INTRODUCTION

Geothermal energy is a low-grade heat source with low-grade heat energy and low energy quality. It is difficult to promote water vapor Rankine cycle power generation, while the organic Rankine cycle (ORC) can select different working fluids according to different grades of the heat source. The low-boiling working medium can be used to promote the expansion of the expander at a lower temperature. The ORC is regarded as an effective power generation method for medium- and low-temperature heat sources.1-5

As shown in Figure 1, the ORC system generally consists of an evaporator, an expander, a condenser, a working fluid pump, and a generator. The research on the ORC system focuses mainly on the selection of working fluids, system parameter analysis, and system optimization. Several studies in the literature have investigated the selection of the appropriate working fluid for low-temperature heat sources. Galloni et al.6 performed experimental investigation of a mini-ORC with a hot source temperature between 75 and 90°C based on R245fa as the working fluid, which can reach a thermal efficiency of approximately 9%, a specific work equal to approximately 20 kJ/kg, and provide an output power somewhat
greater than 1 kW. Dong et al\textsuperscript{7} established an ORC system with the mean-line modeling of a radial turbine coupling model for evaluating the heat source outlet temperature and the performance of the radial turbo expander. In the simulation of different working fluids, when the outlet temperature is lower than 120°C, the use of R245fa output power is the highest. R245fa is a better working medium in a low-temperature organic Rankine circulation system because of its high thermodynamic performance and efficiency, lack of toxicity and non-flammability, and high safety.\textsuperscript{8-11} The influence of the temperature of the heat source in the evaporator of the ORC system on the pinch-point temperature difference has been analyzed and discussed in our previous work.\textsuperscript{12}

The turbo expander is the working component of the organic refrigerant Rankine cycle, and its efficiency and power output directly affect the performance of the system. Rahbar et al\textsuperscript{13,14} carried out a mean-line design coupled with an optimization technique and a real gas formulation approach for the development of an efficient small-scale radial turbine. The maximum radial turbine efficiency of 84% with a power output 15 kW was reported.\textsuperscript{14} Sauret and Gu\textsuperscript{15} performed 1D design and 3D simulation for a radial-inflow turbine working with R143a for different operating conditions including the off-design conditions. The maximum power output and entropic efficiency were 400 kW and 88.45%, respectively, with a blade speed ratio of 0.711, which was slightly above the optimum value of 0.7. Jubori et al\textsuperscript{16} used the multiobjective genetic algorithm (MOGA) optimization technique to achieve optimal turbine design and maximum cycle efficiency. The CFX simulation was carried out using six suitable low-temperature working fluids. The results showed that the maximum entropy efficiency and maximum output power of working medium R123 were 88% and 6.3kW. Fiaschi et al\textsuperscript{17} analyzed the nondesign analysis of the turbo expander and applied six working fluids to the 50-kW expander. The different characteristic curves of the performance were established. According to the selected working fluid, the entropy efficiency range is between 72% and 80%.

The radial turbo expander has the characteristics of high speed, high single-stage expansion ratio, and wide operating range and is often used as a thermal power conversion device for low-power organic Rankine cycle systems. While the impeller is the most important component in the centrifugal turbo expander, its flow characteristics determine the overall performance of the expander. The design of the aerodynamic structure of the impeller directly affects the thermal efficiency and enthalpy of the radial turbo expander and the stability and high efficiency. Due to the extremely complex internal flow of the impeller, the near-wall separation, shock loss, secondary flow, and other problems will occur in the internal working fluid flow of the impeller. The research on the internal flow mechanism of the radial turbo expander is of great significance.

At present, domestic and foreign scholars have performed extensive work on the centrifugal turbo expander through the simulation method using CFD software. The main research on the influence of the guide vane and impeller parameters on the flow field and the optimization of the aerodynamic structure has made good research progress.\textsuperscript{18-22} Sam et al\textsuperscript{23} describes the nozzles and turbine wheel analysis of a high-speed cryogenic turbo expander for helium refrigeration and liquefaction cycles using CFX. The results showed that the performance of the expander is reduced by the flow separation and trailing edge vortices at the nozzle inlet and outlet. A secondary flow occurs in the impeller and interacts with the scraping fluid to form a vortex. Through optimization, the nozzle profile, nozzle trailing edge shape and thickness, turbine blade profile, etc, would improve the turbo expander efficiency. Liu et al\textsuperscript{24} combined the design method of an axial-flow turbine with the characteristics of the centrifugal turbine for nozzle and rotor design of the transonic turbine. The results showed that the output efficiency and the output power were 85.34% and 14.83 kW. The Mach number and pressure distribution were reasonable, and the Mach number at the nozzle outlet reached 1.34.

In summary, through the computer numerical simulation method combined with the continuous analysis of the experimental research, researchers have gained a deep understanding of the internal flow of the centrifugal turbo expander guide vane and impeller, such as secondary flow, vortex, and rim clearance flow. On this basis, it is necessary to optimize the geometry of the guide vane and the impeller to improve the aerodynamic performance of the expander and obtain the most efficient expander.

In this paper, the turbo expander that has been designed is modeled and meshed by the Workbench module in the ANSYS software, and the internal flow path of the expander is numerically simulated by the CFX module. The distribution of the flow field, the vane static pressure, and the Mach number in the guide vane flow channel have been analyzed to provide the basis for subsequent research optimization.

FIGURE 1 Process diagram of a geothermal ORC system
2 | STRUCTURAL DESIGN AND MODELING

In this paper, for low-temperature geothermal energy such as 95°C, R245fa, which is suitable for the temperature of the heat source, has been selected as the organic working fluid for thermal calculation and flow field analysis.

The working condition of the organic working medium in the turbo expander can be illustrated by an enthalpy and entropy diagram, as shown in Figure 2. \( P_0 \) is the inlet pressure of the expander, \( P_1 \) is the outlet pressure of the guide vane of the expander, and \( P_2 \) is the outlet pressure of the impeller of the expander. To simplify the thermodynamic calculation of the turbo expander, the flow in the turbo expander is generally taken as an axially symmetric and adiabatic one-dimensional stable flow. As the figure shows, \( 0^*\sim 1\sim 2 \) represents the process of the isentropic expansion of the gas in the turbo expander. However, due to various energy losses in the flow, the entropy of the gas increases in the expansion process. Therefore, the actual expansion process of the organic working substance in the enthalpy and entropy diagram will be represented by line \( 0^*-0-1-2 \). After considering various losses, the enthalpy at the inlet of the turbo expander impeller is no longer \( h_1 \), but \( h_1 \), the enthalpy at the impeller exit is no longer \( h_2 \), but \( h_2 \), and the difference between the two is the flow loss.

In the turbo expander, the peri-round work performed per unit mass of organic working fluid can be expressed as follows:

\[
\Delta h_u = u_1 c_{1u} - u_2 c_{2u} = \frac{c_1^2 - c_2^2}{2} + \frac{w_1^2 - w_2^2}{2} + \frac{u_1^2 - u_2^2}{2} \tag{1} 
\]

\[
W = m \cdot \Delta h_u \tag{2} 
\]

where \( \Delta h_u \) is the cycle work of the turbo expander, J/kg; \( u_1 \) and \( u_2 \) are the circumferential velocity of the impeller inlet and outlet airflow, respectively, m/s; \( c_{1u} \) and \( c_{2u} \) are the circumferential component velocity of the airflow at the impeller inlet and outlet, respectively, m/s; \( w_1 \) and \( w_2 \) are the relative velocity of the impeller inlet and outlet airflow, respectively, m/s; \( W \) is the cycle power of the turbo expander, kW; and \( m \) is the mass flow rate, kg/s.

The ratio of the cycle work of the turbo expander \( \Delta h_u \) to the isentropic work \( \Delta h_u^* \) is the cycle efficiency of the turbo expander \( \eta_u^* \):

\[
\eta_u = \frac{\Delta h_u}{\Delta h_u^*} \tag{3} 
\]

Formula (1) of the cycle work is substituted into the above equation to obtain.

\[
\eta_u = \frac{(c_1^2 - c_2^2) + (u_1^2 - u_2^2) + (w_1^2 - w_2^2)}{C_a^2} \tag{4} 
\]

or

\[
\eta_u = \frac{2 (u_1 c_{1u} - u_2 c_{2u})}{C_a^2} \tag{5} 
\]

where \( c_a \) is the hypothetical expansion velocity of the turbo and \( \bar{c}, \bar{u}, \bar{w} \) represent the ratio of each velocity \( c_a \), respectively. The cycle efficiency expression will be simplified as.

\[
\eta_u = \bar{c}_1^2 - \bar{c}_2^2 + \bar{u}_1^2 - \bar{u}_2^2 + \bar{w}_1^2 - \bar{w}_2^2 \tag{6} 
\]

or

\[
\eta_u = 2 (\bar{c}_{1u}\bar{u}_1 - \bar{c}_{2u}\bar{u}_2) \tag{7} 
\]

Table 1 shows the initial design parameters of the turbo expander. To save calculation time and storage of design parameters, the refprop physical parameter module embedded in MATLAB was used to program the calculation process, which greatly shortened the computational cycle, improved the design calculation efficiency, and provided optimization for subsequent design parameters. Table 2 shows the main results of the calculation of the design parameters of the turbo expander, providing basic data for the three-dimensional design of the impeller.
In this paper, the design of the blade shape of the turbo expander was applied by the Bezier curve configuration method of the BladeGen submodule in the ANSYS Workbench. The impeller of the turbo expander was designed as a radial-axial flow type. The working fluid expanded from the guide vane and then entered the impeller flow path in the radial direction. After the expansion work, the working fluid flowed out from the axial direction. The flow of the working fluid was a very complicated three-dimensional viscous flow. According to the known impeller aerodynamic parameters and structural parameters, the impeller rim, hub, inlet, and outlet angles were changed by adjusting the end points of the Bezier curve after inputting the structural parameters. The final three-dimensional modeling of the turbo expander impeller is shown in Figure 3.

### 3 | SIMULATION BASIS

#### 3.1 | Controlling equation

In the CFX-Solver embedded in the CFX software, the basic governing equations of three-dimensional flow in a rotating machine are the continuity equation, the momentum equation, and the energy equation. The differential expression of the three governing equations is as follows:

- **Continuity equation:**
  \[
  \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \overline{U}) = 0 \tag{8}
  \]

- **Momentum equation:**
  \[
  \frac{\partial \left( \rho \overline{U} \right)}{\partial t} + \nabla \cdot \left( \rho \overline{U} \overline{U} \right) = -\nabla p + \nabla \cdot (\tau + \overline{S}_M) \tag{9}
  \]

- **Total energy equation:**
  \[
  \frac{\partial \left( \rho h_{tot} \right)}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot \left( \rho \overline{U} h_{tot} \right) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot \left( \overline{U} \cdot \tau \right) + \nabla \cdot \left( \overline{U} \cdot \overline{S}_M \right) + \overline{S}_E \tag{10}
  \]

The total energy equation is suitable for high-speed fluid flow, in which case the function key cannot be ignored. Simultaneously, the CFX software also provides an energy equation, which ignores the kinetic energy term. It may be called the internal energy equation:

\[
\frac{\partial (\rho e)}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho \overline{U} e) = \nabla \cdot (\lambda \nabla T) - p \nabla \cdot \overline{U} + \tau \cdot \nabla \overline{U} + \overline{S}_E + \overline{S}_F \tag{11}
\]

In this paper, the turbo expander is designed to have a high speed with supersonic speed locally for the internal working fluid flow. Therefore, total energy is selected as the calculation model.

#### 3.2 | Turbulence model

The Wilcox \(k-\omega\) model is to solve the entire turbulent region based on the specific dissipation rate and the turbulent kinetic energy equation. The processing model for the wall problem directly integrates the boundary layer instead of using the wall function method, ensuring the calculation accuracy of the near-wall turbulent flow.

SST \(k-\omega\) model\(^{27}\) is a mixture of the Wilcox \(k-\omega\) model and the standard \(k-\varepsilon\) model; Jubori et al\(^{16}\) used the
SST model for the CFD study of a turbine in an organic Rankine cycle. The same was adopted by Sam et al.\textsuperscript{24} for the improvement of flow structures within the nozzle of radial turbo expander. This gave SST model the advantage of the accurate prediction of the near-wall phenomena by the \( k-\omega \) model and of the free stream by the \( k-\varepsilon \) turbulence model.

The Wilcox \( k-\omega \) model has been revised by the eddy viscosity coefficient \( \mu_t \) and can be expressed as:

\[
v_t = \frac{ka_1}{\max(a_1, \omega, SF_2)} \cdot \mu_t = \rho v_t
\]

The transport equation of turbulent kinetic energy \( k \) and turbulent frequency \( \omega \) is:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k U_i) = \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i} \right] + \bar{G}_k - \rho \beta^* \Omega k \quad (13)
\]

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega U_i) = \frac{\partial}{\partial x_i} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] +
\]

\[
\frac{\gamma}{\nu_t} \bar{G}_k - \rho \beta^* \omega^2 + 2 \left(1 - F_1\right) \rho \sigma_{\omega^2} \frac{1}{\nu_t^2} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
\]

The value of these parameters in the model \((\gamma, \beta, \sigma_K, \sigma_\omega)\) can be expressed by the following formula:

\[
\phi = F_1 \phi_1 + \left(1 - F_1\right) \phi_2
\]

Among them, the mixing function \( F_1 \) is defined as:

\[
F_1 = \tanh (\arg_1^2)
\]

The mixed function \( F_2 \) is defined as:

\[
F_2 = \tanh \left( \arg_2^2 \right) \quad (19)
\]

\[
\arg_2 = \max \left( \frac{2 \sqrt{k}}{\beta^* \omega^2 \cdot \rho \gamma^2 \omega} \cdot \frac{500 \mu}{\rho \gamma^2 \omega} \right) \quad (20)
\]

where \( \gamma \) is the distance from the wall. The parameters of the first set \( \phi_1 \) in the model were set as \( \alpha_1 = 0.31, \beta^* = 0.09, \beta_1 = 0.075, \sigma_{k1} = 0.85, \sigma_{\omega1} = 0.85, \gamma_1 = \beta / \beta^* - \sigma_{\omega1} \cdot k^2 / (\beta^*)^{1/2}, \) and \( k = 0.41 \), corresponding to the Wilcox \( k-\omega \) modified model. The parameters of the second set \( \phi_2 \) in the model were set as \( \alpha_2 = 0.31, \beta^* = 0.09, \beta_2 = 0.0828, \sigma_{k2} = 1, \sigma_{\omega2} = 0.856, \gamma_2 = \beta / \beta^* - \sigma_{\omega2} \cdot k^2 / (\beta^*)^{1/2}, \) and \( k = 0.41 \), corresponding to the standard \( k-\varepsilon \) model.

### 3.3 Meshing and grid independence verification

The simulation results of the internal flow field of the turbo expander are related to various factors such as the control equation, the selection of the turbulence model, the accuracy of the guide vane, and the impeller modeling, so more mesh should be used within the acceptable time range to improve the accuracy of the simulation results, ultimately obtaining the optimized simulation results of the flow field. In the previous section, the guide vane and the impeller were modeled by the BladeGen module in the ANSYS Workbench. In this section, the TurboGrid module was used to mesh the vane and impeller and verify the grid independence.

The geometric model of the guide vane and the impeller generated by the BladeGen module was imported into the TurboGrid module for mesh division. The guide vanes and the moving blades both adopted H/J/C/L-Grid topologies. After the topology was selected, the computer performed topology partitioning for the vane and the moving vane path, including multiple blocks such as an inlet, an outlet, and a
The meshing process is repeated, and the mesh quality is detected by the Mesh Analysis function until the optimal mesh is divided. In this paper, the mesh near the blade is encrypted, and the dimensionless number $y^+$ of the first layer is adjusted to be less than 1, ensuring that the first layer of the mesh node is at the sticky bottom. The whole-circle topology of the guide vane and the impeller is shown in Figure 4. The grid-independent verification of the turbo expander model used in the simulation is described in Table 3. The table shows that the difference between the working fluid flow rate and the expander efficiency corresponding to the three meshing results is not significant.

Finally, the second meshing result was adopted. After meshing, the number of single-channel grids of the guide vanes was 247,000. Figure 5A and 5 shows the grid diagram of the single flow path and the whole-circle flow path of the nozzle generated by the TurboGrid module. After grid generation, the grid number of the single flow path of the guide vane is 247,000. Similarly, Figures 6A and 6 is a grid diagram of the impeller with a grid number of 536,000.

3.4 CFX boundary conditions and convergence criteria

In this paper, the boundary conditions and initial conditions of the 50-kW turbo expander vane and impeller were set by the preprocessing module in CFX, as shown in Table 4.

In this paper, the steady-state calculation convergence standard for the field simulation of the guide vane and the impeller of a 50-kW turbo expander was as follows:

1. The global residual convergence was <10^{-4};
2. The entire flow rate of the turbo expander did not change, or the variation in amplitude was <0.5%;
3. In the steady calculation result, the flow field parameters (torque, efficiency, etc) of the turbo expander did not change any more, and the resulting value was close to constant.

4 | RESULTS AND DISCUSSION

4.1 Performance analysis of the entire stage of the turbo expander

This section analyzes the main flow field characteristics within a turbo expander. Figure 7 shows the speed flow figure of the 30%, 50%, 70%, and 90% blade-to-blade (BTB) section of the turbo expander. The four figures show that the flow field of the turbo expander is relatively smooth, the flow field of the vane is not much different, and the entire flow path has no vortex. As the cross-section position increases, a large separation vortex appears on the suction side, mainly because when the working medium is turned from the radial direction to the axial direction, the flow path size becomes large, and the direction changes abruptly. The working medium is affected by the combined force of the hub, the rim, and the blade, and the flow direction is deflected. At the same time, the working fluid is also affected by the torsion angle of the impeller rim and the rim clearance. Therefore, the impeller needs to be optimized during the subsequent design process.

| TABLE 3 | Turbo expander vane and impeller grid independence verification |
|----------|---------------------------------------------------------------|
| Grid 1   | 12.8  | 31.7  | 2.974  | 85.6 |
| Grid 2   | 24.7  | 53.6  | 3.012  | 86.0 |
| Grid 3   | 54.1  | 94.5  | 3.004  | 85.9 |
4.2 Performance analysis of the guide vane of the turbo expander

Figure 8 shows the static pressure change contours of the blade to blade in the flow path of the guide vane. The three maps are the vane root (blade height of 10%), the middle (blade height of 50%), and the tip (blade height of 90%) sections. Comparing the three static pressure contours shows that the working fluid did not have a large eddy during expansion. Analyzing the static pressure cloud diagram of the middle section of the blade discloses that the pressure from the inlet to the outlet of the guide blade decreases uniformly and reaches the minimum value at the outlet. During the whole expansion process, the pressure in the throat of the

| Initial conditions         | Value | Unit | Boundary conditions         | Types of |
|----------------------------|-------|------|-----------------------------|----------|
| Total inlet pressure       | 0.89  | MPa  | Guide vane entrance         | Inlet    |
| Total inlet temperature    | 358.15| K    | Impeller outlet             | Outlet   |
| Static pressure at the exit| 0.211 | MPa  | Vane pressure surface       | Wall     |
| Reference pressure         | 0.1   | MPa  | Guide vane suction surface  | Wall     |
| Rotating speed             | 15 000| r/min| Impeller pressure surface   | Wall     |
| Working fluid flow         | 3     | kg/s | Impeller suction surface    | Wall     |
| Working type               | R245fa|      | Both sides of the calculation domain | Periodic |
| Intake direction           | Radial|      | Static and static interface | Stage    |
| Turbulence model           | SST model |      |                             |          |

TABLE 4 Boundary and initial conditions of guide vane and impeller for 50-kW turbo expander
guide blade passage changes most rapidly. There is apparently a low-pressure zone at the outlet close to the suction surface because the working medium accelerates expansion at the throat of the guide blade. The suction side of the same cross section was disturbed by the trailing edge of the blade to form a low-pressure area.

Figure 9 shows the Mach number contours of the guide vane root (blade height of 10%), the middle (blade height of 50%), and the tip (blade height of 90%) BTB sections. The three figures show that the aerodynamic acceleration of the working fluid in the entire flow channel is better, and no large vortex was generated. After the organic working fluid entered the flow path, the fluid accelerated and expanded rapidly; the speed increased sharply. The Mach number reached the maximum at the throat of the guide vane, at 1.034. From the vane to the exit section, the Mach number had a downward trend, and there was a low-speed zone at the trailing edge, due mainly to the large loss of the beveled portion of the outlet and the wake loss caused by the influence of the leaf thickness at the trailing edge.

Figure 10 shows the variation of the blade surface pressure with relative chord length at the middle (blade height of 50%) sections of the guide vane. When the relative chord length is 0.78, the minimum pressure appears in the middle section of the blade, which is at the throat of the flow path. Before the relative chord length is 0.78, the lateral pressure on the pressure surface remains basically unchanged, and the lateral pressure on the suction surface decreases exponentially, reaching the lowest point at the throat position. After the relative chord length is 0.78, the pressure on the pressure surface drops sharply. Because of the wake loss and shock wave, the pressure oscillates greatly at the trailing edge. On the suction side, the pressure increases first and then decreases, finally reaching the lowest point at the trailing edge.

Figure 11 shows the velocity vector figure and the enlarged view of the trailing edge of the blade flow path. The flow line of the guide vane is relatively smooth, there is no large eddy, and there is a relatively high speed area near the suction surface at the exit, which corresponds to the local low pressure area near the trailing edge of the suction surface.
mentioned above. The partially enlarged view of the trailing edge part shows that due to the influence of the thickness of the trailing edge on the working fluid flow, there are different flow directions at the trailing edge, resulting in local wake loss, which is consistent with the above analysis results and similar to the trailing edge vortex described in the literature where circled in Figure 11(B). This requires the guide vane trailing edge profile and thickness to be modified, so that the pressure difference is uniformly decreasing, which in turn results in a smoother expansion.

4.3 | Performance analysis of the impeller of the turbo expander

The organic working fluid turbo expander designed in this paper had a large expansion, so the enthalpy drop in the
flow path was large. The volume flow rate of the working fluid was high, and the height ratio of the impeller inlet and outlet blades was large. When the working fluid changes from radial flow to axial flow, the flow path area changes rapidly, and flow loss such as the vortex and the secondary flow is easily generated. At the same time, the influence of the absolute velocity of the impeller outlet and the deflection angle of the impeller profile on the flow loss of the working medium on the meridional surface of the impeller should be considered to improve the efficiency of the expander.

Figure 12 shows the static pressure contours of the blade to blade in the impeller flow path. The three images were blade root (blade height of 10%), the middle (blade height of 50%), and the tip (blade height of 90%), respectively. The comparison of the three figures shows that the static pressure distribution in the impeller flow path changed significantly along with the leaf height and the
Airflow changed from the radial flow to the axial flow region; that is, the relative chord length of the impeller is approximately 35%. In the inlet section of the 90% blade height section, a large low-pressure area is generated, and the flow loss was large; however, in the area of the 10% blade height section, the flow loss was not obvious, mainly because the deflection angle of the blade profile increased gradually with the increase in blade height, and the area of the impeller path increases sharply there, causing excessive expansion of the working medium and secondary flow.

At the blade wall boundary layer, secondary flow occurs from the hub to the shroud as the radial component of the Coriolis acceleration of the fluid within the boundary layer is greater than the radial pressure gradient. Similar to the blade wall boundary layer, due to the pressure gradient induced by the curvature effects, the secondary flow occurs from the pressure side to the suction side of the blade, corresponding to the position in the circle of Figure 12C. For this, the impeller profile should be modified by optimizing the blade inlet angle and tip clearance height. There was a high pressure zone at the leading edge of the impeller blade, which was caused by the direct impact of the airflow on the blade after the flow of the guide vane. The wake loss caused a local low-pressure zone at the trailing edge of the impeller. The low-pressure zone gradually weakens with the increase in the blade height. The streams of the pressure side and the suction side of the fluid flow get mixed, which results in the formation of trailing edge vortices, causing the trailing edge loss. The profile along with modifications in trailing edge thickness can reduce the trailing edge loss and thereby improve the performance of the turbo expander. The pressure drop in the impeller flow path was

![Pressure Contours](image)

**Figure 12** The BTB section static pressure contours of the impeller blade (A) Blade height of 10% (B) Blade height of 50% (C) Blade height of 90%
concentrated mainly in the inlet region, the pressure drop in the axial flow section of the impeller path was small, and the flow was relatively gentle; as a result, the expansion process of the working fluid occurred mainly in the front section of the impeller. On both sides of the impeller blade, the pressure on the suction side was always less than the pressure on the pressure side, which was beneficial to the expansion of the working fluid to promote the rotation of the expander.

Figure 13 shows the Mach number change contours of the blade to blade flow surface in the impeller flow path. The three figures show the blade root (blade height of 10%), the middle (blade height of 50%), and the tip (blade height of 90%) sections. According to the comparison of the three figures, the Mach number distribution in the cloud diagram increased with the increase in the blade height, and the Mach number distribution in the tip section was relatively complex. In the area where the airflow changes from radial flow to axial flow, the relative chord strength of the impeller was approximately 35%. There was a local large Mach number area, which was consistent with the low-pressure area on the suction surface in the above static pressure cloud diagram. At the section in the middle and at the root of the blade and at the relative chord strength of approximately 35% of the impeller, the high Mach number value area was obviously weakened, the Mach number distribution is gradually uniform, and the Mach number value on the pressure side was small. In general, the Mach number distribution in the whole impeller path was uniform, and the flow field was smooth.
Figure 14 shows that the surface pressure changes contours along with the profile of the impeller blade. The diagram indicates that the organic working medium expanded in the flow. Figure 15 shows the curve of the blade surface pressure change along with the relative chord length at the middle sections of impeller blade (blade height of 50%). It is clear from the diagram that the organic working medium expanded in the flow.
path of the impeller meridian, and the pressure decreased uniformly in the flow channel direction. The pressure on the pressure surface changed gently along the flow path, and there was no large negative pressure area. When the relative chord length was approximately 30% to the flange, there was a small area of low pressure under the influence of flange clearance leakage loss, which was consistent with the pressure curve. On the suction side of the impeller, there was a secondary flow from the hub to the rim in the axial flow passage. The pressure curve shows that, at the leading edge of the impeller blade, the pressure on the pressure surface and the suction surface varied sharply under the influence of airflow impact, and the pressure on the suction surface was higher than the pressure on the pressure surface. Rahbar et al.29 found that the flow separation near the leading edge affects the flow acceleration downstream. For improved expansion, the difference between the pressure side and the suction side pressure should be more and must be uniformly decreasing, corresponding to the position in the circle of Figure 14A. The pressure curves of the blade heights of the 50% section are relatively consistent with the pressure cloud diagram. In general, there was a large adverse pressure gradient at approximately 30% of the relative chord length, and the pressure fluctuation was large, due mainly to the excessive expansion of the working medium at the throat of the flow path. This phenomenon corresponded to the negative pressure area at the throat on the suction surface in the cloud diagram. After the suction surface had a relative chord length of 40%, the pressure decreased uniformly, and the pressure difference between the suction surface and the pressure surface increased with the blade height.

2. According to the analysis results of the flow field distribution, the blade static pressure, and the Mach number distribution in the guide vane path, the flow field of the guide vane was relatively smooth without large eddy and secondary flow loss, but there was a small range of wake loss at the trailing edge that can affect the flow characteristics within the turbine passage and thereby degrade the performance of the turbo expander. Modifications of the guide vane trailing edge profile and thickness are required to minimize this loss.

3. Based on the analysis of the flow field of the impeller blades, the flow in the designed impeller runner was relatively smooth; however, at the same time, because of the complexity of the impeller path and blade modeling, there were many losses in the preliminary design of the impeller. At the leading edge and trailing edge of the impeller blade, there was impact loss and wake loss. Parameters such as impeller profile, tip clearance height, trailing edge height, and thickness need to be optimized. At the 30% relative chord strength of the impeller, there was excessive expansion, resulting in local low-pressure area and a secondary flow from the hub to the rim. The impeller profile should be modified by optimizing the blade inlet angle, tip clearance height, and other parameters to suppress the secondary flow in the channel.

5 | CONCLUSIONS

In this paper, the controlling equation and the turbulence model were used for the numerical simulation analysis of the turbo expander. According to the main thermal performance parameters of the turbo expander, the aerodynamic parameters of the guide vane and the impeller, structural parameters, and three-dimensional modeling design, the Workbench module in ANSYS software was used to conduct modeling, grid division, and grid independence verification for the designed turbo expander, and the CFX module was used for numerical simulation of the internal flow path of the expander.

Based on the results of the simulations, the following conclusions were reached.

1. The flow field analysis of the turbo expander was carried out. The efficiency of the expander reached 86%, and the power was 58 kW, which was consistent with the design results.

ACKNOWLEDGMENTS

This work was supported by the Ministry of Science and Technology of the People's Republic of China (No. 2017YFB0603300).

ORCID

Wenguang Jia https://orcid.org/0000-0002-5414-9973

REFERENCES

1. Aldrich MJ, Laughlin AW, Gambill DT. Geothermal resource base of the world: a revision of the Electric Power Research Institute's estimate. New Mexico, LA: Los Alamos Scientific Laboratory; 1981.
2. Subbiah S, Natarajan R. Thermodynamic analysis of binary-fluid Rankine cycles for geothermal power plants. Energy Convers Manage. 1988;28(1):47-52.
3. Yamamoto T, Furuhara T, Arai N, Mori K. Design and testing of the organic Rankine cycle. Energy. 2011;26(10):239-251.
4. Badr O, O’Callaghan PW, Probert SD. Rankine-cycle systems for harnessing power from low-grade energy sources. Appl Energy. 1990;36(4):263-292.
5. Zhang J, Zhang W, Li Y, et al. Controller Design for a Heat Exchanger in Waste Heat Utilizing Systems. Advances in Swarm Intelligence. Heidelberg: Springer; 2011:379-386.
6. Galloni E, Fontana G, Staccone S. Design and experimental analysis of a mini ORC (organic Rankine cycle) power plant based on R245fa working fluid. *Energy*. 2015;90:768-775.

7. Dong B, Xu G, Li T, Luo X, Quan Y. Parametric analysis of organic rankine cycle based on a radial turbine for low-grade waste heat recovery [J]. *Appl Therm Eng*. 2017;26:470-479.

8. Rayegan R, Tao YX. A procedure to select working fluids for Solar Organic Rankine Cycles (ORCs). *Renewable Energy*. 2011;36(2):659-670.

9. Lisheng P, Huaixin W. Experimental investigation on performance of an organic Rankine cycle system integrated with a radial flow turbine. *Energies*. 2019;12(4):724.

10. Kajurek J, Rusowicz A, Grzebielec A, et al. Selection of refrigerants for a modified organic rankine cycle. *Energy*. 2019;168:1-8.

11. Chintala V, Kumar S, Pandey JK. A technical review on waste heat recovery from compression ignition engines using organic Rankine cycle [J]. *Renew Sustain Energy Rev*. 2018;81:493-509.

12. Liu B, Liang B, Zhang Y. Heat-source and evaporation temperatures for the organic rankine cycle. *Energy Technol*. 2015;3(7):709-716.

13. Rahbar K, Mahmoud S, Al-Dadah RK, Moazami N. Modelling and optimization of organic Rankine cycle based on a small-scale radial inflow turbine. *Energy Convers Manage*. 2015;91:186-198.

14. Rahbar K, Mahmoud S, Al-Dadah RK, Moazami N. Parametric analysis and optimization of a small-scale radial turbine for Organic Rankine Cycle. *Energy*. 2015;83:696-711.

15. Sauret E, Gu Y. Three-dimensional off-design numerical analysis of an organic Rankine cycle radial-inflow turbine. *Appl Energy*. 2014;135:202-211.

16. Jubori AA, Al-Dadah RK, Mahmoud S, Bahr Ennil A, Rahbar K. Three dimensional optimization of small-scale axial turbine for low temperature heat source driven organic Rankine cycle. *Energy Convers Manage*. 2016;133:411-426.

17. Fiaschi D, Manfrida G, Maraschiello F. Design and performance prediction of radial ORC turbo expanders. *Appl Energy*. 2015;138:517-532.

18. Sauret E, Gu YT. 3D CFD Simulations of a Candidate R143a Radial-Inflow Turbine for Geothermal Power Applications. ASME Power 2014, Power 2014–32158, 28–31 July 2014, Baltimore, USA; 2014: V002T09A013-V002T09A013.

19. Natkaniec CK, Kammeyer J, Seume JR. Secondary Flow Structures and Losses in a Radial Turbine Nozzle. ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition; 2011: 977-987.

20. Putra M, Joos F. Investigation of secondary flow behavior in a radial turbine nozzle. ASME Turbo Expo 2006: Power for Land, Sea, and Air, Barcelona, Spain; 2006.

21. Simpson A, Spence S, Watterson J. Numerical and experimental study of the performance effects of varying vanesless and vane solidity in radial inflow turbine rotors. ASME Turbo Expo 2008: Power for Land, Sea, and Air, Berlin, Germany; 2008.

22. Zhai L, Xu G, Wen J, et al. An improved modeling for low-grade organic Rankine cycle coupled with optimization design of radial-inflow turbine. *Energy Convers Manage*. 2017;153:60-70.

23. Liu Y, Dg H. Design and performance analysis of an ORC transonic centrifugal turbine. *J Mech Sci Technol*. 2019;33(3):1417-1430.

24. Sam AA, Ghosh P. Flow field analysis of high-speed helium turboexpander for cryogenic refrigeration and liquefaction cycles. *Cryogenics*. 2017;82:1-14.

25. Cho SY, Cho CH. An experimental study on the organic Rankine cycle to determine as to how efficiently utilize fluctuating thermal energy. *Renewable Energy*. 2015;80(4):73-79.

26. ANSYS Inc. *ANSYS CFX Release 12.1 User’s Guide*. Canonsburg, PA: ANSYS Inc; 2009.

27. Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*. 1994;32(8):1598-1605.

28. Zangeneh M, Goto A, Harada H. On the design criteria for suppression of secondary flows in centrifugal and mixed flow impellers. *J Turbomach*. 1998;120(4):723.

29. Rahbar K, Mahmoud S, Al-Dadah RK, et al. Mean-line modeling and CFD analysis of a miniature radial turbine for distributed power generation systems. *Int J Low-Carbon Tech*. 2016;11(2):157-168.

**How to cite this article:** Liu B, Wang S, Zhang M, Liu N, Wang C, Jia W. Simulation study on the flow field of guide vane and impeller of turbo expander. *Energy Sci Eng*. 2019;7:2306–2320. [https://doi.org/10.1002/ese3.477](https://doi.org/10.1002/ese3.477)