Numerical Simulation of Turbine Expander in Natural Gas Pressure Energy Power Generation System

Zhan Zhixing ¹,a, Mao Yanqin ¹, Wang Xiaoyue ¹, Li Chaojie ¹, Cai Liang ¹,b*

¹College of Energy and Environment, Southeast University, Nanjing, China
aemail: 220190461@seu.edu.cn
b*Corresponding author: bemail: Cailiang@seu.edu.cn

ABSTRACT: In this paper, the numerical simulation method is used to study the aerodynamic characteristics of the runoff turbine expander, the key equipment in the natural gas pressure energy recovery power generation system. The geometric structure of the turbine expander under the condition of sub-high pressure natural gas expansion was designed and the flow characteristics of the internal flow field during its operation were obtained by numerical simulation. It is found that the expansion process of fluid in the runoff turbine expander mainly occurs in the guide vane channel and the inlet section of the rotor channel. The turbine expander designed in this paper can well meet the working requirements under design conditions, and the flow in the whole channel is good. When the flow channel in the moving impeller changes from radial to axial, the flow field is complex and there is a poor flow. The research results of this paper provide a reference for the design and optimization of runoff turbine expander in natural gas pressure energy power generation system.

1. Introduction
Natural gas as a clean energy has been paid attention to by all countries in the world. According to the data of the International Energy Agency, the global demand for natural gas is expected to increase by 50 % by 2040[1]. In order to meet the increasing demand for natural gas, natural gas pipeline network construction tends to long distance, high pressure and network development. Natural gas in the pipeline network contains a large amount of pressure energy. Taking the West-East Gas Pipeline I as an example, the annual gas supply capacity has exceeded 16 billion m³. If the pressure drops from 8 MPa to 0.4 MPa, the energy recovery is about 2 430 GW·h[2]. At present, the pressure energy recovery and utilization methods of natural gas pipeline network mainly include residual pressure power generation and refrigeration applications, and the residual pressure power generation mainly includes direct expansion power generation and combined cycle power generation. Refrigeration applications mainly include cold storage, ice making, LNG peak shaving, rubber crushing, etc.[3]

At present, scholars in China and abroad have carried out relevant studies on the utilization of natural gas pressure energy. In October 2008, Enbridge, North America, combined expanders with fuel cells to form a DFC-ERG system, and conducted field tests at the Toronto gate station in Canada[4]. In 2009, a British company proposed to install micro turbine expander in natural gas pipeline network for power generation[5]. The research on the utilization of natural gas pressure energy in China started relatively late. Li Xuelai et al. introduced a device for dehydration of natural gas using thermal separators, which can be used as power by the pressure energy of natural gas pipeline network[6]. The research and promotion center of gas wave refrigeration in Dalian University of Technology has...
developed several sets of gas wave refrigeration natural gas dehydration and purification devices.\textsuperscript{[7]} Zhang An’an et al. put forward an integrated scheme of pressure energy generation and cold energy utilization in natural gas high-pressure pipe network to solve the problems of single utilization mode of natural gas pressure energy and low energy efficiency caused by difficult utilization of cold energy in pressure regulating station.\textsuperscript{[8]}

At present, the most widely used pressure energy power generation has been fully developed in foreign countries, while the pressure energy power generation in China is still in its infancy. The research on the utilization of natural gas pressure energy mainly focuses on the system and principle, while the research on the expansion process of pressure energy is less. In this paper, the turbine expander, which is the core component of the pressure energy recovery system, is selected as the research object. Through the combination of theoretical calculation and numerical simulation, an expander suitable for sub-high pressure natural gas pressure regulation is designed. The flow field characteristics of the expander in the process of sub-high pressure gas expansion are analyzed. The influencing factors of the flow field on the performance of the expander are studied, and the direction for the subsequent optimization design is provided.

2. Research object

The typical natural gas pressure energy power generation system is shown in Figure 1. The system consists of heaters, turbine expanders, generators, reheaters and other equipment. In this paper, the key equipment of the system, turbine expander, is selected for structural design.

![Fig.1 Diagram of natural gas pressure energy power generation system](image)

Condition requirements are shown in table 1:

| Table1 Operating parameters of turbine expander |
|-----------------------------------------------|
| inlet total pressure P₀ | P₀/Mpa | 0.80 |
| inlet temperature T₀ | K | 340 |
| outlet back pressure Pᵣ | Mpa | 0.40 |
| mass flow rate | kg/s | 0.80 |

CFturbo software is a professional design software for rotating machinery. It contains a large number of experimental data and empirical equations, which can quickly and accurately complete the design and modeling of the expander. In this paper, CFTurbo software is used to complete the structural design and 3D modeling of the turbine expander through repeated optimization of design parameters.

| Table2 Design parameters of impeller structure |
|-----------------------------------------------|
| impeller diameter | mm | 155 |
| output power | kW | 40.65 |
| blade number | | 12 |
| rated speed | rad/min | 26000 |
| isentropic efficiency | | 83% |
3. Numerical simulation method

3.1 controlling equation

The SST k-ω turbulence model is used to simulate the internal turbulent motion of the expander. The flow follows the mass conservation law, momentum conservation law and energy conservation law. The mathematical description of each controlling equation is as follows[9]:

1) Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0$$  \hspace{1cm} (1)

2) momentum equation:

$$\frac{\partial (\rho \mathbf{U})}{\partial t} + \frac{\partial (\rho \mathbf{U} \mathbf{U})}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \sigma_k \right) \frac{\partial k}{\partial x_j} \right] + \bar{p} - \rho \omega k$$

$$\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \sigma_\omega \right) \frac{\partial \omega}{\partial x_j} \right] + c_1 \rho \frac{\alpha}{\mu_t} - c_4 \rho \omega^2 + 2(1 - F_1) \rho \sigma_\omega \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

$$\mu_t = \rho \frac{\alpha_k}{\max(\alpha_k, \alpha F_2)}$$  \hspace{1cm} (3)

In the formula, $k$ - turbulent kinetic energy; $u_j$ - velocity component, $m/s$; $x_j$ - coordinate vector, $m$; $\mu$ - fluid viscosity, $Pa \cdot s$; $\mu_t$ - coefficient of eddy viscosity; $\omega$ - Special turbulent dissipation rate; $\sigma_k, \sigma_\omega$ - Prandt number corresponding to turbulent kinetic energy $k$ and Special turbulent dissipation rate $\omega$; $\bar{p}$ - pressure term; $F_1, F_2$ - blending function; $S$ - The constant term of shear force tensor; $c_1, c_2, a_1$ - empirical constant;

3) Energy conservation equation

$$\frac{\partial (\rho h^*)}{\partial t} + \nabla (\rho \mathbf{U} h^*) = \nabla (\lambda \nabla T) + \nabla (\tilde{T} \cdot \mathbf{S}_m^*) + \tilde{S}_E$$

In the formula, $\mathbf{S}_m^*$ - momentum source term; $S_E$ - Energy source term; $h^*$ - Total specific enthalpy, $J/kg$; $\lambda$ - thermal conductivity, $W/(m \cdot K)$; $\tilde{T}$ - stress tensor;

3.2 Mesh division and boundary conditions

3.2.1 mesh division. In this paper, TurboGrid software and ICEM CFD software are used to mesh rotor blade, guide blade and volute respectively, as shown in Fig.3 In the division, the areas with complex flow conditions such as rotor runner and volute tongue are encrypted. Overall grid quality is above 0.35 and the number of grid cells is 3256642.
3.2.2 Boundary condition setting. In this paper, the numerical simulation analysis of the whole expander is carried out, and the boundary conditions are set as follows: The fluid region is divided into rotor region and stationary region. The stationary region includes volute flow channel, stator flow channel and impeller outlet region. The rotor region adopts transient calculation. The working speed $n$ is 26000 rad/min and the working medium is ideal air. The inlet boundary is given total temperature and total pressure. Outlet boundary, given outlet static pressure. All solid walls are non-slip and adiabatic boundary conditions.

4. Analysis of calculation results

4.1 Flow field performance analysis of turbine expander
Firstly, the flow field of the whole expander is analyzed. Fig.4 shows the flow line diagram of the whole expander. It can be seen from the figure that the flow field of the whole expander is relatively smooth, and there is no large separation and vortex. The volute, guide impeller and moving impeller are well coordinated. However, the flow field near the volute tongue is relatively disordered, which is because the working fluid diverges when flowing through the volute tongue. Most of the fluid flows clockwise along the volute, and then flows from the volute outlet to the guide vane channel. A small fraction of the fluid flows backward counterclockwise along the volute from the volute tongue, thus forming a turbulent flow and vortex near the volute tongue.
Fig. 4 shows the velocity streamlines of 10% blade height (blade root), 50% blade height (blade middle) and 90% blade height (blade tip) from blade to blade section of expander. It can be seen from the figure that the flow field of the guide vane has no great difference at different blade heights, and there is no separation and vortex in the flow channel. At the same time, the streamlines at the junction of the guide vane and the rotor are coherent, and the guide vane and the rotor are well coordinated. However, with the increase of the cross-section position, large eddy and irregular disturbance flow appear on the suction side of the rotor channel, which is mainly caused by the following reasons:

1) The flow direction of working fluid suddenly changes from radial to axial, and the direction changes sharply. The channel area increases suddenly, and the excess is not stable enough.

2) In the flow process, the impeller rotates at a high speed, and the fluid impacts the pressure surface of the impeller to drive the impeller to rotate. At the same time, the flow is deflected by a variety of complex forces such as the impeller, the hub and the rim.

3) The internal flow field of the impeller is very complex. 90% of the blade height is close to the rim. With the increase of the blade height, the deflection angle of the blade type increases, and the clearance of the rim has a great influence on the flow.

(a) 10% blade height  (b) 50% blade height
4.2 Performance analysis of expander impeller

Fig. 6 shows the static pressure contours of three blade-to-blade sections, namely, 10% blade height (blade root), 50% blade height (blade middle), and 90% blade height (blade tip) in the impeller flow passage. It can be seen from the three figures that when the airflow changes from radial to axial in the flow channel, a low-pressure zone is generated near the suction surface, and the low-pressure zone is larger at 90% blade height, which is consistent with the previous analysis. This is mainly because the flow channel direction changes from radial to axial, and the flow situation is complex. At the same time, the sudden increase in the channel area causes excessive expansion, resulting in a low-pressure zone. There is a high pressure area at the impeller inlet, which is caused by the direct impact of airflow on the pressure surface after it flows out of the guide vane. There is a small range of low pressure area at the trailing edge of the blade root, which is because the flow loss caused by a certain thickness of the trailing edge when the airflow flows out of the impeller. This part of loss can be reduced by adjusting the trailing edge curve. On the whole, the fluid pressure drop in the impeller flow channel mainly occurs in the inlet section, that is, it mainly occurs in the radial flow channel, and the fluid pressure drop in the axial flow channel is not obvious. The fluid pressure on the pressure side of the impeller flow channel is always greater than that on the suction side, which shows that the blade structure is reasonable and is conducive to the rotation of the impeller.
Fig. 6 Static pressure contours of blade to blade section of impeller

Fig. 7 shows the velocity contours of three blade-to-blade sections, namely, 10 % blade height (blade root), 50 % blade height (blade middle), and 90 % blade height (blade tip) in the impeller flow passage. It can be seen from the three figures that the distribution of velocity contours is gradually confused with the increase of blade height, which is due to the increase of blade deflection angle with the increase of blade height. At the same time, the flow near the tip of the blade is more complex due to the existence of flange clearance, which leads to the distribution of velocity contours near the tip of the blade is more complex. At the impeller inlet, there is a local high-speed region on the suction side, and there is a local low-speed region behind him. The position of the low-speed region is consistent with that of the low-pressure region analyzed above. Overall, the fluid velocity in the impeller passage increases uniformly along the meridian channel direction, and the fluid is basically accelerated from the inlet to the outlet of the impeller, and the flow field is relatively smooth.
Fig. 8 shows the surface static pressure contours of the impeller blade surfaces. It can be seen from the figure that the static pressure of the blade pressure surface decreases uniformly along the meridian channel direction, and the stress condition is good. On the suction side of the blade, some low pressure zones appear when the middle part of the blade changes from radial direction to axial direction, which is consistent with the previous analysis results. At the same time, the force conditions of the blades in different flow channels are slightly different but basically the same, indicating that the volute structure is relatively reasonable, the volute and guide vanes are well coordinated, and the gas is relatively uniform when flowing out of the volute outlet. Overall, when the fluid expands and works in the runner channel, the force condition of blade is good.

In summary, the overall performance of the moving impeller designed in this paper is good, and the flow in the flow channel is relatively smooth. There is no disturbance and vortex affecting the flow on the pressure side, but there are some disordered flow areas on the suction side, which affects the aerodynamic performance of the impeller. There is room for further optimization of the impeller structure.

4.3 Analysis of meridian channel of expander

Fig. 9 is the pressure contours of the expander meridian channel, and fig. 10 is the pressure change curve of the fluid along the channel. It can be seen from the figure that the fluid expands in the meridian channel of the expander, and the pressure decreases continuously along the meridian channel, and the fluid pressure decreases rapidly in the front of the meridian channel, indicating that the expansion of the fluid mainly occurs in the guide vane channel and the inlet section of the rotor channel. There is a low pressure zone in the middle of the midday channel, which is in line with the area analyzed earlier. There is a large low pressure area at the end of the meridian flow channel, which is due to the gas has a certain angular velocity when flowing out of the impeller outlet, and the flow is spiral in the outlet section of the expander. The pressure at the center of the spiral is low to form a low pressure area. With the development of flow, the low pressure area gradually disappears and the flow is restored to regular flow.
4.4 Comparison of design and simulation values

Table 3 is the comparison between the design value and the simulation value of the runoff turbine expander designed in this paper. It can be seen from the table that the error of simulation results is within 6 %, which is basically consistent with the design value. This shows that the expander meets the design requirements and can work well under the design conditions.

Table 3 Comparison of expander performance parameters

|                         | mass flow rate (kg/s) | output power (kW) | isentropic efficiency (%) |
|-------------------------|-----------------------|-------------------|----------------------------|
| design value            | 0.80                  | 40.65             | 81.7%                      |
| simulation value        | 0.81                  | 42.72             | 84.8%                      |
| error                   | 1.25%                 | 5.09%             | 3.79%                      |

5. Conclusions

In this paper, the turbine expander in natural gas pressure energy power generation system is taken as the research object, and its aerodynamic design and three-dimensional modeling are carried out. The
working performance and flow field characteristics under design conditions are studied by numerical simulation. The results show that:

1. The turbine expander designed in this paper has an output power of 42.72 kW and isentropic efficiency of 84.8% under design conditions, which meets the design requirements.

2. Through the analysis of the impeller flow field, it is found that the flow field in the designed impeller channel is relatively smooth, and the pressure and velocity distribution are reasonable. However, due to the complex structure of the impeller and many factors affecting the flow, there are local poor flow in the impeller channel.

3. The overall working characteristics of the impeller are good. The fluid pressure on the pressure side of the impeller channel is always greater than that on the suction side, which is beneficial to the rotation of the impeller. The pressure on the blade pressure surface decreases uniformly along the meridian flow channel, and the force condition of blade is good.

4. In the turbine expander, the expansion process of working fluid mainly occurs in the guide vane flow channel and the radial flow channel of the impeller flow channel. At the same time, when the runner changes from radial to axial, the flow loss is generated due to various factors. This should be considered in the subsequent optimization.

References

[1] Lo Cascio, E., Von Friesen, M.P., Schenone, C. (2018) Optimal retrofitting of natural gas pressure reduction stations for energy recovery. J. Energy. 153: 387-399.

[2] (2018) Some opinions of the State Council on promoting the coordinated and stable development of natural gas. J. Gazette of the State Council of the People's Republic of China, 26: 17-20.

[3] Li, j.j., (2011) Study on the recovery and application of cold energy in the process of natural gas pressure regulation in urban gate station. D. chongqing university.

[4] Pogorski, S., (2008) Enbridge pipeline to ultra-clean power generation project installation of a turbo expander in a North American gas distribution system. C. In:47th Annual GGA Gas Measurement School.

[5] Vasiljev, V.Y., Kiselev, O.M., (2009) Method of utilization of gas expansion energy and utilization power installation for implementation of this method: US,2009/0272115A1[P].

[6] Li, X.L., Huang, Q.F., (2001) Heat Separation Technology and Energy Recovery J. Energy Development and Conservation in Fujian, 2001,3:57-60.

[7] Zhao, H.L., Hou, Y., Zhang, M., (2005) Discussion on refrigeration method of natural gas liquefaction plant J. natural gas industry, 25(10):120-123.

[8] Zhan, A.A., Feng, Y.T., Lin, D., Li, Q., Wu, J.Z., (2020) Gas-electricity demand response model of microgrid considering comprehensive utilization of natural gas pressure energy. J. power system automation, 44(08):19-27.

[9] Wei, Y.J., Luo, K.Q., Tan, J.J., Tang, L., W, J., (2020) Study on hydraulic characteristics of vertical slit fishway based on three turbulence models J. hydroelectric energy, 38(10):71-74+78.