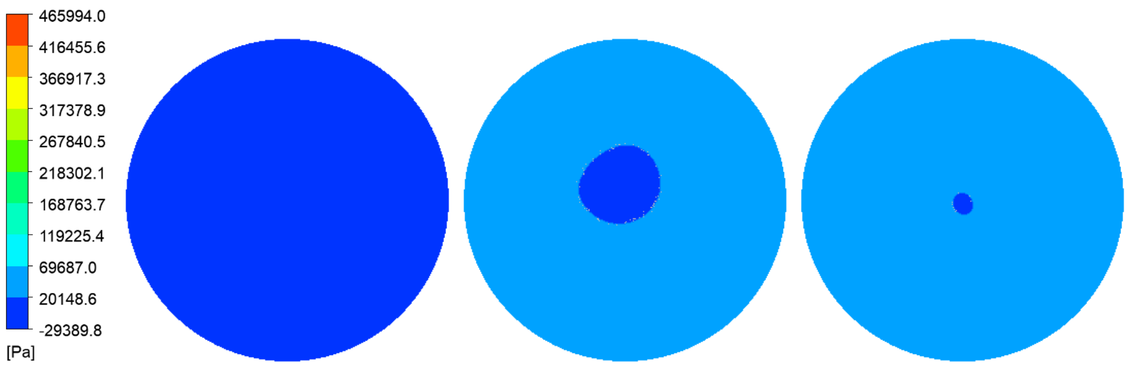
Figure 5: Pressure Contours of Centrifugal Pump Turbine Impeller under Different Working Conditions.

After analyzing the pressure contours of the centrifugal pump turbine impeller under three typical working conditions (low, medium, and high), it is found that in all working condition ranges, a distinct high-pressure zone is generated in the outer blade area of the impeller. Near the middle of the impeller, that is, where the pump suction inlet and the turbine discharge outlet are connected, the pressure is significantly low. As the operation intensity increases, the flow rate of the incoming medium gradually increases, and the pressure at the outer periphery of the impeller rises synchronously. This phenomenon is very obvious during high-load operation. The pressure gradient at the outer periphery of the impeller becomes larger, and the pressure value around the turbine exhaust port continues to rise.

3.4 Pressure Contour of Turbine Outlet Section

The analysis of the pressure field distribution at the centrifugal pump turbine outlet is of crucial importance. Its results can reveal the pressure distribution law at the outlet section, which is helpful for understanding the flow velocity changes and flow patterns. The pressure contour intuitively presents the pressure gradient, contributing to the identification of complex flow regions such as vortices. Through the pressure field distribution, the regions with uneven force on the impeller can be accurately located, and the structure can be improved targetedly to reduce vibration and enhance efficiency.

This analysis is the basis of fluid mechanics research, which can verify the CFD model and provide data support for the optimization of energy recovery systems in fields such as new energy, promoting the research on the dynamic characteristics of fluid in pumps and the development of engineering applications. Figure 6 shows the pressure contours of the centrifugal pump turbine outlet section under various working conditions.



(a) Low Working Condition (b) Medium Working Condition (c)High Working Condition

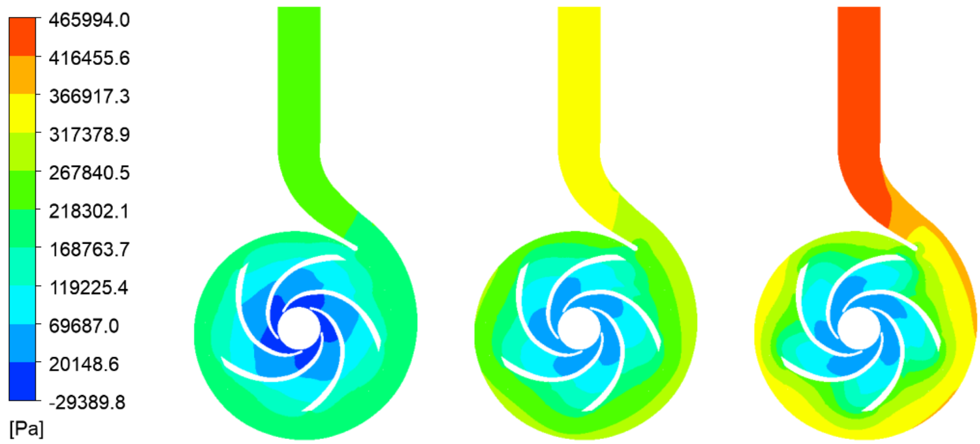
Figure 6: Pressure Contours of Centrifugal Pump Turbine Outlet Section under Different Working Conditions.

For the pressure contours of the outlet section of the centrifugal pump turbine under low, medium, and high working conditions, the pressure distribution decreases gradually from the edge to the center, and the pressure in the central region decreases significantly. With the increase of operation intensity and incoming flow rate, the peripheral pressure field is obviously enhanced, and the area of the central low-pressure region is gradually reduced.

With the increase of flow rate, the motion state of the fluid in the outlet section is improved. The peripheral pressure value is always relatively high, and the low-pressure region at the center of the flow passage is obviously reduced. This method can effectively improve the efficiency of the outlet section and simultaneously suppress the potential risk of cavitation.

3.5 Overall Pressure Contour

The overall pressure contours of the centrifugal pump turbine under different working conditions are shown in Figure 7.



(a) Low Working Condition (b) Medium Working Condition (c)High Working Condition

Figure 7: Overall Pressure Contours of Centrifugal Pump Turbine under Different Working Conditions.

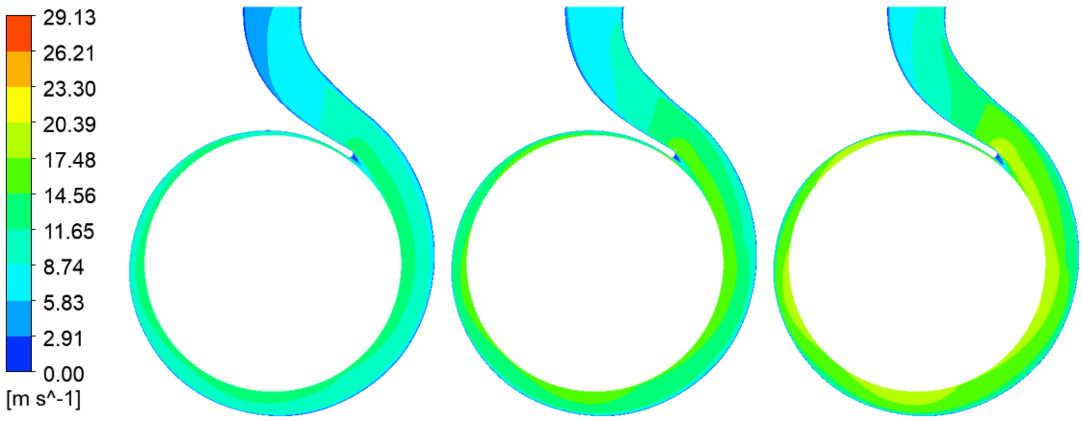
When analyzing the overall pressure contours of the centrifugal pump turbine under low, medium, and high working conditions, several key change trends emerge. By observing the pressure contour at the section where x = 0, the pressure at the centrifugal pump outlet (turbine inlet) reaches the maximum value. During the flow stage where water enters the impeller through the volute, the pressure gradually decreases, and near the centrifugal pump inlet (turbine outlet), the pressure drops to a critically low level.

When the working conditions change, transitioning from low load to high load, the increase in incoming flow rate causes a significant increase in the inlet pressure of the centrifugal pump turbine. The pressure inside the volute and the pressure of the impeller rise synchronously. Increasing the flow rate can improve the flow characteristics of the fluid, and the increase in pressure parameters can improve the working efficiency of the centrifugal pump, effectively resist the potential harm of cavitation, and optimize the stable state of the pump under different operating environments.

3.6 Velocity Contour of Volute

With the help of the velocity contour of the centrifugal pump turbine volute, the velocity gradient of the fluid in the cavity can be effectively displayed, facilitating the exploration of the change law of flow characteristics.

As a key tool for fault diagnosis and a core approach for fluid dynamics research, the velocity contour can determine abnormal phenomena in the flow process, diagnose faults such as volute blockage, and based on this, quantitatively evaluate the working characteristics of the pump. It serves as an effective reference for analyzing the law of fluid behavior and promotes the upgrading of pump machinery. The velocity contour of the centrifugal pump turbine volute under different working conditions is shown in Figure 8.



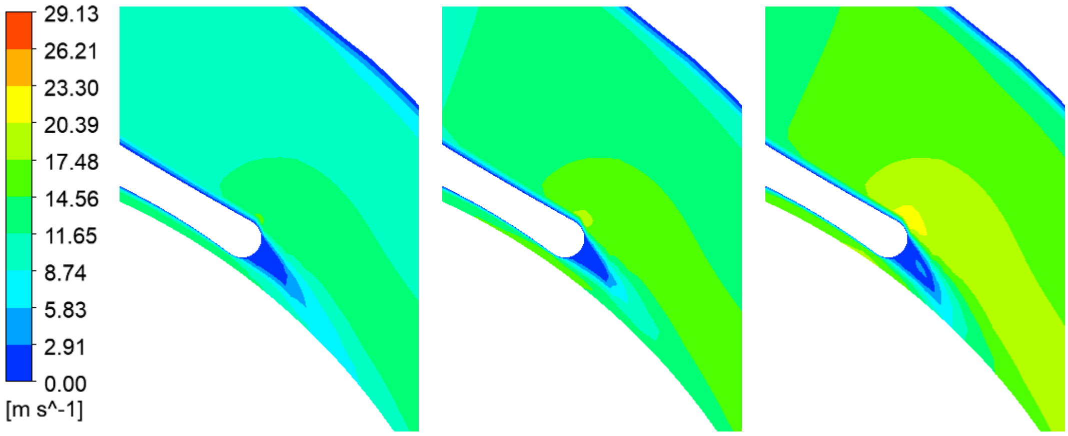
(a) Low Working Condition (b) Medium Working Condition (c) High Working Condition

Figure 8: Velocity Contour of Centrifugal Pump Turbine Volute Under Different Working Conditions.

An analysis is conducted on the velocity contours of the centrifugal pump turbine volute under low, medium, and high working conditions. It is found that the velocity gradient in the initial flow channel region is relatively small, and the fluid accelerates gradually within the volute flow channel. The flow velocity is low in the area where the volute and the impeller are not connected, while the fluid velocity near the impeller increases significantly. As the operating condition becomes stronger and the flow rate of the incoming medium increases, the flow velocity in the volute channel increases sharply. The increase in flow velocity can improve the overall working efficiency of the pump, ensuring that the fluid is fully delivered to the impeller and then discharged smoothly by the volute.

3.7 Velocity Contour of Tongue

Velocity contour analysis can clearly identify the velocity distribution pattern of the fluid near the tongue, providing support for the analysis of flow states. It can be used as an effective method for fault analysis. If deviations between the flow velocity in the tongue region and the normal value are detected, or abnormal situations such as pipeline blockage exist, identifying these conditions in advance can enable timely measures to be taken, reducing losses and maintenance costs. The velocity distribution contours of the tongue section under various working conditions are shown in Figure 9.



(a) Low Working Condition (b) Medium Working Condition (c) High Working Condition

Figure 9: Velocity Contour of Centrifugal Pump Turbine Tongue Under Different Working Conditions.

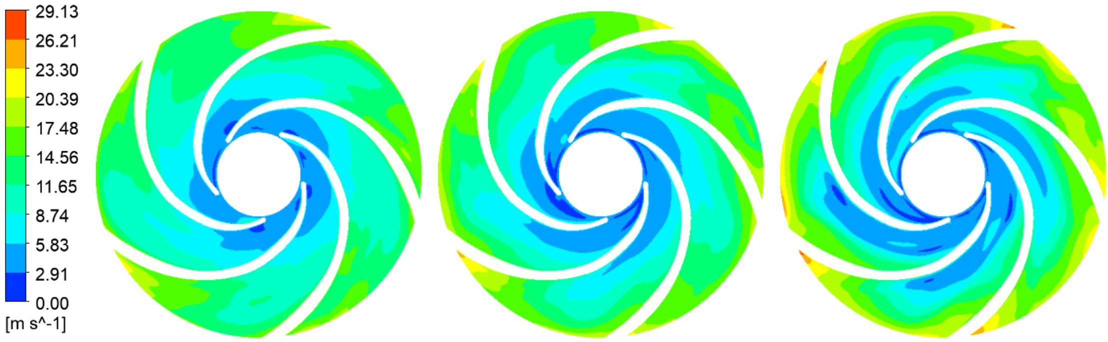
For three typical working conditions (low, medium, and high), after analyzing the velocity contour at the volute tongue, it is found that the flow velocity at this location is significantly low. Moreover, as the distance to the tongue decreases, the flow velocity further decreases. The acceleration effect of the fluid at the tongue is not saturated, which easily leads to a decline in flow efficiency. According to different working conditions, the velocity field at the tongue is very sensitive to changes in flow rate. Especially when the working condition reaches the peak stage, the flow velocity will climb to the maximum value.

With the increase of operating parameters, the inlet flow rate increases, and the flow velocity in the tongue area increases significantly. This change can effectively suppress local loss phenomena and improve the overall efficiency of the centrifugal pump.

3.8 Velocity Contour of Impeller

When evaluating the performance of a centrifugal pump, it is necessary to focus on studying the characteristics of the velocity contour in the turbine impeller region. The contour can intuitively present the fluid velocity distribution inside the impeller, facilitating the understanding of fluid flow characteristics. Based on the contour, the changes in the flow velocity gradient at the inlet and outlet of the impeller can be analyzed, demonstrating the dynamic distribution of the fluid inside the impeller. Furthermore, specific regions with different fluid motion velocities can be identified, effectively detecting potential cavitation or efficiency decline phenomena.

In fault diagnosis, velocity contours also play a role. If there is local abnormality in the flow velocity distribution spectrum, it indicates problems such as blockage in the system. Analyzing these contours allows for preventive measures to be taken before problems occur, reducing maintenance costs and equipment downtime. The velocity distribution contour of the impeller effectively enhances the working efficiency and operational reliability of the centrifugal pump. The collected data is of great significance for industry technological progress and disciplinary development. The variation of the velocity contour of the centrifugal pump turbine volute tongue with working conditions is shown in Figure 10.



(a) Low Working Condition (b) Medium Working Condition (c) High Working Condition

Figure 10: Velocity Contours of Centrifugal Pump Turbine Impeller Under Different Operating Conditions.

Based on the velocity contours of the centrifugal pump turbine under three operating conditions, the different flow velocity distributions at various parts of the impeller are revealed. Near the blade profile, the fluid velocity on the non-water outlet side is significantly higher than that on the near-water outlet side, and the flow velocity in the part adjacent to the water outlet section is relatively slow, forming a velocity profile transitioning from the peripheral high-speed region to the central low-speed region. The internal flow field distribution of the impeller is jointly influenced by geometric features and fluid dynamic parameters.

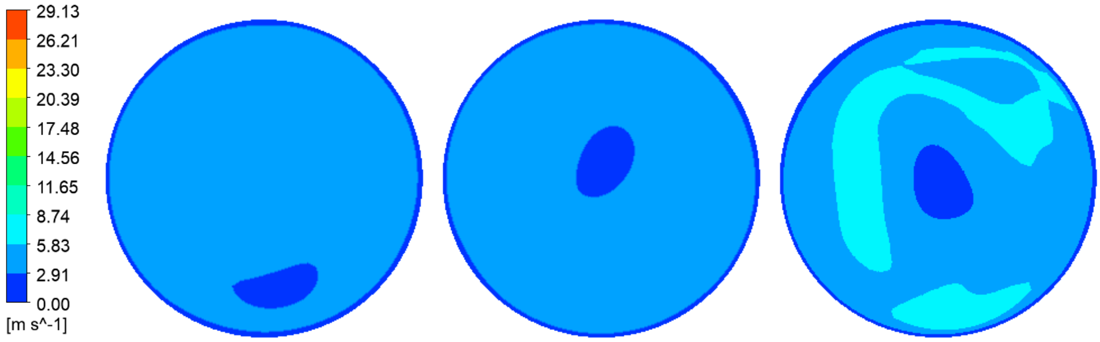
The continuous increase of incoming flow rate leads to the continuous rise of working conditions, and the impeller speed will increase synchronously. In the change from low to high working conditions, the flow rate increases monotonically, resulting in the gradual expansion of the low-speed region in the middle of a single blade. This phenomenon will directly affect the efficiency of the pump. If the pump is under a large working load, the increase in flow rate may cause a decline in flow performance, thereby affecting the working characteristics of the pump.

3.9 Velocity Contour of Turbine Outlet Section

The velocity contour, in the form of an intuitive color gradient, accurately presents the fluid velocity gradient distribution at the outlet section of the centrifugal pump turbine. Through the transition and comparison of different colors, it clearly shows the variation trend of flow velocity from the center to the edge and from the high-pressure region to the low-pressure region, providing visual assistance for the analysis of flow mechanisms.

This visual analysis can not only help identify high-speed regions, low-speed regions, and vortex positions, providing a basis for optimizing impeller design, reducing energy loss, and improving turbine efficiency, but also clearly show the spatial evolution trajectory of flow velocity and pressure, finely depict the characteristics of the flow field, and contribute to the in-depth analysis of flow mechanisms.

The velocity contours of the water outlet section of the centrifugal pump turbine under different working conditions are shown in Figure 11.



(a) Low Working Condition (b) Medium Working Condition (c) High Working Condition

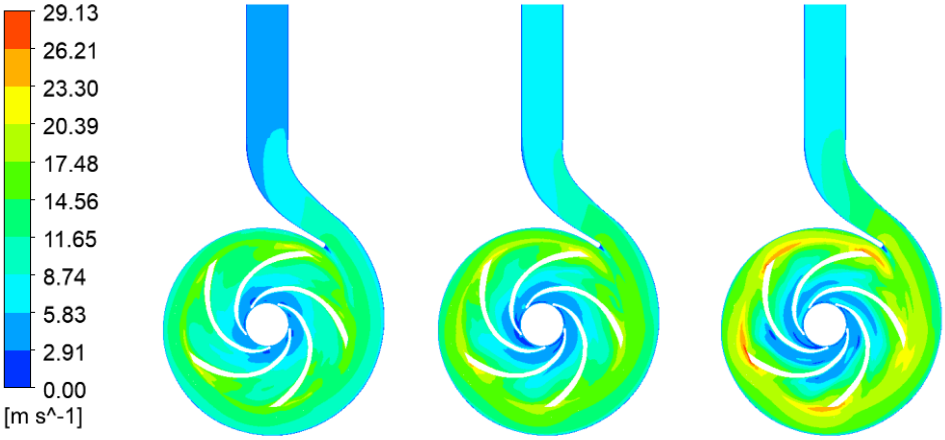
Figure 11: Velocity Contours of the Water Outlet Section of Centrifugal Pump Turbine Under Different Operating Conditions.

An analysis of the velocity contours of the water outlet section of the centrifugal pump turbine under different operating conditions (low, medium, and high) reveals significant features. Regardless of changes in operating conditions, the water flow velocity in this region is mostly relatively low, especially in the central part of the water outlet, where the flow velocity reaches the minimum value. The decrease in flow velocity in this region will affect the overall efficiency of the pump, causing the liquid at the outlet to stagnate, thereby affecting the overall working efficiency of the pump.

As the operating condition parameters increase, that is, the inlet flow rate rises, the flow velocity at the discharge port increases sharply under high system load, and the flow velocity at the end also increases significantly. The hydraulic conduction characteristics are enhanced. Regulating the flow rate to an appropriate range can optimize the water outlet performance of the pump and make the flow field distribution more uniform.

3.10 Overall Velocity Contour

The velocity contour can effectively present the velocity distribution of the fluid inside the centrifugal pump turbine, and can accurately identify the characteristics of the flow field and potential flow-related hazards. Step-by-step analysis and optimization based on the velocity contour can improve the energy efficiency of the centrifugal pump and enhance its reliability. The overall velocity contours of the centrifugal pump turbine corresponding to different operating conditions are shown in Figure 12.



(a) Low Working Condition (b) Medium Working Condition (c) High Working Condition

Figure 12: Overall Velocity Contours of Centrifugal Pump Turbine Under Different Operating Conditions.

The velocity contours confirm that there are significant differences in fluid flow within the centrifugal pump turbine under low, medium, and high operating conditions. The flow velocities in the inlet and outlet pipes of the pump are generally not high. When the water flow enters the volute instantaneously, the fluid velocity at the impeller edge gradually increases, forming a flow velocity gradient. During the flow of fluid from the outer edge of the blades to the inlet and outlet sections, the flow velocity drops to the minimum value near the outlet section, which may cause partial energy loss and interfere with the operating characteristics of the pump.

As the operating condition improves, that is, the incoming flow rate increases, the flow velocity in the inlet section increases significantly, and the flow velocities in the volute and impeller increase remarkably. The low-velocity region near the impeller axis becomes more elongated. Appropriately increasing the flow rate can enhance the flow stability of the impeller, which is beneficial to the improvement of the pump's conveying efficiency.

4. CONCLUSIONS

This study focuses on the low-specific-speed pump as a turbine under low, medium, and high operating conditions. By analyzing the pressure contours and velocity contours of each structure of the centrifugal pump, and through multi-aspect comparisons and experimental data, the following conclusions are drawn:In terms of pressure characteristics: The pressure at the outlet of the centrifugal pump (inlet of the turbine) is the highest at the x=0 section. The pressure gradually decreases when the fluid flows from the volute to the impeller, and drops to a critically low level at the inlet of the centrifugal pump (outlet of the turbine). When the flow rate increases, the inlet pressure, volute pressure, and impeller pressure rise simultaneously, which can improve the flow characteristics of the centrifugal pump, increase efficiency, resist cavitation, and optimize the operational stability of the pump. In terms of velocity characteristics: There are significant differences in fluid flow under different operating conditions. The flow velocities in the inlet and outlet pipes are generally low. After the fluid enters the volute, the flow velocity at the impeller edge gradually increases to form a velocity gradient, and the flow velocity is the lowest near the outlet section, which is prone to energy loss. When the flow rate increases, the flow velocities in the inlet section, volute, and impeller all increase significantly, and the low-velocity region at the impeller axis becomes more elongated. Appropriately increasing the flow rate can enhance the flow stability of the impeller and improve the conveying efficiency of the centrifugal pump.

This study clarifies the pressure and velocity characteristic laws of low-specific-speed centrifugal pump turbines under different working conditions, providing certain theoretical support for their design optimization, retrofitting application, efficiency improvement, and stable operation. Through this research and analysis, in-depth exploration and development of similar centrifugal pump-related studies can be carried out subsequently to further promote the development of this field.

ACKNOWLEDGMENT

The research was financially supported by the national college students science and technology innovation project (No.202411488043).

REFERENCES

1. Tang Y C. Research on the Influence of Guide Vane and Volute Inlet Section Diameter on Flow Noise of Pump-as-Turbine[D]. Lanzhou University of Technology, 2024.
2. Zhang Y. Analysis of Hydrodynamic Characteristics of High-Speed Pump-as-Turbine[D]. Chang'an University, 2023.
3. Ji X, Wu M L, Wei Y, et al. System Optimization of Power Generation Device for Centrifugal Pump-as-Turbine[J]. New Technology & New Process, 2015; (09): 44-47.
4. Meng B Y. Research on the Influence of Volute Geometric Shape on the Performance of High-Specific-Speed Centrifugal Pump[D]. Chang'an University, 2023.

5. Liang Z Y. Research on Internal Flow Field and Entropy Production of Multistage High-Pressure Water Injection Centrifugal Pump[D]. Shandong University, 2022.
6. Sun X, Zeng F K, Zhou F L, et al. Numerical Calculation of Steady Flow in Pump-as-Turbine[J]. Journal of Hunan University of Technology, 2021; 35(01): 39-47.
7. Yang R F. Internal Flow Field Analysis and Multi-Objective Optimization of Low-Specific-Speed Multistage Centrifugal Pump[D]. Lanzhou University of Technology, 2023.
8. Song D Y. Research on the Influence of Volute Tongue Geometric Parameters on Turbine Performance and Internal Energy Loss[D]. Lanzhou University of Technology, 2024.
9. Shao H. Numerical Calculation and Experimental Research on Impeller of Mixed-Flow Pump-as-Turbine[D]. Jiangsu University, 2020.
10. Zhang D S, Qi B, Zhao R J, et al. Optimal Design of Hydraulic Model for Seawater Desalination Energy Recovery Turbine[J]. Journal of Drainage and Irrigation Machinery Engineering, 2021; 39(07): 649-654.
11. Zhu H L, Yang M Z, Wang Z L, et al. Flow Field Characteristic Experiment of Low-Specific-Speed Axial Flow Pump[J]. Ship Engineering, 2025; 36(02): 112-121.