Investigation on Interaction between Geometry and Performance and Design of S-CO$_2$ Radial Inflow Turbine’s Vaneless Inlet Volute

Chang Zhu$^1$, Wen-Yang Shao$^1$, Zi-Yue Ma$^1$, Guo-Chuan Lv$^2$, Xiao-Fang Wang$^*$

$^1$ School of Energy and Power Engineering, Dalian University of Technology, Dalian 116024, China
$^2$ NEIAS Commercial Aircraft Company Limited, AVIC, Nanjing 211100, China
$^*$ Corresponding Author

Email: dlwxf@dlut.edu.cn

Abstract. The inlet volute is one of the key components that guide the working fluid into the radial inflow turbine. It has a significant effect on the performance of the radial inflow turbine. The conventional design criteria for the vaneless inlet volute mainly focuses on the ideal gas, which does not apply to real gas cases, such as supercritical carbon dioxide(S-CO$_2$). In this paper, an MW-class S-CO$_2$ radial inflow turbine is studied. Firstly, the inlet volute design of the radial inflow turbine with a vaned stator is completed, and the flow in the whole machine is calculated using CFD simulations. Then, based on the inlet boundary conditions of the impellers, the vaneless inlet volute with different throat areas and cross-sectional area distributions are investigated. The results indicate that the size of the counter rotating vortexes in the vaneless inlet volute increases with the throat area increasing, and is also influenced apparently by the cross-sectional area distribution. The volute throat area has little effect on the circumferential distribution of the outlet airflow angle but changes the average value of the airflow angle. The cross-sectional area distribution of the volute influences the circumferential distribution of the outlet flow angle. The result indicates that volute with a small-scale convex profile has a uniform outlet flow field. Compared to the turbine with the vaned stator, the radial dimension of the vaneless volute turbine obtained in the final design is significantly reduced, and the total-total efficiency at the design point is 85.62%, which meets the design requirements. Moreover, it also has good performance at off-design operating conditions.

Nomenclature

- $F$ Area
- $G$ Mass flow rate
- $L$ Volute outlet width
- $P_t$ Total pressure
- $R$ Radius
- $V_{ar}$ Absolute radial velocity
- $V_t$ Absolute tangential velocity
- $V_{sy}$ Absolute velocity
- $\alpha$ Airflow angle
- $c$ Velocity
- $r$ Radial direction
- $\theta$ Tangential direction
- $\phi$ Circumferential azimuth angle
- $\eta_{t-t}$ Total-total efficiency
- $\rho$ Density
- $\phi$ Velocity loss coefficient
- $l$ Volute outlet
1. Introduction
In recent years, the Brayton cycle with S-CO$_2$ as the working fluid has become a research hot spot due to its high cycle heat efficiency, compactness and environmental protection[1-2]. When the cycle power level is lower than 10 MW, radial flow turbines are often used instead of the axial flow turbine due to the blade height limitation[3]. Compared to the turbine with the vaned stator, the radial dimension of the vaneless volute turbine is smaller, which is more conducive to the structural arrangement of the machine. Therefore, the vaneless volute turbine capable of reducing the radial dimension needs to be studied under the premise of ensuring efficiency. The vaneless inlet volute serves to connect the turbine directly to the outer pipeline, and whether the flow field uniformity at the volute outlet directly affects the performance of the impeller significantly.

Many researchers have investigated the vaneless volute. Yansheng Li et al. [4] considered the radial velocity variation of the flow in the vaneless volute, when integrating the mass flow over each volute cross-section, and proposed a simplified two-dimensional analysis method. Hara K et al. [5] conducted an experimental study on a vaneless volute and measured the flow field in the volute and the boundary layer of the sidewall. They found that the circumferential non-uniformity of the volute outlet flow field is greatly affected by the internal boundary layer of the volute. Spence S.W.T. et al. [6] conducted experimental tests on a series of radial inlet volutes with and without guide vanes. The vaneless volute turbine showed better performance and its measurement efficiency value had increased by 4.5% under the same conditions as the others. The above researches on the vaneless inlet volute mainly focus on incompressible fluid. However, there are few studies on real gas cases, such as S-CO$_2$ with high density and high specific heat capacity. For S-CO$_2$, the design method of the volute does not provide a clear range in the selection of the velocity loss coefficient. And the investigation of the interaction between geometry and performance of the S-CO$_2$ volute is also lacking. Moreover, the computational domain simulated in most researches on the S-CO$_2$ radial inflow turbine only consists of the vane and the impeller, and the simulation of the inlet volute is always neglected.

In this paper, an MW-class S-CO$_2$ radial inflow turbine is studied. Firstly, an inlet volute of the radial inflow turbine with a vaned stator is designed, and the performance of the whole machine is calculated using CFD simulations. Then, based on the inlet boundary conditions of the impeller, various vaneless inlet volute models are obtained by changing the throat area and cross-sectional area distribution. Then, the interaction between geometry and performance of the vaneless volute is investigated in detail. Finally, the vaneless inlet volute matching the target impeller is obtained.

2. Volute Design Method
The one-dimensional design method developed by Yansheng Li et al.[4] is adopted. The radial inflow turbine volute is designed based on the blade height and the estimated inlet airflow angle of the vaned stator, and the cross-section of the volute is taken as a circular cross-section. The main design parameters of the radial inflow turbine required for the design method are shown in Table.1.

Firstly, the flow is assumed to be steady and adiabatic. Approximately, the angular momentum of the flow in the inlet volute varies according to the constant circulation flow:

\[ c_{\omega \theta} R_\theta \theta = c_{\omega i} R_1 = \text{const} \]  \( (1) \)

The relationship between the mass flow rate \( G_\theta \) and the azimuthal angle \( \theta \) is as follows:

\[ G_\theta = G(\theta_{\text{max}} - \theta) / \theta_{\text{max}} \]  \( (2) \)

Four additional relationships can be obtained from the continuity equation:

\[ G = c_{\omega i} \rho_i F_1 \]  \( (3) \)

\[ F_1 = 2 \pi R_1 L \]  \( (4) \)

\[ G_\theta = c_{\omega \theta} \rho \theta F_\theta \]  \( (5) \)

\[ \tan \alpha_\theta = c_{\omega \theta} / c_{r \theta} \]  \( (6) \)
By solving equations (1-6) simultaneously, the distribution of the cross-sectional area $F_\theta$ with the azimuth angle $\theta$ can be obtained:

$$F_\theta / R_\theta = 2\pi L \varphi \rho_1 (\theta_{\text{max}} - \theta) / \rho_\varphi \theta_{\text{max}} \tan \alpha_1$$  \hspace{1cm} (7)

The flow velocity is low in the scroll of the vaned stator’s volute, and the density of S-CO$_2$ is very high. It can be considered that the density of S-CO$_2$ in the volute is constant, which is also in line with the calculation results of numerical simulations. Therefore, equation (7) can be simplified as follows:

$$F_\theta / R_\theta = 2\pi L \varphi (\theta_{\text{max}} - \theta) / \theta_{\text{max}} \tan \alpha_1$$  \hspace{1cm} (8)

It can be found from equation (8) that the ratio of the cross-sectional area $F_\theta$ to the core radius $R_\theta$ has a positive correlation with the azimuthal angle $\theta$. Besides, the velocity loss coefficient $\varphi$ at the volute inlet recommended by Baines [7] ranges from 0.85 to 0.95.

The preliminary design of the vaneless inlet volute is based on the two-dimensional design method [4], but it lacks the selection range of the velocity loss coefficient for the real gas. Therefore, the derivation of the method is not specifically described here. The geometric parameters of both the volute with vaned stator and the preliminary design of the vaneless inlet volute are shown in Table 2.

Table 1 Main parameters of an MW-class S-CO$_2$ radial inflow turbine.

| Item                  | Unit | Parameter |
|-----------------------|------|-----------|
| Inlet total temperature | K    | 773.15    |
| Inlet total pressure  | MPa  | 20        |
| Outlet total pressure | MPa  | 8.20      |
| Mass flow rate        | kg/s | 16        |
| Rotational speed      | rpm  | 50000     |
| Impeller inlet blade height | mm  | 5.20      |
| Impeller inlet radius | mm   | 69        |
| Total-total efficiency | %   | >85       |

Table 2 Geometric parameters of the volute with vaned stator and the preliminary design of the vaneless inlet volute.

| Item                  | Unit | Volute with vaned stator | Vaneless inlet volute |
|-----------------------|------|--------------------------|-----------------------|
| Volute outlet radius  | mm   | 87                       | 69                    |
| Volute outlet width   | mm   | 5.20                     | 5.20                  |
| F/R (Throat)          | /    | 23.44                    | 7.04                  |
| Throat centroid radius| mm   | 122.13                   | 88.91                 |
| Throat area           | mm$^2$| 2862.80                  | 625.80                |

3. Numerical Simulation

In this paper, the numerical simulation of the S-CO$_2$ radial inflow turbine with a vaned stator is carried out by the commercial CFD software NUMECA. Then, based on the inlet boundary conditions of the vaned turbine impeller, a series of vaneless inlet volute models with different volute throat areas and cross-sectional area distributions are designed and analyzed. Then, the interaction between geometry and performance of the vaneless inlet volute is investigated. The thermodynamic properties of S-CO$_2$ are interpolated from the physical property tables generated by the TABGEN during the numerical simulations. The SA model is used to simulate the turbulent flow. The computational grids of vanes and impellers are generated by AutoGrid5, and the grid of volute is generated by IGG. The resulting grids satisfy the turbulence model’s requirement (SA model requires $\gamma^+\leq10$) and the grid independence study is performed to obtain the proper mesh number. For the numerical simulation of the whole
machine, the total temperature and the total pressure are imposed as the inlet boundary conditions, and the outlet condition is set with static pressure; for the numerical simulation of the individual volute model, the inlet conditions are specified by the total temperature and the total pressure, and the outlet condition is specified by the mass flow rate.

On basis of the geometric data of the SNL(Sandia National Laboratory) radial inflow turbine given in the literature [8], different turbulence models were adopted to simulate the turbine in the previous works of the authors, and the numerical simulations were compared with the SNL’s experimental results. The results showed that the SA mode is accurate and reliable for the simulation of the S-CO₂ radial inflow turbine.

4. Results and Discussions

4.1. Analysis of the volute with the vaned stator

The results of three-dimensional simulations with or without the inlet volute are shown in Table.3. According to the authors’ design experience, the total pressure of the vane inlet is adjusted to 19.85 MPa by considering the total pressure in the inlet volute. It can be found that the estimated total pressure of the vane inlet is slightly lower than the calculated CFD value. The loss caused by the inlet volute only reduces the total-total efficiency by 0.94%. In summary, the inlet volute designed by the one-dimensional method has little effect on the performance of the S-CO₂ radial inflow turbine under the design condition.

| Item                  | Unit | Vane+Impeller | Volute+Vane+Impeller | Change rate |
|-----------------------|------|---------------|----------------------|-------------|
| Vane inlet total pressure | MPa  | 19.85         | 19.94                | 0.45%       |
| Mass flow rate        | kg/s | 16.06         | 16.10                | 0.25%       |
| Impeller outlet total pressure | MPa  | 8.20          | 8.20                 | 0           |
| Total-total efficiency | %    | 89.05         | 88.21                | -0.94%      |

Figure.1 shows the velocity field and the airflow angle distribution at the vaned stator mid-span section. It can be seen from the figure that the flow inside the vaned stator is relatively uniform, and the airflow angle at the volute outlet is well-matched with the vane inlet blade angle. The inlet volute with guide vane guides the flow well towards the impeller and no obvious flow separation can be seen. In summary, the inlet volute designed by the one-dimensional method matches well with the guide vane and the impeller under the design condition.

Figure.2 shows the performance curve of the radial inflow turbine total-total efficiency and mass flow rate as functions of the total-total expansion ratio. It can be seen that the total-total efficiency of the radial inflow turbine is higher than 0.8 under the working conditions of the study (The total-total expansion ratio ranges from 1.8 to 3). In the vicinity of the design point, the turbine efficiency is not significantly affected by the operating conditions. With the total-total expansion ratio increasing, the mass flow rate of the turbine increases gradually and finally keeps nearly constant. In summary, the radial inflow turbine composed of the designed inlet volute, guide vane and impeller has good performance under off-design operating conditions.

Figure.3 and Figure.4 show the velocity field and the airflow angle distribution of the vaned stator mid-span section under two off-design points with the total-total expansion ratio of 1.81 (small flow condition) and 2.97 (large flow condition). It can be seen that the flows inside the vaned stator under two off-design points are almost as uniform as under the design condition. The change of the velocity in the inlet volute is not obvious. This is because the internal velocity of the inlet volute is much
smaller compared with that in the vaned stator, which result in less obvious change of the velocity under off-design conditions. The same reason applies to the airflow angle at the outlet of the volute. The airflow angle distributions at the volute outlet are basically unchanged, matching well with the guide vane inlet blade angle.

Figure.1 The velocity field and the airflow angle distribution at the vaned stator mid-span section

Figure.2 The performance curve of the radial inflow turbine total-total efficiency and mass flow rate as functions of the total-total expansion ratio

Figure.3 The velocity field and the airflow angle distribution at the vaned stator mid-span section with total-total expansion ratio of 1.81

Figure.4 The velocity field and the airflow angle distribution at the vaned stator mid-span section with total-total expansion ratio of 2.97

4.2. Analysis of the vaneless inlet volute
The design of the vaneless inlet volute in the paper referenced to the two-dimensional design method proposed in the literature[4]. However, under the inlet and outlet conditions of the design point, the mass flow rate of the vaneless radial inflow turbine increases to 18.22 kg/s, which is 13.88% higher than the design mass flow rate. It means that the preliminary design of the vaneless inlet volute does not match the target impeller because of the lack of the determination of the real gas velocity loss coefficient. It has been determined that the section of the inlet volute is circular. On the basis of the preliminary design result, several models of vaneless inlet volute are obtained by changing the throat area and the cross-sectional area distribution. It should be noted that only the vaneless volute is
simulated during the parameter exploration in this section in order to reduce the computation time. For the numerical simulation of the individual volute model, the inlet conditions are specified by the total temperature and the total pressure, and the outlet condition is specified by the mass flow rate.

**Figure.5** Cross-sectional area distributions curve of the vaneless inlet volute

**Figure.6** Total pressure loss coefficient of vaneless inlet volute as a function of the throat area

**Figure.7** Outlet velocity of vaneless inlet volute as a function of the throat area

4.2.1. *Influence of the vaneless inlet volute throat area.* As shown in Figure.5, four new volute models are obtained by adjusting the throat area to 500-800 mm², with the same cross-sectional area distribution. Figure.6 shows the influence of the volute throat area on the total pressure loss coefficient. It can be found that the total pressure loss coefficient decreases with the volute throat area increasing. Figure.7 shows the variations of the volute outlet velocity as a function of the throat area. The axial velocity is very small and can be ignored. It can be found that the volute outlet velocity decreases with the throat area increasing. Because of the same cross-section area distribution applied, the change of the volute outlet radial velocity is not obvious. And the change of the volute outlet velocity caused by the throat area mainly results from the outlet tangential velocity. Therefore, the average airflow angle of the volute outlet also shows a trend of decrease with increasing the throat area.

Figure.8 shows the velocity field at the volute mid-span section with different throat areas. Figure.9 shows the static pressure contours at the volute mid-span section with different throat areas. It can be seen from the figures that the flow in the two different volutes is good, and there is no obvious flow separation phenomenon.

Figure.10 shows the streamlines of the secondary flow at several circumferential positions (45° to 225°) in the vaneless inlet volute with different throat areas. It can be found that the secondary flow in the volute is obvious. The results indicate that there are two pairs of counter rotating vortexes, one of which moves along the radial direction to the outlet of the volute, and the other pair of vortexes gradually reduces. The size of the counter rotating vortexes decreases in the circumferential direction and finally they almost disappear at the circumferential section of 225°. At the 45° circumferential sec-
Figure.8 Velocity field at the volute mid-span section with different throat areas

Figure.9 Static pressure contours at the volute mid-span section with different throat areas

In the 135° circumferential section, the volute with a throat area of 500 mm² still has two pairs of counter rotating vortexes, but there is only one pair of counter rotating vortexes in the other two volute models. At the 135° circumferential section, the volute with a throat area of 500 mm² still has two pairs of counter rotating vortexes, but in contrast, only one pair of counter rotating vortexes is left at the same position for the other three volute models. This indicates that the influence range of the two pairs of counter rotating vortexes is larger in the vaneless inlet volute with smaller throat area. In addition, at any circumferential section, the size of the vortexes increases with the volute throat area increasing. In summary, with the volute throat area increasing, the size of the counter rotating vortexes increases but the influence range of the two pairs of counter rotating vortexes, on the contrary, decrease.

Figure.10 Streamlines of the secondary flow at different azimuthal positions in the vaneless inlet volute with different throat areas

Figure.11 shows the axial average airflow angle distributions at the outlet of the vaneless inlet volutes with different throat areas in the circumferential direction. It can be seen that volute throat area has a significant effect on the outlet flow angle, but almost does not affect the distribution trend of the flow angle in the circumference direction. With the circumferential position ranging from 30° to 140°, there is a fluctuation in the flow angle, and the fluctuation amplitude increases with the volute throat area increasing. In the range of the circumferential angle from 320° to 30°, there is a sharp change in the flow angle. This phenomenon is due to the mixing of high-speed flow and low-speed flow
downstream of the volute tongue. It can be found that the vaneless volute with a throat area of 600 mm² in Figure.11 has a relatively good flow angle distribution near the volute tongue. This shows that the uniformity of the flow angle near the volute tongue improves first and then deteriorates with increasing the volute throat area.

**Table.4** The numerical simulations of the volutes with different cross-sectional area distributions

| Item                        | Unit     | Line1  | Line2  | Line3  | Line4  |
|-----------------------------|----------|--------|--------|--------|--------|
| Total pressure loss coefficient |          | 0.198  | 0.213  | 0.228  | 0.235  |
| Outlet average radial velocity | m/s     | 75.89  | 76.05  | 76.50  | 79.08  |
| Outlet average tangential velocity | m/s     | 285.22 | 299.38 | 307.89 | 312.74 |
| Outlet average velocity     | m/s     | 295.95 | 308.92 | 317.53 | 323.00 |
| Outlet average airflow angle | °       | 75.10  | 75.75  | 76.05  | 75.81  |

**Figure.13** Velocity field at the volute mid-span section with different cross-sectional area distributions

**Figure.14** Static pressure contours at the volute mid-span section with different cross-sectional area distributions

4.2.2. **Influence of the cross-sectional area distribution of the vaneless inlet volute.** The optimum volute throat area is readjusted according to the parameter exploration results presented in the above section. In this section, the volute throat area is kept constant, and the effect of the cross-sectional area distribution is studied by comparing the performance of four volutes with different cross-sectional area distributions. Table.4 shows the numerical simulations of four vaneless inlet volutes. It can be found that the volute total pressure loss coefficient increases as the shape of the cross-sectional area...
distribution changes from concave to convex. The outlet average velocity also shows the same results. However, for the average flow angle, a maximum value could be expected as the shape of the cross-sectional area distribution changes from concave to convex.

Figure 13 and 14 show the velocity field and the static pressure contours at the mid-span section of the volutes with different cross-sectional area distributions. The results indicate that the flow in the two different volutes is good, and there is no obvious flow separation.

Figure 15 shows the streamlines of the secondary flow at the different azimuthal positions in the vaneless inlet volute with different cross-sectional area distributions. It can be found that the secondary flow inside the vaneless volute is basically the same as the results presented in the previous section. Apart from the circumferential position of 225°, the size of the counter rotating vortexes increases as the shape of the cross-sectional area distribution changes from concave to convex. At the 135° circumferential section, the vaneless inlet volute models corresponding to line3 and line4 still have two pairs of counter rotating vortexes, but only one pair of counter rotating vortexes is observed at the corresponding section of the other two volute models. This indicates that the influence range of the two pairs of counter rotating vortexes is larger in the vaneless inlet volute with a more convex profile. In summary, the size of the counter rotating vortexes increases as the shape of the cross-sectional area distribution changes from concave to convex, and it is the same case for the influence range of the two pairs of counter rotating vortexes.

Figure 15 Streamlines of the secondary flow at different azimuthal positions in the vaneless inlet volute with different cross-sectional area distributions

Figure 16 Effect of the cross-sectional area distribution on the axial average flow angle at the outlet of the volute

Figure 16 shows the effect of the cross-sectional area distribution on the axial average flow angle at the outlet of the volute. It can be seen that the uniformity of the outlet axial average flow angle distribution in the circumferential direction improves first and then deteriorates with the shape of the cross-sectional area distribution changes from concave to convex. For the spiral part, with the cross-sectional area distribution changes from concave to convex, the outlet axial average flow angle of the first half increases gradually, while the second half decreases gradually. For the flow near the volute tongue, the uniformity of outlet axial average flow angle distribution improves first and then deteriorates. According to the results in Table 4, the axial average flow angle at the vaneless volute outlet corresponding to line3 has the most uniform distribution in the circumferential direction, and the numerical result of the flow angle best matches the impeller in the four volute models. In summary,
when the volute inlet area is constant, the vaneless inlet volute with a small-scale convex profile has a relatively uniform outlet flow angle distribution.

4.3. Analysis of the final vaneless inlet volute

The final design of the vaneless inlet volute is obtained by adjusting the inlet area and the cross-sectional area distribution. The performance comparison of the radial inflow turbine with guide vanes and the one with vaneless volute is shown in Table.5. It can be found that the total-total efficiency of the vaneless volute turbine is lower than the turbine with the vaned stator, but it still meets the design requirements (> 85%). The inlet total pressure of the vaneless volute turbine is 2.34% less than that of the turbine with the vaned stator, which shows the former’s stator loss is higher than that of the latter. The mass flow rate of the vaneless volute turbine decreases with the lower total pressure of the impeller inlet. However, the radial dimension of the vaneless volute turbine is significantly smaller

| Item                        | Unit       | Vaned stator+Impeller | Volute+Impeller | Change rate |
|-----------------------------|------------|-----------------------|-----------------|-------------|
| Impeller inlet total pressure | MPa       | 19.19                 | 18.74           | -2.34%      |
| Mass flow rate              | kg/s       | 16.10                 | 15.95           | -0.93%      |
| Outlet total pressure       | MPa        | 8.20                  | 8.20            | 0           |
| Total-total efficiency      | %          | 88.21                 | 85.62           | -2.93%      |
| Maximum radius              | mm         | 152.33                | 101.22          | -33.55%     |

**Figure.17** Velocity field and the static pressure distribution on the impeller meridian surface

**Figure.18** Velocity field and the static pressure distribution at the impeller mid-span section

**Figure.19** Performance curves of the vaneless volute turbine total-total efficiency and mass flow rate with total-total expansion ratio
than the turbine with the vaned stator, and the maximum radius is reduced by 33.55%. In summary, the final design of the vaneless volute meets the requirements at the design point.

Figure 17 and 18 show the velocity field and the static pressure contours on the impeller meridian surface and the mid-span section. It can be seen that no obvious flow separation occurs near the impeller leading edge, which indicates that S-CO₂ flowing through the vaneless volute flows well.

Figure 19 shows the performance curves of the vaneless volute turbine, total-total efficiency and mass flow rate with total-total expansion ratio. It can be seen that the total-total efficiency of the vaneless volute turbine is higher than 0.8 under the working conditions of the study. In the vicinity of the design point, the turbine efficiency is not significantly affected by the change of the operating conditions. With the increase of the total-total expansion ratio, the mass flow rate of the turbine increases gradually, but its increasing ratio decreases obviously. In summary, the vaneless volute turbine has good performance under off-design operating conditions.

5. Conclusions
In this paper, based on the two-dimensional design method, several vaneless inlet volute models are obtained by changing the volute throat area and the cross-sectional area distribution. Three dimensional CFD simulations are performed to investigate the interaction between the geometry and the performance of the vaneless inlet volute. Finally, the vaneless inlet volute matching with the target impeller is obtained. The conclusions are as follows:

1) Under the same cross-sectional area distribution, the total pressure loss coefficient of the volute gradually decreases with the volute throat area increasing. With the volute throat area increasing, the size of the counter rotating vortexes increases but the influence range of the two pairs of counter rotating vortexes decreases.

2) The volute throat area has little effect on the circumferential distribution trend of the outlet flow angle, but obviously affects the value of the flow angle. The uniformity of the flow angle near the volute tongue improves first and then deteriorates with increasing the volute throat area.

3) With the same volute throat area, the total pressure loss coefficient of the volute gradually increases with the cross-sectional area distribution changes from concave to convex. The size of the counter rotating vortexes increases as the shape of the cross-sectional area distribution changes from concave to convex, and it is the same case for the influence range of two pairs of counter rotating vortexes.

4) The uniformity of the outlet flow angle distribution in the circumferential direction improves first and then deteriorates with the shape of the cross-sectional area distribution changes from concave to convex, and the vaneless inlet volute with small-scale convex profile has a relatively uniform outlet flow angle distribution.

5) The radial dimension of the vaneless volute turbine obtained in the final design is significantly reduced, and the total-total efficiency of the design point is 85.62%, which meets the design requirements and also has good performance under off-design operating conditions.

Acknowledgments
The research is supported by National Key Research and Development Project(2016YFB0600104) and Youth Program of National Natural Science Foundation of China(51606026).

References
[1] Crespi F, Gavagnin G, Sánchez D, et al. Supercritical carbon dioxide cycles for power generation: A review[J]. Applied Energy, 2017, 195:152-183.
[2] Garg P, Kumar P, Srinivasan K. Supercritical carbon dioxide Brayton cycle for concentrated solar power[J]. Journal of Supercritical Fluids, 2013, 76(4):54-60.
[3] Sienecki JJ, Moisseeytsev A, Fuller RL, et al. Scale Dependencies of Supercritical Carbon Dioxide Brayton Cycle Technologies and the Optimal Size for a Next-Step Supercritical CO₂ Cycle Demonstration[C]//SCO2 Power Cycle Symp 2011.
[4] Yansheng Li. Simplified two-dimensional analysis method for radial inflow turbine’s vaneless inlet volute[J]. Journal of University of Shanghai for Science and Technology, 1982,(4): 13-33.

[5] Hara K, Furukawa M, Inoue M. Behavior of three-dimensional boundary layers in a radial inflow turbine volute[J]. Journal of turbomachinery, 1994,116(3): 446-452.

[6] Spence SWT, Rosborough RSE, Artt DW, et al. A direct performance comparison of vaned and vaneless stators for radial turbines[J]. Journal of turbomachinery-transactions of the ASME, 2007, 129(1): 53–61.

[7] Moustapha H, Zelesky M F, Baines N C, et al. Axial and radial turbines[M]. White River Junction, VT: Concepts NREC, 2003.

[8] Vilim. Development and validation of a radial inflow turbine model for simulation of the SNL S-CO₂ split-flow loop.[J]. Office of Scientific & Technical Information Technical Reports, 2012.