Research Article

Numerical Simulation on the Influence of Rotating Speed on the Hydraulic Loss Characteristics of Desalination Energy Recovery Turbine

Bing Qi , Desheng Zhang , Qi Zhang, Mengcheng Wang , and Ibra Fall

National Research Center of Pumps, Jiangsu University, Zhenjiang, Jiangsu 212013, China

Correspondence should be addressed to Desheng Zhang; zds@ujs.edu.cn

Received 16 March 2021; Revised 23 July 2021; Accepted 3 August 2021; Published 2 September 2021

Academic Editor: Roman Gabl

Copyright © 2021 Bing Qi et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

The performance of energy recovery turbine (ERT) directly determines the cost and energy consumption of reverse osmosis desalination. In order to study the performance and loss mechanisms of ERT under different conditions, the external characteristics and the losses of different components were quantitatively analyzed. The loss mechanisms of each component in the turbine were revealed through the comparative analysis of the internal flow field. The results show that the efficiency is 2.2% higher than that at the design speed when turbine runs at $n = 22000$ r/min. The impeller losses account for more than 67% of the total losses. The impeller loss is mainly observed at the leading edge. The vortex on the pressure side of the leading edge is caused by the impact effect, while the vortex on the suction side of the leading edge is caused by the flow separation. With the increase in the rotating speed, the loss caused by flow separation in impeller decreases obviously. The volute loss is mainly observed near the tongue, which is caused by the flow separation at the tongue. The design of the tongue is very important to the performance of the volute. The turbulent kinetic energy (TKE) and loss decrease with the increase in the rotating speed. The loss in the draft tube is mainly observed at the inlet core. With the increase in the rotating speed, the turbulence pulsation and the radial pressure fluctuation amplitude reduce. Therefore, the turbine can be operated at the design or slightly higher than the design rotating speed under the condition that both the hydraulic condition and the intensity are satisfied, which are conducive to the efficient utilization of energy.

1. Introduction

Seawater desalination by reverse osmosis is one of the means to solve the global shortage of fresh water resources. At present, it is developing towards the trend of reducing project cost and operating cost. In desalination systems without any energy recovery equipment, the booster pump that pressurizes the seawater to pass through the desalination membrane accounts for about 70% of the total system energy consumption [1]. Corresponding energy recovery devices need to be developed to reduce the high cost of high energy consumption and improve the energy utilization efficiency [2]. Therefore, energy recovery device, as the key supporting equipment of reverse osmosis desalination technology, gradually becomes the focus of research [3].

At present, the turbine is used to recover and reuse the energy of the high-pressure brine which could not pass through the reverse osmosis membrane has become a common energy recovery method in reverse osmosis desalination system [4]. The energy recovery turbine (ERT) has the advantages of simple structure, wide operating range, and low cost [5]. Existing data show that the ERT can reduce the energy consumption of reverse osmosis desalination by about 60% [6]. The ERT has broad application prospects in industrial production, so developing an efficient ERT and applying it to reverse osmosis seawater desalination system will have great value and significance.

The research on the reverse turbine mainly focuses on three aspects: turbine selection, design and optimization, internal loss mechanism, and unsteady pressure fluctuation characteristics. Large quantities of studies have been

conducted to predict the best efficiency point of PAT with different research methods [7–9]. Singh and Nestmann [10] studied the influence of impeller inlet rounding on turbine performance of centrifugal and mixed flow turbine by experimental method. It was found that impeller rounding helps to improve the efficiency of the whole operating condition of the turbine, with an increase range of 1–3%. Wang et al. [11] proposed the theoretical design method of turbine special impeller on the premise of keeping the structural parameters of volute unchanged and verified the feasibility of the design method of turbine special impeller by combining numerical simulation and experiment. Wang et al. [12] used the inverse design method to optimize the impeller of a pump as a turbine and proposed that minimizing the profile loss coefficient benefits the turbine mode efficiency. Liu et al. [13, 14] used the method of experiment and numerical calculation to study the internal flow and loss mechanism of pump as turbine at the best efficiency point in two modes. The reason why the efficiency in turbine mode is lower than that in pump mode and the loss mechanism are clarified, and the method of impeller optimization based on loss theory was proposed. Lin et al. [15] investigated the flow separation characteristics in a PAT impeller under the BEP condition by numerical method and used surface friction lines and flow topological structure to diagnose the flow separation at the surface of the blade. Lin et al. [16] used the entropy dissipation theory to study the energy loss mechanism of each part of a centrifugal pump when it was operating under different flow conditions. It was found that the entropy dissipation theory can not only accurately calculate the loss of each part of the turbine, but also determine the location and mode of water loss. Ghorani et al. [17] studied the energy loss mechanism of a pump as turbine for the first time from the perspective of entropy generation theory and the second law of thermodynamics and pointed out that the turbulent term in entropy generation is the main factor causing the energy loss in the turbine. Li et al. [18] pointed out that mass flow mainly affects the position of pressure fluctuation peak, and the number of blades mainly affects the main frequency of pressure fluctuation. Yang et al. [19] found that there is an optimal radial clearance to achieve the highest efficiency. Han et al. [20, 21] studied the influence of rotating speed on tip leakage vortex of mixed flow pump and the evolution characteristics of leakage vortex.

Many achievements have been made in the research of ERT, and it has become an essential trend to add ERT in seawater desalination system to reduce the energy consumption and cost of water production. However, the flow and pressure of seawater through desalination membrane are easily affected by seasonal changes, which will affect the flow and pressure of turbine entrance. On the other hand, the operating conditions are easily affected by the load of high-pressure pump terminals. So the operating conditions of ERT are complex and changeable. Therefore, it is of great value and significance to study the performance and loss distribution of the turbine under different operating conditions and to reveal its internal flow characteristics and loss

mechanism. In this study, a seawater desalination ERT was taken as the research object. The performance of the turbine and the main loss distribution of key components under different rotating speeds and variable conditions were calculated by combining experimental and numerical methods, and the loss mechanisms were studied to provide guidance for the design and optimization of ERT.

2. Reverse Osmosis Desalination Technology and ERT

2.1. Reverse Osmosis Desalination Technology. Reverse osmosis desalination uses a semipermeable membrane, which only allows the solvent to pass through but not the solute to pass through and uses the pressure difference as the driving force to separate the seawater from the fresh water. When the pressure exceeds the osmotic pressure of the membrane, the seawater passes through the desalination membrane to get fresh water at the low pressure side of the membrane and concentrated seawater at the high pressure side, i.e. concentrated saline water [22]. Reverse osmosis desalination technology is one of the mainstream technologies of modern and future desalination, which currently accounts for 65% [23] of global desalination installed capacity. In the process of reverse osmosis desalination, energy consumption is needed to increase seawater pressure to overcome osmosis pressure to complete desalination. In the process of reverse osmosis desalination without energy recovery device, the energy consumption is about 8–10 kW h/m³. However, the energy consumption of reverse osmosis desalination can be greatly reduced to 3–4.5 kW h/m³ [24] by recovering concentrated saline water with high pressure. Therefore, the research and development of energy recovery device has become a very important part in the development of reverse osmosis desalination.

2.2. Working Principle of ERT. Figure 1 shows the operation principle of an energy recovery turbine unit. The seawater enters from the inlet of the front boost pump. The seawater is supplied with a certain pressure energy through the front boost pump. The seawater then flows into the unit from the inlet of the booster pump, and it is further pressurized by the boost pump. Then sea water flows into the reverse osmosis membrane module for desalination. Some of it passes through the desalination membrane and becomes fresh water; the other becomes concentrated saline with high pressure. This part of high-pressure concentrated saline water flows into the turbine. Because the turbine and booster pump are coaxially connected, it drives the booster pump to run, thus achieving the purpose of energy recovery. As shown in Figure 2, it is a three-dimensional model of a turbine-type energy recovery unit, which mainly consists of boost pump, ERT, and spindle. In this paper, the ERT is extracted as the main research object to study its hydraulic performance and energy loss mechanism at different rotating speeds.

FIGURE 1: Flowchart of reverse osmosis desalination process.

FIGURE 2: ERT model. (1) Flow control valve for concentrated brine; (2) signage; (3) thrust bearing; (4) single vortex design of brine pipe; (5) turbine impeller blades; (6) rotor shaft; (7) flow path of hydraulic thrust bearing center; (8) impeller of booster pump; (9) single-ended blind; (10) pump volute; (11) base; (12) seal ring; (13) inlet of integral machine; (14) standard material; (15) outlet of integral machine.

3. Physical Model and Mesh

3.1. Physical Model of Turbine. Table 1 shows the main design parameters of an ERT, where the specific speed N_{sd} is calculated as follows:

$$N_{sd} = \frac{3.65n_d\sqrt{Q_d}}{H_d^{0.75}}. \quad (1)$$

The meanings and units of each variable in equation (1) are the same as those in Table 1.

A hydraulic model of the ERT is shown in Figure 3. The hydraulic model was selected and designed according to the literature [19]. The turbine hydraulic model mainly consists of 4 parts: volute, impeller, chamber, and draft tube. In order to ensure that the fluid fully enters the volute and flows out

TABLE 1: Main design parameters of ERT.

Parameter	Value
Design flow rate Q_d (m ³ /h)	50.9
Inlet pressure (MPa)	5.5
Rotating speed n_d (r/min)	20000
Specific speed N_{sd}	76
Number of impeller blades Z	6
Impeller inlet diameter D_I (mm)	75

FIGURE 3: Hydraulic model of ERT.

of the draft tube and to ensure the accuracy of the following numerical calculations, the suction chamber and draft tube were extended appropriately.

3.2. Mesh Arrangement. In this paper, a hexahedral structured mesh was used to discrete the entire computational domain based on ANSYS-ICEM. In order to ensure the accuracy of calculation and to accurately simulate the characteristics near the wall, the meshes near the impeller blade and volute tongue wall were partially refined. In addition, the Y-type topology was used to generate the mesh of impeller calculation domain inlet. As shown in Figure 4(a), the mesh diagram of turbine impeller and Y Plus distribution of blade were shown, in which the height of the first cell on the blade wall was 0.02 mm, and the maximum Y^+ value was 34. Figure 4(b) shows the mesh of volute.

4. Numerical Method and Setting

4.1. Numerical Method. The computational fluid dynamics code of CFX 19.0 is employed in the present work. The shear stress transfer (SST) $k-\omega$ turbulence model that takes into account the advantages of $k-\epsilon$ turbulence model and $k-\omega$ turbulence model was used in steady simulation [25–27]. The SST $k-\omega$ model includes the turbulent kinetic energy equation (2) and the turbulent frequency equation (3):

FIGURE 4: Turbine mesh generation. (a) Impeller grid and Y^+ values of blades. (b) Spiral case mesh.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{u}_i k)}{\partial x_i} = \bar{P}_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i} \right], \quad (2)$$

$$\begin{aligned} \frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \bar{u}_i \omega)}{\partial x_i} &= \left[\frac{5}{9} F_1 + 0.44(1 - F_1) \right] \frac{1}{v_t} \bar{P}_k - \beta \rho \omega^2 \\ &+ \frac{\partial}{\partial x_i} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] \\ &+ 2(1 - F_1) \frac{\rho \sigma_{\omega 2}}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}. \end{aligned} \quad (3)$$

The calculation of Reynolds stress in this model is based on the Bossinger hypothesis in the following equation [17]:

$$\overline{u'_i u'_j} = -\nu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) + \frac{2}{3} k \delta_{ij}. \quad (4)$$

The boundary condition of the turbine inlet was set as mass flow, the outlet was set as opening flow, and the reference pressure was set as 1 atm. The performance curve of the turbine was obtained by changing the mass flow value of the turbine inlet. The surface roughness of the chamber near the front and back covers of the impeller was set as 50 μm . The connection between the static domain and the rotating domain was interface. The second-order upwind scheme was used to discretize the turbulent kinetic energy, specific dissipation rate, and momentum. The convergence criterion was defined as that the calculation was considered to be

convergent when the convergence accuracy of root mean square residuals was less than 10^{-5} .

4.2. Independence Test of Mesh Density. In this numerical calculation process, in order to ensure the accuracy of numerical calculation, the influence of mesh density on the numerical results was considered. In order to evaluate the influence of mesh density on the numerical results, the turbine model was divided into four groups of different numbers of grids (meshes 1 to 5). As shown in Figure 5, the influence of mesh number on turbine head was shown. The global mesh number ranges from 2.64 million to 4.96 million. It can be seen that when the mesh number exceeds 4.0×10^6 , the head becomes stable. The head error calculated by mesh 4 and mesh 5 was 0.03%. Therefore, the influence of mesh density on turbine calculation results could be ignored when the mesh number exceeds 4.5×10^6 . To calculate accurately and save time, mesh 4 was used in the present study.

4.3. Simulation Accuracy Validation. In order to verify the reliability of the numerical calculation method, it is necessary to calculate and test the performance of the turbine. However, because the rotating speed of the original turbine is very high, the current laboratory conditions and costs cannot meet. Therefore, the performance of a turbine prototype is tested and calculated. The rotating speed of the turbine prototype is 1450 rpm. As shown in Figure 6, the schematic diagram of the turbine prototype performance

FIGURE 5: Head versus mesh number.

FIGURE 6: Schematic diagram of turbine performance test bench.

test was shown. During the test, the pressure and flow signals were collected and processed by signal box and displayed and read on the computer. In this paper, the flow rate of the boost pump system and the output pressure of boost pump were regulated by the frequency converter, so as to change the parameters at the input of the turbine, and the speed of the turbine was controlled by adding a load to the dynamometer, thus the performance characteristic curves of the turbine at different speeds were obtained. As shown in Figure 7, the turbine prototype was shown, in which the measuring range of pressure sensor was 1.6 MPa and the measuring accuracy was 0.01 Pa. The pressure-bearing range of electromagnetic flowmeter was 0–1.6 MPa, and the measuring accuracy was $10^{-4} \text{ m}^3/\text{h}$.

Figure 8 shows the external characteristic curve of the numerical calculation and experimental test of the turbine prototype. 11 flow rates from $13.96 \text{ m}^3/\text{h}$ to $31.32 \text{ m}^3/\text{h}$ were selected to calculate and test the corresponding head and efficiency. It can be observed that the change trends of the numerical calculation performance are basically consistent with the experimental results. There are still some errors between the numerical and the experimental results. These can be attributed to two reasons: the geometries used for

FIGURE 7: Turbine field test.

computation are not as totally same as that for an experiment, and there is a certain error on the friction loss and volume loss between the numerical and the experimental results. However, the overall error ratio of numerical

FIGURE 8: Numerical simulation and experimental test results of turbine model machine.

calculation and experiment is relatively small. The errors of numerical and experimental back-curved PAT head and efficiency under the design condition are 3.4% and 2.1%, respectively. The errors of numerical and experimental front-curved PAT head and efficiency under the design condition are 2.8% and 2.1%, respectively. Therefore, the selected mesh and numerical methods are qualified to guarantee the accuracy and reliability of this present work.

5. Result and Discussion

5.1. Performance of Turbine under Variable Conditions at Different Speeds. The performance characteristics of turbines under different flow conditions at different speeds were obtained. The overall performance of the turbine at four different speeds ($n = 16000$ r/min, $n = 18000$ r/min, $n = 20000$ r/min, and $n = 22000$ r/min) was studied.

Figure 9(a) shows the flow rate-head distribution of a turbine that operates at different rotating speeds. Obviously, the head increases with the increase in the rotating speed at the same flow condition, which indicates that the ability of the turbine to convert the energy of the fluid into mechanical energy increases with the increase in the rotating speed. The head of the turbine increases with the increase in the flow rate at the same speed.

Figure 9(b) shows the distribution of the flow rate-shaft power characteristic curve of a turbine running at different speeds. It can be seen that the shaft power increases with the increase in flow rate at the same speed. The higher the turbine speed, the greater the shaft power under the same flow condition, and the closer to the design condition, the greater the difference of shaft power between adjacent rotating speeds than the small flow condition, which indicates that the closer the turbine is to the design condition, the higher the shaft power provided by the turbine is, the higher the energy converted into useful power is. Under design conditions, the shaft power of turbine running at $n = 22000$ r/min is 8.4% higher than that at design speed.

Figure 9(c) shows the flow-efficiency curve of a turbine that operates at four different speeds. It can be seen from the figure that the maximum efficiency points of the turbine

deviate from the design conditions at all four speeds. The lower the speed, the farther the high-efficiency points that deviate from the design conditions. When the operating speed of the turbine exceeds the design speed, the maximum efficiency of the turbine decreases slightly, but its high-efficiency range improves slightly. Under design flow rate conditions, the efficiency of a turbine at $n = 22000$ r/min is 2.2% higher than that at the design speed. Therefore, the efficiency of the turbine will be improved if the operating speed of the turbine is higher than that of the design speed in a certain range.

5.2. Loss Distribution of Turbines under Variable Operating Conditions. The performances at the design speed ($n = 20000$ r/min), one low speed ($n = 18000$ r/min), and one high speed ($n = 22000$ r/min) were selected, respectively, to make a more comprehensive and concise comparison and analysis. And then the losses in turbine key components such as volute, impeller, and draft tube were calculated numerically.

$$\begin{aligned}\Delta h_V &= \frac{Pt_{V_in} - Pt_{V_out}}{\rho \cdot g}, \\ \Delta h_{DT} &= \frac{Pt_{DT_in} - Pt_{DT_out}}{\rho \cdot g}, \\ \Delta h_I &= \frac{P_{in} - P_{out}}{\rho \cdot g} - \Delta h_V - \Delta h_{DT} - \Delta h_C,\end{aligned}\quad (5)$$

where Δh_V is the loss in the volute, m; Pt_{V_in} and Pt_{V_out} are separately the total pressure at the inlet and outlet of the volute, Pa; Δh_{DT} is the loss in the draft tube, m; Pt_{DT_in} and Pt_{DT_out} are separately the total pressure at the inlet and outlet of the draft tube, Pa; Δh_I is the loss in the impeller, m; P_{in} and P_{out} are separately the input and output power of the turbine, W; and Δh_C is the loss in the chamber, m.

As shown in Figure 10(a), the loss distributions under variable operating conditions of the volute were shown. The DOP in the figure means the design operating point. It can be seen that the loss in the volute increases gradually with the

FIGURE 9: Performance curve. (a) Head H versus flow rate. (b) Shaft power P versus flow rate. (c) Efficiency η versus flow rate.

increase in flow rate. The loss in the volute decreases with the increase in speed under the same flow rate condition. The change of loss in the volute decreases more obviously with the increase in speed when the operating flow rate condition is more than $45 \text{ m}^3/\text{h}$.

Figure 10(b) shows the loss distribution of draft tube under variable operating conditions at different speeds. It can be seen that the loss in draft tube is closely related to the efficiency of the turbine. The loss in draft tube is the smallest when the efficiency of the turbine reaches the maximum at the corresponding speed. The loss decreases and then increases with the increase in flow rate, and the internal loss of draft tube decreases with the increase in speed at the DOP. The internal loss of draft tube reaches the minimum at high rotating speed, and it is 36.2% lower than the design speed.

Figure 10(c) shows the loss distribution of a turbine impeller under variable operating conditions at different speeds. It can be seen that the internal loss of the impeller is significantly higher than that combined of other

components of the turbine. In addition, the internal loss of the impeller is lower when the efficiency of the impeller reaches the highest speed. The internal loss of the turbine impeller plays a decisive role on the performance of the turbine. The loss in the impeller is 4.7% lower than those at the design speed when the turbine is operating at high speeds. Therefore, the hydraulic performance of a turbine can be improved when it is properly operating at high rotating speeds.

As shown in Figure 11, the proportion of loss to each component of a turbine operating at design points at different speeds is shown. It can be seen that at the same speed, the proportion of loss of each component in the turbine from the highest to the lowest is impeller, volute, draft tube, and chamber. The loss of impeller is much higher than that of other parts, accounting for more than 67% of the whole turbine internal loss, and the internal loss of impeller and volute increases with the increase in rotating speed. But the loss ratio of draft tube and chamber under the $n = 22000 \text{ r/min}$ is much different from other

FIGURE 10: Loss distribution of turbine components. (a) Volute loss. (b) Draft tube loss. (c) Impeller loss.

conditions. Losses in the chamber are mainly due to disc friction and leakage losses. As the rotating speed increases, the angular velocity and the pressure difference between the inlet and outlet increase, so that the disc friction and the leakage losses in the chamber increase. The reasons for the obvious reduction in the losses in the draft tube at high speed will be analyzed in Section 5.5.

Combined with Figure 10, it can be seen that, with the increase in speed, the loss in the impeller gradually decreases, but the proportion of the loss in the impeller gradually increases. This shows that, with the increase in speed, the influence of impeller on turbine performance is more obvious.

5.3. Internal Flow Characteristics of Volute

5.3.1. Distribution of the TKE in Volute. Figure 12 shows the TKE distribution at the middle section of volute under different rotating speeds. It can be seen that the turbulent kinetic energy in the volute is mainly concentrated in the downstream of the volute tongue. The main flow from the volute inlet is separated at the tongue, and the flow from the volute and the separated flow converge at the downstream of the tongue, which result in a greater turbulence intensity. The turbulence intensity and kinetic energy decrease gradually with the increase in rotating speed. These show that the loss in the volute is mainly

FIGURE 11: Loss ratio distribution of turbine under design condition. (a) $n = 18000$ r/min. (b) $n = 20000$ r/min. (c) $n = 22000$ r/min.

FIGURE 12: Distribution of the TKE at the middle section of volute.

FIGURE 13: The centrifugal TKE distribution of volute outlet. (a) Schematic diagram in middle section of volute. (b) Circumferential TKE distribution.

concentrated in the downstream of the tongue, and the tongue design has a very important impact on the performance of the volute.

5.3.2. The Circumferential TKE Distribution of the Volute Outlet. In order to study the distribution characteristics of pressure and TKE at the outlet of volute, Figure 13(a)

shows the section diagram of volute. This figure shows the location of the volute outlet. Meanwhile, in order to study the distribution of the circumferential pressure on the outlet of the volute, the corresponding exit of Section VIII of the volute is defined as 0 degrees and increases gradually along the clockwise direction. The outlet position corresponding to the tongue of the volute is 338 degrees.

Figure 13(b) shows the circumferential TKE distribution at the volute outlet. It can be seen that at the same speed and different circumferential positions, the TKE at the outlet of volute fluctuates obviously in the circumferential directions of 80°, 138°, 265°, and 338° and the fluctuation amplitude also increases with the increase in circumferential angle. Moreover, the TKE at the outlet of the volute from 265° to 338° is significantly higher than that at other positions. Combined with the analysis of the velocity and TKE distribution in the volute outlet in Section 5.3.1, it can be seen that the high TKE of the volute outlet in this region is due to the increase in the flow velocity due to the reduction in the cross-sectional area of the volute on the one hand, and the obvious effect of the volute tongue on the fluid there. At different rotating speeds and circumferential positions, the peak values of TKE at the volute outlet at 80°, 138°, 265° and 338° decrease with the increase in rotating speed, while the TKE at other positions has little difference.

5.4. Internal Flow Characteristics of Impeller

5.4.1. Velocity Distribution in Impeller. As shown in Figure 14, the velocity distribution in the impeller passage at different speeds is shown, which shows the flow pattern in the impeller passage. It can be seen that there is a low-velocity region on the pressure surface and suction surface of the blade inlet, which is the main reason for the loss in the impeller.

There are obvious low-velocity regions on the pressure surface and suction surface of the impeller blade, and the interaction between the fluid and the blade is strong under the rotating speed of 18000 r/min. The low-velocity region on the pressure surface of the blade is caused by the impact effect between the blade and fluid, and the low-velocity region on the suction surface of the blade is caused by the separation of the fluid. However, with the increase in rotating speed, the velocity distributions on the pressure surface and suction surface of the blade change differently. The flow impact effect on the pressure surface of the blade is aggravated. A vortex is generated at the pressure surface of the blade marked with red dash. But the separation effect on the suction surface of the blade is weakened under the rotating speed of 20000 r/min. The low-velocity region at the suction surface of the blade marked with purple dash in the figure has been significantly improved. The flow impact effect on the blade pressure surface is further aggravated when the rotating speed increases to 22000 r/min. Two vortexes are generated at the blade pressure surface marked with red dash in the figure, but there is no obvious flow separation phenomenon on the suction surface of the blade. Combined with Figure 11, this shows that, with the increase in the rotating speed, the loss

caused by flow separation is obviously improved. The flow pattern in the impeller is different with the change of rotating speed, which is mainly caused by the incomplete matching between the relative flow angle and the blade setting angle [15].

5.4.2. TKE Distribution from Inlet to Outlet of Impeller.

Figure 15 shows the TKE distribution from the inlet to the outlet of the impeller. The coefficient L^* at the impeller inlet is defined as 0, and the coefficient L^* at the impeller outlet is defined as 1. It can be seen from the figure that the TKE in the impeller first increases gradually, and the maximum value of the TKE appears at $L^* = 0.2$, and then the TKE in the impeller decreases gradually and reaches the minimum value at $L^* = 0.8$ and then increases gradually from $L^* = 0.8$ to the impeller outlet. At different speeds, the distribution trend of TKE in the impeller is similar, but with the increase in turbine speed, the TKE at the same position in the impeller decreases. This shows that the appropriate increase in the rotating speed is helpful to improve the TKE in the impeller and reduce the turbulent intensity.

5.5. Internal Flow Characteristics of Draft Tube

5.5.1. Streamline and TKE Distribution in Draft Tube.

Figure 16 shows the streamline and TKE distribution in the draft tube at different speeds. It can be seen that the maximum TKE of draft tube is mainly densely observed at the middle of the draft tube inlet and gradually decreases along the radial and axial directions. A large vortex is formed in the middle of the draft tube inlet and gradually dissipates along the streamline direction at $n = 18000$ r/min. There is no vortex in the middle of the draft tube inlet, but the flow pattern is still poor at $n = 20000$ r/min. The flow pattern in the middle of the draft tube inlet is obviously improved, and the flow is uniform at $n = 22000$ r/min. This shows that the flow pattern in the draft tube can be improved and the turbulent intensity can be reduced by properly increasing the turbine speed.

5.5.2. Velocity and Radial Pressure Distribution near Draft Tube Inlet.

In order to study the velocity and radial pressure distribution near the draft tube inlet at different speeds, the impeller-draft tube axial surface schematic diagram is shown in Figure 17(a). DT_{out} in the figure represents the outlet of the draft tube, and $X = -0.02$ is the interface between impeller and volute. In order to study the velocity and radial pressure distribution near the draft tube inlet, the velocity of the YOZ plane where the line e is located in the figure is analyzed, and the pressure of the line e along the Y direction is analyzed. Figure 17(b) shows the TKE distribution of the YOZ plane where the straight line “ e ” is located. The turbulent kinetic energy at the draft tube inlet is mainly concentrated in the center of the tube and gradually decreases along the radial direction. With the increase in rotating speed, the area of turbulent kinetic energy in the center of the tube decreases gradually, and the turbulent kinetic energy in other positions also decreases correspondingly. This also shows that the energy loss in the

FIGURE 14: Streamline and velocity distribution in the impeller passage.

FIGURE 15: TKE distribution of the impeller blade.

FIGURE 16: Distribution of streamline and TKE of the draft tube.

FIGURE 17: Velocity and radial pressure distribution at draft tube inlet. (a) Axial projection diagram of impeller-draft tube. (b) Distribution of the TKE near draft tube inlet. (c) Radial pressure distribution at draft tube inlet.

draft tube is mainly concentrated in the center of the tube. With the increase in the rotating speed, the turbulence intensity in the draft tube gradually decreases and the energy loss decreases. Figure 17(c) shows the radial pressure distribution along the Y direction of the straight line “e” in the draft tube. It can be seen that at the same speed, the pressure near the draft tube inlet first decreases and then increases along the radial direction. The pressure value reaches the minimum value at the center line of the draft tube, and the pressure distribution is symmetrical with respect to the center line of the draft tube. At different speeds, the pressure distribution near the draft tube inlet tends to be similar, reaching the minimum at the center line of the draft tube. However, with the increase of speed, the radial fluctuation amplitude of the pressure near the draft tube inlet decreases and the pulsation weakens. It also shows that the flow at the draft tube inlet can be effectively improved and the steady flow of fluid in the draft tube can be ensured by properly increasing the turbine operating speed.

6. Conclusions

The energy loss and internal flow characteristics of an ERT under different conditions are systematically observed and analyzed. The conclusions based on detailed analyses can be drawn as follows:

- (1) The shaft power and head of the turbine increase with the increase in the flow rate, but the efficiency of the turbine first increases and then decreases with the increase in the flow rate. At different rotating speeds, the head and shaft power of the turbine increases with the increase in the rotating speed. The best efficiency point tends to the larger flow rate condition with the increase in the rotating speed.
- (2) From the highest to the lowest, the losses of the turbine happen in the impeller, volute, draft tube, and cavity, respectively. The loss of impeller accounts for more than 67% of the total loss and decreases with the increase in the rotating speed. This is

because with the increase in the speed, the impact effect of the blade pressure surface occupies the main reason for the loss

- (3) The loss in the volute is mainly concentrated in the downstream of the tongue, and the tongue design has a very important impact on the performance of the volute. The loss in the volute decreases with the increase in the rotating speed. The loss proportion of draft tube is the smallest, and the loss is mainly concentrated in the inlet center. With the increase in the rotating speed, the turbulent kinetic energy and radial pressure pulsation in draft tube decrease, so the loss decreases gradually.
- (4) The turbine can be operated at the design or slightly higher than the design rotating speed under the condition that both the hydraulic condition and the intensity are satisfied, which is conducive to the efficient utilization of energy.

Data Availability

The data used to support the findings of this study are included within the article.

Conflicts of Interest

The authors declare that they have no known conflicts of interest or personal relationships that could have appeared to influence the work reported in this study.

Authors' Contributions

Bing Qi prepared the original draft, performed experiment, and conceptualized the study and was responsible for software. Desheng Zhang conceptualized the study and was responsible for study supervision and methodology. Qi Zhang reviewed and edited the manuscript and investigated the data. Mengcheng Wang reviewed and edited the manuscript. Ibra Fall reviewed and edited the manuscript and provided guidance.

Acknowledgments

This work was supported by the National Natural Science Foundation of China (Grant no. 2017YFC0404201) and State Key Laboratory of Water Pump (Jiangsu University). The authors thank Ibra Fall for the revision of the language of the paper.

References

- [1] W. A. Raja and R. W. Piazza, *Reverse Running Centrifugal Pumps as Hydraulic Power Recovery Turbines for Seawater Reverse Osmosis Systems*, Elsevier, Amsterdam, Netherlands, 1981.
- [2] A. Bennett, "Advances in desalination energy recovery," *World Pumps*, vol. 2015, no. 7-8, 2015.
- [3] B. Qi, *Studies on Characteristics and Application of Energy Recovery Device for Seawater Reverse Osmosis Desalination System*, TianJin University, Tianjin, China, 2012.
- [4] X. Xu, "Application of hydraulic turbine in petrochemical industry," *General machinery*, vol. 3, pp. 23–27, 2013.
- [5] Z. Bi, J. Yang, and W. Zhao, "Status of research of hydraulic turbines for energy recovery," *Fluid Machinery*, vol. 42, no. 8, pp. 41–45, 2014.
- [6] C. Xu, P. Zhang, and S. Song, "Characteristics and application of energy recovery hydraulic turbine," *General machinery*, vol. 8, pp. 32–34, 2015.
- [7] S.-S. Yang, S. Derakhshan, and F.-Y. Kong, "Theoretical, numerical and experimental prediction of pump as turbine performance," *Renewable Energy*, vol. 48, pp. 507–513, 2012.
- [8] M. Liu, L. Tan, and S. Cao, "Theoretical model of energy performance prediction and BEP determination for centrifugal pump as turbine," *Energy*, vol. 172, pp. 712–732, 2019.
- [9] T. Lin, Z. Zhu, X. Li, J. Li, and Y. Lin, "Theoretical, experimental, and numerical methods to predict the best efficiency point of centrifugal pump as turbine," *Renewable Energy*, vol. 168, pp. 31–44, 2021.
- [10] P. Singh and F. Nestmann, "Internal hydraulic analysis of impeller rounding in centrifugal pumps as turbines," *Experimental Thermal and Fluid Science*, vol. 35, no. 1, 2010.
- [11] T. Wang, C. Wang, F. Kong, Q. Gou, and S. Yang, "Theoretical, experimental, and numerical study of special impeller used in turbine mode of centrifugal pump as turbine," *Energy*, vol. 130, 2017.
- [12] P. Wang, M. Vera-Morales, M. Vollmer, M. Zangeneh, B. S. Zhu, and Z. Ma, "Optimisation of a pump-as-turbine runner using a 3D inverse design methodology," *IOP Conference Series: Earth and Environmental Science*, vol. 240, 2019.
- [13] Z. H. Liu, M. Shinji, and K. Miyagawa, "Internal flow and loss mechanisms of specific speed 160 m-kW shroudless hydro turbine," *IOP Conference Series: Earth and Environmental Science*, vol. 163, 2018.
- [14] Z. H. Liu, T. Yukawa, K. Miyagawa, T. Fahimi, and M. Tahani, "Characteristics and internal flow of a low specific speed pump used as a turbine," *IOP Conference Series: Earth and Environmental Science*, vol. 240, 2019.
- [15] T. Lin, X. Li, Z. Zhu, R. Xie, and Y. Lin, "Investigation of flow separation characteristics in a pump as turbines impeller under the best efficiency point condition," *Journal of Fluids Engineering*, vol. 143, no. 6, 2021.
- [16] T. Lin, X. Li, Z. Zhu, J. Xie, Y. Li, and H. Yang, "Application of entropy dissipation to analyze energy loss in a centrifugal pump as turbine," *Renewable Energy*, vol. 163, pp. 41–55, 2021.
- [17] M. M. Ghorani, M. H. Sotoude Haghighi, A. Maleki, and A. Riasi, "A numerical study on mechanisms of energy dissipation in a pump as turbine (PAT) using entropy generation theory," *Renewable Energy*, vol. 162, pp. 1036–1053, 2020.
- [18] D. Li, Y. Sun, Z. Zuo, S. Liu, H. Wang, and Z. Li, "Analysis of pressure fluctuations in a prototype pump-turbine with different numbers of runner blades in turbine mode," *Energies*, vol. 11, no. 6, 2018.
- [19] S.-S. Yang, H.-L. Liu, F.-Y. Kong, B. Xia, and L.-W. Tan, "Effects of the radial gap between impeller tips and volute tongue influencing the performance and pressure pulsations of pump as turbine," *Journal of Fluids Engineering*, vol. 136, no. 5, 2014.
- [20] Y. Han and L. Tan, "Dynamic mode decomposition and reconstruction of tip leakage vortex in a mixed flow pump as turbine at pump mode," *Renewable Energy*, vol. 155, pp. 725–734, 2020.

- [21] Y. Han and L. Tan, "Influence of rotating speed on tip leakage vortex in a mixed flow pump as turbine at pump mode," *Renewable Energy*, vol. 162, pp. 144–150, 2020.
- [22] T. Yu, *Development and Application of Key Technologies and Equipments for Large-Scale Reverse Osmosis Desalination*, Zhejiang University, Hangzhou, China, 2019.
- [23] H. Nassrullah, S. F. Anis, R. Hashaikh, and N. Hilal, "Energy for desalination: a state-of-the-art review," *Desalination*, vol. 491, 2020.
- [24] Z. Liu, Y. Xiao, C. Wang et al., "Hydraulic performance analysis of a RO seawater desalination turbo energy recovery device," *Journal of hydropower*, vol. 11, pp. 12–29, 2020.
- [25] X. Su, S. Huang, X. Zhang, and S. Yang, "Numerical research on unsteady flow rate characteristics of pump as turbine," *Renewable Energy*, vol. 94, pp. 488–495, 2016.
- [26] Y. Hao and L. Tan, "Symmetrical and unsymmetrical tip clearances on cavitation performance and radial force of a mixed flow pump as turbine at pump mode," *Renewable Energy*, vol. 127, pp. 368–376, 2018.
- [27] J. Pei, M. K. Osman, W. Wang, J. Yuan, T. Yin, and D. Appiah, "Unsteady flow characteristics and cavitation prediction in the double-suction centrifugal pump using a novel approach," *Proceedings of the Institution of Mechanical Engineers–Part A: Journal of Power and Energy*, vol. 234, no. 3, pp. 283–299, 2020.