Design and Numerical Simulation of a High Speed Centrifugal Compressor

Qingyang Yu¹, Peng Wu², Yelin Deng¹

¹School of Rail Transportation, Soochow University, Suzhou, 215006, China
²School of Mechanical And Electrical Engineering, Soochow University, Suzhou, 215006, China
E-mail: yelin.deng@suda.edu.cn

Abstract. The issue of energy consumption has attracted worldwide attention, therefore the performance requirements of rotating machinery are getting higher and higher in modern commercial run. Active magnetic bearings have the advantages of high speed and low friction, so they are widely used in high-speed rotating machines such as high-speed centrifugal compressors. A stage of impeller and diffuser was designed. Numerical simulations of the flow field were performed using CFX. The pressure and velocity field in the model stage (impeller and diffuser) were obtained and analyzed. The results show that the model stage has uniform velocity field distribution and regular flow field, and the performance parameters can well meet the industrial requirements.

1. Introduction
In modern commercial run, due to the mechanical friction of the traditional motor that supports the rotor with mechanical bearings, the existing compressors are limited in speed. The compressors that use magnetic levitation bearings have extremely low frictional resistance at high or super high speed, and they can maintain stable performance under low temperature or high temperature [1-3]. The impeller is the core component of a centrifugal compressor. How to achieve energy saving and high efficiency is the main content of research on centrifugal compressor [4-5]. The current study designs a high-speed compressor. Through the simulation and analysis of the airflow inside the centrifugal compressor, the complicated curved surface and airflow flow characteristics of the impeller during its rotation are understood in detail. The current paper is organized in three parts. First, a centrifugal compressor is designed by the CF turbo. Second, the 3D flow field of the stage is simulated utilizing CFD software. Third, the flow characteristics of the compressor is discussed, with particular emphasis on velocity field of impeller and diffuser.

2. Three-dimensional model design
Considering the actual manufacturing process, the designed impeller blades are Ruled surface blades in the current paper, which are used especially to enable flank milling for manufacturing. The mean surface is generated by spatial movement of a straight line. To reduce the flow loss, the NACA6509 blades are used, and they are made of titanium alloy. The three-dimensional model of CF turbo impeller and diffuser is shown in Figure 1.
The current paper designs a high-speed centrifugal compressor, with considering the material and key size structure and speed of the impeller and diffuser. The ternary flow impeller has twisted blades, whose shapes are determined according to the flow requirements. Therefore, the ternary impeller has stable and efficient characteristics. The paper uses a unshroud impeller with splitter blades. And the specific design parameters are presented in Table 1.

## Table 1. Impeller parameters

| Impeller parameters                      | Value   |
|------------------------------------------|---------|
| Rated pressure ratio                    | 3       |
| Rated speed (r / min)                   | 40000   |
| Inlet temperature (K)                   | 293     |
| Total inlet pressure (Pa)               | 101325  |
| Impeller inlet diameter (mm)            | 138     |
| Impeller inlet hub diameter(mm)         | 28      |
| Number of main blades                   | 9       |
| Number of shunt blades                  | 9       |
| Impeller outlet diameter (mm)           | 223     |
| Impeller exit installation angle(°)     | 38      |
| Specific power                          | 3.9     |

3. **Theoretical basis of numerical simulation**

### 3.1. Governing Equations

1. **Mass Conservation Equation**

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{U}) = 0
\]

2. **Momentum Conservation Equation**

\[
\frac{\partial (\rho \vec{U})}{\partial t} + \nabla \cdot (\rho \vec{U} \vec{U}) = -\nabla p + \nabla \tau + \vec{S}_M
\]

\[
\tau = \left[ \mu \nabla \cdot \vec{U} + \left( \nabla \cdot \vec{U} \right)^T - \frac{2}{3} \delta \vec{U} \right]
\]

Among them, \( \vec{S}_M \) is the momentum source term, \( \tau \) is stress tensor.

3. **Energy Conservation Equation**

\[
\frac{\partial (\rho T)}{\partial t} + \nabla \cdot (\rho \mu T) = \nabla \cdot \left( \frac{\lambda}{c_p} \text{grad} T \right) + S_h + \phi
\]

### 3.2. Turbulence Model

Turbulence is a continuous pulsation of the fluid velocity, temperature, pressure, and other parameters that change with time and space. Its essence is a three-dimensional unsteady vortex motion.

1. **SST Model**
The SST model is derived from changes in the standard k-ε model. SST k-ε is a hybrid model, which retains the advantages of the model near the wall and uses the model far away from the wall. The specific formula is as follows,

\[ v_t = \frac{a_1 k}{\max (a_1 \omega; \Omega F_2)} \]  \hspace{1cm} (5)

\[ \frac{D \rho k}{D t} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \sigma_k \mu_t \right) \frac{\partial k}{\partial x_i} \right] + \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k \]  \hspace{1cm} (6)

\[ \frac{D \rho \omega}{D t} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \sigma_\omega \mu_t \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\gamma}{v_t} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta \rho \omega^2 + 2(-F_1) \rho \sigma_\omega^2 \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \]  \hspace{1cm} (7)

Among them, \( \Omega \) is the vorticity, \( F_1 \) and \( F_2 \) are mixed functions, which are functions of the distance from the point to the wall surface.

4. Numerical simulation

4.1. Calculation settings

1. Mesh: The mesh is a unstructured mesh. The leading and trailing edges of the impeller blades and diffuser blades have been refined. The total number of grid points is 13.04 million. The mesh is shown in Figure 2.

2. Boundary conditions: Adiabatic, non-slip boundaries are used on the wall.

3. Computational setup: The dynamic and static interface settings of the impeller and diffuser are set using Frozen Rotor, and the convergence criterion is 1E05.

5. Compressor flow characteristics analysis

5.1. Analysis of pressure field characteristics

Figure 3 represents that the overall pressure distribution in the impeller is relatively uniform. Along the direction of fluid flow, the static pressure on the pressure surface of the impeller increases uniformly, and the pressure at the turning of the suction surface at 1/2 of the diffuser blade drops suddenly. When the pressure reaches the end of the diffuser blade, the pressure rises sharply. This phenomenon of a sharp decrease in pressure increases with the generation of shock waves, which will lead to the loss of kinetic energy. From the pressure nephogram, it is indicated that along the direction of fluid flow, the fluid pressure inside the volute is slightly lower than the fluid pressure outside, and the pressure is lower near the volute, but the overall distribution is regular.
5.2. Analysis of flow field characteristics

The impeller streamline diagram and compressor streamline diagram are shown in Figures 4 and 5, respectively. The reason why there are some irregular flow lines at the impeller blade tip clearance in Figure 4 is the leakage and flow of the tip clearance, but it indicated that the tip leakage loss are little in the impeller. The flow velocity in the diffuser increases sharply at the corresponding sudden pressure change and secondary flow occurs at the trailing edge of individual blades, which is due to the influence of the reverse pressure gradient. The effect of the diffuser blade at different positions on the flow field at the outlet is large, but the flow separation is small, and the overall flow is stable.

![Figure 4. Impeller streamline diagram](image)

![Figure 5. Compressor streamline diagram](image)

5.3. Analysis of velocity field characteristics

The speed vector diagram of the impeller is shown in Figure 6. The flow angle of attack at the leading of the blade and the front mounting angle of the blade has a low coefficient of variation. Due to the high speed, it can be seen from the figure 6 that the high-speed section is mainly concentrated on the inside of the main blade, and the speed is significantly reduced after the splitter blade, and the speed at the tip is higher than the surrounding speed.

The speed vector diagram of the diffuser is shown in Figure 7. The deviation of the flow angle of attack at the leading edge of the diffuser blade is basically zero, which causes very little flow loss. The stability of the flow at the outlet of the diffuser greatly affects the performance of the compressor. The distribution of the field relatively intuitively reflects the stability of the flow at the outlet of the diffuser. It can be seen from the figure that the flow at the diffuser outlet is evenly distributed, and there is basically no separation in the flow channel to minimize loss.

![Figure 6. Impeller speed vector](image)

![Figure 7. Vector of diffuser speed](image)

6. Conclusion

The numerical results show that the actual pressure ratio of the compressor meets the design requirements. Through numerical simulation of the impeller, it can be seen that the flow path of the impeller is relatively narrow, the working distance of the impeller is long, the flow separation is limited, and the efficiency is high. An unavoidable leakage occurs at the top of the impeller, resulting in secondary flow. However, there is small loss of flow due to the low intensity of secondary flow. In addition, this impeller has advantages of air-distribution uniformity and reasonable pressure distribution. Through the simulation analysis of the diffuser, the deviation of the flow angle of attack...
at the leading edge of the diffuser blade is basically zero, and the air flow at the blade trailing edge tends to be stable, which reduces the flow loss. And the design is reasonable.

High-speed centrifugal compressors with ternary flow theory for blade design and excellent magnetic suspension bearings are bound to have very important application value and broad application prospects in practical industrial production.

References
[1] Tian Li. Research on the Fault Tolerant Control System of Active Magnetic Bearing [D]. Huazhong University of Science and Technology, 2019.
[2] Jing Tian. Structural Optimization Analysis of high Speed Axial Magnetic Bearings[D]. Shangdong University, 2019.
[3] Reza Safaeian, Hossein Heydari. Study effect of the back irons in passive magnetic bearings with alternating radially magnetized rings by using 2D analytical method[J]. Compel, 2019, 38(6).
[4] Ziliang Li, Ge Han, Xingen Lu, Enliang Huang, Shengfeng Zhao. Improving the operating range using a centrifugal compressor with a tandem impeller[J]. Aerospace Science and Technology, 2019.
[5] Evandro L.L. Pereira, Cesar J. Deschamps. Numerical analysis and correlations for radial and tangential leakage of gas in scroll compressors[J]. International Journal of Refrigeration, 2019.