

Article

Numerical Study on Hydraulic Loss Characteristics in an Azimuth Waterjet Propulsion

Zikai Lv  and Puyu Cao *

Fluid Machinery Engineering Technology Research Center, Jiangsu University, No. 301 Xuefu Road, Zhenjiang 212013, China

* Correspondence: mafatu1988@ujs.edu.cn; Tel.: +86-158-6298-5295

Abstract

To address the low efficiency and unclear internal loss mechanisms of azimuth waterjet propulsion (AWP) systems operating under shallow and complex flow conditions, this study investigates an AWP unit at 950 rpm with a thrust of 1.63 kN. Steady numerical simulations are conducted under mooring and low-speed conditions, focusing on thrust coefficient, impeller efficiency, pump efficiency, and diffuser flow characteristics, with comparisons to a conventional mixed-flow pump. The results show that the propeller hydraulic efficiency at the design condition is approximately 52%, significantly lower than the 80–93% typical of mixed-flow pumps. The diffuser contributes nearly 80% of the total hydraulic loss, dominated by secondary flow effects. From the perspective of radial equilibrium in the guide vanes, secondary flow development is closely linked to spanwise momentum non-uniformity and deviation from equilibrium. The inclined outflow from the impeller induces strong spanwise imbalance, while the nearly 180° turning in the diffuser suppresses conventional force terms and establishes a pressure-gradient-dominated inertial balance associated with streamline curvature. This mechanism drives transverse migration and entrainment, promoting the formation of counter-rotating vortex pairs and secondary flows. Four major vortex concentration regions are identified, where interactions between secondary flow and recirculation generate complex three-dimensional vortex structures, including induced and spiral separation vortices. These vortices locally block the flow passage, causing pressure fluctuations and energy dissipation. The mid-span region of the guide vanes is identified as the primary location of loss accumulation. These findings provide theoretical and engineering guidance for diffuser optimization in AWP systems. It should be noted that the present study is based solely on numerical simulations, and no experimental validation for the investigated AWP configuration is currently available. Future experimental studies are needed to further verify the predicted hydraulic performance and flow structures.



Academic Editor: Davide Lengani

Received: 29 May 2026

Revised: 5 July 2026

Accepted: 8 July 2026

Published: 13 July 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

Keywords: azimuth; waterjet propulsion; hydraulic loss characteristics; efficiency

1. Introduction

Owing to its distinctive structural configuration and control mechanism, the azimuth waterjet propulsion (AWP) system is increasingly recognized as a promising alternative to conventional waterjet propulsion in shallow waters, wetlands, and other complex aquatic environments. The diffuser and nozzles of the AWP can rotate omnidirectionally through 360°, enabling thrust to be generated in any direction without altering the vessel's attitude. Furthermore, the entire propulsion system is installed flush with the ship's bottom, which

not only reduces additional hydrodynamic drag but also significantly lowers the risk of collisions with floating debris, riverbeds, and underwater obstacles. This configuration enhances operational reliability and environmental adaptability under extremely shallow-water conditions [1]. These characteristics provide the AWP with notable advantages in terms of maneuverability, operational capability in spatially constrained environments, and flexibility in hull integration, thereby attracting increasing attention from both engineering practitioners and academic researchers.

The AWP was first proposed by Josef Becker (Schottel founder) at Schottel GmbH [2] and has since been applied to a variety of vessel types, including ferries, tugs, yachts, research vessels, and patrol boats [3]. In recent years, this propulsion concept has also been adopted in Autonomous Surface Vehicles (ASVs) for applications such as wetland environmental monitoring and ecological conservation [4]. At present, leading international manufacturers of AWP systems include Schottel GmbH, Veth Propulsion, Jastram GmbH & Co. KG, and Tees White Gill. Their products exhibit notable differences in suction inlet layout, impeller configuration, diffuser structure, and nozzle arrangement. Odetti et al. [4] conducted a systematic comparison of the structural features of these products, while Yan Peng et al. [5] further reviewed their development history and typical configurations.

From the perspective of structural configuration, the propulsion systems developed by Veth Propulsion and Jastram GmbH & Co. KG exhibit considerable similarity. Both employ axial-flow impellers combined with rear nozzle structures. The principal distinction lies in the orientation of the impeller axis: in the former, the axis is inclined approximately 17° relative to the vertical direction, whereas in the latter it remains strictly vertical. During operation, water passes through the impeller, where it gains energy before entering the diffusion passage and ultimately being discharged through the rear nozzle. The impeller-diffuser assembly rotates about the vertical axis to produce omnidirectional thrust. Publicly available information indicates that both companies have gradually reduced or discontinued research, development, and production of such thrusters in recent years.

The AWP developed by Tees White Gill [6] also adopts a vertically arranged impeller; However, the suction inlet is located on the side of the hull. In this configuration, water enters the propulsor laterally and is discharged through a bottom nozzle. Although this design offers advantages in terms of structural compactness, it imposes strict constraints on hull form design, particularly limiting the transverse dimensions of the hull in the vicinity of the propulsor.

In contrast, the AWP produced by Schottel GmbH [2] employs a mixed-flow impeller with characteristics approaching those of a centrifugal impeller. The impeller axis is vertically oriented, and water is drawn vertically from the bottom inlet. After energy conversion within the diffuser chamber, the flow is discharged through the bottom nozzle. Multiple nozzles are typically arranged at the base of the diffuser, and thrust vector control is achieved through a judicious configuration of nozzle area ratios and injection directions. Compared with other configurations, this design offers advantages in structural compactness, hull adaptability, and engineering maturity, thereby becoming the dominant configuration in the current market.

Compared with traditional axial-flow waterjet propulsion systems [7–17], systematic research on AWP systems remains relatively limited. Odetti et al. [4,18] designed an AWP specifically for extremely shallow water environments such as rivers, lakes, and marshes. The fundamental configuration resembles that of Schottel-type products, employing a closed impeller and utilizing three-dimensional printing technology to manufacture hydraulic components. Thrust characteristics and thrust deduction coefficients at different speed and rotational speed conditions were measured using a bollard pull testing system. Liu Wenhao [19] designed a compact AWP based on the parameters of a patrol vessel and

optimized its key hydraulic parameters by computational fluid dynamics methods. The effects of operating speed and blade numbers on propulsion performance were systematically analyzed and subsequently validated through experimental testing.

Regarding ship maneuverability, Carreño et al. [3] and Moreno et al. [20] investigated patrol vessels equipped with AWP systems and developed mathematical models to evaluate their maneuvering performance and associated characteristics. With respect to internal flow mechanisms, Zhang et al. [21] studied the momentum distribution within the diffuser nozzle. Their results demonstrated that, through appropriate selection of the nozzle area ratio and incorporation of guide vanes, the lateral force could be reduced by up to 56% without any reduction in total thrust. Yan Peng et al. [1] further analyzed the influence of rotational speed on overall performance through numerical simulations under multiple operating conditions. The entropy generation rate method was employed to quantitatively evaluate the magnitude and spatial distribution of internal losses, and four primary sources of loss within the diffuser region were identified.

In general, existing studies have primarily focused on overall performance evaluation, maneuvering control modeling, and localized structural optimization of waterjet propulsors. However, systematic investigations of internal flow loss mechanisms in AWP systems remain scarce, particularly comprehensive analyses that integrate regional loss quantification with multi-scale flow structure identification.

To address this research gap, the present study systematically investigates the internal flow loss characteristics of a self-designed AWP using numerical simulation. First, the geometric configuration and operating principle of the propulsor are introduced. Subsequently, the numerical model and computational methodology are described, including the construction of the computational domain, the grid generation strategy, the selection of the turbulence model, and the specification of boundary conditions. On this basis, the external performance of the propulsor is analyzed and the internal flow losses are quantitatively evaluated. The underlying loss mechanisms are then interpreted by examining the evolution of three-dimensional flow structures in conjunction with the radial equilibrium equation. Furthermore, a detailed analysis of the internal flow field is conducted, focusing on the static pressure distribution, limiting streamlines, and vortex structures in key flow regions in order to identify the dominant loss mechanisms. Finally, the principal conclusions and their engineering implications are summarized.

2. Configurations and Numerical Setup

2.1. Geometry of Waterjet Pump and Working Principle

The sketch of AWP is illustrated in Figure 1a. The AWP primarily consists of an impeller and a diffuser, as illustrated in Figure 1b. The impeller has a rated rotational speed of 950 rpm and rotates clockwise when viewed from bottom to top. The maximum outer diameter of the diffuser is 1204 mm, and the principal geometric parameters are summarized in Table 1. Fluid enters the propulsor from the bottom and gains both kinetic and pressure energy after passing through the mixed-flow impeller. The flow then moves circumferentially from the impeller outlet into the diffuser, where the gradual expansion of the channel cross-section converts kinetic energy progressively into pressure energy. Upon entering the nozzle section, the channel area contracts rapidly, reconverts pressure energy into kinetic energy, and the fluid is ultimately discharged as a high-velocity jet. The thrust generated by the propulsor results from the momentum difference between the outflow and the incoming flow, with the thrust direction opposite to the jet direction at the nozzle outlet. The diffuser can rotate about the impeller axis, enabling omnidirectional control of the thrust vector. As shown in Figure 1b, three nozzles with unequal outlet areas are arranged at the bottom of the diffuser. This design follows the findings of Zhang

et al. [21], which indicate that an appropriate nozzle area ratio can effectively reduce thrust misalignment and enhance the stability of the thrust vector. Detailed geometric parameters of the propulsor are listed in Table 2.

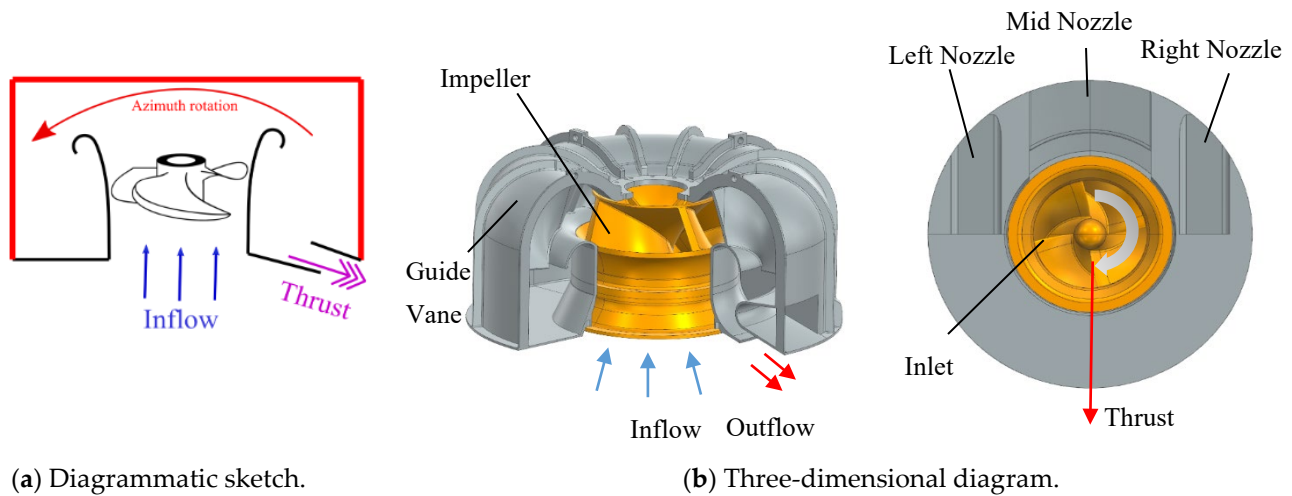


Figure 1. Main components of azimuth waterjet propulsion and working principle.

Table 1. Summary of global parameters.

Parameter	Value
Impeller rotational speed (rpm)	950
Propulsion outer diameter (mm)	1204
Motor power (kW)	280
Bollard Thrust (kN)	1.63

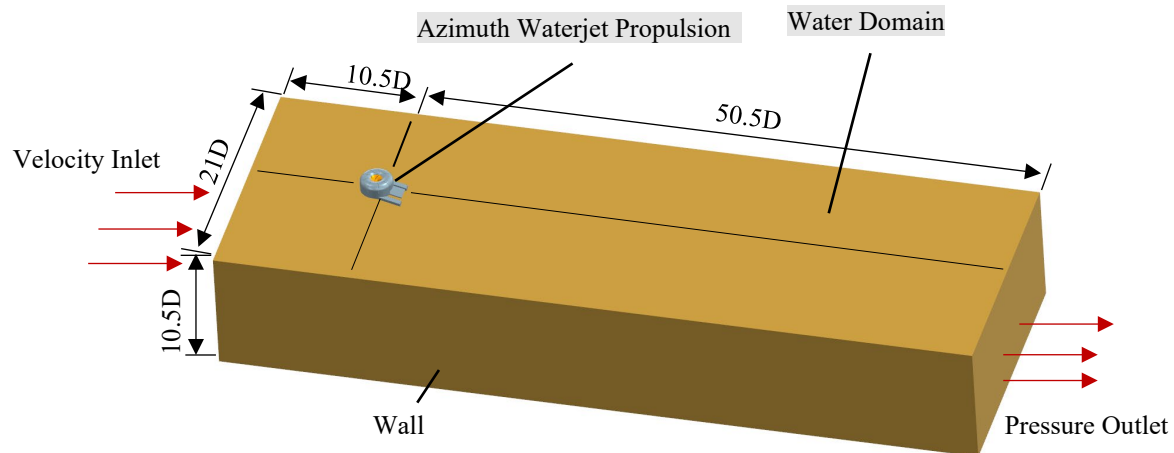
Table 2. Summary of geometric parameters.

Parameter	Value
Impeller inlet diameter (mm)	450
Impeller type	Mixed flow
Number of blades	4
Reference diameter D (mm)	540
Nozzle area (mm ²)	70,653
Left nozzle area ratio (%)	29
Mid nozzle area ratio (%)	35.5
Right nozzle area ratio (%)	35.5

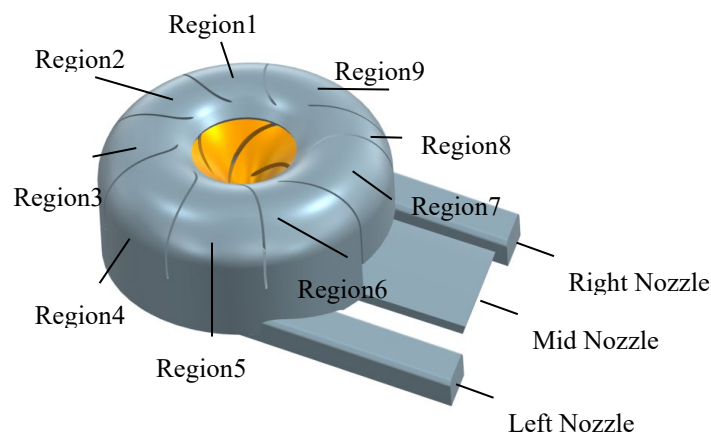
2.2. Computational Domain and Mesh Generation

To systematically investigate the external performance and internal flow mechanisms of the AWP, a three-dimensional computational domain encompassing both the propulsor and the surrounding fluid region was established, as illustrated in Figure 2a. Following the domain sizing approach proposed by Yan Peng et al. [1], the impeller inlet diameter (D) was adopted as the characteristic length. The upstream boundary was positioned at a distance of $10.5D$ from the propulsor center, while the downstream boundary was located $50.5D$ away, thereby minimizing the influence of outlet boundary conditions on the internal flow field. The transverse width and vertical depth of the computational domain were set to $21D$ and $10.5D$, respectively, with the propulsor located at the transverse center. Figure 2b illustrates the detailed layout of the internal stator domain within the propulsor. To facilitate subsequent quantitative analysis of loss distribution and its origins, the internal flow region

was divided into nine subregions, labeled Region 1 to Region 9 in a counterclockwise direction, starting from the passage adjacent to the right nozzle.



(a) Computational domain and boundary conditions.



(b) Naming of each region for azimuth waterjet propulsion.

Figure 2. Computational domain and boundary conditions.

The computational grid was generated using the polyhedral meshing algorithm in the commercial CFD software ANSYS Fluent 2022 R1 Meshing. Polyhedral meshes are increasingly favored in engineering flow simulations because of their superior numerical stability and convergence characteristics in complex geometries. Compared with conventional tetrahedral or hexahedral meshes, polyhedral meshes can significantly reduce the number of elements and computational resource requirements while maintaining solution accuracy [22,23].

To ensure that the numerical results were independent of grid resolution, a grid independence study was conducted. Head and thrust were selected as the primary evaluation metrics, and simulation results obtained with different mesh densities were compared, with those from Grid 4 serving as the normalization reference, as shown in Figure 3. The differences in head and thrust between Grid 0 and Grid 1 are approximately 0.2% and 7%, respectively. In contrast, the differences among Grid 1, Grid 2, Grid 3, and Grid 4 are all below 1%.

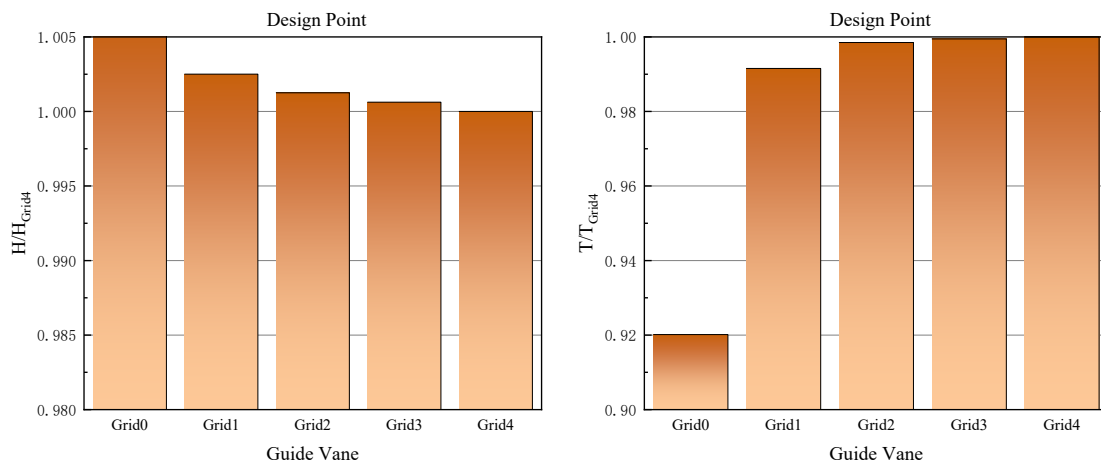


Figure 3. Grid independence verification.

Further error analysis of the numerical simulations was conducted using the Grid Convergence Index (*GCI*) method. The *GCI* approach provides a standardized and quantitative evaluation of discretization errors and numerical convergence in CFD simulations [24,25]. The corresponding formulations are expressed as follows.

The grid convergence error ε is defined as:

$$\varepsilon = \frac{f_1 - f_2}{f_1} \quad (1)$$

$$\varepsilon_{21} = f_2 - f_1$$

where f_1 and f_2 represent the converged solutions obtained from the fine and coarse grids, respectively.

The grid refinement ratio $r_{h,k+1}$ is defined as:

$$r_{h,k+1} = \frac{h_k}{h_{k+1}} \quad (2)$$

where h_k denotes the representative grid spacing of a given mesh, and h_{k+1} is the average grid spacing after one level of refinement.

The characteristic grid size for each mesh can be calculated by:

$$h_k = 3 \sqrt{\frac{\sum_{i=1}^{N_A} \Delta V_i}{N_k}} \quad (3)$$

where N_k is the total number of nodes in the k -th grid system, and ΔV_i represents the volume of each computational cell.

For the same physical model, the number of grid nodes varies among different mesh systems. Therefore, the grid refinement ratio can be simplified as Equation (4):

$$r_{h,k+1} = \frac{h_k}{h_{k+1}} = 3 \sqrt{\frac{N_{k+1}}{N_k}} \quad (4)$$

where N_{k+1} denotes the total number of nodes in the refined mesh.

The apparent order of convergence m is defined as:

$$m = \frac{\ln\left(\frac{\varepsilon_{32}}{\varepsilon_{21}}\right)}{\ln(r)} \quad (5)$$

where ε_{32} represents the convergence error between the third and second mesh systems, while ε_{21} denotes the convergence error between the second and first mesh systems.

The Grid Convergence Index (*GCI*) is calculated as:

$$GCI = F_s \frac{|\varepsilon|}{r^m - 1} \quad (6)$$

where F_s is the safety factor. When the *GCI* is estimated using two grid levels, $F_s = 3$; when three or more grid levels are employed, $F_s = 1.25$. The refinement ratio r is taken as 2. The apparent order of convergence m is determined from the first three mesh systems, yielding $m_H = 1.58$ and $m_T = 3.36$.

The *GCI* results for the five mesh systems are summarized in the following tables.

As shown in Tables 3 and 4, the *GCI* values calculated using both the head and thrust coefficients are below 3%, and gradually decrease with increasing grid density, indicating satisfactory grid convergence. Considering both numerical accuracy and computational cost, the selected mesh scheme was deemed sufficiently reliable for the subsequent simulations. Grid 1 was selected as the baseline mesh for all subsequent simulations. The cell counts for each region were listed in Table 5. As illustrated in Figure 4, local mesh refinement was applied near the leading edge, trailing edge, and blade fillet regions to more accurately resolve high-shear flows and localized separation phenomena. The total number of cells in the final computational domain was approximately 3.59×10^6 .

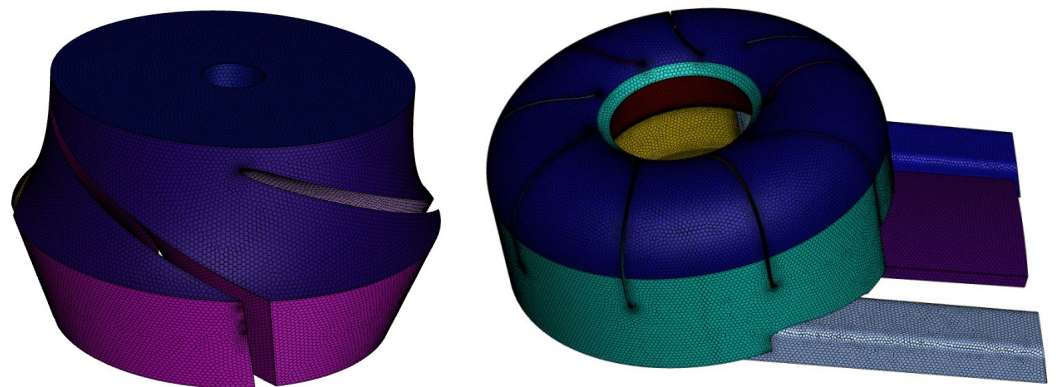


Figure 4. Computational meshes.

Table 3. *GCI* calculation based on the head coefficient.

	H	ε_H	m_H	$GCI_H/\%$
Grid0	16.06	0.003736	1.58	0.23
Grid1	16.00	0.00125	1.58	0.07
Grid2	15.98	0.000626	1.58	0.04
Grid3	15.97	0.000626	1.58	0.04
Grid4	15.96	—	1.58	—

Table 4. *GCI* calculation based on the thrust coefficient.

	T	ε_T	m_T	$GCI_T/\%$
Grid0	14,023	0.077587	3.36	1.046
Grid1	15,111	0.007015	3.36	0.094
Grid2	15,217	0.000986	3.36	0.013
Grid3	15,232	0.000525	3.36	0.007
Grid4	15,240	—	3.36	—

Table 5. Summary of mesh numbers in different domain.

Domain	Mesh Numbers
Impeller	1,270,000
Diffuser	1,570,000
Water domain	750,000
Total	3,590,000

2.3. CFD Method and Boundary Conditions

The numerical simulations were performed using ANSYS Fluent 2022R1. The SST $k-\omega$ turbulence model was adopted. The all y^+ wall treatment which is suitable for a wide range of near-wall mesh densities was applied. Pressure-velocity coupling was handled using the SIMPLEC algorithm, and second-order upwind discretization schemes were applied to the transport equations for turbulent kinetic energy and turbulent dissipation rate. The SST $k-\omega$ turbulence model was selected because of its proven performance in turbomachinery flows involving adverse pressure gradients and flow separation. However, the diffuser flow considered in this study exhibits strong curvature, secondary-flow development, and localized three-dimensional separation. As a Reynolds-averaged turbulence model, SST $k-\omega$ primarily resolves the time-averaged flow field and may not fully capture transient vortex interactions and turbulence anisotropy. Therefore, the present results should be interpreted as representing the dominant mean-flow structures and loss mechanisms.

Since the present work focuses on the evaluation of mean hydraulic performance and time-averaged loss characteristics of the AWP system, the steady-state MRF approach was considered sufficiently accurate and computationally efficient. The impeller region was defined as a rotating domain, while the remaining regions were treated as stationary. The boundary conditions were illustrated in Figure 2. The inlet of the computational domain was specified as a velocity inlet with a design velocity of 1 m/s. The Reynolds number based on the impeller diameter and inlet velocity is approximately 5.3×10^5 . A turbulence intensity of 3%. Both the outer boundary of the computational domain and the three nozzle outlets were set as pressure outlets with a reference pressure of 1 atm. All remaining boundaries were defined as no-slip wall conditions.

3. Results and Discussion

3.1. Performance Curve

The AWP system can be analogously treated as a pump, in which the interface between the impeller and the external water domain serves as the pump inlet, while the three nozzles act as the pump outlet. Because thrust is the most critical performance metric for such propulsion systems, Equation (7) is introduced to non-dimensionalize the thrust, enabling consistent comparison with other waterjet propulsion devices. The pump head is calculated from the mass-averaged total pressure difference between the inlet and outlet sections, and the hydraulic efficiency of the pump is subsequently determined using Equation (8). A similar approach is applied to evaluate the impeller efficiency. The numerical results, shown in Table 6, indicate that the thrust coefficient, pump efficiency, and impeller efficiency exhibit clear trends with increasing advance speed.

Under the mooring condition (advance speed of 0 m/s), the thrust coefficient reaches its maximum value of 0.73 and gradually decreases as the advance speed increases. The mooring thrust coefficient is approximately 0.13 lower than that reported for another AWP studied in reference [1]. The impeller efficiency remains consistently above 89%, reaching a peak value of 96% at the design speed of 1 m/s, after which it gradually decreases with further increasing speed. In contrast, the overall hydraulic efficiency of the pump attains its maximum at the design speed but reaches only 52%, indicating substantial energy losses

within the diffuser section. The corresponding loss ratio ranges from approximately 41% to 45%. It is significantly lower than the typical hydraulic efficiency range (80–93%) of conventional mixed-flow pumps equipped with guide vanes [26,27], further highlighting the severity of internal flow losses within the diffuser.

Table 6. Relationship of CFD predicted thrust coefficient, impeller efficiency and pump efficiency vs. ship speed.

	0 m/s	1 m/s	2 m/s	4 m/s	6 m/s	8 m/s	10 m/s
KT	0.73	0.72	0.69	0.63	0.62	0.60	0.58
Pump efficiency (%)	51.3	51.8	51.4	49.2	47.5	47.2	46.5
Impeller efficiency (%)	95.9	96.1	95.1	93.2	92.2	90.7	89.0

$$K_T = \frac{T}{\rho n^2 D^4} \quad (7)$$

$$H = \frac{\rho g Q H}{P} \quad (8)$$

In these equations, T denotes thrust, ρ is the fluid density, n is the rotational speed, D is the reference diameter, Q represents the flow rate, H is the head, P is the input power, and g is the gravitational acceleration.

3.2. Proportion of Losses in Each Computational Domain

A quantitative comparison of the losses in the impeller and diffuser regions (Figure 5a) show that the primary hydraulic losses in the AWP are concentrated in the diffuser. This observation is highly consistent with the pump efficiency distribution characteristics shown in Table 6. To further clarify the physical origins of these losses, the loss estimation model proposed by Zhong Xu [28] is employed to quantify the various loss components inside the diffuser. According to this model, the internal losses within the flow passage can be categorized into four primary components: friction loss, shock loss, secondary flow loss, and wake loss. Friction loss originates from wall shear stress and viscous dissipation. Shock loss is caused by incidence mismatch between the incoming flow and blade angle. Secondary flow loss arises from three-dimensional cross-passage motions induced by pressure gradients and flow turning. Wake loss is generated by the mixing and dissipation of low-momentum fluid in the blade trailing-edge wake. The corresponding estimation formulas are provided in Equations (9)–(12).

$$h_f = \iint \lambda \rho \frac{v^2}{2} dS \quad (9)$$

$$h_c = k(v_e^2 - v_h^2) \quad (10)$$

$$h_m = \xi_m \frac{v^2}{2g} \quad (11)$$

$$h_s = C_1 \frac{(v_\infty - v_{1q})^2}{2g} \quad (12)$$

In these equations, h_f represents the friction loss, where λ denotes the friction resistance coefficient and v is the characteristic fluid velocity. h_c refers to the secondary flow loss, in which k is the secondary flow loss coefficient, while v_e and v_h represent the velocities at the hub and shroud, respectively. The wake loss h_m is characterized by the wake loss coefficient ξ_m and the velocity v_2 downstream of the blade trailing edge. The shock loss h_s is defined using the shock loss coefficient C_1 , where $v_{1\infty}$ denotes the relative velocity at the

blade leading edge under undisturbed conditions and v_{1q} represents the velocity after flow displacement by the blade.

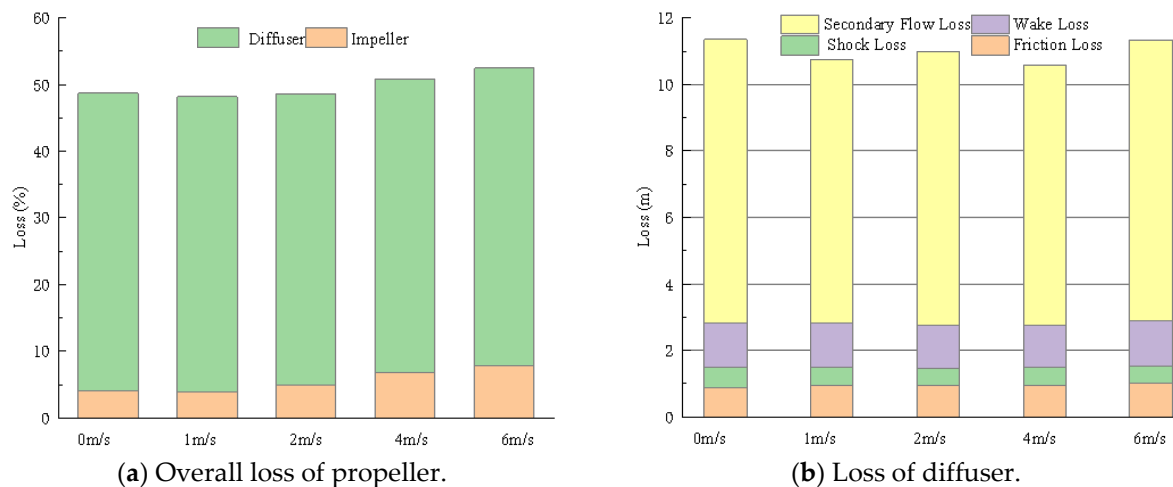


Figure 5. Loss of impeller diffuser and various types of losses inside the diffuser.

The empirical loss model proposed in Ref. [28] was originally developed for centrifugal compressor diffusers. However, the model fundamentally characterizes hydraulic loss mechanisms associated with rotating diffuser flows, including friction, diffusion-induced separation, and secondary-flow losses, which are governed primarily by flow curvature, adverse pressure gradients, and boundary-layer development rather than compressibility effects. In the present azimuth waterjet propulsion system, the flow operates under low-Mach-number incompressible conditions, where compressibility effects are negligible. Therefore, the dominant loss mechanisms remain physically consistent with those described in Ref. [28].

Based on the applicability assessment of the present model to the current geometry, the corresponding coefficients were determined as follows. The characteristic velocity v was defined as the impeller inlet velocity. The friction coefficient λ was set to 0.02, the empirical coefficient k was assigned a value of 0.06, the coefficient ζ_m was taken as 1, and the coefficient C_1 was specified as 0.16. Figure 5b presents the quantitative distribution of the various loss components within the diffuser under different advance speeds (0, 1, 2, 4, and 6 m/s). Taking the design speed of 1 m/s as an illustrative example, the secondary flow loss is approximately 7.8 m, accounting for 72% of the total loss. The wake loss is about 1.7 m, representing 16%; the shock loss is roughly 0.4 m, contributing 4%; and the friction loss is approximately 0.8 m, accounting for 8%. These results indicate that secondary flow loss is the dominant component within the diffuser, whereas the other loss mechanisms collectively contribute only a relatively small proportion.

From a mechanistic perspective, secondary flow within guide vane passages is typically driven by the static pressure gradient across the passage, accompanied by the migration and separation of low-energy fluid. Therefore, the subsequent analysis focuses on the internal flow structure within the guide vane region to elucidate the formation mechanism and evolutionary characteristics of secondary flow losses.

3.3. Spanwise Velocity

It should be noted that the geometry of the guide vane in the AWP differs substantially from that of conventional mixed-flow pumps. As illustrated in Figure 6a, the primary flow path in a conventional mixed-flow pump transitions smoothly along the axial direction and exits in a consistent orientation, resulting in a relatively continuous flow trajectory with limited deflection. In contrast, the main flow within the AWP undergoes significant

spatial torsion after passing through the diffuser. The flow enters axially but ultimately exits in the opposite direction. In the mid-span region of the guide vane, the incoming flow experiences a nearly 180° deflection, exhibiting pronounced three-dimensional curvature characteristics.

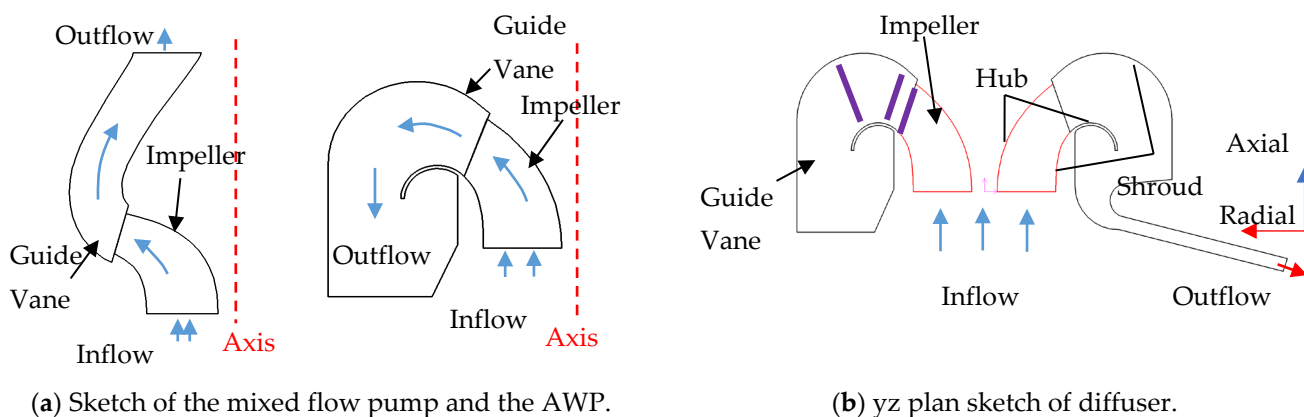


Figure 6. Sketch of the middle section between the traditional mixed flow pump and the azimuth waterjet propulsion.

It should also be noted that, due to the downward orientation of the impeller axis and the geometric turning of the diffuser passage, the hub side of the impeller corresponds to the shroud side of the diffuser passage in the three-dimensional configuration, while the shroud side of the impeller corresponds to the hub side of the diffuser passage, as illustrated in Figure 6b. In this study, Region 3 is selected as a representative case to compare the internal flow characteristics at three critical sections: the impeller outlet, the guide vane inlet, and the midspan section of the guide vane (indicated by the purple curve in Figure 6b).

The spanwise velocity distributions are presented in Figure 7. At the impeller outlet, the spanwise velocity increases monotonically from the hub to the shroud, with an inflection point observed at a span of approximately 0.35. Upon entering the guide vane inlet, the overall spanwise momentum decreases significantly: at span = 0, the velocity drops from 2.6 m/s to 0.4 m/s, while at span = 1 it decreases from 6 m/s to 1.6 m/s. However, in the midspan region of the guide vane, the spanwise imbalance intensifies again within the span range of 0.55–1, reaching -3.7 m/s at the hub. Consequently, the spanwise momentum within the guide vane passage exhibits a trend of initial attenuation followed by subsequent amplification.

The renewed enhancement of spanwise velocity provides the direct dynamic driving mechanism for the formation of counter-rotating vortices (CV). Counter-rotating vortices (CV) are vortex pairs with opposite rotational directions formed by spanwise momentum imbalance and secondary-flow migration within the guide-vane passage. The interaction between the two vortices enhances transverse entrainment and mixing, leading to increased local energy dissipation and loss generation. High-energy fluid in the mid-shroud region migrates laterally in the spanwise direction and, under the influence of the spanwise pressure gradient across the blade passage, converges toward the hub region. Meanwhile, the intensified spanwise momentum near the hub interacts strongly with the mainstream flow, thereby inducing CV structures and their associated three-dimensional vortical patterns. Therefore, it can be inferred that the development of secondary flows and CV within the guide vane are closely related to the non-uniform spanwise velocity distribution at the impeller outlet and its subsequent redistribution within the guide vane passage.

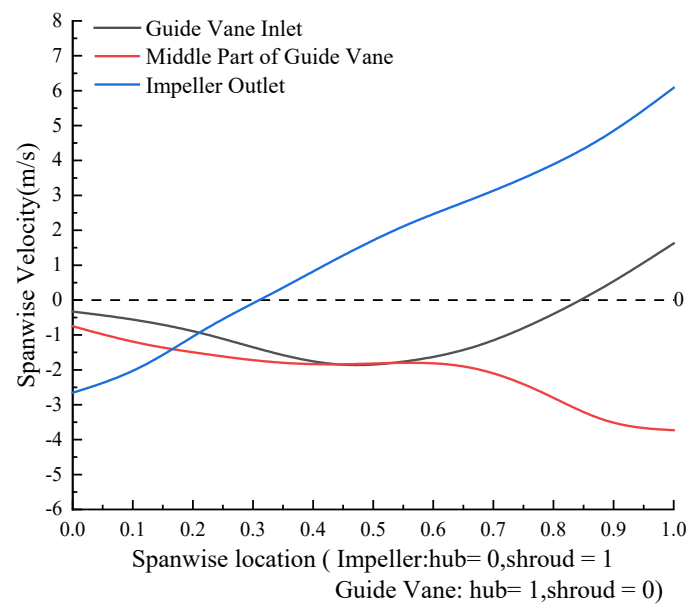


Figure 7. Spanwise velocity map at three positions, positive values are in the positive z-axis direction.

3.4. Radial Equilibrium Equation

To further investigate the influence of the nearly 180° bending of the guide vane on the internal flow of the AWP, the radial equilibrium equation is introduced. Based on the assumptions of axisymmetry, steady flow, negligible axial velocity gradients, and negligible body forces, the radial equilibrium equation (Equation 13) can be expressed as follows:

$$F_r + \frac{1}{\rho} \frac{dp}{dr} = \frac{v_u^2}{r} + \frac{v_m^2}{r_c} \quad (13)$$

In this equation, F_r represents the radial component of the blade surface lift reaction force; dp/dr denotes the spanwise pressure gradient from the shroud to the hub; $v_u^2/2$ corresponds to the centripetal force required for the fluid to maintain rotational motion in the absolute reference frame; and v_m^2/r_c represents the centripetal force associated with the curvature of the meridional streamline.

Originally developed for flow-field analysis in axial-flow pumps, the radial equilibrium equation fundamentally describes the balance among inertial forces, body forces, and pressure gradients in the direction normal to the primary flow. When extended to mixed-flow impellers, the inconsistency between the mainstream direction and the geometric radial direction modifies the physical interpretation of its terms. Specifically, F_r becomes the spanwise component of the blade surface lift reaction. Because the pressure gradient term remains oriented in the radial direction, the conventional centrifugal force term associated with rotational motion no longer dominates the equilibrium. Instead, due to the altered streamline curvature, the inertial term induced by streamline curvature becomes the primary balancing component. Consequently, with the rotational centrifugal force term effectively diminished, a new equilibrium is established between the inertial force associated with streamline curvature and the spanwise pressure gradient, giving rise to distinctive flow characteristics within the AWP. Therefore, Equation (13) can be rewritten as Equation (14).

$$F_r + \frac{1}{\rho} \frac{dp}{dr} = \frac{v_m^2}{r_c} \quad (14)$$

Although the flow field in the diffuser exhibits a certain degree of asymmetry and local recirculation, which deviates from the assumptions of the classical radial equilibrium equation, Equation (14) is not intended to serve as a rigorous governing equation for the

three-dimensional flow field in the present study. Instead, it is employed as a first-order physical interpretation of the dominant relationship between the radial pressure gradient and centrifugal effect associated with streamline curvature. Although local flow separation and secondary flow may introduce deviations from the simplified formulation, the relative variation of the terms on both sides of the equation can still provide qualitative insight into the evolution of flow behavior within the diffuser.

Based on the three characteristic cross-sections, shown in Figure 6b, a quantitative comparison between the spanwise pressure gradient term and the centrifugal force term induced by streamline curvature is presented in Figure 8. The results indicate that, in the impeller outlet region, the two terms differ significantly in magnitude. The centrifugal force term remains close to zero across the span range of 0–0.5 and gradually increases between 0.5 and 1.0. Meanwhile, its deviation from the pressure gradient term continues to increase, reaching a maximum difference of approximately 600 m/s^2 at the shroud (span = 1). It suggests the presence of residual spanwise momentum at the impeller outlet, which is consistent with the non-uniform spanwise velocity distribution discussed in Section 3.3. At the guide vane inlet, the peak values of both terms decrease, indicating substantial dissipation of spanwise momentum within the rotor-stator interaction region. Within the span range of 0–0.7, the pressure gradient term is significantly weakened, suggesting a localized regulating effect near the guide vane inlet. Nevertheless, a considerable discrepancy between the two terms persists within the span range of 0.7–1.0, indicating that the spanwise momentum imbalance is not fully eliminated but instead propagates downstream into the guide vane passage with the main flow. At this stage, the net driving force becomes negative, corresponding to fluid motion directed toward the hub.

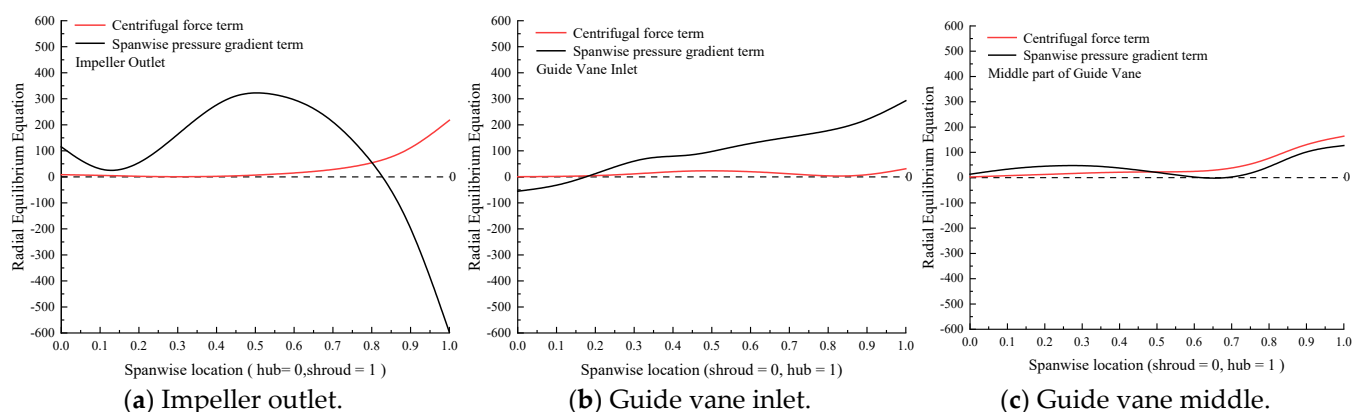


Figure 8. Radial Balance Curve Diagram.

In the midspan region of the guide vane, the magnitudes of the two terms become comparable within the span range of 0–0.7, indicating partial redistribution of spanwise momentum. However, within the span range of 0.7–1.0, although the two terms remain similar in magnitude, they do not converge to zero. This suggests that the spanwise momentum imbalance is not completely eliminated but instead evolves into a quasi-equilibrium state, in which the reverse flow reorganizes into stable secondary flow structures. This observation is consistent with the increasing trend of spanwise velocity near the hub reported in Section 3.3.

Combining the results from Figures 9 and 10, it is evident that neither the inlet nor the midspan region of the guide vane effectively eliminates the spanwise momentum inherited from the impeller outlet. Moreover, the presence of the nearly 180° bend removes the conventional rotational centrifugal force term from the radial equilibrium equation. As a result, persistent discrepancies between the spanwise pressure gradient and the

centrifugal driving force re-emerge within the guide vane passage, leading to the formation of secondary flow structures and new characteristic flow patterns. Under these conditions, the large spatial deflection acts as the macroscopic structural feature that drives both the redistribution of spanwise momentum and the persistent imbalance of radial forces.

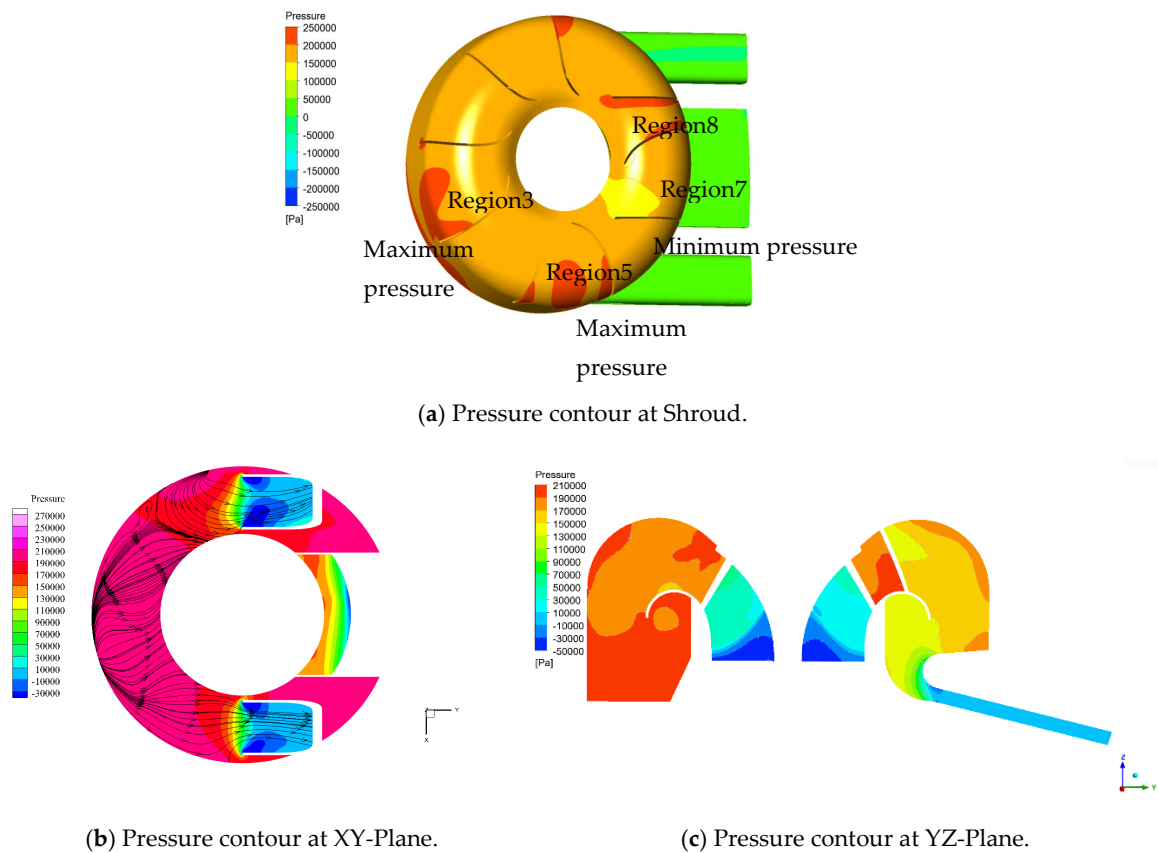


Figure 9. Pressure contours at different locations of the diffuser.

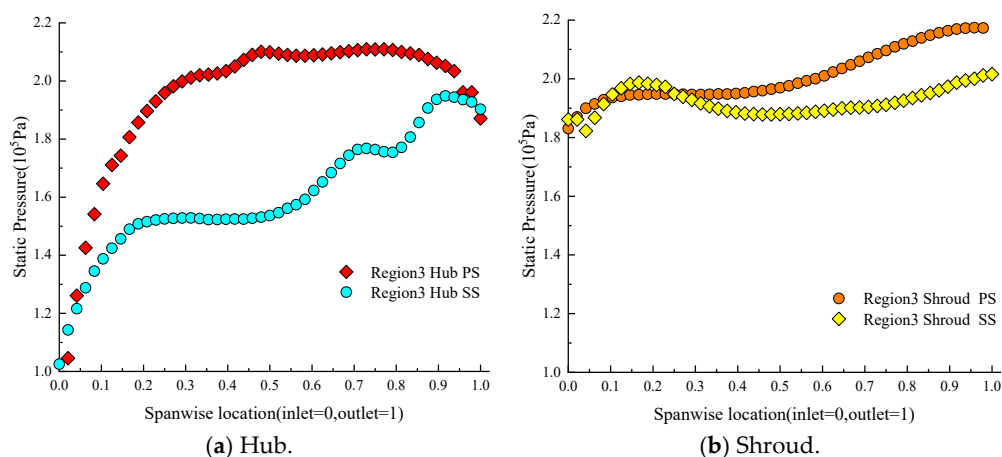


Figure 10. Static pressure curves of pressure and suction surfaces in region 3.

Consequently, the nearly 180° turning of the flow direction in the midspan region of the guide vane (Figure 6b) alters the overall flow topology, preventing the complete dissipation of the spanwise momentum inherited from the impeller. Instead, it further promotes the development of secondary flows and vortical structures, thereby intensifying energy dissipation. This flow behavior corresponds directly to the high loss proportion

identified in the diffuser section and provides a structural-dynamic coupling explanation for the underlying high-loss mechanism in the AWP.

3.5. Analysis of Internal Flow Losses in Region 3

To identify regions with significant static pressure gradients within the diffuser, Figure 9 presents static pressure contour distributions on several representative cross-sections. The results indicate pronounced non-uniformity in the internal flow field. As shown in Figure 9a, local high-pressure extrema is observed in Region 3, Region 5, and Region 8, whereas a low-pressure extreme appears in Region 7. The static pressure contours and streamline distribution on the xy-plane (Figure 9b) further reveal the presence of flow blockage and backflow structures in several passages. The yz-plane shown in Figure 9c, located between Region 3 and Region 7, highlights the asymmetric characteristics and localized extrema of the static pressure field inside the diffuser. Based on these observations, four passages exhibiting significant pressure anomalies—Region 3, Region 5, Region 7, and Region 8—were selected for detailed investigation of their internal flow structures. Furthermore, given the similar characteristics of Region 3 and Region 5, Regions 3, 7, and 8 were ultimately selected for detailed analysis of their internal flow structures.

From a theoretical standpoint, the primary function of a diffuser is to convert the kinetic energy of the fluid into static pressure through the gradual expansion of the flow passage area. Consequently, the static pressure is expected to increase monotonically along the flow direction, while the flow velocity decreases accordingly. However, due to the presence of the nozzle structure downstream of the diffuser, the static pressure distribution near the diffuser trailing edge may be affected by nozzle-induced acceleration, which can produce localized pressure reductions. Figure 10 illustrates the static pressure distribution along the streamwise direction in Region 3. The horizontal axis represents the normalized position (0–1) from inlet to outlet, while the vertical axis denotes the static pressure. Although the overall trend generally conforms to the theoretical expectation of pressure recovery, noticeable local fluctuations are present. Specifically, as shown in Figure 10a, the pressure surface (PS) exhibits mild fluctuations near the 0.5-span location, whereas the suction surface (SS) remains relatively smooth within the spanwise range of 0.2–0.5 but displays pronounced oscillations between 0.5 and 0.9 span. In addition, Figure 10b shows that the SS presents an S-shaped pressure variation within the span range of 0–0.2, indicating the presence of localized negative work effects associated with complex flow interactions.

To further clarify the physical mechanisms responsible for these pressure fluctuations, the internal flow structures are examined from three perspectives: limiting streamline topology, flow separation behavior, and vortex structure identification.

As shown in Figure 11, the limiting streamlines on the shroud surface clearly reveal that the inlet vortex on the PS produces a significant blockage effect within the passage. After interacting with the incoming flow, the backflow vortex undergoes circumferential deflection and impinges on the adjacent SS at a relatively large incidence angle. This interaction triggers flow separation in the mid-span region of the SS and generates a spiral separation vortex around the spiral node (yellow in Figure 11c). Meanwhile, the large-scale backflow on the PS interacts with the mainstream flow to form vortical structures with opposite rotational directions (green in Figure 11c). These vortices subsequently develop downstream and evolve into a counter-rotating vortex pair (CV), as illustrated in Figure 11c.

The CV occupies a large portion of the flow passage. The vortex located on the SS side (yellow) drives low-energy fluid to migrate spanwise from the shroud toward the hub, as indicated by the streamline convergence region on the SS side in Figure 11a. This migration induces open flow separation and subsequently develops into the induced vortex identified

in Figure 11c (red). Meanwhile, the vortex located on the PS side (green) partially blocks the PS-side passage, forcing the incoming flow to interact with the backflow in the mid-channel region. This interaction generates a spiral separation vortex in the PS-hub corner region.

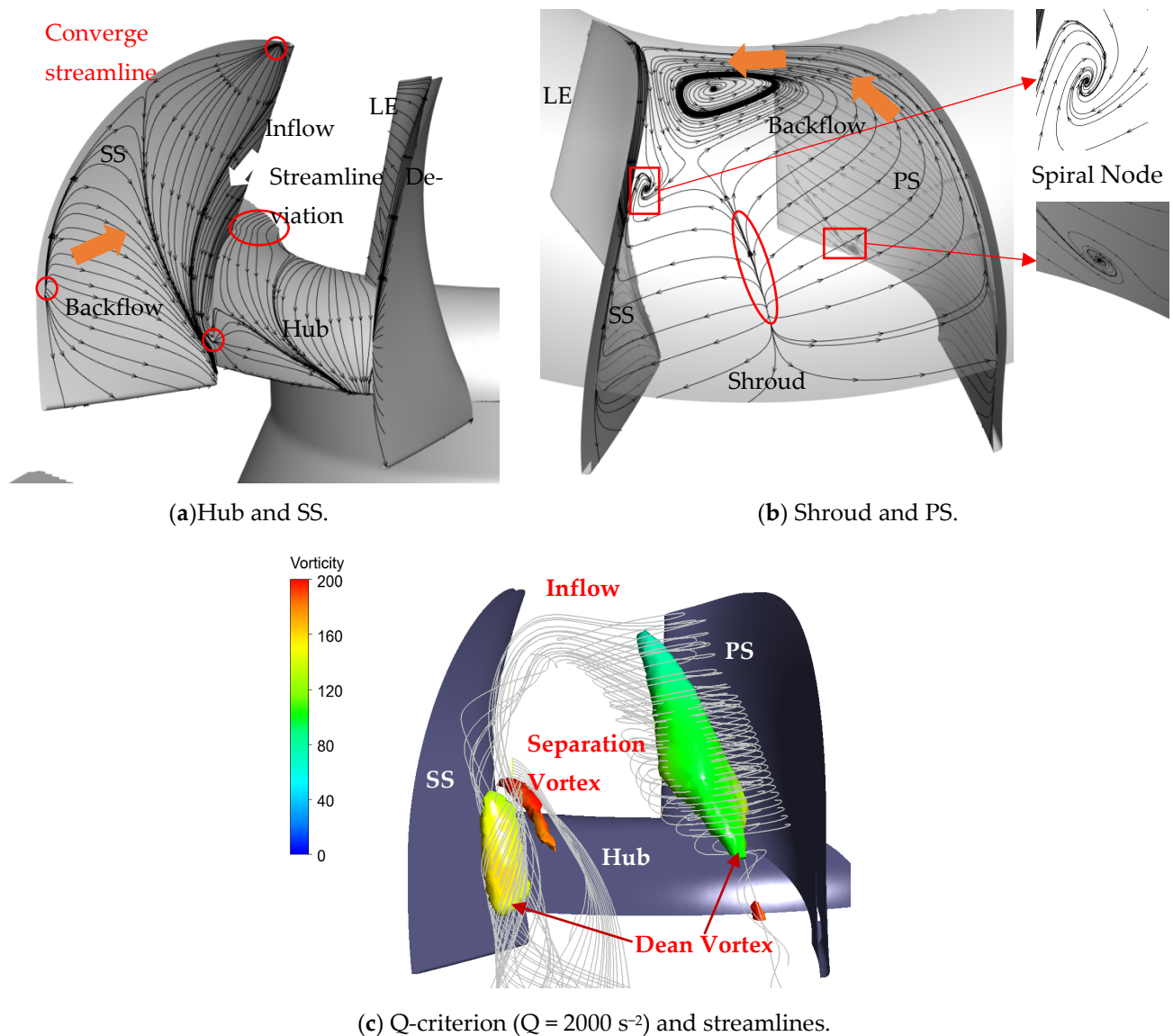


Figure 11. Region 3 Extreme Streamline and identification of vortices using the Q-criterion.

Based on these observations, a vortex-structure model can be established consisting of four characteristic vortical components:

- I—*inlet recirculation vortex*;
- II—*counter-rotating vortex pair* (PS-side green vortex and SS-side yellow vortex);
- III—*induced vortex* (SS hub region);
- IV—*spiral separation vortex* (PS hub region).

By correlating these vortex structures with the shroud static pressure curve shown in Figure 10b, a clear relationship between vortex evolution and static pressure variation can be identified. The inlet recirculation vortex (I) produces local blockage and flow stagnation within the passage, resulting in a peak in the SS-side static pressure curve at span ≈ 0.2 . Subsequently, the formation of the SS-side vortex (yellow vortex II) is accompanied by the development of a localized low-pressure region, causing a rapid decrease in static pressure downstream of the peak. As the CV continues to develop downstream, the effective

flow area becomes further restricted, leading to renewed pressure accumulation and a corresponding increase in the SS static pressure curve.

In the hub region, the spiral separation vortex (IV) induces noticeable fluctuations in the PS-side static pressure curve around span ≈ 0.5 . As illustrated in Figure 11a, the strong streamline deviation—and even local flow reversal—occurs near the SS inlet region close to the hub. In this region, centrifugal forces act in the same direction as the ambient pressure gradient, producing a superimposed effect that causes a rapid increase in SS-side static pressure at the inlet. Subsequently, the SS-side vortex (yellow vortex II) transports low-energy fluid from the shroud toward the hub, where it mixes with upstream high-pressure fluid. This process reduces the pressure growth rate and results in the plateau-like behavior observed in the SS static pressure curve within the span range of 0.2–0.5. The formation of the induced vortex (III) corresponds to a localized low-pressure region, which coincides with the minimum of the SS static pressure curve at approximately span ≈ 0.8 . After the induced vortex breaks down, the low-energy fluid transported by the wake and secondary flow accumulates near the outlet region, partially blocking the passage and causing the static pressure to rise again. This behavior is consistent with the upward trend of the SS pressure curve downstream of span ≈ 0.8 .

3.6. Analysis of Internal Flow Losses in Region 7 and Region 8

Figure 12 presents the static pressure distribution characteristics along the flow direction within Region 7 and Region 8. Regions 7 and 8 form two geometrically distinctive zones inside the diffuser: the suction surface (SS) of Region 7 and the pressure surface (PS) of Region 8 both correspond to vertically oriented blades. This geometric configuration arises from the nozzle arrangement requirements, where the left and right sidewalls of the central nozzle align with these two vertical blades. Consequently, the cross-sectional area of the flow passage in Region 7 is slightly larger than that in conventional regions, whereas in Region 8 the passage area is smaller and progressively contracts along the streamwise direction. As a result, the static pressure distribution in both regions deviates markedly from typical diffuser flow behavior.

Specifically, as shown in Figure 12a,c, the static pressure on the PS of Region 7 decreases within the spanwise range of 0–0.2, followed by a gradual recovery between spans of 0.2 and 0.6. In contrast, in Region 8, the PS static pressure exhibits only a slight increase near the inlet and then decreases continuously along the streamwise direction, particularly beyond span ≈ 0.4 , reflecting the dominant influence of the contracting passage. On the SS side, Region 7 exhibits relatively smooth variation across the spanwise interval of 0.2–0.8, whereas in Region 8, a pressure increase occurs only within a narrow spanwise range of 0–0.05, followed by a monotonic decrease with slight recovery near span ≈ 0.95 . Figure 12b,d further reveal that, in Region 7, a local minimum appears on the PS at approximately span = 0.03, where the static pressure falls below that on the corresponding SS, indicating the presence of localized negative work on the blade. Beyond approximately span = 0.3, the static pressure on the PS continues to decline, suggesting that the diffusing effect is disrupted and remains effective only within the limited spanwise range of 0–0.3. In contrast, Region 8 shows no pronounced local minimum on the PS; instead, the pressure decreases in a more continuous manner due to geometric contraction. On the SS side, Region 7 exhibits a sharp pressure increase at the inlet followed by a gradual decrease within 0–0.2 span, while Region 8 shows a continuous pressure decline over a wider spanwise range due to persistent flow acceleration in the converging passage.

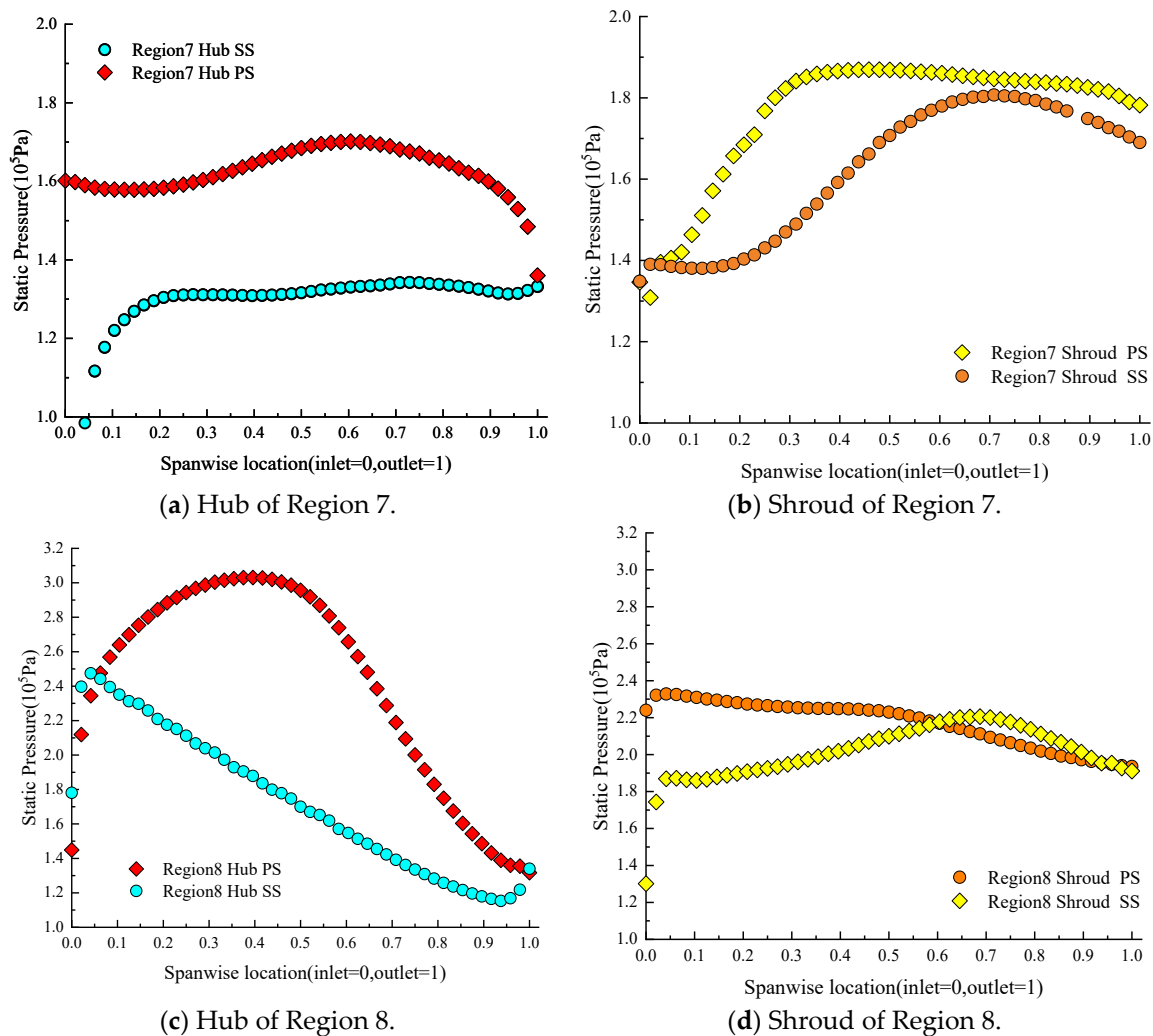


Figure 12. Static pressure curves of pressure and suction surfaces in region 7 and Region 8.

As illustrated in Figure 13a,b, the limiting streamlines in Region 7 reveal a pronounced recirculation vortex structure near the inlet on the PS side in the shroud region. Compared with Region 3, the recirculation vortex in Region 7 is positioned closer to the SS inlet. This shift is attributed to the enlarged flow passage downstream of Region 7, which enhances the diffusion capability and lowers the local static pressure, thereby inducing an upstream displacement of the recirculation vortex. In contrast, as shown in Figure 13c,d, Region 8 no longer exhibits a bidirectional recirculation pattern. The converging streamlines disappear, and only a unidirectional recirculation from the PS toward the SS remains. A distinct recirculation zone forms near the SS inlet, while the flow is subsequently convected downstream along the blade. After mixing with the incoming flow in Region 7, the recirculated fluid undergoes circumferential deflection and impinges on the adjacent SS at a higher incidence angle. This interaction induces flow separation and entrainment in the upper part of the SS, resulting in the formation of a spiral separation vortex. In Region 8, however, although recirculation persists in the shroud region, it remains largely confined near the shroud and propagates downstream in a relatively stable manner due to the absence of strong inflow mixing.

Furthermore, a cross-regional flow coupling exists between the PS of Region 7 and the SS of Region 8. Because Region 8 has a smaller flow passage area, it generally maintains a higher static pressure level. High-energy fluid originating from the SS of Region 8 migrates into the PS of Region 7, where it interacts with the large-scale recirculation and

the mainstream flow. This interaction generates a counter-rotating vortex pair (CV), as illustrated in Figure 14, and produces a spiral node at the PS inlet.

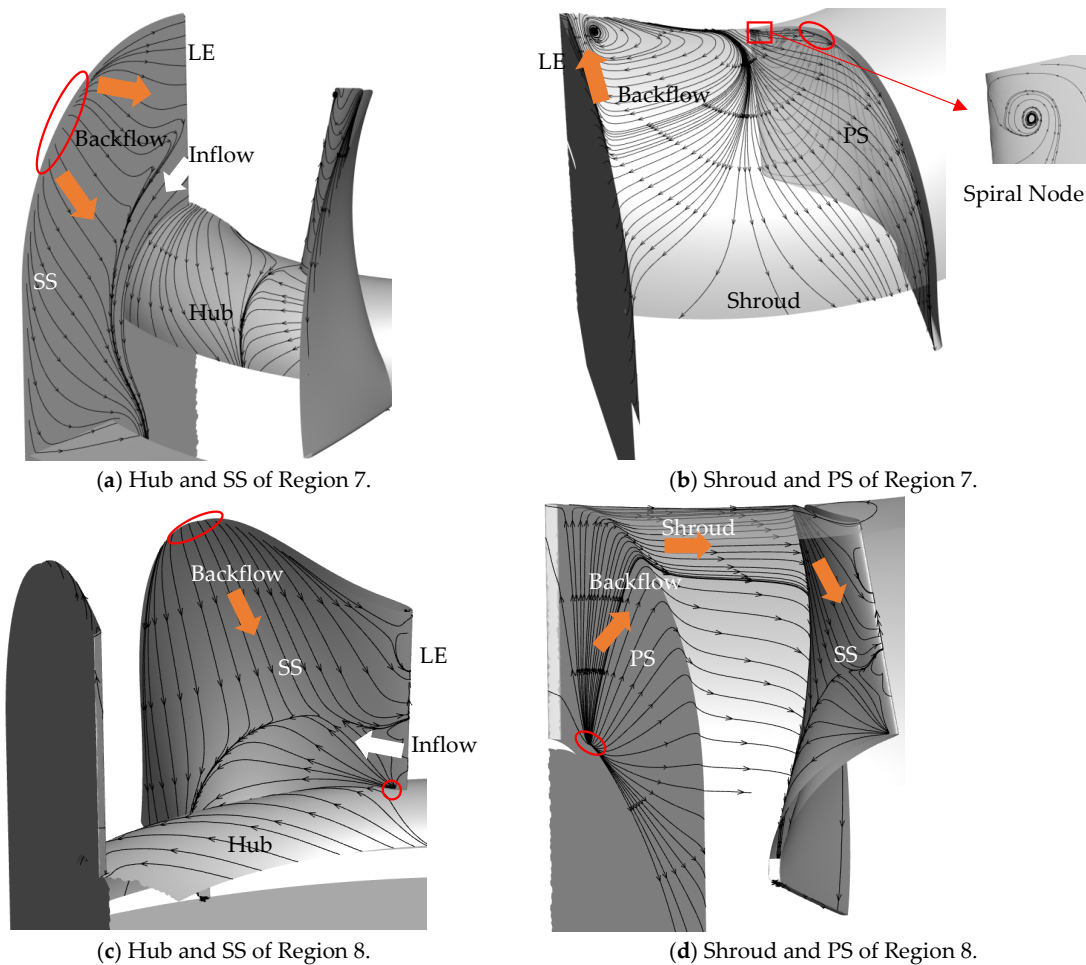


Figure 13. Region 7 and Region 8 Extreme Streamlines.

Based on these observations, a vortex-structure model can be established. In Region 7, three characteristic vortical components are identified: (I) counter-rotating vortex pair, (II) induced vortex in the SS shroud region, and (III) spiral separation vortex in the PS shroud region. In contrast, the vortex structure in Region 8 is comparatively simpler, dominated by a channel vortex formed near the inlet due to the interaction between incoming flow and recirculated fluid, without forming a strong CV system.

By correlating the shroud static pressure curves (Figure 12b,d) with the identified vortex structures, a clear correspondence between vortex evolution and static pressure variation can be identified. In Region 7, the spiral node at the PS inlet produces localized blockage and stagnation, resulting in the minimum pressure at span ≈ 0.03 . Due to cross-regional recirculation from Region 8, the direction of the recirculating flow in Region 7 shifts near span ≈ 0.3 , after which the fluid moves downstream. The persistent blockage effect of the spiral separation vortex causes a continuous decline in static pressure beyond this location. In Region 8, the continuous contraction of the passage leads to a sustained decrease in PS static pressure, while the SS pressure variation is influenced by recirculation-induced energy redistribution.

In the hub region, the static pressure distribution also reflects the differing flow mechanisms. In Region 7, the inlet vortex causes a gradual decrease in SS pressure over the spanwise range of 0–0.2, while the flow remains relatively stable without significant separation. In Region 8, the decreasing passage area leads to a continuous decline in SS

pressure, whereas on the PS, the convergence of incoming flow produces a pressure rise followed by a decrease, forming a saddle point near span ≈ 0.4 .

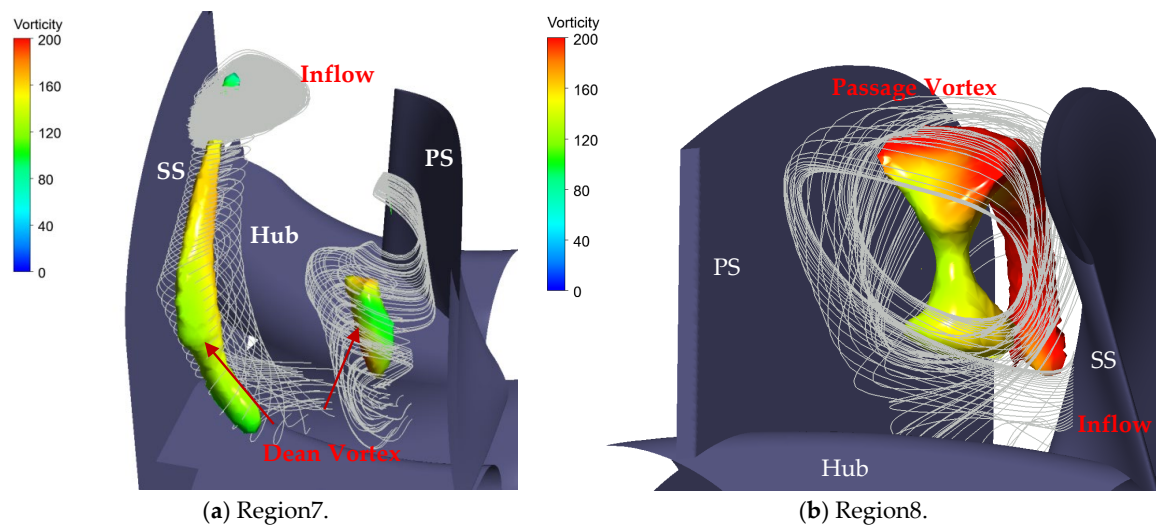


Figure 14. Identification of vortices using the Q-criterion ($Q = 2000 \text{ s}^{-2}$) and streamlines.

4. Discussion

Internal flow loss characteristics and the underlying mechanisms within a self-designed azimuth waterjet propulsion (AWP) system are systematically investigated. Numerical results indicate that the hydraulic efficiency at the design conditions is only 52%, which is significantly lower than the 80–93% range of conventional mixed-flow pumps. Further quantification reveals that the diffuser contributes approximately 80% of the total hydraulic losses, among which secondary flow losses are dominant. Accordingly, detailed analyses of pressure, vortex structures, and secondary flows are carried out in regions 3, 7, and 8 within the diffuser.

From a mechanistic perspective, the development of secondary flow in the diffuser is closely associated with the non-uniform spanwise distribution of momentum and the deviation from spanwise equilibrium. The geometric inclination at the outlet of the mixed-flow impeller produces a pronounced non-uniform spanwise velocity distribution. This imbalance remains observable at the diffuser outlet and the guide vane inlet, and progressively accumulates toward the hub side within the guide vane. Simultaneously, the diffuser passage involves a nearly 180° spatial turning, which substantially enhances streamline curvature. Under these conditions, the conventional radial equilibrium relationship—characterized by a balance between centrifugal force and pressure gradient in a rotating reference frame—evolves into a new inertial equilibrium state, which dominated by the interaction between the pressure gradient and curvature-induced inertial forces. However, this balance does not eliminate the spanwise momentum difference inherited from the impeller outlet. Instead, it promotes additional transverse migration and entrainment within the guide vanes, thereby providing the driving mechanism for the formation of counter-rotating vortex pairs.

From the perspective of flow structures, the aforementioned spanwise imbalance directly triggers complex three-dimensional vortex systems within the diffuser. In regions 3 and 7, a clear convergence streamlines are observed in the mid-shroud region. The fluid recirculates toward both the pressure side (PS) and suction side (SS), and undergoes cross-span migration near the shroud at the guide vane inlet. After merging with the mainstream, local flow separation occurs, leading to the formation of induced vortices. The interaction between these induced vortices and the recirculation vortices originating from the shroud

side forms counter-rotating vortex pairs distributed along the PS and SS, which become the dominant structures responsible for local blockage and energy dissipation.

In contrast, region 8 exhibits a different flow mechanism. The flow passage contracts along the streamwise direction, and the pressure side consists of a vertical blade. As a result, a passage-scale recirculation from the PS toward the SS occurs at the guide vane inlet, forming a spiral separation structure that propagates downstream. This region is characterized by coupled effects of flow passage contraction and channel-scale recirculation, leading to intensified losses.

In summary, the guide vane region is the primary location where spanwise momentum imbalance accumulates, spanwise disequilibrium intensifies, and large-scale vortex structures persist. For regions 3 and 7, particularly attention should be directed to the spanwise pressure distribution within the guide vanes and the optimization of the overall channel curvature. For region 8, emphasis should be placed on the influence of flow passage contraction patterns and vertical blade configurations on flow recirculation and separation structures. These findings provide a reliable physical basis and practical guidance for identifying loss mechanisms and optimizing diffuser design of azimuth waterjet propulsion systems.

A limitation of the present study is that the conclusions are derived solely from numerical simulations without experimental validation for the investigated configuration. Future experimental studies are required to further verify the predicted hydraulic performance and flow structures. It should be noted that the present analysis is based on a steady MRF framework. Consequently, transient phenomena associated with vortex shedding and flow instability may not be fully resolved.

Author Contributions: Methodology, Z.L.; software, Z.L.; formal analysis, P.C.; investigation, P.C.; data curation, P.C.; writing—original draft preparation, Z.L.; writing—review and editing, Z.L.; project administration, P.C. All authors have read and agreed to the published version of the manuscript.

Funding: This work was supported by the National Key Research and Development Program (Grant No. 2021YFC2801504), “Unveiling the list of commanders” key research and development projects in Wenling City (Grant No. 2023G00015), Taizhou Science and Technology Bureau (Project No. 23sf06), the National Natural Science Foundation of China (Grant No. 52279086).

Data Availability Statement: The data that support the findings of this study are available from the corresponding author upon reasonable request.

Conflicts of Interest: The authors declare no conflicts of interest.

Abbreviations

The following abbreviations are used in this manuscript:

AWP	Azimuth waterjet propulsion
CV	Counter-rotating vortex pairs
PS	Pressure surface
SS	Suction surface
D	Reference diameter
H	Head
K_T	Thrust coefficient
η	Efficiency
h_f	Friction loss
h_c	Secondary flow loss
h_m	Wake loss
h_s	Shock loss

References

1. Yan, P.; Zhang, R.; Meng, K.; Geng, H.; Zhou, J.; Zhang, Y.; Chen, J. Numerical investigation of internal flow characteristics in an azimuth waterjet propulsion at different ship speeds. *Appl. Ocean Res.* **2025**, *154*, 104406. [\[CrossRef\]](#)
2. SCHOTTEL GmbH. Schottel Product Guide. Available online: <https://www.schottel.de/en/portfolio/products/product-details/spj-schottel-pumpjet> (accessed on 22 January 2026).
3. Carreño, J.E.; Mora, J.D.; Pérez, F.L. Mathematical model for maneuverability of a riverine support patrol vessel with a pump-jet propulsion system. *Ocean Eng.* **2013**, *63*, 96–104. [\[CrossRef\]](#)
4. Odetti, A.; Altosole, M.; Bruzzzone, G.; Caccia, M.; Viviani, M. Design and construction of a modular pump-jet thruster for autonomous surface vehicle operations in extremely shallow water. *J. Mar. Sci. Eng.* **2019**, *7*, 222. [\[CrossRef\]](#)
5. Yan, P.; Zhang, R.; Zhou, J.; Zhang, Y.; Chen, J. On the internal flow mechanism of an azimuth waterjet propulsion system. *Ship Boat* **2025**, *36*, 122.
6. Tees White Gill. Vertical Shaft Units. Available online: <http://www.teesgillthrusters.com/products/vertical-shaft-units> (accessed on 22 January 2026).
7. Park, W.G.; Yun, H.S.; Chun, H.H.; Kim, M.C. Numerical flow simulation of flush type intake duct of waterjet. *Ocean Eng.* **2005**, *32*, 2107–2120. [\[CrossRef\]](#)
8. Park, W.G.; Jang, J.H.; Chun, H.H.; Kim, M.C. Numerical flow and performance analysis of waterjet propulsion system. *Ocean Eng.* **2005**, *32*, 1740–1761. [\[CrossRef\]](#)
9. Gao, H.; Lin, W.; Du, Z. Numerical flow and performance analysis of a water-jet axial flow pump. *Ocean Eng.* **2008**, *35*, 1604–1614. [\[CrossRef\]](#)
10. Fang, F.H.; Peng, W.; Dazhuan, W.; Leqin, W. Numerical study on the stall behavior of a water jet mixed-flow pump. *J. Mar. Sci. Technol.* **2014**, *19*, 438–449. [\[CrossRef\]](#)
11. Cao, P.; Wang, Y.; Kang, C.; Li, G.; Zhang, X. Investigation of the role of non-uniform suction flow in the performance of water-jet pump. *Ocean Eng.* **2017**, *140*, 258–269. [\[CrossRef\]](#)
12. Huang, R.; Ye, W.; Dai, Y.; Luo, X.; Wang, Y.; Du, T.; Huang, C. Investigations into the unsteady internal flow characteristics for a waterjet propulsion system at different cruising speeds. *Ocean Eng.* **2020**, *203*, 107218. [\[CrossRef\]](#)
13. Zhao, H.; Wang, F.; Wang, C.; Lu, H.; Zhu, B. Study of driving force characteristics in the formation of dominant unstable flow events in water jet propulsion pumps. *Ocean Eng.* **2023**, *289*, 116291. [\[CrossRef\]](#)
14. Zheng, F.; Zhang, X.; Chen, T.; Huang, B.; Xu, J.; Wang, G. Strong transient characteristics in axial flow waterjet pump during rapid starting period with special emphasis on saddle zone. *Ocean Eng.* **2023**, *269*, 113506. [\[CrossRef\]](#)
15. Jiao, W.; Wang, Y.; Cheng, L.; Zhao, H. Experimental study on thrust pulsation characteristics of water jet propulsion pump units. *Ocean Eng.* **2023**, *284*, 115079. [\[CrossRef\]](#)
16. Jiao, W.; Zhao, H.; Cheng, L.; Yang, Y.; Li, Z.; Wang, C. Nonlinear dynamic characteristics of suction-vortex-induced pressure fluctuations based on chaos theory for a water jet pump unit. *Ocean Eng.* **2023**, *268*, 113429. [\[CrossRef\]](#)
17. Jiao, W.; Zhao, H.; Cheng, L.; Wang, Y. Numerical and experimental study on the hydrodynamic characteristics of the hub vortex at the nozzle of a water jet propulsion pump unit. *Phys. Fluids* **2023**, *35*, 062102. [\[CrossRef\]](#)
18. Odetti, A.; Altosole, M.; Bibuli, M.; Bruzzzone, G.; Caccia, M.; Viviani, M. Advance speed-hull-pump-jet interactions in small ASV. *Prog. Mar. Sci. Technol.* **2020**, *5*, 197–206.
19. Liu, W. Design and Performance Study of a Bottom-Mounted Azimuth Waterjet Propulsor. Master's Thesis, Jiangsu University of Science and Technology, Jiangsu, China, 2016.
20. Carreño Moreno, J.; Sierra Vanegas, E.; Jimenez Gonzalez, V. Ship manoeuvrability: Full scale trials of Colombian navy riverine support patrol vessel. *Ship Sci. Technol.* **2011**, *5*, 69–86. [\[CrossRef\]](#)
21. Zhang, R.; Zhou, M.; Yan, P.; Zhou, J. Research on the momentum distribution mechanism within the diffuser chamber nozzles of the azimuth waterjet propulsion system. In Proceedings of the ISOPE International Ocean and Polar Engineering Conference, Ottawa, ON, Canada, 19–23 June 2023.
22. Ibraheem, A. Evaluating the efficiency of polyhedral mesh elements in solving the problem of the flow around ship's rudder. *Int. J. Eng. Manag. Sci.* **2021**, *6*, 241–256. [\[CrossRef\]](#)
23. Zhang, H.; Tang, S.; Yue, H.; Wu, K.; Zhu, Y.; Liu, C.; Li, C. Comparison of computational fluid dynamic simulation of a stirred tank with polyhedral and tetrahedral meshes. *Iran. J. Chem. Chem. Eng.* **2020**, *39*, 311–319.
24. Baker, N.; O'Sullivan, P. Verification of Multiphase Flow CFD Simulation and Grid Convergence Index (GCI) Analysis. In Proceedings of the 30th International Manufacturing Conference (IMC30), Dublin, Ireland, 3–4 September 2013.
25. Yang, L.; Fang, S. Grid independence verification of curved rectangular duct based on FLUENT. *J. Beibu Gulf Univ.* **2023**, *38*, 63–69. [\[CrossRef\]](#)
26. Su, C.; Zhang, Z.; Zhu, D.; Tao, R. Enhancing the operating efficiency of mixed-flow pumps through adjustable guide vanes. *Water* **2025**, *17*, 423. [\[CrossRef\]](#)

27. Zhang, H.; Meng, F.; Cao, L.; Li, Y.; Wang, X. The influence of a pumping chamber on hydraulic losses in a mixed-flow pump. *Processes* **2022**, *10*, 407. [[CrossRef](#)]
28. Xu, Z. *Principles of Centrifugal Compressors*; China Machine Press: Beijing, China, 1990; pp. 32–56.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.