

Article

Draft Tube Wake Vortex Evolution and Suppression in a Pump as Turbine with Splitter Blades Based on a Modified Burgers Vortex Model

Chenguang Wang ^{1,2}, Wang Zheng ³, Yingxiao Shi ³, Hua Liu ^{2,*}, Dazhuan Wu ¹ and Qiaorui Si ^{3,*} 

¹ College of Energy Engineering, Zhejiang University, Hangzhou 310027, China; wangchenguang@zju.edu.cn (C.W.); wudazhuan@zju.edu.cn (D.W.)

² Zhuhai Gree Electric Appliance Co., Ltd., Zhuhai 000651, China

³ Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, Zhenjiang 212013, China; zhwang0227@stmail.ujs.edu.cn (W.Z.); syx200204@163.com (Y.S.)

* Correspondence: lh7711@126.com (H.L.); siqiaorui@ujs.edu.cn (Q.S.); Tel.: +86-139-5280-1593 (H.L.); +86-136-5529-3881 (Q.S.)

Abstract

Owing to its efficient energy recovery capability, the pump as turbine (PAT) has attracted considerable attention and has been widely applied in micro hydropower systems. However, under off-design conditions, large-scale helical vortex ropes are readily induced in the draft tube, causing severe hydraulic losses and flow instability. Because existing theoretical models do not account for the slip effect at the impeller outlet, this study combines vortex dynamics theory with numerical simulation and introduces a correction coefficient to develop a Burgers vortex-based analytical wake vortex model for a PAT with splitter blades. The model is verified by its ability to predict the peak tangential velocity and radial decay trend of the vortex core. In addition, the influence of draft tube configuration on vortex rope evolution is revealed using the Liutex vortex identification method and enstrophy analysis. The results show that the geometric curvature of the elbow draft tube induces vortex rope breakup and high energy dissipation. Finally, entropy production theory is used to quantitatively evaluate the vortex suppression benefit and hydraulic loss caused by deflector plates. The results indicate that the transverse deflector plate (TDP) provides a significantly better suppression effect than the longitudinal deflector plate (LDP) by disrupting the circumferential continuity of the vortex rope. Although increasing the insertion depth of the deflector plate improves vortex suppression, it induces non-negligible local high-entropy production on the upstream-facing surface (US). This study clarifies the physical mechanism of wake vortices in a PAT with splitter blades and provides theoretical guidance for efficient PAT operation and wake vortex control.

Keywords: pump as turbine; splitter blade; draft tube vortex; Burgers vortex theory; deflector plate



Academic Editor: Nadia Penna

Received: 7 July 2026

Revised: 4 August 2026

Accepted: 5 August 2026

Published: 7 August 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.

1. Introduction

With the rapid development of the global economy, the demand for renewable energy continues to increase worldwide [1,2]. The development and utilization of hydropower resources are essential for alleviating energy shortages, improving the global ecological environment, and promoting sustainable development [3,4]. Owing to its compact structure, low cost, and ease of maintenance, the PAT has gradually replaced conventional

pressure-reducing valves and has been widely applied in urban water supply networks and micro hydropower systems [5–7]. However, as application scenarios become increasingly complex, the operating efficiency and reliability of PATs have become key constraints affecting overall system performance [8].

Existing studies on PAT performance optimization have mainly focused on structural improvement and internal flow field evolution in core components, such as the volute and impeller [9,10]. In contrast, relatively limited attention has been paid to the draft tube, a key downstream flow-passing and energy recovery component. In fact, when a PAT operates under off-design conditions, the fluid often carries strong residual circulation after leaving the impeller trailing edge. After this high-speed swirling flow enters the draft tube, it can readily evolve into a large-scale helical draft tube vortex rope [11]. This vortex rope not only blocks the flow passage and induces severe hydraulic losses, but also causes low-frequency pressure pulsations, threatening the safe and stable operation of the unit [12].

To clarify the formation mechanism and evolution of draft tube vortices, numerous studies have examined their flow characteristics using computational fluid dynamics (CFD) simulations and vortex identification methods. Yang et al. [13] and Anup et al. [14] demonstrated that the inception location and pulsation frequency of vortex ropes inside PATs are strongly correlated with the velocity circulation at the impeller outlet. Liu et al. [15] investigated the effects of tip clearance on pressure pulsation intensity and vortex characteristics in a mixed-flow PAT under pump mode, and found that the flow rate substantially affects vortex structure and trajectory. Zhou et al. [16] combined CFD with visualization experiments to identify cluster vortices and central vortex rope structures in the draft tube of a pump-turbine, revealing the mechanism by which rotating stall in guide vane passages induces vortex evolution in the vaneless region. Lai et al. [17] established and experimentally verified the relationship between vortex rope formation, development, and breakdown in a pump-turbine and the precession frequency and swirl intensity. Regarding the spatial evolution and energy dissipation of vortex ropes, Wang et al. [18] found that wake vortices undergo spatial and frequency-domain attenuation of kinetic energy during downstream evolution, and that periodic disturbances induced by impeller rotation intensify the vorticity response. Xu et al. [19] further combined the Omega vortex identification method with entropy production theory and confirmed that vortex structure aggregation is a direct cause of local high energy loss. Xue et al. [20] investigated the contributions of vortex structures with different vector directions to pressure pulsations and found that circumferential and axial vortex evolution is the main cause of complex low-frequency pressure pulsations.

To reveal the generation and dissipation mechanisms of wake vortices at the mechanistic level, researchers have introduced vortex dynamics theory to develop analytical wake vortex models. Wake vortex dynamics theory has been continuously improved, evolving from the classical Burgers axisymmetric line-vortex model to the viscous line-vortex models of Rosenhead and Saffman, which incorporate circulation [21–24]. To account for the strong shear and boundary-layer interference effects in practical hydraulic machinery, Escudier et al. [25] proposed a modified Burgers vortex model. By fitting the tangential velocity distribution within a cylindrical vortex tube, this model effectively enhanced the accuracy of the classical formulation in characterizing the outer weak vorticity region. Duinmeijer et al. [26] further validated the applicability of this model to free-surface vortex flow fields through stereo PIV measurements. Nevertheless, the classical Burgers vortex model and its derived forms are predominantly based on idealized axisymmetric vortex assumptions. For a PAT with splitter blades, the distinctive long-short blade configuration and strong rotor–stator interaction (RSI) can induce pronounced non-axisymmetric wake distortion. Therefore, realistic unsteady inflow boundary conditions of the PAT must be incorporated

into the classical theoretical framework, so as to establish a theoretical analytical wake vortex model that accurately reflects the characteristics of splitter blade impellers.

On the basis of clarifying the internal flow evolution mechanism in the draft tube, suppressing wake vortex evolution and weakening low-frequency pressure pulsations through effective flow-control strategies have become research hotspots in rotating machinery [27,28]. Kumar et al. [29] systematically reviewed the inception and evolution of draft tube vortex ropes and their adverse effects on unit stability, and pointed out that optimizing draft tube geometry or adding internal flow disturbing structures is an effective approach for vortex rope suppression. In addition, studies have shown that asymmetric draft tubes, fins, and wall protrusions can suppress vortex ropes to some extent and reduce pressure pulsations [30–32]. In recent years, researchers have conducted in-depth studies on specific flow control structures. Shrestha et al. [33] performed multi-objective optimization of the geometric parameters of J-grooves on the draft tube wall using combined numerical and experimental methods, establishing design criteria for suppressing swirling flow with a relatively small resistance penalty. Wang et al. [34] installed an airfoil-shaped stabilizing ring in the draft tube of a Francis turbine and found that it significantly enhanced flow stability under wide-load conditions. Among various control elements, deflector plates have attracted considerable attention because of their simple structure and high reliability. Zhou et al. [35] arranged baffles circumferentially in the draft tube to forcibly block the strong swirling flow at the impeller outlet. The baffles substantially reduced the tangential velocity of the flow field and effectively disrupted vortex-rope continuity. Joy et al. [36] installed adjustable deflector plates in the draft tube and found that an appropriate deflector plate angle could reduce hydraulic losses and improve hydraulic performance. However, while deflector plates interrupt swirling flow, they inevitably induce non-negligible local hydraulic losses. Existing studies still lack a quantitative trade-off analysis between vortex suppression effectiveness and hydraulic cost. Moreover, for PATs with splitter blades and complex internal flow fields, the influence mechanism of deflector plate spatial arrangement on wake vortex dissipation and overall machine performance remains unclear.

Based on the above research gaps, this study takes a PAT with splitter blades as the research object and conducts a systematic investigation by combining vortex dynamics theory with numerical simulation. Considering the slip effect at the impeller outlet, a modified Burgers vortex analytical model suitable for a PAT with splitter blades is established. By investigating the effects of different deflector plate arrangements on wake vortex suppression, the relationship between vortex suppression benefits and local hydraulic losses is quantitatively evaluated. This study aims to clarify the flow characteristics and suppression mechanism of wake vortices in a PAT with splitter blades. The findings provide theoretical support and engineering guidance for flow control and performance optimization of this type of PAT.

2. Numerical Methodology

2.1. Governing Equations and Turbulence Model

Fluid motion in the flow field follows the conservation laws of mass, momentum, and energy. Because the present study analyzes the PAT under isothermal conditions, the energy equation is not considered in the flow field simulation.

(1) Continuity equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

where ρ is the fluid density, and t is time. When the fluid is incompressible, its density remains constant, and the equation above reduces to:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (2)$$

(2) Momentum equation

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_i} + S_{mi} \quad (3)$$

where p is the fluid static pressure, S_{mi} is the source term added to the equation, and τ_{ij} is the stress tensor. For a Newtonian fluid, the stress tensor can be expressed as:

$$\tau_{ij} = \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] - \frac{2}{3} \mu \frac{\partial u_l}{\partial x_l} \delta_{ij} \quad (4)$$

where μ is the fluid viscosity and δ_{ij} is the Kronecker delta. When $i = j$, $\delta_{ij} = 1$; when $i \neq j$, $\delta_{ij} = 0$.

In this study, the Shear Stress Transport (SST) k - ω turbulence model is employed to solve the Reynolds-Averaged Navier–Stokes (RANS) equations. By introducing a blending function F_1 , this model uses the standard k - ω model in the near-wall region and transitions to the k - ϵ model in the far-field mainstream region. Because the SST k - ω model can effectively capture turbulent features in complex geometries, it has been widely used in numerical simulations. Therefore, the SST k - ω model is selected to solve the internal flow field of the PAT, and its governing equations are as follows:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho k \omega \quad (5)$$

$$\begin{aligned} \rho \frac{\partial \omega}{\partial t} + \rho u_j \frac{\partial \omega}{\partial x_j} = & \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\omega}{k} \left(\alpha \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho k \omega \right) \\ & + (1 - F_1) 2\rho \frac{1}{\omega \sigma_{\omega 2}} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (6)$$

where the turbulent viscosity and the turbulence production term can be expressed as:

$$\mu_t = \rho \frac{a_1 k}{\max(a_1 \omega, SF_2)} \quad (7)$$

$$P_k = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \left(3\mu_t \frac{\partial u_k}{\partial x_k} + \rho k \right) \quad (8)$$

where P_k is the turbulence production term caused by viscous forces. In the above equations, each constant is a blend of inner and outer constants.

$$\varphi = \varphi_1 F_1 + \varphi_2 (1 - F_1) \quad (9)$$

The blending functions F_1 and F_2 can be expressed as:

$$F_1 = \tanh \left\{ \left\{ \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\mu}{y^2 \omega \rho} \right), \frac{4\rho k}{CD_{k\omega} \sigma_{\omega 2} y^2} \right] \right\}^4 \right\} \quad (10)$$

$$CD_{k\omega} = \max \left(2\rho \frac{1}{\sigma_{\omega 2} \omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right) \quad (11)$$

$$F_2 = \tanh \left\{ \left[\max \left(\frac{2\sqrt{k}}{\beta^* \omega y}, \frac{500\mu}{y^2 \omega \rho} \right) \right]^2 \right\} \quad (12)$$

where $\sigma_{\omega 2}$ and β^* are blending functions, and y is the distance from the calculation point to the wall.

2.2. Geometric Model and Computational Scheme

The research object of this study is a PAT with splitter blades operating at a specific speed of 16.5. The PAT primarily consists of an impeller, volute, chamber, wear rings, and inlet and outlet passages, as illustrated in the three-dimensional computational model in Figure 1. To enhance internal flow stability under off-design conditions and suppress backflow and secondary flow structures at the impeller outlet, the PAT adopts a splitter blade design. The alternating arrangement of long and short blades optimizes the pressure distribution within the flow passages and improves the work transfer from the fluid to the blades. In addition, the splitter blades indirectly affect the velocity slip and residual circulation at the impeller outlet by altering the inlet flow distribution, blade loading, and the merging process of the subdivided flow passages. The relevant PAT parameters are listed in Table 1.

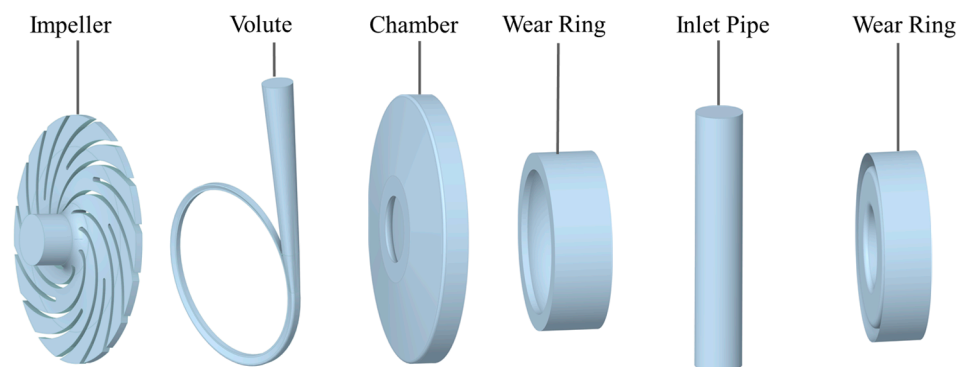


Figure 1. Computational domain of the PAT.

Table 1. Parameters of the PAT.

Design Parameter	Value
Rated flow rate Q_d (m ³ /h)	28
Rated rotational speed n (r/min)	1500
Specific speed n_s	16.5
Head H (m)	100
Impeller outlet diameter D_1 (mm)	55
Impeller inlet diameter D_2 (mm)	265
Number of long blades Z_1	8
Number of splitter blades Z_2	8
Inlet width of volute b_3 (mm)	12
Base circle diameter of volute D_3 (mm)	275

In practical engineering applications, the draft tube configuration must satisfy both hydraulic performance requirements and installation constraints. Therefore, two typical draft tube configurations were designed in this study, as shown in Figure 2, where the main geometric dimensions are indicated. The elbow draft tube, one of the most widely used configurations, redirects the flow while also serving as a diffuser. To ensure a fair comparison, the geometric parameters of the gradually expanding section in the straight conical draft tube were kept consistent with those of the elbow draft tube. In the elbow

draft tube, the axial length of the straight conical section is $L_s = 125$ mm, and the draft tube inlet diameter is $D_{in} = 55$ mm, yielding $L_s/D_{in} = 2.27$. This geometric matching allows the differences in vortex evolution between the two draft tubes to be attributed primarily to the curvature of the elbow. However, L_s was fixed in this study and was not parametrically optimized.

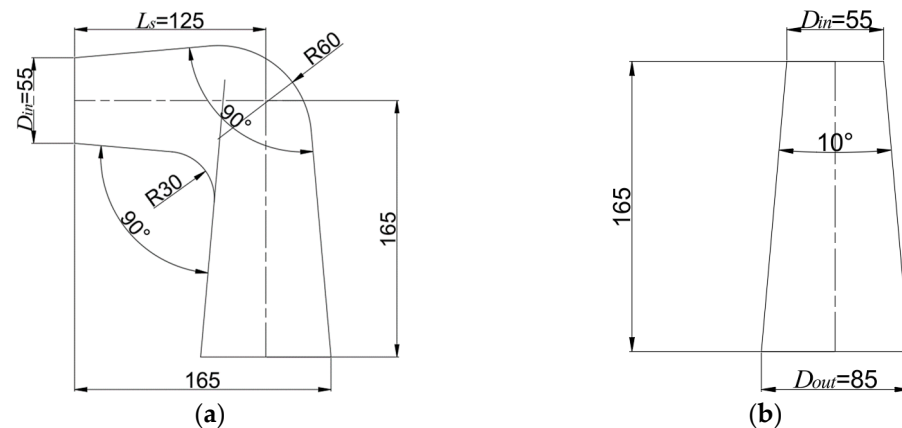


Figure 2. Schematic diagram of different draft tube configurations. All linear dimensions are in mm. (a) Elbow draft tube; (b) Straight conical draft tube.

In addition, to suppress the spiral vortex rope inside the draft tube and thereby improve flow field stability, four control schemes were designed in the straight conical section of the elbow draft tube. The deflector plate orientation, namely the transverse deflector plate (TDP) and longitudinal deflector plate (LDP), and the insertion depth, 15 mm or 30 mm, were selected as geometric variables. The transverse and longitudinal arrangements correspond to directions perpendicular and parallel to the symmetry plane of the elbow section, respectively. The different control schemes are shown in Figure 3. Moreover, in practical engineering applications, installing the deflector plates at the inlet of the straight conical section facilitates their fabrication, installation, and subsequent maintenance, while enabling early intervention in the impeller outlet swirl after it enters the draft tube. The same axial position was adopted for all control schemes to isolate the effects of the deflector plate orientation and insertion depth. The axial position of the deflector plates was not parametrically optimized in this study.

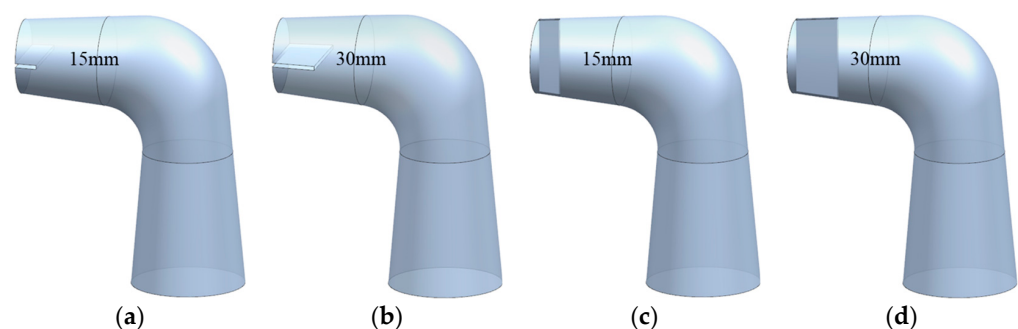


Figure 3. Different control schemes for the draft tube vortex rope. (a) TDP-15; (b) TDP-30; (c) LDP-30; (d) LDP-30.

2.3. Mesh Generation and Grid Independence Verification

After completing the geometric modeling of the PAT computational domain, ANSYS ICEM (version 2021 R1) was used to generate meshes for all flow passage components of the PAT. Compared with unstructured meshes, structured meshes have significant advan-

tages in convergence speed, numerical dissipation control, and grid orthogonality, thereby improving computational accuracy and reducing computational resource consumption. Consequently, a fully structured mesh was applied to the entire PAT flow domain. Figure 4 illustrates the mesh generation results for the global computational domain of PAT with splitter blades.

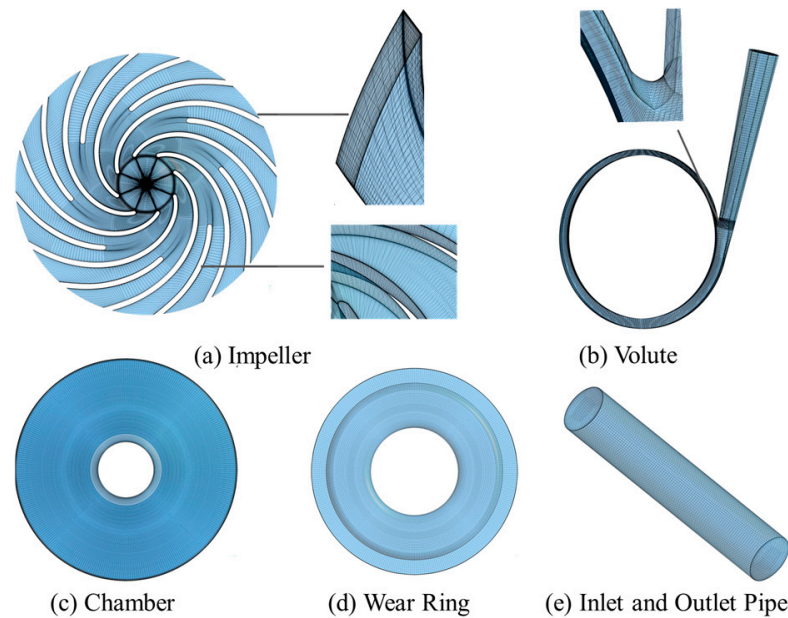


Figure 4. Mesh generation of the PAT.

For the draft tube computational domain with deflector plates installed, structured meshes were also applied. To accurately capture the complex flow characteristics between the deflector plates and tube walls, local refinement was performed on the deflector plate surfaces and near-wall regions. This refinement enables accurate simulation of velocity gradients and shear flow behavior within the near-wall viscous sublayer. Figure 5 shows the mesh generation results of the two arrangement schemes.

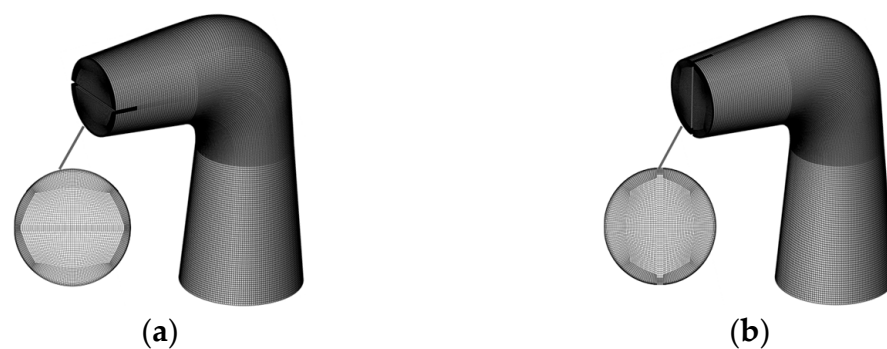


Figure 5. Mesh generation of different arrangement schemes. (a) TDP-15; (b) LDP-15.

To reduce the influence of mesh count on numerical simulation results, a grid independence verification was conducted, and multiple schemes with different grid counts were generated. Figure 6 presents the calculation results of the PAT under various mesh counts. As the mesh number increases, the output power of the PAT increases and then gradually stabilizes, indicating that the dependence of the calculation results on the mesh is progressively weakened. Considering both accuracy requirements and computational cost, the total grid count for the PAT was ultimately selected as 7.4 million.

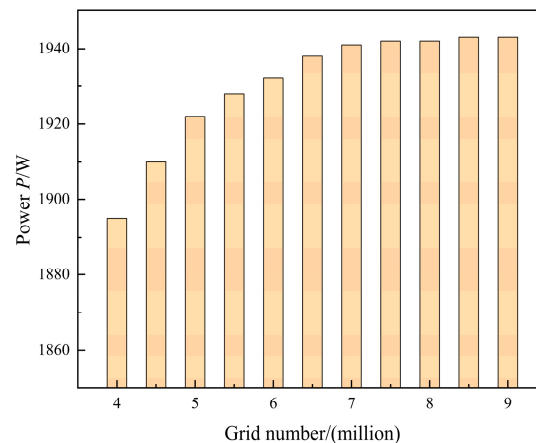


Figure 6. Grid independence verification.

2.4. Numerical Computation Settings

Numerical simulations were conducted using ANSYS Fluent. The working fluid was clear water under normal temperature and pressure. A pressure inlet boundary condition was specified at the inlet with a constant total pressure of 1 MPa, while a mass flow outlet boundary condition was applied at the outlet. Interfaces were used for data exchange between different components, and all walls were set as no-slip walls. The residual convergence tolerance was set to 10^{-5} . A steady state simulation was first performed, and its results were used as the initial condition for the subsequent transient simulation. For the transient simulation, the time step size was set to 3.333×10^{-4} s, corresponding to a 3 degrees of impeller rotation per time step, and a total of 10 revolutions were simulated. Other settings are listed in Table 2.

Table 2. Numerical simulation parameters.

Item	Setting
Fluid medium	Water
Inlet boundary condition	Pressure-inlet
Outlet boundary condition	Mass flow-outlet
Maximum residual value	10^{-5}
Wall condition	No-slip wall
Discretization scheme for convective terms	Second-order

2.5. Burgers Vortex Theory

In the complex turbulent flow within the PAT, the processes of energy transfer and dissipation are highly dependent on the mechanisms of generation, evolution, and break-down of vortex structures. The Burgers vortex model is widely used as an idealized analytical model for characterizing typical vortex tube structures in viscous fluids. This model provides an analytical solution for the stable vortex tube structure within a viscous incompressible fluid under the action of an axisymmetric stretching strain field. The Burgers vortex is generally defined in an axisymmetric, steady, incompressible flow field, reflecting the equilibrium relationship among axial stretching, radial compression, and viscous dissipation.

In cylindrical coordinates (r, θ, z) , the velocity field of the Burgers vortex can be expressed as:

$$V_r = -\alpha r, \quad V_z = 2\alpha z, \quad V_\theta = \frac{\Gamma}{2\pi r} \left(1 - e^{-\frac{\alpha r^2}{2\nu}}\right) \quad (13)$$

where V_r , V_θ , and V_z is the radial, azimuthal, and axial velocity components, respectively; α is the strain rate, characterizing the intensity of the applied axial strain; ν is the kinematic viscosity of the fluid; and Γ is the total circulation.

In the Burgers vortex, V_θ characterizes the rotational motion of the vortex and is central to model analysis. For axisymmetric steady flow, the axial vorticity depends only on the radial coordinate r , i.e.,

$$\omega_z = \omega(r) \quad (14)$$

Under the combined action of the strain field and viscosity, the axial vorticity satisfies the convection-diffusion balance equation. Its analytical solution can be written as:

$$\omega(r) = \omega_0 \exp\left(-\frac{\alpha r^2}{4\nu}\right) \quad (15)$$

Equation (15) indicates that the vorticity of the Burgers vortex exhibits an exponential decay distribution along the radial direction, with vorticity mainly concentrated near the vortex core. Its characteristic radius is typically determined jointly by the strain rate and viscosity.

$$r_c \sim \sqrt{\frac{4\nu}{\alpha}} \quad (16)$$

When the viscous effect intensifies, the vortex core radius increases, leading to more dispersed vorticity distribution. Conversely, when the strain rate increases, the vortex structure becomes more concentrated. Based on the relationship between circulation and tangential velocity under axisymmetric conditions, the tangential velocity distribution can be derived as:

$$V_\theta(r) = \frac{\Gamma}{2\pi r} \left[1 - \exp\left(-\frac{\alpha r^2}{4\nu}\right)\right] \quad (17)$$

where Γ is the total circulation.

The tangential velocity distribution of the Burgers vortex has the following characteristics. At the vortex core center, V_θ increases approximately linearly with r because of viscous smoothing. In the region far from the vortex core, the exponential term tends to zero, and V_θ gradually approaches the free vortex form. Thus, the Burgers vortex behaves as a forced vortex in the near field and transitions to a free vortex in the far field.

When the fluid exits the impeller of PAT, it often fails to maintain an ideal irrotational discharge, inevitably carrying a relatively high residual circulation. This residual circulation evolves into a large-scale helical wake vortex rope within the draft tube, as illustrated in Figure 7. Under the assumption of a steady and axisymmetric ideal flow field, the complex flow pattern at the impeller outlet can be decomposed into two fundamental components: uniform axial flow along the mainstream direction and circumferential rotational motion about the central axis. The superposition of these two components, together with the geometric constraints imposed by the draft tube walls, ultimately forms an axisymmetric helical vortex structure with significant axial stretching. This idealized wake vortex model closely resembles the Burgers vortex model. Therefore, Burgers vortex theory is introduced as the analytical foundation for constructing a wake vortex model suitable for describing vortex dynamics inside the draft tube of PAT with splitter blades.

However, the classical Burgers vortex model assumes an absolutely irrotational external region outside the vortex core. In actual PAT operation, wall friction in the draft tube and radial constraints prevent the external region from maintaining an irrotational state, causing significant changes in the flow velocity. To correct this theoretical limitation,

Escudier et al. proposed a modified Burgers vortex analytical model that considers wall confinement effect, expressed as:

$$V_{\theta} = \frac{\Gamma_0}{2\pi r} \left[1 - \exp\left(-\frac{r^2}{r_0^2}\right) \right] + \frac{1}{2}\omega r \quad (18)$$

where Γ_0 is the characteristic circulation at the vortex core radius r_0 , and ω is a parameter characterizing the mean vorticity in the external region of the vortex core.

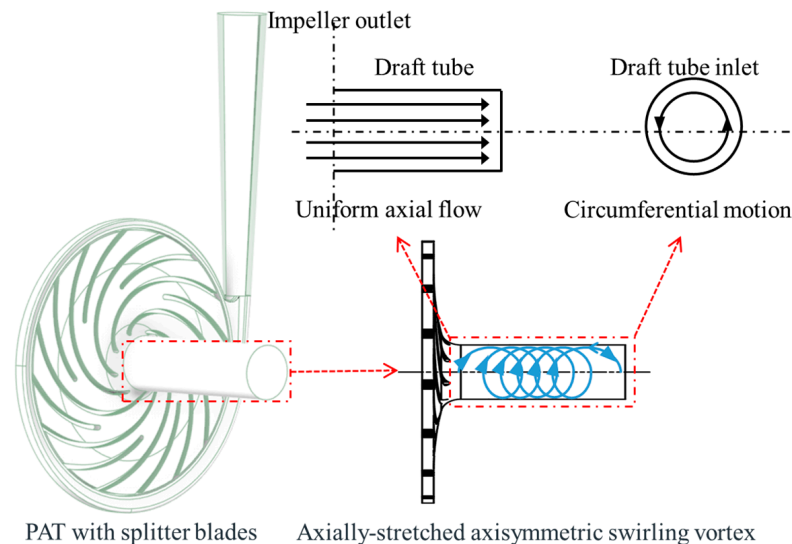


Figure 7. Wake vortex flow pattern of the PAT.

3. Experimental Method and Validation

3.1. Experimental Instruments and Uncertainty Analysis

To verify the accuracy of the numerical method, a test rig for the PAT was established in this study. The three-dimensional model of the test bench is shown in Figure 8a, and the experimental site is shown in Figure 8b. The water circulation loop of the test rig mainly consists of valves, an electromagnetic flowmeter, a water tank, pipelines, and the PAT. During the experiments, the front booster pump and pipeline control valves, were jointly regulated to provide an inlet pressure higher than 1 MPa for the PAT under different flow rates. The rotational speed of the booster pump is adjusted using a frequency converter, thereby regulating the flow rate and head required by the PAT at different rotational speeds.

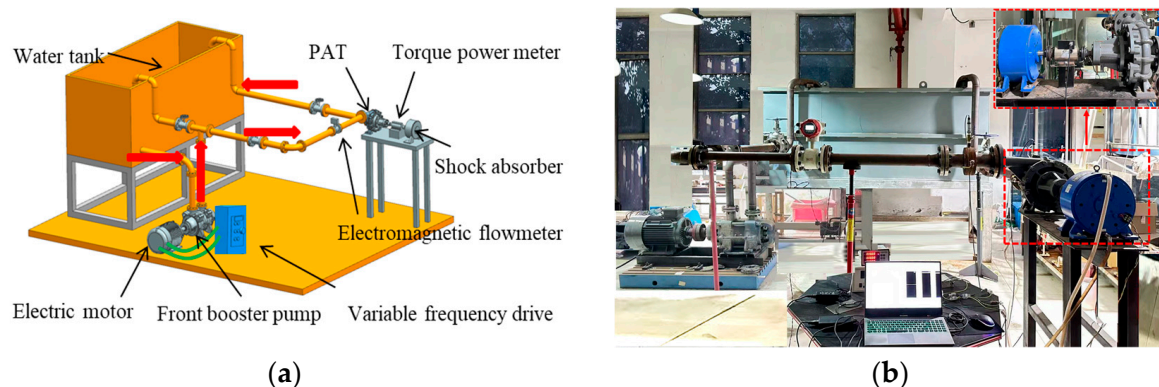


Figure 8. PAT experimental test bench. (a) 3D model of the experimental bench; (b) Experimental bench schematic diagram.

The experimental flow rate was measured using an electromagnetic flowmeter equipped with a KROHNE IFC300 signal converter, with a measurement range of 0–60 m³/h. The experimental flow-rate range of 20–45 m³/h corresponds to 33–75% of the full-scale range. Therefore, all experimental operating conditions were within the effective measurement range of the flowmeter and were well above its lower measurement limit.

The pressure difference was measured using pressure sensors arranged at the inlet and outlet of the PAT. A torque meter is used to synchronously acquire the rotational speed and shaft torque during PAT operation. All sensor signals were fed into the data acquisition and processing system for real-time display, analysis, and storage, from which the efficiency, head, and output shaft power of the PAT were obtained. The performance parameters of the PAT were calculated using the following equations:

$$P = M \frac{2\pi n}{60} \quad (19)$$

$$H = \frac{P_{out} - P_{in}}{\rho g} \quad (20)$$

$$\eta = \frac{P}{\rho g Q H} \times 100\% \quad (21)$$

where M is the torque output of the PAT, n is the rotational speed, P_{in} and P_{out} are the total pressures at the PAT inlet and outlet, respectively, Q is the flow rate, and H is the head.

Uncertainty refers to the unavoidable deviation between a measurement result and the true value of the measured quantity caused by measurement errors and limitations in measurement conditions [37]. The total uncertainty of the test system consists of systematic uncertainty E_s and random uncertainty E_r . The systematic uncertainty is mainly determined by the accuracy class of the measuring instruments, whereas the random uncertainty is calculated from the mean value and standard deviation of the measurement samples [38,39]. The relevant expressions are as follows:

$$E = \pm \sqrt{E_s^2 + E_r^2} \quad (22)$$

$$E_s = \pm \sqrt{\left(\frac{f_m}{k_u}\right)^2 + \left(\frac{f_c}{k_u}\right)^2} \quad (23)$$

$$E_r = \pm \frac{t_{n-1} S_x}{\bar{x} \sqrt{n}} \times 100\% \quad (24)$$

where f_m and f_c are uncertainties from different sources of the measuring instruments, x_i is the experimentally measured value, S_x is the sample standard deviation, $\bar{x}$ is the sample mean value, t_{n-1} is the confidence interval, and n is the number of measurements.

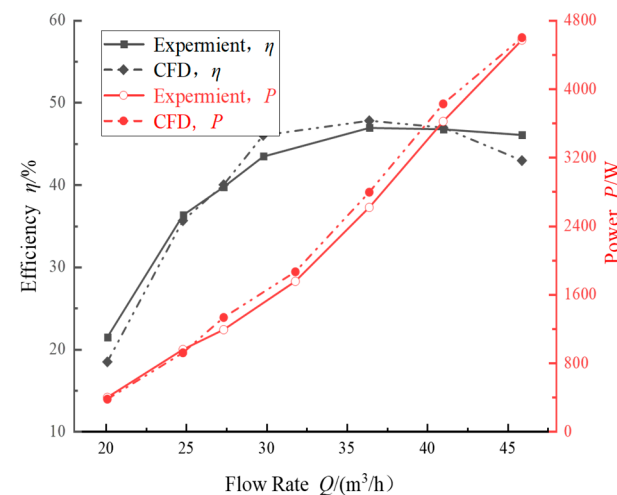
Table 3 presents the uncertainties of the experimental parameters. The combined uncertainties of flow rate, head, torque, power, and efficiency are $\pm 0.19\%$, $\pm 0.17\%$, $\pm 0.19\%$, $\pm 0.23\%$, and $\pm 0.18\%$, respectively. The systematic uncertainties of flow rate and head are caused by the accuracy errors of the electromagnetic flowmeter and pressure sensors. The uncertainties of the power and torque are mainly affected by the transmission error of the torque meter. Efficiency is a calculated quantity from multiple parameters, and its combined uncertainty is strongly affected by the accumulated errors. The combined uncertainty of each measurement is controlled within $\pm 0.25\%$, ensuring the reliability of the experimental results.

Table 3. Uncertainty of experimental parameters.

Notation Uncertainty	Q (m ³ /h)	H (m)	M (N·m)	P (W)	η (%)
E_s	$\pm 0.16\%$	$\pm 0.07\%$	$\pm 0.09\%$	$\pm 0.12\%$	$\pm 0.11\%$
E_r	$\pm 0.10\%$	$\pm 0.15\%$	$\pm 0.17\%$	$\pm 0.20\%$	$\pm 0.14\%$
E	$\pm 0.19\%$	$\pm 0.17\%$	$\pm 0.19\%$	$\pm 0.23\%$	$\pm 0.18\%$

3.2. Experimental Validation

To verify the accuracy of the numerical simulation, Figure 9 compares the experimental and numerical results of the PAT under different flow rates. As the flow rate increases, the PAT power monotonically rises, whereas the efficiency first increases and then decreases. At around 28 m³/h, the measured power and efficiency of the PAT are 2.21 kW and 46.2%, respectively, while the numerical simulation results are 2.31 kW and 47.5%. At this flow rate, the PAT efficiency reaches its maximum value. Overall, the experimental and numerical results agree well, with errors below 5%, demonstrating the accuracy and reliability of the numerical simulation method adopted in this study [40].

**Figure 9.** Comparison of experimental and numerical simulation results.

4. Results and Discussion

4.1. Modified Analytical Model of the Wake Vortex Based on Burgers Vortex Theory

The draft tube vortex in a PAT is generated by impeller rotation. Therefore, the input parameters of the analytical model should be replaced with the outlet boundary conditions of the PAT. Based on the velocity triangle theory at the impeller outlet, the theoretical circulation at the impeller outlet is introduced. Figure 10 shows the decomposition of the impeller outlet velocity triangle.

In actual operation, owing to the finite number of impeller blades, strong interactions between relative vortices and passage vortices occur within the flow passages of the PAT. As a result, velocity slip inevitably occurs when the fluid leaves the blade trailing edge, as shown in Figure 11. Under low-flow conditions ($\beta < \beta_{2\infty}$), the smaller meridional velocity V_m causes the theoretical absolute velocity $V_{2\infty}$ to have a positive tangential component. The slip velocity component ΔV_{u2} is superimposed in the same direction, leading to a sharp increase in both the intensity and dissipation of the draft tube vortex. Under the rated condition, $\Delta V_{u2} \approx 0$, the outlet swirl is approximately zero, and the vortex intensity is the weakest. However, under high-flow conditions ($\beta > \beta_{2\infty}$), the larger V_m causes $V_{2\infty}$ to produce a negative tangential component, while the positive ΔV_{u2} provides an opposite

compensation effect in the vector composition, thereby effectively suppressing the vortex intensity. For a PAT, because the fluid must overcome hydraulic losses in the flow passages and the slip effect reduces the actual work capacity of the impeller below the theoretical value, the actual operating head consumed by the PAT is necessarily greater than the theoretical Euler head based on ideal inviscid flow.

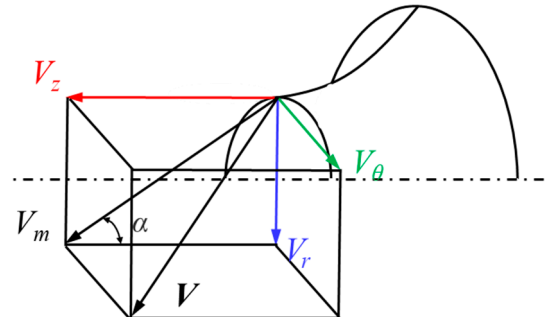


Figure 10. Schematic decomposition of the impeller outlet velocity triangle.

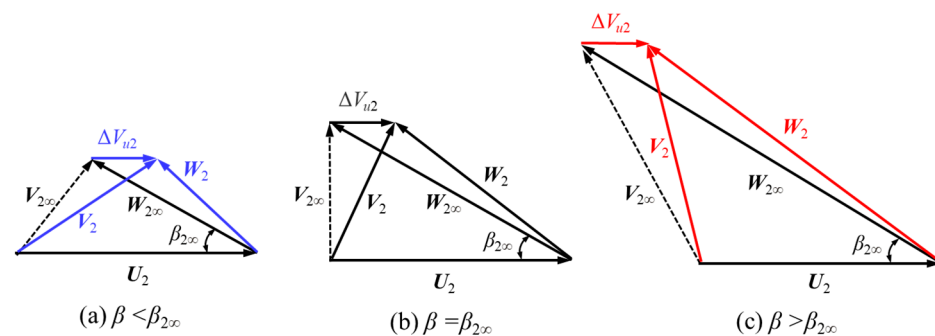


Figure 11. Velocity triangle at the impeller outlet of the PAT.

The actual head of the PAT, H , is defined based on the absolute pressure difference between the inlet and outlet sections of the flow field. As the fluid passes through the PAT passages, it must overcome strong fluid shear, boundary friction, and dissipation induced by large-scale wake vortices. The portion that is ultimately converted into mechanical energy is defined as the effective output head, H_t , which can be calculated from the hydraulic efficiency η as follows:

$$H_t = H \cdot \eta \quad (25)$$

According to the angular momentum theorem, the Euler head, H_e , corresponding to the actual deflection of the velocity vector as the fluid passes through the impeller inlet and outlet sections, can be calculated from the section-averaged velocity components extracted from the CFD flow field:

$$H_e = \frac{U_1 V_{u1} - U_2 V_{u2}}{g} \quad (26)$$

where U is the circumferential velocity, and V_{u1} and V_{u2} are the circumferential components of the absolute velocity at the impeller inlet and outlet, respectively.

Because flow separation, secondary flow, and enstrophy dissipation occur within the impeller passages, part of H_e still cannot be converted into effective mechanical output. Therefore, the relationship $H > H_e > H_t$ is satisfied. The difference between H_e and H_t represents the non-ideal flow loss induced by the slip effect and viscous dissipation of vortical flow. Accordingly, a non-ideal flow dissipation coefficient, S , is introduced to quantitatively characterize the degree of energy attenuation:

$$S = \frac{H_e - H_t}{H_e} \quad (27)$$

As shown in Table 4, S decreases monotonically with increasing flow rate. At $0.6Q_d$, a strong positive swirl exists at the impeller outlet, which severely weakens the work capacity of the impeller. Consequently, S reaches its maximum value of 0.1465. At $1.0Q_d$, the outlet circulation approaches zero, and the slip induced dissipation is minimized. As the flow rate further increases to $1.2Q_d$, although the outlet circulation Γ_{out} increases again, its direction is opposite to that of U_2 . Therefore, S decreases to 0.0536.

Table 4. Outlet circulation and dissipation coefficient of the PAT.

Flow Rate	H (m)	H_t (m)	H_e (m)	Γ_{out}	S	k
$0.6Q_d$	31.39	11.01	13.01	0.4784	0.1465	0.22
$0.8Q_d$	36.24	18.10	20.80	0.1250	0.1296	0.30
$1.0Q_d$	43.72	24.11	26.94	0.0498	0.1050	0.38
$1.2Q_d$	51.75	28.69	31.13	0.2270	0.0785	0.52
$1.4Q_d$	64.14	34.35	36.29	0.4128	0.0536	0.60

Based on the preceding analysis, this study proposes a wake vortex analytical model for a PAT with splitter blades that accounts for slip induced dissipation, as expressed in Equation (28). On the basis of the classical Burgers vortex analytical model, a correction coefficient k is introduced to modify the impeller outlet circulation, thereby representing the proportion of the actual effective circulation after slip induced dissipation relative to the theoretical circulation.

$$V_\theta = k \frac{\Gamma_{out}}{2\pi r} \left[1 - \exp\left(-\frac{r^2}{r_c^2}\right) \right] + \frac{1}{2}\omega r \quad (28)$$

where r_c is the vortex core radius, and $1/2\omega r$ is used to characterize the rotational driving effect at the impeller outlet. In Equation (28), $\Gamma_{eff} = k\Gamma_{out}$ is defined as the corrected effective circulation. The coefficient k represents the fraction of the theoretical circulation at the impeller outlet that is retained in the coherent axisymmetric wake vortex after attenuation by velocity slip, viscous diffusion, turbulent mixing, and secondary-flow interactions. The correction coefficient k is applied only to the circulation-driven Burgers vortex term. The correction coefficient k is calibrated using the flow dissipation coefficient S , as shown in Equation (29). For each operating condition, the non-ideal flow dissipation coefficient S_i was calculated using Equation (27). The corresponding correction coefficient k_i was then determined by minimizing the normalized root mean square error between the analytical predictions and the CFD results within the vortex-core-dominated region. Based on the (S_i , k_i) data listed in Table 4, a linear regression relationship was established, as expressed in Equation (29). The regression yielded a coefficient of determination R^2 of 0.9941 and a root mean square error of 0.0107, indicating a strong negative linear correlation between k and S within the present calibration range.

$$k = 0.8285 - 4.136S, 0.0536 \leq S \leq 0.1465 \quad (29)$$

It should be noted that Equation (29) is a semi-empirical correlation developed for the PAT investigated in this study, which has a specific speed of 16.5 and a particular splitter blade configuration. Its applicability is limited to $0.0536 \leq S \leq 0.1465$. The specific speed, outlet blade angle, and blade number can all affect velocity slip and residual circulation at the impeller outlet. Furthermore, the monotonic increase in k with flow rate is associated with the decrease in the relative non-ideal dissipation within the impeller. Under low flow conditions, flow separation, secondary flow, and interactions between passage vortices result in a relatively high value of S , indicating substantial attenuation of the effective outlet circulation and hence a small value of k . As the flow rate increases, the throughflow

becomes better aligned with the blade passages and the relative difference between H_e and H_t decreases. Consequently, a larger proportion of the outlet circulation is retained, resulting in an increase in k .

To verify the accuracy of the analytical model, Figure 12 compares the V_θ distributions on a characteristic section at $1.0Q_d$ obtained from the classical Burgers vortex model, the proposed analytical model, and CFD simulation. The classical Burgers vortex model directly employs the theoretical circulation at the impeller outlet. However, in an actual PAT, velocity slip caused by the finite number of blades, viscous dissipation, and flow separation reduce the effective circulation that sustains the coherent wake vortex structure. Consequently, the classical Burgers vortex model overestimates the peak V_θ by approximately 76% compared with the numerical simulation result. In contrast, the analytical model with the correction coefficient introduced ($k = 0.38$) agrees well with the CFD simulation. The model accurately captures the typical two region structure of the wake vortex: in the inner forced-vortex core region ($r/r_\theta < 0.35$), the circumferential velocity increases linearly with radius, whereas in the outer free-vortex region ($r/r_\theta > 0.35$), it decays approximately inversely with radius.

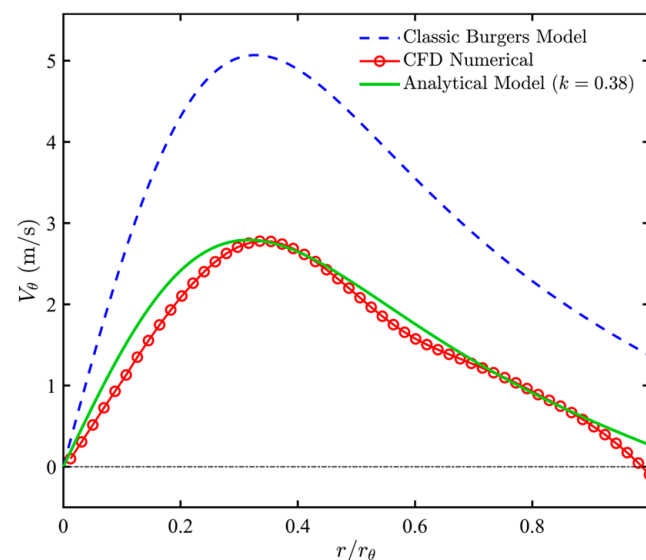


Figure 12. Comparison between the analytical model and numerical results at $1.0Q_d$.

It should be clarified that the velocity profiles predicted by the analytical model were compared with the CFD results rather than directly validated against experimental measurements. To further examine the applicability of the model under off-design conditions, Figure 13 compares the analytical predictions and numerical simulation results of the wake vortex circumferential velocity at $0.6Q_d$ and $1.4Q_d$. Overall, the analytical model accurately captures the peak velocity and decay trend in the vortex core region under both operating conditions. As the flow rate increases, the correction coefficient k increases from 0.22 to 0.60. It should be noted that local deviations occur between the analytical prediction curves and the numerical results in the outer region at $1.4Q_d$ ($r/r_\theta > 0.4$) and in the near-wall region at $0.6Q_d$ ($r/r_\theta > 0.6$). At $1.4Q_d$, the deviation is particularly pronounced near $r/r_\theta \approx 1$. This is mainly attributed to the strong interaction between the wall boundary layer and the mainstream shear layer, which induces asymmetric secondary flow and separated vortices beyond the descriptive capability of the axisymmetric analytical model. Nevertheless, the proposed analytical model maintains high prediction accuracy in the vortex core dominated region ($r/r_\theta < 0.4$), with peak velocity errors controlled within 5%.

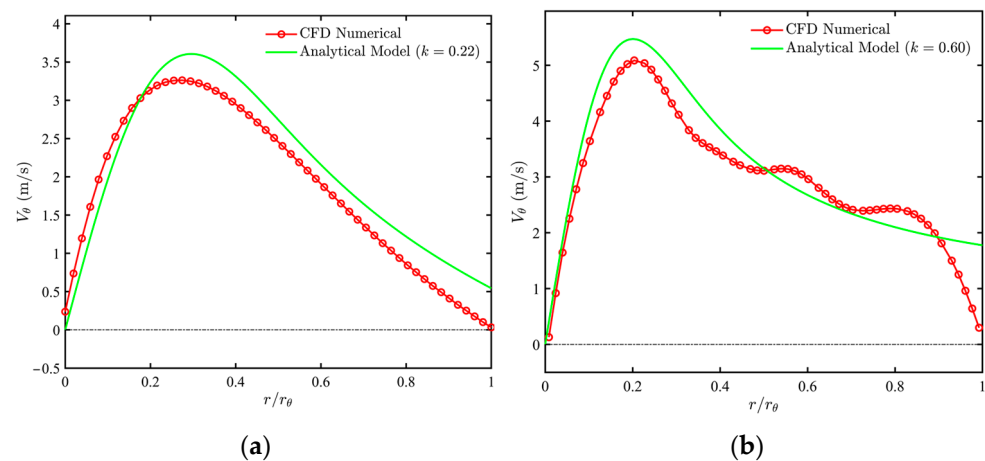


Figure 13. Comparison between the analytical model and numerical simulation results. (a) $0.6Q_d$; (b) $1.4Q_d$.

4.2. Wake Vortex Evolution in Draft Tubes with Different Configurations

Because the flow within the PAT draft tube involves strong shear induced by the impeller wake and wall boundary layers, Liutex is more suitable for the present analysis than the classical Q and λ_2 criteria. In addition to identifying regions dominated by rotational motion, Liutex extracts the local rotation axis from the velocity gradient tensor and quantifies the rigid-body rotational strength after removing the contribution of shear. It therefore enables clearer differentiation between coherent rotation and shear deformation and provides a more physically meaningful representation of the spatial continuity, breakup, and fragmentation of the helical vortex rope. Accordingly, the Liutex method was adopted to analyze vortex structure evolution in different draft tube configurations and deflector plate schemes.

Figure 14 presents the three-dimensional vortex isosurfaces (Liutex $R = 165$) of the two draft tube configurations under different flow rates. The isosurfaces are colored by the Liutex-Omega parameter. An Omega value close to 1 indicates a stronger dominance of rigid body rotation in the local fluid region. For the elbow draft tube, the vortex structures at $0.8Q_d$ are mainly characterized by locally discrete and fragmented wake vortices. At $1.0Q_d$, the vortex structures become more concentrated and are mainly distributed from the inlet of the straight conical section to the elbow corner, with enhanced rotational intensity of the vortex core. As the flow rate increases to $1.2Q_d$, the vortex rope expands markedly in scale. The helical vortex rope penetrates the inlet straight conical section and extends downstream of the elbow. Notably, the flow direction deflection in the elbow section disrupts the continuity of the vortex rope, causing local breakup and reorganization. The vortex structures then re-aggregate near the elbow outlet, forming a thick, continuous, and strong helical vortex rope. For the straight conical draft tube, because of its favorable axisymmetric geometry, the vortex rope exhibits a more regular helical development pattern under all operating conditions, extending downstream while uniformly winding around the central axis.

Enstrophy is used to characterize the intensity of rotational motion and vortical dissipation in the flow field. Its value reflects the rotational intensity of vortex structures in a given region and is commonly used to identify high hydraulic loss regions inside hydraulic machinery. The expression for enstrophy is given as:

$$\Omega = \frac{1}{2} \int |\omega|^2 dV \quad (30)$$

where $|\omega|$ is the magnitude of the vorticity vector.

Figure 15 shows the enstrophy distribution on the mid-plane of the two draft tube configurations. At $0.8Q_d$ and $1.0Q_d$, high enstrophy regions are concentrated near the draft tube inlet. When the flow rate increases to $1.2Q_d$, the high enstrophy regions spread toward the middle and downstream sections of the draft tube. This indicates that intense turbulent dissipation induced by the draft tube vortex under high-flow conditions is the main cause of the decline in PAT hydraulic performance. In the elbow draft tube with geometric curvature, the high enstrophy regions are closely attached to the inner arc wall of the elbow section, indicating that the transverse adverse pressure gradient in this region induces severe separation of the inner side boundary layer and leads to strong fluid shear. In contrast, in the straight conical draft tube, as the flow passage expands axisymmetrically, the high enstrophy is released more smoothly and gradually decays toward the middle and downstream sections.

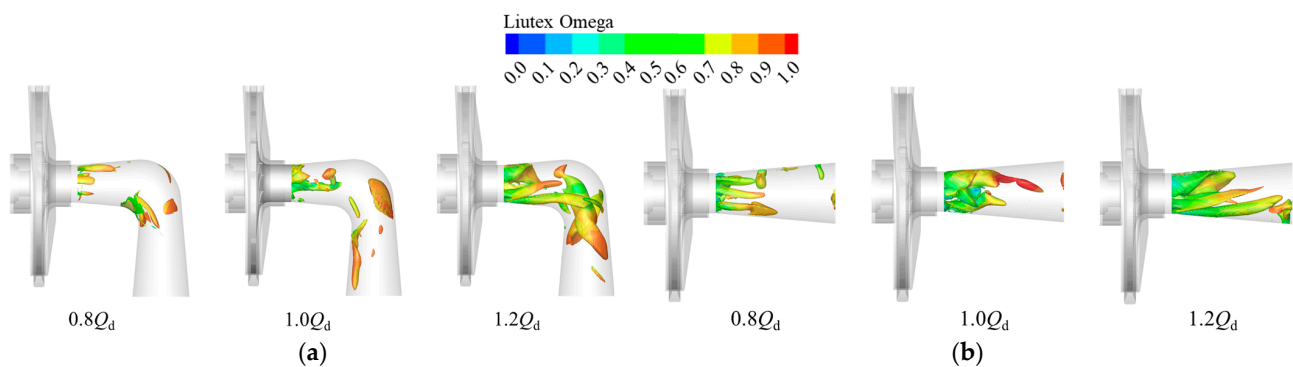


Figure 14. Vortex structure distributions in draft tubes with different configurations. (a) Elbow draft tube; (b) Straight conical draft tube.

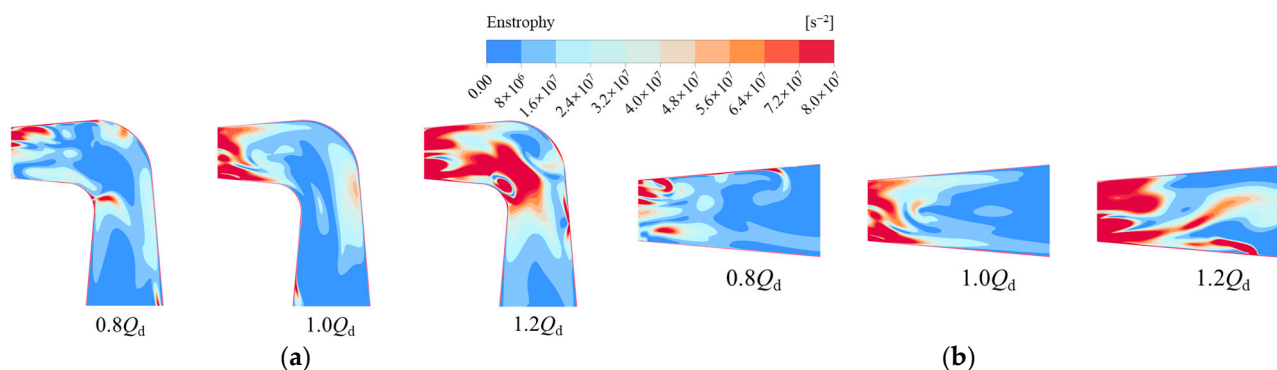


Figure 15. Enstrophy distribution on the mid-plane of draft tubes with different configurations. (a) Elbow draft tube; (b) Straight conical draft tube.

To reveal the flow characteristics inside the draft tube, several monitoring sections were arranged along the flow direction, as shown in Figure 16. For the elbow draft tube, monitoring sections A1–A6 were placed in the inlet section, elbow section, and outlet diffuser section to capture the pressure gradient variation and secondary flow development during the flow process. For the straight conical draft tube, four sections, B1–B4, were uniformly selected along the axial direction to analyze the velocity distribution and pressure recovery process in the gradually expanding section.

Figure 17 shows the enstrophy distribution on characteristic sections along the mainstream direction in the two draft tube configurations. The inlet sections in both configurations exhibit large-area, high-intensity, and irregularly distributed enstrophy accumulation regions. In the elbow draft tube, the cross-sectional expansion of the diffuser section rapidly weakens the mainstream kinetic energy. As a result, the area of the high enstrophy region

decreases when the flow enters the front part of the elbow section. In the straight conical draft tube, as the flow passage area expands uniformly, the axial velocity and vortical flow remain relatively stable. From section B1 to B2, the high enstrophy region migrates rapidly toward the wall and decays markedly. By sections B3 and B4, the flow field is mainly occupied by low dissipation regions.

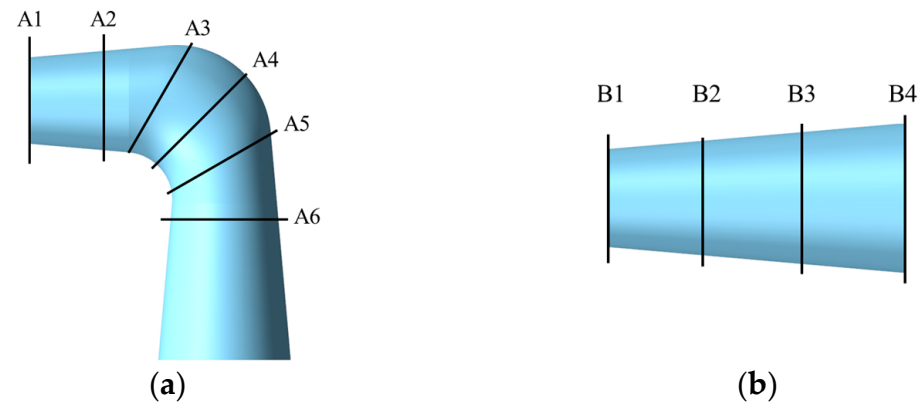


Figure 16. Distribution of monitoring sections in the draft tube. (a) Elbow draft tube; (b) Straight conical draft tube.

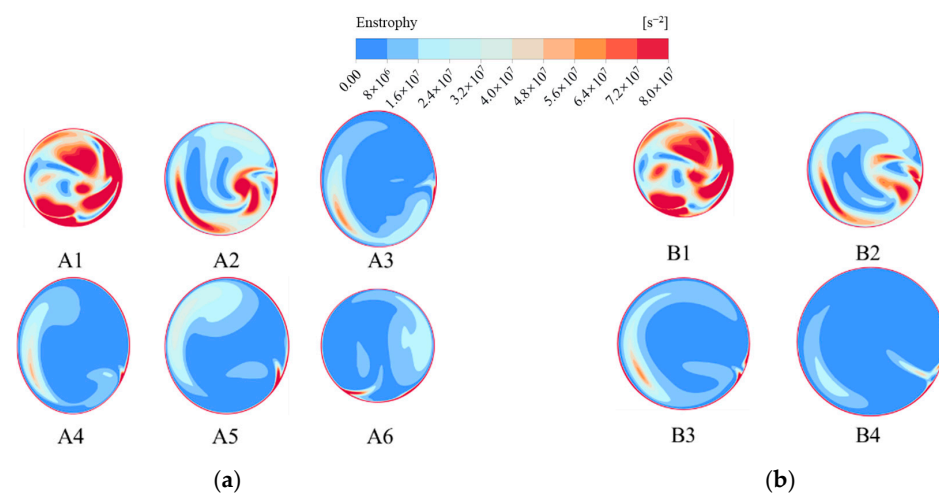


Figure 17. Sectional enstrophy distributions in draft tubes with different configurations. (a) Elbow draft tube; (b) Straight conical draft tube.

To quantitatively compare the degree of rotational intensity accumulation on different sections along the flow direction and thereby evaluate the influence of different draft tube configurations on the streamwise evolution of vortex energy, Figure 18 presents the area-weighted average enstrophy curves for the two draft tube configurations. Overall, dominated by the high residual circulation and absolute kinetic energy at the impeller outlet, the overall dissipation intensity in each configuration increases with increasing flow rate. The dissipation curve of the straight conical draft tube shows a decreasing trend, indicating that the symmetric gradually expanding geometry can smoothly guide the high swirling flow to achieve natural dissipation and kinetic energy recovery, reducing it to the lowest global energy level at B4. Although the area expansion in the inlet section of the elbow draft tube effectively weakens the absolute kinetic energy of the fluid, under high flow conditions through the elbow section, the adverse pressure gradient and flow separation induced by sharp flow-direction deflection cause strong fluctuations in the local dissipation intensity.

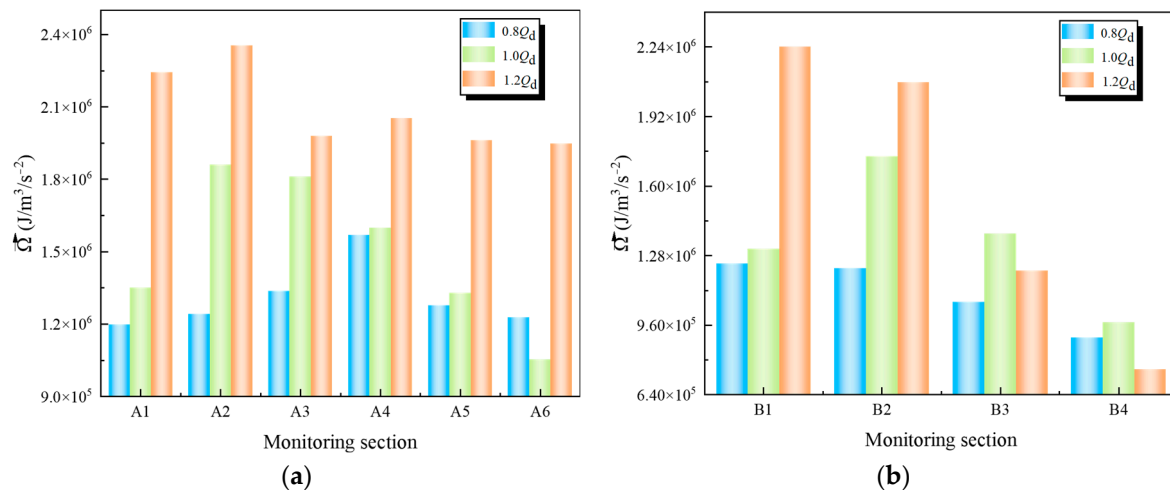


Figure 18. Streamwise distribution curves of area-averaged enstrophy at each monitoring section. (a) Elbow draft tube; (b) Straight conical draft tube.

4.3. Wake Vortex Suppression Based on Deflector Plates

The analysis in Section 4.2 shows that, compared with the straight conical draft tube, the elbow draft tube causes complex evolution behaviors, such as breakup of the helical vortex rope in the elbow section, owing to its special geometric configuration. These behaviors further induce broadband hydraulic excitation and flow instability, posing a potential threat to the safe and stable operation of the PAT unit. Accordingly, this section takes the elbow draft tube as an example and installs deflector plates in its straight conical section to disrupt the circumferential continuity of the vortex rope, thereby suppressing its evolution and improving flow field stability. The geometric parameters of each deflector plate arrangement have been detailed in Section 2.2 and are shown in Figure 3.

To quantitatively reveal the suppression effect of the deflector plates on the vortex structures inside the draft tube, three-dimensional isosurfaces of $\text{Liutex } R = 125$ were extracted, as shown in Figure 19. In the original scheme, a complete and spatially continuous eccentric helical vortex rope is formed in the straight conical section under the influence of residual circulation at the impeller outlet. The vortex rope then extends along the spatial curvature of the elbow section into the downstream transverse diffuser section. After the TDP is introduced, the circumferential continuity of the helical vortex rope is interrupted. In the TDP-15 scheme, the continuous vortex rope in the straight conical section undergoes obvious local breakup before entering the elbow section, and the length of the residual vortex rope in the downstream transverse diffuser section is significantly reduced. When the insertion depth is increased to 30 mm, the best control effect is achieved. The original continuous helical vortex rope is completely broken into small-scale and spatially dispersed shed vortex clusters. In contrast, because the LDP is parallel to the axial direction of the draft tube, a continuous vortex rope of a certain scale still remains in the straight conical section, and the volume of the vortex isosurface is clearly larger than that in the TDP scheme with the corresponding insertion depth.

Figure 20 shows the local dissipative entropy production distribution on the mid-plane of the draft tube under different control schemes. The high-entropy production regions are mainly concentrated in the wake region of the deflector plate and near the inner arc wall of the elbow section. In the TDP-15 scheme, the high-entropy production region appears as a banded distribution extending along the wake direction of the deflector plate, with energy dissipation mainly concentrated in the local downstream region of the deflector plate in the straight conical section. When the insertion depth is increased to 30 mm, a large high-entropy production region forms in the straight conical section and extends

upstream to the vicinity of the leading edge of the deflector plate. Compared with the TDP schemes, the LDP produces weaker transverse disturbance to the mainstream flow. Its wake separation intensity is clearly lower, and the high-entropy production region is mainly confined to a small-scale shear layer near the deflector plate.

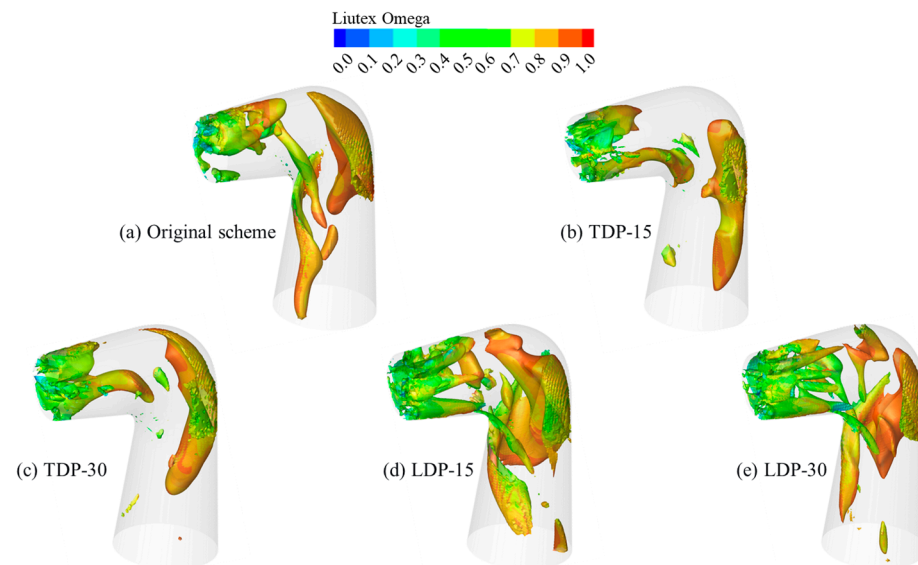


Figure 19. Vortex structure distributions under different suppression schemes.

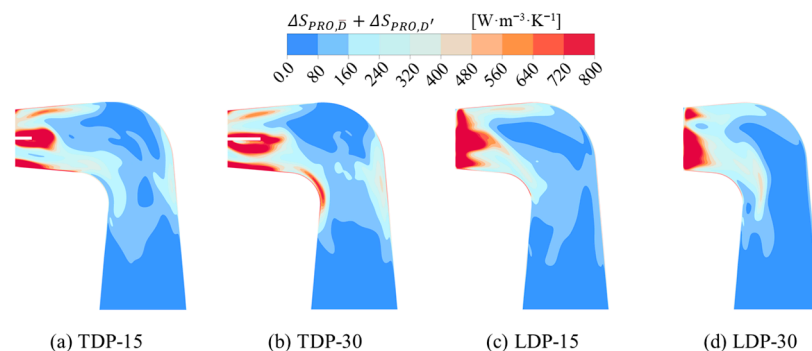


Figure 20. Local entropy production distributions under different control schemes.

Figure 21 shows the entropy production distribution in the draft tube under different control schemes. In all schemes, local entropy production dominates the total entropy production, whereas wall entropy production accounts for only a small proportion. This indicates that, after the deflector plates are installed, energy dissipation in the draft tube is not mainly caused by streamwise friction between the fluid and the wall. Instead, it primarily originates from the strong turbulent fluctuations in the wake region of the deflector plate and viscous friction associated with vortex breakup. In addition, the total entropy production of the TDP-30 scheme is significantly higher than that of the other schemes. This is mainly because this control scheme consumes the large rotational kinetic energy carried by the helical vortex rope at the draft tube inlet by inducing higher entropy production, thereby weakening the vortex rope and producing a slight improvement in the overall hydraulic efficiency of the PAT.

To quantitatively evaluate the irreversible energy dissipation at solid-wall boundaries under different schemes, the wall entropy production rates on the draft tube wall and deflector plate surfaces were compared. Figure 22 shows the wall entropy production distributions on the draft tube wall (DTW), the upstream-facing surface (US), and the downstream-facing surface (DS) of the deflector plate under the four schemes. After the

deflector plates are installed, the entropy production on the original DTW remains low in all schemes, whereas the deflector plate surfaces dominate solid wall frictional dissipation. This indicates that the deflector plates improve the flow pattern and reduce the vortex rope, but also induce different degrees of wall-related energy loss. In addition, the US in the TDP schemes produces relatively large energy dissipation, which increases sharply with increasing insertion depth. In particular, the entropy production on the US in the TDP-30 scheme reaches 2.2 W/K and becomes dominant. This indicates that increasing the insertion depth enhances vortex rope suppression, but also causes considerable energy loss on the US. In contrast, in the LDP schemes, the wall entropy production on both sides of the deflector plate is more uniformly distributed and is significantly lower than the US peak in the TDP schemes. Combined with the preceding analysis, the LDP schemes have a limited suppression effect on the vortex rope. The disturbance imposed by the deflector plate on the mainstream flow is relatively weak and fails to effectively reorganize the vortex rope topology. Therefore, their overall improvement in macroscopic hydraulic performance is inferior to that of the TDP schemes.

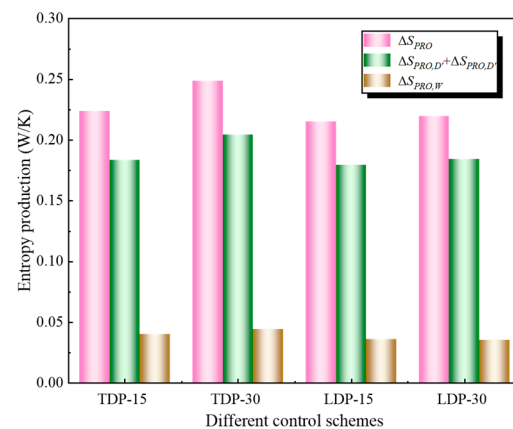


Figure 21. Entropy production distributions in the draft tube under different control schemes.

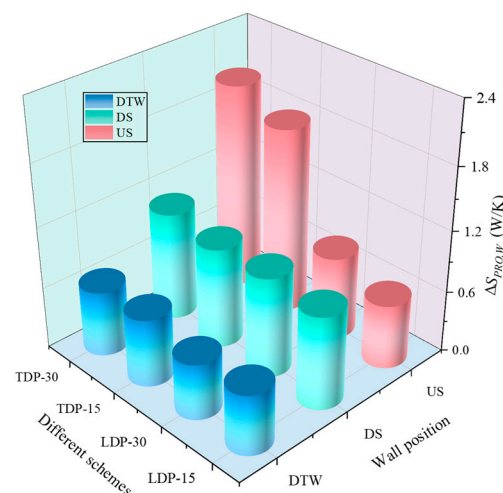


Figure 22. Wall entropy production distributions under different suppression schemes.

5. Conclusions

This study takes a PAT with splitter blades as the research object and systematically investigates the evolution mechanism and suppression method of wake vortices inside the draft tube by combining Burgers vortex theory, numerical simulation, and the Liutex vortex identification method. The main conclusions are as follows:

- (1) A modified Burgers wake vortex analytical model suitable for a PAT with splitter blades was established by considering the velocity slip effect at the impeller outlet and viscous dissipation of vortical flow. Compared with the classical Burgers vortex model, the proposed model significantly improves the prediction accuracy of the vortex core tangential velocity. At $1.0Q_d$, the classical model overestimates the peak tangential velocity by approximately 76% compared with the numerical simulation result. After introducing the correction coefficient $k = 0.38$, the analytical prediction agrees well with the CFD result. Under the $0.6Q_d$ and $1.4Q_d$ operating conditions, the model can still accurately capture the peak velocity and radial decay trend in the main vortex core region, with peak velocity errors controlled within 5%. This indicates that the model has good adaptability to different operating conditions and good prediction accuracy.
- (2) The draft tube configuration has a significant influence on the spatial evolution and energy dissipation of wake vortices. As the flow rate increases from $0.6Q_d$ to $1.2Q_d$, the helical vortex rope in the elbow draft tube expands markedly in scale. Owing to the effects of geometric curvature and flow-direction deflection in the elbow section, the sectional enstrophy becomes unevenly distributed along the flow direction, and local energy dissipation is enhanced. In contrast, owing to its favorable diffusion and flow-straightening effects, the straight conical draft tube continuously weakens the high-energy vortex rope along the axial direction, and its flow field uniformity is significantly better than that of the elbow draft tube.
- (3) Deflector plates can effectively disrupt the circumferential continuity of the helical vortex rope inside the draft tube, thereby suppressing wake vortex development. The vortex suppression effect of the TDP is clearly superior to that of the LDP, among which the TDP-30 scheme shows the strongest vortex breakup and dissipation effect. However, increasing the insertion depth of the deflector plate induces additional hydraulic losses. Wall entropy production analysis shows that the wall entropy production on the upstream-facing surface of the deflector plate in the TDP-30 scheme reaches 2.2 W/K , making it dominant among all wall regions. In contrast, the wall entropy production on both sides of the LDP is more uniformly distributed, but its disturbance to the mainstream flow and vortex rope topology is weaker. Therefore, its overall vortex suppression effect and hydraulic performance improvement are inferior to those of the TDP.

This study is limited to a semi-empirical relationship between the correction coefficient k and the flow dissipation coefficient S , established for a PAT with a specific speed of 16.5 and a specific splitter blade configuration. Its applicability is restricted to the geometry and calibration range considered herein. Future research will extend and recalibrate the modified model for different specific speeds and splitter blade configurations and employ PIV or LDA to measure local velocity distributions within the draft tube.

Author Contributions: C.W.: Review and editing, methodology, data curation. W.Z.: Writing—original draft, formal analysis. Y.S.: Software, methodology. H.L.: Validation, visualization. D.W.: supervision, resources. Q.S.: Investigation, funding acquisition, methodology. All authors have read and agreed to the published version of the manuscript.

Funding: This work was supported by the Guangdong Special Support Program for High-level Talents (2024JC08L126), the Special fund of technological innovation for carbon peak and carbon neutrality in Jiangsu Province (BE2022032-3).

Data Availability Statement: The raw data supporting the conclusions of this article will be made available by the authors on request.

Conflicts of Interest: Authors Chenguang Wang and Hua Liu were employed by the company Zhuhai Gree Electric Appliance Co., Ltd. The remaining authors declare that the research was conducted in the absence of any commercial or financial relationships that could be construed as a potential conflict of interest.

Abbreviations

The following abbreviations are used in this manuscript:

PAT	Pump as turbine
TDP	Transverse deflector plate
LDP	Longitudinal deflector plate
CFD	Computational fluid dynamics
RSI	Rotor–stator interaction
SST	Shear stress transport
RANS	Reynolds-Averaged Navier–Stokes
US	Upstream surface
DS	Downstream surface
DTW	Draft tube wall

References

1. Chang, J.X.; Li, Y.Y.; Yuan, M.; Wang, Y.M. Efficiency evaluation of hydropower station operation: A case study of Longyangxia station in the Yellow River, China. *Energy* **2017**, *135*, 23–31. [\[CrossRef\]](#)
2. Nautiyal, H.; Varun; Kumar, A. Reverse running pumps analytical, experimental and computational study: A review. *Renew. Sustain. Energy Rev.* **2010**, *14*, 2059–2067. [\[CrossRef\]](#)
3. Cavazzini, G.; Santolin, A.; Pavesi, G.; Ardizzon, G. Accurate estimation model for small and micro hydropower plants costs in hybrid energy systems modelling. *Energy* **2016**, *103*, 746–757. [\[CrossRef\]](#)
4. Xu, Z.; Zheng, Y.; Kan, K.; Chen, H.X. Flow instability and energy performance of a coastal axial-flow pump as turbine under the influence of upstream waves. *Energy* **2023**, *272*, 127121. [\[CrossRef\]](#)
5. Qi, B.; Zhang, D.S.; Geng, L.L.; Zhao, R.J.; van Esch, B.P.M. Numerical and experimental investigations on inflow loss in the energy recovery turbines with back-curved and front-curved impeller based on the entropy generation theory. *Energy* **2022**, *239*, 122426. [\[CrossRef\]](#)
6. Zhang, Y.N.; Zhang, Y.N.; Wu, Y.L. A review of rotating stall in reversible pump turbine. *Proc. Inst. Mech. Eng. Part C-J. Mech. Eng. Sci.* **2017**, *231*, 1181–1204.
7. Zhang, Y.N.; Chen, T.; Li, J.W.; Yu, J.X. Experimental Study of Load Variations on Pressure Fluctuations in a Prototype Reversible Pump Turbine in Generating Mode. *J. Fluids Eng.-Trans. ASME* **2017**, *139*, 074501. [\[CrossRef\]](#)
8. Fecarotta, O.; Aricò, C.; Carravetta, A.; Martino, R.; Ramos, H.M. Hydropower Potential in Water Distribution Networks: Pressure Control by PATs. *Water Resour. Manag.* **2015**, *29*, 699–714.
9. Kim, J.; Oh, K.; Pyun, K.; Kim, C.; Choi, Y.; Yoon, J. Design optimization of a centrifugal pump impeller and volute using computational fluid dynamics. *IOP Conf. Ser. Earth Environ. Sci.* **2012**, *15*, 032025. [\[CrossRef\]](#)
10. Yang, W.; Xiao, R.F. Multiobjective Optimization Design of a Pump-Turbine Impeller Based on an Inverse Design Using a Combination Optimization Strategy. *J. Fluids Eng.-Trans. ASME* **2014**, *136*, 014501.
11. Rode, B.R.; Kumar, A. Effects of vortex rope formation in draft tube of reversible pump-turbine—A review. *J. Energy Storage* **2024**, *99*, 113250. [\[CrossRef\]](#)
12. Zhou, T.X.; Yu, X.D.; Chen, S.; Zhang, J.; Xu, H. An intelligent identification method of draft tube vortex rope based on dynamic feature extraction and random forest: Application to a prototype pump-turbine. *J. Energy Storage* **2024**, *102*, 114227. [\[CrossRef\]](#)
13. Yang, F.; Li, Z.B.; Yuan, Y.; Lin, Z.K.; Zhou, G.X.; Ji, Q.W. Study on vortex flow and pressure fluctuation in dustpan-shaped conduit of a low head axial-flow pump as turbine. *Renew. Energy* **2022**, *196*, 856–869. [\[CrossRef\]](#)
14. Anup, K.C.; Lee, Y.H.; Thapa, B. CFD study on prediction of vortex shedding in draft tube of Francis turbine and vortex control techniques. *Renew. Energy* **2016**, *86*, 1406–1421. [\[CrossRef\]](#)
15. Liu, Y.B.; Tan, L. Tip clearance on pressure fluctuation intensity and vortex characteristic of a mixed flow pump as turbine at pump mode. *Renew. Energy* **2018**, *129*, 606–615. [\[CrossRef\]](#)
16. Zhou, J.Y.; Xu, L.C.; Zhang, Y.Q.; Zheng, Y.; Li, C.Y.; Zang, W.; Zhang, Z. Analysis of vortex evolution in the stall region of pump-turbines in pump mode: Based on model testing and CFD. *Renew. Energy* **2026**, *256*, 124410. [\[CrossRef\]](#)

17. Lai, X.D.; Chen, X.M.; Liang, Q.W.; Ye, D.X.; Gou, Q.Q.; Wang, R.T.; Yan, Y. Experimental and numerical investigation of vortex flows and pressure fluctuations in a high-head pump-turbine. *Renew. Energy* **2023**, *211*, 236–247. [\[CrossRef\]](#)
18. Wang, Y.Q.; Cheng, L.; Feng, X.S.; Teng, H.B.; Yuan, L.G.; Ma, X.F. Investigation of vortex turbulent structures and multiscale energy transfer characteristics in the draft-tube vortex of a pump-as-turbine based on flow field model reduction and energy cascade analysis. *Energy* **2026**, *357*, 141226. [\[CrossRef\]](#)
19. Xu, Z.; Zheng, Y.; Kan, K.; Pavesi, G.; Rossi, M.; Xu, L.C.; Yan, X.T. Vortex evolution and energy loss of an axial flow Pump-As-Turbine (PAT) with the influence of upstream waves. *Energy* **2025**, *322*, 135726. [\[CrossRef\]](#)
20. Xue, Y.L.; Wang, S.; Wang, W.; Cao, L.; Shi, G.T.; Hao, Z.K. Study on the relationship between pressure pulsation and rigid vortex in the hump area of reversible pump turbine. *J. Energy Storage* **2025**, *132*, 117907. [\[CrossRef\]](#)
21. Burgers, J.M. A mathematical model illustrating the theory of turbulence. *J. Adv. Appl. Mech.* **1948**, *1*, 171–199. [\[CrossRef\]](#)
22. Rosenhead, L. The spread of vorticity in the wake behind a cylinder. *Proc. R. Soc. A* **1930**, *127*, 590–612. [\[CrossRef\]](#)
23. Hite, J.E., Jr.; Mih, W.C. Velocity of air-core vortices at hydraulic intakes. *J. Hydraul. Eng.* **1994**, *120*, 284–297. [\[CrossRef\]](#)
24. Saffman, P.G. Vortex dynamics. In *Theoretical Approaches to Turbulence*; Springer: Berlin/Heidelberg, Germany, 1992; pp. 263–277.
25. Escudier, M.; Bornstein, J.; Maxworthy, T. The dynamics of confined vortices. *Proc. R. Soc. A* **1982**, *382*, 335–360. [\[CrossRef\]](#)
26. Duinmeijer, A.; Oldenziel, G.; Clemens, F. Experimental study on the 3D-flow field of a free-surface vortex using stereo PIV. *J. Hydraul. Res.* **2020**, *58*, 105–119. [\[CrossRef\]](#)
27. Wang, L.; Cui, J.Y.; Shu, L.F.; Jiang, D.H.; Xiang, C.; Li, L.W.; Zhou, P.J. Research on the Vortex Rope Control Techniques in Draft Tube of Francis Turbines. *Energies* **2022**, *15*, 9280. [\[CrossRef\]](#)
28. Pang, S.J.; Zhu, B.S.; Shen, Y.D.; Chen, Z.M. Study on suppression of cavitating vortex rope on pump-turbines by J-groove. *Appl. Energy* **2024**, *360*, 122843. [\[CrossRef\]](#)
29. Kumar, S.; Cervantes, M.J.; Gandhi, B.K. Rotating vortex rope formation and mitigation in draft tube of hydro turbines—A review from experimental perspective. *Renew. Sustain. Energy Rev.* **2021**, *136*, 110354. [\[CrossRef\]](#)
30. Zhou, X.; Wu, H.G.; Cheng, L.; Huang, Q.S.; Shi, C.Z. A new draft tube shape optimisation methodology of introducing inclined conical diffuser in hydraulic turbine. *Energy* **2023**, *265*, 126374. [\[CrossRef\]](#)
31. Daniels, S.J.; Rahat, A.A.M.; Tabor, G.R.; Fieldsend, J.E.; Everson, R.M. Shape optimisation of the sharp-heeled Kaplan draft tube: Performance evaluation using Computational Fluid Dynamics. *Renew. Energy* **2020**, *160*, 112–126. [\[CrossRef\]](#)
32. Wang, X.; Yang, J.F.; Huang, X.W.; Wang, W.Q. Using bionic tubercles to control swirling flow instabilities of a hydraulic turbine during the load rejection process. *Energy* **2024**, *311*, 133354. [\[CrossRef\]](#)
33. Shrestha, U.; Choi, Y.D. Suppression of flow instability in the Francis hydro turbine draft tube by J-groove shape optimization at a partial flow rate. *J. Mech. Sci. Technol.* **2021**, *35*, 2523–2533. [\[CrossRef\]](#)
34. Wang, X.; Luo, X.; Li, S.H.; Wang, W.Q. Enhancing operational flexibility of large hydraulic turbines through a passive draft-tube stabilizing ring. *Int. J. Heat Fluid Flow* **2026**, *120*, 110429. [\[CrossRef\]](#)
35. Zhou, X.; Wu, H.G.; Shi, C.Z. Numerical and experimental investigation of the effect of baffles on flow instabilities in a Francis turbine draft tube under partial load conditions. *Adv. Mech. Eng.* **2019**, *11*, 1687814018824468. [\[CrossRef\]](#)
36. Joy, J.; Raisee, M.; Cervantes, M.J. Hydraulic Performance of a Francis Turbine with a Variable Draft Tube Guide Vane System to Mitigate Pressure Pulsations. *Energies* **2022**, *15*, 6542. [\[CrossRef\]](#)
37. Derakhshan, S.; Nourbakhsh, A. Experimental study of characteristic curves of centrifugal pumps working as turbines in different specific speeds. *Exp. Therm. Fluid Sci.* **2008**, *32*, 800–807. [\[CrossRef\]](#)
38. Liang, A.; Li, H.C.; Zhang, W.W.; Yao, Z.F.; Zhu, B.S.; Wang, F.J. Study on pressure fluctuation and rotating stall characteristics in the vaneless space of a pump-turbine in pump mode. *J. Energy Storage* **2024**, *94*, 112385. [\[CrossRef\]](#)
39. Gong, R.Z.; Wang, H.J.; Chen, L.X.; Li, D.Y.; Zhang, H.C.; Wei, X.Z. Application of entropy production theory to hydro-turbine hydraulic analysis. *Sci. China-Technol. Sci.* **2013**, *56*, 1636–1643. [\[CrossRef\]](#)
40. Adu, D.; Du, J.; Darko, R.O.; Antwi, E.O.; Khan, M.A.S. Numerical and experimental characterization of splitter blade impact on pump as turbine performance. *Sci. Prog.* **2021**, *104*, 0036850421993247. [\[CrossRef\]](#) [\[PubMed\]](#)

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.