Design of a High Pressure Ratio Centrifugal Compressor

Pei-yuan LI
Department of Energy and Power Engineering, Tsinghua University, Beijing 100084, China

Keywords: Design, High pressure ratio, Centrifugal compressor, Impeller, Diffuser.

Abstract. High pressure ratio centrifugal compressor has gained widely attention recently. In this paper, a pressure ratio 11 centrifugal compressor with an inlet mach number of about 1.6 is preliminary designed by computer. The complex shock boundary layer interaction, low energy region in the vicinity of the casing and the outlet flow conditions are analyzed. The vaneless and vaned diffusers are also designed. CFD results show that the vaned diffuser can effectively improve the compressor efficiency.

Introduction

In recent years, with the development of small gas turbines and helicopter engines, the centrifugal compressor has gained widely attention[1-3]. Compared with axis compressor, centrifugal compressor has higher ability to transport power with the same rotation speed and mass flow. So the centrifugal compressor has been generally used in helicopter and small plane engines. Due to single stage compressor ratio can't meet the requirement, so gas turbine mostly adopt two stage centrifugal compressor or multi-stage axis compressor with single stage centrifugal compressor. However, the design not only makes the gas circuit more complicated, but also makes the gas turbine bigger, raises the weight and decreases thrust-weight radio. Because of the above disadvantages in the multi-stage structure, so research on the high single stage pressure ratio of centrifugal compressor has important significance for improving the entire performance of the gas turbine.

With the development of related disciplines such as aerodynamic and materials science, the single stage pressure ratio of centrifugal compressor continues to rise and the research of internal flow analysis becomes more. In 1976, Eckardt[4] firstly used L2F method to measure the Jet-wake phenomenon in the outlet of a centrifugal compressor with pressure radio of 2.1. Karin[5] also adopts L2F method to measure the a centrifugal compressor with pressure radio of 4. C Hah[6] performed 3D numerical simulation on it. Later on, Krain[7,8] measured the shock wave in the inlet of centrifugal compressor which has inlet tip mach number of 1.3 and the pressure radio of 6.1. C Hah[9] made numerical simulation on it. Y Senoo[10] measured the shock wave and pressure distribution in the inlet of centrifugal compressor with pressure radio of 10. Hirotaka Higashimori[11,12] measured the centrifugal compressor with pressure radio of 11 and analyzed the flow characteristic.

The vaned diffuser as one of three coral parts of the high pressure radio centrifugal compressor, has the crucial influence on entire stage efficiency and blocking flow. There are lots of research on the transonic flow in the vaned diffuser inlet. Seloochi Lbaraki[13] made PIV measurement on the vaned diffuser with inlet mach number of 0.95 and made CFD numerical simulation. Krain[14] made test and CFD numerical research on the unsteady flow of the vaned diffuser with inlet mach number of 0.9. The research on the vaned diffuser with transonic inlet is seldom.

The paper designed a centrifugal compressor with pressure radio up to 11. Its inlet mach number reached 1.6. Using the CFD software[15-22] to make numerical simulation and analyzed the flow characteristic. Then, the design of vaned diffuse was completed based on the calculation results of the impeller outlet.
The Method and Results of Impeller Design

The design process of centrifugal compressor consists of two parts, 1D aerodynamic design and 3D impeller modeling. 1D aerodynamic design has been relatively mature after these years’ development. The tradition design method and TEIS method are adopted frequently. As for the tradition design method, the gas loss in the impeller is divided into skin friction loss, blade loading loss, leakage loss, recirculation loss, disk friction loss and so on. For the different losses, the corresponding empirical formula is given based on the test and CFD numerical simulation. In the design process, the loss of each part can be calculated according to the design parameters such as mass flow and pressure radio. Then the required inlet and outlet diameter can be calculated. For every kind loss, there are many empirical formulas to be choose from, but every empirical formula has its adjustment for different impeller. So it is important to choose empirical formula for different design demand and it will determine the accuracy of tradition 1D design method result. Then tradition 1D design method has higher requirements for designer's experience. For the sake of avoiding the disadvantages of tradition 1D design method, Japikse[23] proposed TEIS method in 90s. In this method, the gas flow in the impeller is divided into two parts with the throat as the boundary: (1) the front segment is from inlet to rear and (2) the posterior segment is from throat to outlet. Each segment is regarded as a simple nozzle or diffuser. The flow in the posterior segment is divided into main stream and the second flow. The flow in the main stream is regarded as ideal flow. All the losses occurred in the secondary flow area [24]. In the design process, 4 relevant parameters are given to complete the design. Although there are only 4 relevant parameters, many results should be tested for the design accuracy. No matter it is tradition design method or TEIS method, the slip factor played a vital role. Because the slip factor directly affects the work made by the impeller. Similar to the loss model, slip factor also has many models to be selected. The difficult point of 1D design is how to choose the appropriate model according to the characteristics of the design impeller. After the 1D pneumatic design is finished, the shape of the arc, the thickness distribution and the patten of the stacking line in the impeller are determined in the 3D modeling program. Finally, the design of the impeller will be accomplished. After the shape of the impeller is determined, the CFD numerical simulation will be used to simulate the impeller. If the calculated results agree well with the design demand, then the preliminary design is done. Then the design of vane diffuser and the impeller optimization can be carried out based on the calculation results. If the result can’t meet the design demand, then the calculated results will be analyzed again. If the impeller design is unreasonable to make the internal loss bigger. Then it is necessary to redesign the impeller using 3D impeller modeling. If the impeller geometry parameter is wrong, then it is necessary to recalculate the 1D pneumatic design. Repeat until the design is completed.

The impeller structure is shown in Figure 1. The impeller design rotation speed is 50000rpm. The design pressure ratio is 11. In the design process, the double splitter blade is added in the large blade. It makes the blade fewer in the impeller inlet to increase the inlet area and the blocking flow. The blade increases in the impeller middle part to reduce the blade load. The blade number increases in the outlet which can enlarge slip factor and improve the impeller work. It also improves the flow condition in the outlet of the impeller. The impeller outlet pressure ratio is 13.6 and the efficiency is 85.7%. The meridional view of centrifugal compressor is shown in Figure 2.
The Internal Flow Analysis

The relative mach number contours on 90% span is given in Figure 3. In the design condition, an oblique shock wave is produced in the sanction surface of the large blade and extended to the inlet leading edge of the adjacent blade. There is a small passage shock wave behind the oblique shock wave, which takes up almost the whole passage. Behind the passage shock wave, the gas decelerated uniformly. But from the front of the second flow to the impeller outlet, there is an obvious low speed area. In the near blockage condition, the oblique shock wave in the sanction surface of the large blade increase a little, and the intense of the passage shock wave increases shapely over the first oblique shock wave. The position moves to the leading edge of the first shunt blade and takes up the whole passage. In the near stall condition, the intense of oblique shock wave changes not much. But the passage shock wave completely disappeared to make the whole passage decelerated uniformly.

For analyzing the gas flow in the passage, ten cross sections like Figure 2 are taken along the flow direction on the meridional flow surface. Figure 4 shows the design point, the relative mach number distribution and the entropy contours on different sections.

In the section A of impeller inlet, the relative mach number is the largest on the pressure surface near the rim. The closer the sanction surface, the smaller the mach number. It is because the far away from the sanction surface to the oblique shock wave. The loss mainly occurred near the rim and the
hub. In the section B, there is an obvious velocity interface near the rim. This is because the gas, which is near the pressure surface, has crossed the oblique shock wave. So the flow is subsonic. The gas far away from the pressure surface is on the hypersonic flow condition, because it is near the shock wave. The closer the sanction surface, the lower the mach number. In the section C, the subsonic flow area becomes larger and two hypersonic flow areas are produced. The hypersonic area near the sanction surface is in the front of the oblique shock wave, because the intense of oblique shock wave is large, so the mach number is large. In the middle of the flow passage, the hypersonic area near the rim is behind the oblique shock wave. In the front of the passage shock wave, due to the weak of the passage shock wave, the Mach number is low. The obvious low speed area appears near the rim. The loss in the rim becomes larger.

**The Design and Analysis of Vaned Diffuser**

Figure 5 shows the absolute mach number and absolute flow angle contours on impeller outlet section J. It can be seen the gas absolute mach number is about 1.3 in the impeller outlet. There is a small low speed area near the rim. The absolute flow angle is about 650. The farther from the rim, the bigger the angle. The angle sudden decreases negative near the rim, indicating that there is a backflow near the rim. From the above analysis, it can be seen that the gas speed is hypersonic at the vaned diffuser inlet. And the absolute flow angle distributed uniformly. These all make the vane diffuser design more difficult.

![Figure 5. Absolute mach number and absolute flow angle contours on impeller outlet.](image)

The vaned diffuser structure is shown in Figure 6. The vaned diffuser has two parts including radial diffuser and axis diffuser. The relative mach number contours on 50% span is shown in Figure 7. It can be seen the mach number is 1.3 in the vaned diffuser inlet. The flow before the throat is hypersonic flow and behind is subsonic. The diffusive process is basically completed in the front of radial diffuser. There is a low speed area near the tail pressure surface. The main function of axis diffuser is to make the gas flow axial in the outlet. The parameter comparison of vaneless and vaned diffusers are shown in Table 1. When the vaned diffuser is used, the outlet pressure radio is 11.14 and the efficiency is 76.2%. It satisfied the design demand.

![Figure 6. Vaned diffuser structure.](image)  ![Figure 7. Relative mach number contours on 50% span.](image)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Radial diffuser outlet</th>
<th>Axial diffuser outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure radio</td>
<td>11.67</td>
<td>11.14</td>
</tr>
<tr>
<td>Pressure loss coefficient</td>
<td>0.587</td>
<td>0.413</td>
</tr>
<tr>
<td>Total pressure loss coefficient</td>
<td>0.273</td>
<td>0.292</td>
</tr>
<tr>
<td>Efficiency</td>
<td>0.782</td>
<td>0.762</td>
</tr>
<tr>
<td>Flow angle</td>
<td>18°</td>
<td>90°</td>
</tr>
</tbody>
</table>
Conclusions

The paper adopted the 1D pneumatic design procedure and 3D modeling procedure to accomplish the preliminary design of the centrifugal compressor with the pressure ratio of 11 and the vaned diffuser. It also analyzed the relative mach number and entropy distribution on different sections in the design condition. The result shows that there is a complex wave structure in the front section of the high pressure radio centrifugal compressor impeller. The low speed area near the impeller rear rim which is simulated by the CFD needs further research. In the vaned diffuser, the diffusive process is mainly completed in the radial diffuser and the main function of axis diffuser is to make the gas flow axial in the outlet.

References


