Numerical Simulation of Internal Flow Field in Turbo-Expander

Chaojie Li1,a, Yanqin Mao1, Xiaoyue Wang1, Zhixing Zhan1, Liang Cai1,b*

1College of Energy and Environment, Southeast University, Nanjing, Jiangsu, 210096, China

aemail: 220190526@seu.edu.cn

b*Corresponding author’s e-mail: Cailiang@seu.edu.cn

Abstract. As everyone pays more attention to energy consumption, it is very meaningful to use natural gas pressure energy for power generation and turbo-expander is an important part of power generation devices. In this paper, the turbo-expander model for pressure energy generation is meshed and numerically simulated based on fluent, and the pressure distribution and velocity distribution in the turbo-expander are obtained. The volute profile is Archimedes spiral, and the impeller is modeled by cfturbo. The main conclusions are as follows: when the number of grids is more than 2.2 million, the simulation results are less affected by the number of grids. The internal basin of the turbo-expander has obvious pressure gradient and velocity gradient. Due to the negative pressure at the elbow of the inlet pipe of the centrifugal effect, the existence of the blade leads to the change of the flow direction. Different watershed planes have different pressure and velocity distributions. The velocity and pressure of the watershed plane near the impeller outlet and the volute outlet are often smaller, but the flow vortex is more intense.

1. Introduction

As everyone pays more attention to energy consumption, more and more scholars pay more attention to the recovery of natural gas pressure energy. It is very meaningful to use natural gas pressure energy for power generation and turbo-expander is an important part of power generation devices. Turbo-expander is a machine that converts pressure energy into other forms of energy when high pressure gas expands, which is widely used in the acquisition of cooling capacity and pressure energy generation. As important components of turbo-expander, the design and flow field analysis of volute and impeller are very important to improve the performance of turbo-expander. Anna et al[1] analyzed the huge energy loss in the process of gas pipeline transportation. The energy loss in the process of decompression can be first converted into mechanical energy, and then converted into electrical energy through the generator. An oil-free twin-screw expander was designed for high-pressure gas pressure energy recovery device, and the rotor, bearing and other components were discussed. The feasibility and convenience of screw expander in the recovery and utilization of natural gas pressure energy were proved by experiments. Li et al[2] proposed a new system for the recovery of natural gas pressure energy, and combined the by-product energy and low-grade heat of the pressure regulating station. The integrated system consists of two subsystems. The natural gas expansion subsystem can recover the pressure energy of natural gas. The Rankine cycle subsystem can recover the cold energy and low-grade heat of natural gas, so as to maximize the recovery of natural gas pressure energy. Deymi et al[3] used turbine expansion system to replace the surge station to use high pressure energy for the production of electricity and freshwater and
achieved good results. Golchoobian et al[4] studied the feasibility of using turbo-expander to replace expansion valve in natural gas pressure regulating station, so as to effectively utilize the energy lost by gas pipeline pressure regulation and finally achieve good results. Li Qiusheng[5] studied the influence of the relative position of diversion shell and impeller on the performance of submersible pump. Guo Xiang[6] takes IS80-65-160 centrifugal pump as the research object, analyzes the change of internal flow and external characteristics of centrifugal pump, and puts forward the partition method of steady performance curve.

In natural gas pressure energy power generation projects, turbo-expanders are often used as core components. The turbo-expander model studied in this paper is mainly used for pressure energy generation. The analysis of internal flow field is helpful to understand the shortcomings of volute and impeller structure, which is convenient for future improvement and saves experimental cost.

2. Geometric model and numerical model

In this paper, the turbo-expander is mainly used for pressure energy power generation. The high-pressure gas is inflowed from the volute inlet to promote the impeller rotation. The impeller crankshaft drives the magnetic coupling and generator to complete the power generation. The modeling of the volute used is mainly based on the constant velocity normal line proposed by Stepanoff[7]. Assuming that the fluid inside the volute is uniform and the velocity is constant, the Archimedes spiral can be obtained. The profile equation adopted is shown below.

\[ R_\varphi = R_2 e^{2\pi B c_2 u R_2 \varphi} = R_2 \left[ 1 + m \varphi + \frac{1}{2!} (m \varphi)^2 + \frac{1}{3!} (m \varphi)^3 + \ldots \right] \]  \hspace{1cm} (1)

When the rotor speed is small, the power function expanded by the above equation is only the first two, and the general expression of Archimedes spiral can be obtained. Then, the following equation can be obtained by appropriate deformation. The values of m, A and B affect the clearance between volute and impeller. Fig.1 is the Archimedes spiral line corresponding to different m values, and the inner circle is the outer diameter contour of the impeller. From inside to outside, the corresponding m values are 0.05, 0.1, 0.15 and 0.2 respectively.

\[ R_\varphi = A R_2 (1 + m \varphi) + B \]  \hspace{1cm} (2)

Fig. 1 Archimedes spiral

Based on the Archimedes spiral equation, the volute is modeled by UG, as shown in Fig.2, where A = 1.02, R2 = 110mm, m = 0.06, B = 0. Fig.3 is the 220mm impeller generated by cfturbo, and the blade height is 10mm.
Numerical simulation should satisfy continuity equation, mass conservation, momentum conservation and energy conservation. For incompressible fluid continuity equation is as follows:

$$\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z} = 0 \quad (3)$$

The mass conservation equation is as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = S_m \quad (4)$$

The momentum conservation equation is as follows:

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \quad (5)$$

The energy conservation equation is as follows:

$$\frac{\partial}{\partial t} (\rho T) + div (\rho \vec{u} T) = div \left( \frac{k}{c_p} \nabla T \right) + S_T \quad (6)$$

Turbulence control equation uses RNG k-ε two-equation model, where k equation is as follows:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k - \rho \varepsilon \quad (7)$$

ε equation is as follows:

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon}^* G_k \frac{\varepsilon}{k} - C_{2\varepsilon} \frac{k}{\varepsilon^2} \quad (8)$$

In the formula, k is turbulent kinetic energy, $G_k$ is turbulent kinetic energy generation rate, $\varepsilon$ is turbulent dissipation rate, $\mu_{eff}$ is equivalent viscosity coefficient, and $\mu_{eff} = \mu + \mu_\tau$, $C_{1\varepsilon} = C_{1\varepsilon} - \frac{\eta (1 - \frac{\rho^2}{\rho_\tau^2})}{1 + \beta q^2}$, $\alpha_k$, $\alpha_\varepsilon$, $C_{1\varepsilon}$, $C_{2\varepsilon}$ are all empirical values.

3. Grid Independence Verification

This paper mainly simulates the internal flow field of turbo-expander and the changes of flow field in different basin planes. Some parameters in the profile equation of the volute model are as follows: $A = 1.02$, $r = 110$ mm, $m = 0.06$, $B = 0$. The outer diameter of the impeller is 220mm and the blade height is 10mm.

In order to minimize the influence of grid number on simulation results, grid independence verification is needed to reduce the error factor of grid number on simulation results. Ensure inlet flow 0.5kg/s, outlet pressure 0.4MPa and rotor speed 600rpm unchanged. The ratio of inlet pressure to outlet pressure is monitored when the number of turbine model grids is 52w, 108w, 160w, 226w and 280w respectively. The turbulence model used in the simulation is RNG k-ε model, flow inlet, pressure outlet, and non-slip boundary conditions. As shown in Fig.4, when the grid mass is above 220 w, the turbine expansion ratio is less affected by the number of grids. Therefore, in order to ensure the accuracy of the simulation calculation and the less use of computational resources, the following simulation analysis shows that the number of grids is about 2.2 million.
Fig. 4 Relationship between number of grids and expansion ratio

Fig. 5 is the impeller mesh and fig. 6 is the volute mesh. The number of grids is about 2.2 million, and the quality is above 0.3. There is an interface between rotating domain and stationary domain.

Fig.5 Impeller region grid  
Fig. 6 Volute region grid

4. Methods and Numerical Simulation

4.1 Methods and Material

In the simulation process, the steady-state mrf model is used. In order to facilitate the later comparison with the experiment and consider the safety, air is selected instead of natural gas for simulation. The turbulence model used in the numerical simulation is the RNG $k-\varepsilon$ model, the mass flow inlet boundary, the pressure outlet boundary, and the expandable wall function.

In this paper, the turbo-expander is designed and modeled based on Archimedes equation, which is used for pressure energy generation. When the high pressure gas flows from the inlet, the volute shrinks and drives the impeller to rotate, and the crankshaft drives the magnetic coupling and generator to generate electricity successfully. Because the torque of the device is too large, the rotor speed is only about 600rpm during the experiment. It is necessary to reasonably analyze the change of the internal flow field of the turbo-expander to improve the experimental efficiency.

4.2 Internal flow field analysis of turbo-expander

Fig. 7 and Fig. 8 are the pressure distribution and velocity vector distribution when inlet flow is 0.5kg/s and outlet pressure is 0.4Mpa. Turbo-expander impeller speed is 600rpm, inlet pressure is 0.508Mpa, expansion ratio is 1.27. When the airflow passes through the inlet, due to the sudden change of airflow velocity direction, the elbow has a strong centrifugal force, and the pressure drops to the negative pressure. At the same time, it is accompanied by a large velocity distribution, and then tends to be stable. When the airflow drives the impeller to rotate in a steady state, there will be vortexes in the basin between the blades, and the vortex intensity near the inlet section is stronger and the pressure is greater. With the reduction of cross section, the pressure tends to decrease gradually. Fig. 9 is the internal
streamline diagram of the turbo-expander. The inlet pressure of the impeller is large, and then flows out through the outlet of the impeller. The pressure near the outlet of the impeller is relatively flat.

![Pressure nephogram of central axis of impeller inlet center](image1)

**Fig. 7** Pressure nephogram of central axis of impeller inlet center

![Velocity vector cloud map of central axis of impeller inlet center](image2)

**Fig. 8** Velocity vector cloud map of central axis of impeller inlet center
4.3 Analysis of plane flow field in different basins of turbo-expander

The outer diameter of the impeller used in this paper is 220 mm, the blade height is 10 mm, the z coordinate of the blade height center is 0.015 m, and the inlet diameter of the turbo-expander is 25 mm, and the height is 150 mm. Fig. 10 shows the distribution of basin pressure corresponding to $z = 0.03\text{m}$, 0.04 m, 0.05 m, 0.06 m, 0.07 m and 0.08 m respectively. When the $z$ coordinate of the plane increases, the maximum pressure value in the plane basin decreases gradually, which is because the airflow pushes the impeller to rotate and collides with the impeller to dissipate the pressure energy. The airflow outflow from the impeller is a process of energy attenuation for the air, which converts into the mechanical energy of the impeller. When the $z$ coordinate increases, the pressure gradient in the plane basin also gradually decreases and tends to the outlet pressure. However, when $z$ is 0.07 m, the maximum pressure corresponding to the plane is higher, which is related to the backflow effect of the outlet section, but the overall pressure distribution is at a low level.

![Flow lines in basin of turbo-expander](image)

![Pressure distribution](image)
Fig. 10 shows the velocity distribution of the river basin when $z = 0.03m, 0.04m, 0.05m, 0.06m, 0.07m$ and $0.08m$ respectively. With the increase of $z$ coordinate, the maximum velocity value of the corresponding river basin plane decreases gradually, accompanied by flow vortex under the action of centrifugal force. The velocity distribution is larger in the region near the impeller outer diameter. When the $z$ value is large, when the basin plane is close to the outlet, it will be affected by the gas backflow and the impeller outlet, forming a huge vortex centered on the impeller crankshaft, and a local small vortex is distributed around it.
Fig. 11 The plane velocity distribution of turbo-expander in different basins
5. Conclusion
Based on the results and discussions presented above, the conclusions are obtained as below:

1. The internal basin of the turbo-expander has obvious pressure gradient and velocity gradient. Due to the negative pressure at the elbow of the inlet pipe of the centrifugal effect, the existence of the blade leads to the change of the flow direction. There is a flow vortex between the blades and the vortex near the inlet is larger.

2. After the fluid flows into the volute, it flows into the impeller inlet, and then flows out through the impeller. Different watershed planes have different pressure and velocity distributions. The velocity and pressure of the watershed plane near the impeller outlet and the volute outlet are often smaller, but the flow vortex is more intense.

3. Research on the performance of turbo-expander is helpful to improve the efficiency of natural gas pressure energy generation and realize energy recovery and utilization. If time and conditions permit, it is necessary to study the working conditions under various cross-sections in more detail. It is worthy of further study to study the effect of the gap between the volute and the impeller or the effect of different flow conditions on the performance of the turbo-expander and the efficiency of power generation.

Reference
[1] Anna D, Yuli W, Yi G, et al. Development and application of screw expander in natural gas pressure energy recovery at city gas station[J]. Applied Thermal Engineering, 2018, 142:665-673.
[2] Li C. Optimal design and thermo-economic analysis of an integrated power generation system in natural gas pressure reduction stations[J]. Energy Conversion and Management, 2019, 200.
[3] Deymi M, Dadpour D, Khadem J. Using the potential of energy losses in gas pressure reduction stations for producing power and fresh water[J]. Desalination, 2021, 497:114763.
[4] Golchoobian H, Taheri M H, Saedodin S. Thermodynamic analysis of turboexpander and gas turbine hybrid system for gas pressure reduction station of a power plant[J]. Case Studies in Thermal Engineering, 2019, 14:100488.
[5] Li Q, Hao S, Zhang R, Cai Y. Influence of relative position of diversion shell and impeller on performance of submersible pump [ J ]. Coal mine machinery, 2021,42 ( 06 ) : 72-74.
[6] Guo X, Shao C, Zhu R, Dai Y, Huang X, Gu B. Numerical simulation of steady flow in centrifugal pump [ J ].Coal mine machinery, 2010,31 ( 01 ) : 50-52.
[7] Stepanoff, 1957, Centrifugal and Axial Flow Pumps:Theory, Design and Application[M]. Wiley, New York.