DESIGN AND ANALYSIS OF A HIGH PRESSURE RATIO
CENTRIFUGAL COMPRESSOR WITH THREE DIFFUSERS

Ge Han¹,², Lei Guo³, Ziliang Li¹,², Chengwu Yang¹,², Shengfeng Zhao¹,², Xingen Lu¹,²
1. Key Laboratory of Light-duty Gas-turbine/Institute of Engineering Thermophysics, Chinese Academy of Sciences
2. University of Chinese Academy of Sciences
hange898525@sina.com
Beijing, China

ABSTRACT
Ever-growing compressor total pressure ratio is one of the main tendencies in centrifugal compressor development. However, single stage high pressure ratio centrifugal compressor design is a tough problem since both stage stall margin and adiabatic efficiency will be decreased under high pressure ratio conditions. In this present work, a pressure ratio 8.1 single stage centrifugal compressor is designed with three different radial diffusers to control the shock waves and flow separations in diffuser passages. These three radial diffusers are cascade diffuser, wedge diffuser and a kind of modified wedge diffuser. Stage performances are obtained by numerical method. Due to the arbitrary controlled blade angle and thickness of modified wedge diffuser, it can control the shock wave and flow separation better than the other two diffusers. Thus, the impeller with modified wedge diffuser has the highest stage total pressure ratio and adiabatic efficiency. Then the centrifugal compressor stage with modified wedge diffuser is tested on a high speed compressor rig. Compressor map of this centrifugal compressor is obtained for both design and off-design speedlines. The design object of this high pressure ratio compressor is achieved. Through this work, physical insight into flows in this high pressure ratio centrifugal compressor and test results of the compressor stage are obtained to give some guidelines on design of high pressure ratio centrifugal compressors.

INTRODUCTION
Due to their high pressure ratio, wide operating range, simple structures and low manufacturing costs, centrifugal compressors are commonly used in turbochargers as well as helicopter engines and other small gas turbine applications (Krain, 2005). With the growing necessity of increasing specific output power and decreasing fuel consumption, the high pressure ratio, high efficiency centrifugal compressor design has been encouraged in recent years. Krain (Krain et al, 2007) improved the centrifugal compressor total pressure ratio from 5.9 to 6.2 and the adiabatic efficiency of this compressor was increased by 2%. However, the surge margin of low rotating speed was reduced. Then an inverse design was done by Zangeneh (Zangeneh et al, 2011) on this total pressure ratio 6.2 centrifugal compressor. Three dimensional blades were used instead of ruled surface blades to change the load and vorticity streamwise distributions and the stacking curve. Pressure ratio at design speed was increased and adiabatic efficiency was improved by 2% at all rotating speeds. A centrifugal compressor with pipe diffuser was tested and analysed by Sugimoto (Sugimoto et al, 2014) and the highest pressure ratio of 7.75 with a corresponding isentropic efficiency of 80% had been achieved at the design point flow rate of 4.05 kg/s. Detailed flow in a total pressure ratio 11.0 single stage centrifugal compressor was studied by both numerical and experimental method (Higashimori et al, 2004; Higashimori et al, 2007) and it was used on MG-5 serious turboshaft engines (Uchida et al, 2003). High pressure ratio single stage centrifugal compressor was designed and tested by Motor Sich and it was used on the MS-500 serious turboshaft engines equipped on ANSIAT helicopter (Jain, 2011). The pressure ratio of this compressor was about 11.5 and it achieves a polytropic efficiency of 80%.

As the total pressure ratio of centrifugal compressor increases, it is hard to achieve high efficiency (Oana et al, 2004; Japikse and Osborne, 1994) and surge margin of compressor stage will be decreased. So it is difficult to design a high pressure ratio centrifugal compressor. This is because the shock wave is strong in both impeller and diffuser in high pressure ratio centrifugal compressors. The shock wave and boundary layer interaction also induces flow separation and causes large losses and flow instability. So it is important to control the shock waves in high pressure ratio centrifugal compressor and reduce losses to improve stage efficiency and surge margin.

The present work is aimed at designing and analysing a total pressure ratio 8.1 single stage centrifugal compressor.
for a small gas turbine engine. Impeller with three different diffusers is designed with an in-house code. The three diffusers are cascade diffuser, wedge diffuser and modified wedge diffuser. Then internal flow in the impeller and three diffusers are analysed through numerical method. The shock wave and losses in three diffusers is compared in detail. Finally, the stage with modified wedge diffuser which has the best performance is tested on a compressor rig.

**COMPRESSOR DESIGN**

**Preliminary Design**

A total pressure ratio 8.1 single stage centrifugal compressor is needed in a small turbo engine. It is consist of impeller, radial diffuser and axial diffuser. The meridional contour of the stage is plotted in Fig.1. The preliminary design and the 3D blade forming of both the impeller and diffusers are done with an in-house code.

![Fig.1 Meridional contour of centrifugal compressor](image.png)

**Table 1 Compressor Specifications**

<table>
<thead>
<tr>
<th>Compressor Stage</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Design mass flow</td>
<td>2 kg/s</td>
</tr>
<tr>
<td>Total Pressure ratio</td>
<td>8.1</td>
</tr>
<tr>
<td>Adiabatic efficiency</td>
<td>80%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Impeller</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotating speed</td>
<td>54000 r/min</td>
</tr>
<tr>
<td>Specific speed</td>
<td>0.886</td>
</tr>
<tr>
<td>Number of blades</td>
<td>11full+1splitter</td>
</tr>
<tr>
<td>Exit diameter</td>
<td>217mm</td>
</tr>
<tr>
<td>Back sweep angle</td>
<td>30°</td>
</tr>
<tr>
<td>Exit tip speed</td>
<td>614m/s</td>
</tr>
<tr>
<td>Exit blade height</td>
<td>7mm</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Radial Diffuser</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades</td>
<td>23</td>
</tr>
<tr>
<td>Leading edge radius</td>
<td>116mm</td>
</tr>
<tr>
<td>Trailing edge radius</td>
<td>155.5mm</td>
</tr>
<tr>
<td>Inlet metal angle</td>
<td>13°</td>
</tr>
</tbody>
</table>

One dimensional calculation, using semi-empirical correlations (Van den Braembussche, 2012; Japikse, 1988), has been performed for both impeller and diffuser. The two zone model is used to describe the flow in impeller in preliminary design and the slip factor relation of Wiesner (Wiesner, 1967) is adopted. The efficiency of the stage is determined by the loss model. And the losses evaluated in the stage including incidence loss, blade loading loss, friction loss, tip clearance loss and throat blockage loss. The effects of shock waves are considered in the blockage loss in terms of Mach number correction. Surge for the stage is predicted by a critical flow angle at diffuser inlet and the value of the angle is based on our own design experiences. After the preliminary design, the specifications of this centrifugal compressor are listed in Table 1.

Three dimensional blades of both impeller and diffusers are formed based on the results of preliminary design.

**Impeller Blade Design**

A backswep unshrouded impeller is designed for this compressor. The hub and shroud curve of the impeller is optimized to control the flows. The slope and curvature of hub and shroud in meridional surface is shown in Fig.2. The slope of impeller shroud is small near leading edge and increases along the streamwise. Meanwhile, the curvature of shroud is small near leading edge and trailing edge of the blades but large in the middle of the impeller. It is propitious to eliminate the Mach number at blade leading edge near shroud and reduce the reverse flow at impeller outlet near shroud.

![Fig.2 Slope and curvature of impeller contour](image.png)
shroud traces. Then the thickness distribution is applied on the blade. A linear connection between points of both traces generated a ruled surface. The streamwise blade angle distribution and blade thickness distribution are provided in Fig.3, and the picture of the impeller is shown in Fig.4. The main blade shroud leading edge blade angle varies slowly to decrease the acceleration of flow, aiming at reduce the shock intensity. In order to decrease the inlet relative Mach number, the blade leading edge thickness is thin and the leading edge is designed as elliptical arc.

**Diffuser Design**

Three different kinds of radial diffusers are taken into consideration in this stage, which are cascade diffuser, wedge diffuser and a new kind of modified wedge diffuser. Cascade diffuser is usually used in high pressure centrifugal compressors. And the requirement for high efficiency and compact size has led to widespread use of wedge diffusers in highly loaded transonic centrifugal stages (Filipenco, 2000). So these two kinds of diffusers are chosen for this compressor. However, both cascade diffuser and wedge diffuser could not develop the blade arbitrarily. Thus a new kind of modified wedge diffuser is used.

The cascade diffuser has a MCA blade profile, which could be seen in Fig.5(a) The maximum thickness of the blade is about 2mm with 0.6mm leading edge and trailing edge. The wedge diffuser has a 0.2mm leading edge radius and the divergence angle of the diffuser passage is 5° to control flow separation in diffuser passage. As shown in Fig.5(b), the pressure side corner near trailing edge of the vane is rounded by line and circular arc instead of using a blunt trailing edge to reduce the wake behind the wedge diffuser. The modified wedge diffuser has arbitrary controlled blade angle and thickness distribution along streamwise. Its blade is shown in Fig.5(c).

![Fig.3 Blade angle and thickness](image)

![Fig.4 Tested impeller](image)

**Fig.5 Three radial diffusers**

The axial diffuser is a row of MCA blades. The geometrical parameters of the axial diffuser blade are shown in Table 2.

<table>
<thead>
<tr>
<th>Table 2 Parameters of axial diffuser blade</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\beta_{LE}/^o$</td>
</tr>
<tr>
<td>-----------------</td>
</tr>
<tr>
<td>60</td>
</tr>
</tbody>
</table>

**NUMERICAL METHOD**

The numerical results for this high pressure centrifugal compressor are evaluated with three-dimensional, steady state, viscous CFD solver FINE/Turbo. This code is a cell-centered, finite volume, multi-block structured solver for the
compressible Navier-Stokes equations. In this study, the Reynolds-averaged Navier-Stokes (RANS) equations in conservative formulation are spatially discretized based on a cell central explicit finite volume scheme under relative cylindrical coordinate rotating together with the reference frame. For the temporal discretization, the forth-order Runge-Kutta scheme is applied. In this code a wide variety of turbulence models are provided for turbulence closure. And in this work the turbulence is modeled by one equation Spalart-Allmaras model which features numeration accuracy for the calculation of viscous boundary layer turbulent flow and separated flows of small or medium scale (Spalart and Allmaras, 1992).

The discretization of both impeller and diffusers are accomplished using a structured multi-block meshing strategy employed by the AutoGrid mesh generator from NUMECA. The mesh is generated and optimized by the Row Wizard in AutoGrid to increase the quality of periodical matched grids. An H&I topology is used for the impeller to improve orthogonality of the mesh. And the so-called HOH topology is used for diffusers. To properly capture the viscous and turbulent effects close to the compressor walls, the mesh discretization is performed such that a $y^+$ less or equal to 2 is obtained at the walls. The mesh quality of the three rows is good with the minimum skewness angle 18.4°. The computational domain consists of a single passage of impeller, radial diffuser and axial diffuser. Grid with $2.4 \times 10^6$ nodes is used for the whole compressor after study of mesh independence. Normally about $1.2 \times 10^6$ nodes is used to build the impeller, $0.7 \times 10^6$ nodes for the radial diffuser and $0.5 \times 10^6$ nodes for the axial diffuser. The computational grid for compressor stage with modified wedge diffuser is shown in Fig. 6.

The working fluid is air ideal gas. The total pressure (101325Pa), total temperature (288.2K), and flow angle (axial intake) are specified at the impeller inlet. The domain exit average static pressure is adjusted to make the operating condition of compressor shift from choke toward stall. Nonslip and adiabatic wall boundary conditions are applied to both the blades and hub/casing walls. The mixing plane is used between the impeller/radial diffuser and radial diffuser/axial diffuser to combine them as a single computational domain. Surge is predicted based on convergence difficulty.

**NUMERICAL ANALYSIS**

**Stage Performance**

The total pressure ratio and adiabatic efficiency against the non-dimensional corrected mass flow rate at 100% rotating speed of the compressor stages with three different diffusers are shown in Fig. 7. The mass flow rate is non-dimensionalized by the design mass flow. As shown in Fig. 7, the compressor stage with modified wedge diffuser has higher efficiency and pressure ratio than the compressor stages with the other two diffusers. The pressure ratio and peak efficiency of stages with both wedge diffuser and cascade diffuser are almost identical. There is an improvement of total pressure ratio for 0.4 by using modified wedge diffuser. Meanwhile, an approximately 2.4% maximum adiabatic efficiency increase is achieved. The stages with modified wedge diffuser and cascade diffuser have almost the same surge mass flow, but stage with wedge diffuser has large surge mass flow. Although all the three diffusers have almost the same geometrical throat size, stage with modified wedge diffuser has larger choking mass flow due to small throat blockage.

![Fig.6 Computational grids](image1)

![Fig.7 Performance comparison](image2)
By comparing the performances of stages with three diffusers, it is obvious that the stage with modified wedge diffuser is the optimum choice for this high pressure ratio centrifugal compressor.

**Loss mechanisms in diffuser and impeller**

There is a big difference when the centrifugal compressor impeller equipped with different diffusers. So there is necessity to figure out the reasons. In order to guarantee a fair comparison between the stages with different diffusers, the same value of $m/\pi$, similar as the work done by impeller (Yang et al., 2003; Marsan et al., 2012), is required. It can be seen in Fig. 7 that there is a circled operating point on each of the speedlines with different diffusers. These circled operating points corresponding to the peak efficiency point of wedge diffuser have the same value of $m/\pi$. Mach number and radial velocity contours for circled operating points in three diffusers are shown in Fig.8.

![Mach number and flow separation in diffusers](image)

**Fig.8 Mach number and flow separation in diffusers**

As seen in Fig.8, there is a high Mach number region from diffuser inlet to throat. A shock wave is formed at the diffuser leading edge from hub to shroud. The Mach number of wedge diffuser in this region is the highest among the three diffusers. For the cascade diffuser, the inlet Mach number is smaller than wedge diffuser since there is a deceleration effect caused by the cambered diffuser blade. The blade angle and thickness distributions between leading edge and throat of the modified wedge diffuser are designed elaborately in order to control the shock and reduce Mach number. And this results in the lowest Mach number at modified wedge diffuser leading edge. High Mach number flow can induce strong shock in diffuser and cause large losses. The low Mach number in modified wedge diffuser is apt to decrease shock losses compared with the other diffusers.
Low velocity flow appears at pressure side of cascade diffuser near hub. And the radial velocity also becomes negative at cascade diffuser pressure side corresponding to the low Mach number region. That means the flow separates at cascade diffuser pressure side and reverses to the inlet of diffuser. For the wedge diffuser, flow separates at both pressure and suction side of vane. Since the shock wave in wedge diffuser is strong, the separation near suction side is mainly due to the interaction of shock and boundary layer. Moreover, because of the thick trailing edge, reverse flow also occurs after vane trailing edge. However, flow in modified wedge diffuser only separates slightly at the middle of pressure side and after trailing edge near hub. The midspan flow improves a lot compared with the flow field at 5% span for all the three diffusers. The flow separation still exists at cascade diffuser pressure side, but the region is smaller than 5% span. In wedge diffuser and modified wedge diffuser, flow separation is eliminated in diffuser passages. And the reverse flow after trailing edge is also depressed. The pressure side flow separation becomes smaller at 95% span in cascade diffuser. The reverse flow is near trailing edge at pressure side. The flow field for wedge diffuser and modified wedge diffuser at 95% span is similar to the flow field at midspan, except the attenuation of inlet Mach number. There is a noticeable reverse flow region at diffuser inlet for all the three diffusers at 95% span. This is induced by the wake of impeller. Flow separation in diffuser passage can cause large losses, thus modified wedge diffuser is tend to get lower losses than the other two diffusers.

As the modified wedge diffuser has low shock losses and small flow separation in diffuser passage, it has high performance. Though the flow separation near cascade diffuser pressure side is severe, the surge margin is not decreased a lot. Whereas, when the operating point moves towards small flow rate condition, the incidence at wedge diffuser leading edge becomes larger and the flow separation at suction side becomes more serious. The increased suction side separation finally leads to an early surge.

Since the stage with modified wedge diffuser has best performance. The flow in the impeller for the stage with modified wedge diffuser at the circled operating point in Fig. 7 is illustrated in Fig.9 and Fig.10.

Fig. 9 shows the computational relative Mach number contours at 95% span near casing. The passage shock is located in front of the main blade leading edge and the calculated maximum relative Mach number is about 1.4. This shock starts from adjacent main blade suction side and oblique towards the other adjacent main blade pressure side. After this shock the flow becomes subsonic before inducer throat. As flow goes into the channel between the two main blades, it accelerates again and another shock is generated in the channel, but the intensity of this shock wave is low. By changing the impeller leading edge shape, blade angle distribution and shroud contour shape, the shock in the impeller is reasonably controlled and an adiabatic efficiency of 90% is achieved.

![Fig.9 Relative Mach number contour at impeller 95% span](image)

There is a low velocity region in the impeller passage near outlet in Fig.9. And this can also be seen in Fig.10. The jet and wake structure is obvious at the impeller outlet. The district wake flow as present in the exit of impeller close to the suction side near shroud causes a large absolute flow angle from radial direction. The losses are mainly caused by the wake flow. The jet located at the hub/pressure side corner and results in a relatively small absolute flow angle. So it is hard to design the diffuser under such a non-uniform impeller outflow. This may be improved in the future.

![Fig.10 Impeller outflow](image)

**TEST OF COMPRESSOR**

**Test Facility**

The compressor is tested on a high speed rig shown in Fig.11. The impeller is powered by an 800kW motor, and the rotational speed is controlled by changing the frequency of the power supplied to the motor. A gear box with a speed ratio 21.67 is used between the motor and compressor to increase the rotating speed from 3000rpm up to 65000rpm. The test compressor details are shown in Fig.12. The centrifugal compressor stage is experimentally investigated with the modified wedge diffuser and axial diffuser. After the axial diffuser, the flow turns to the radial direction in an exhaust casing and discharged into a volute radially.
The major parameters measured during the testing are the total/static pressure and temperature at the inlet/outlet of the compressor, mass flow rate, rotational speed, ambient pressure and ambient temperature. The total and static temperature are measured by thermocouples with an error of ±1 °C, the total and static pressure are measured by piezoelectric pressure sensors with a relative error of less than 0.3%, the mass flow rate is measured using the inlet bellmouth with a relative error within ±0.5%, and the rotating speed is measured by an electromagnetic transducer with a relative error within ±0.15%. The inlet station is placed before the impeller inlet, whereas the outlet measurement station is located downstream of the axial diffuser. The mass flow rate in the compressor is adjusted by two sensitive backpressure valves on the pipe downstream of volute. The last point before the audible surge is regarded as the surge point on the performance map.

**Test Results**

Fig. 13 provides experimental measured compressor map for the single stage centrifugal compressor at design and off-design rotating speeds. The total pressure ratio and the adiabatic efficiency are plotted against the non-dimensional mass flow. The design point is marked on the map. It can be seen from these two figures that the design total pressure ratio is 8.137 with the adiabatic efficiency about 80.5%. The surge margin is about 15.5%. However, the mass flow rate of design point on the map is a little smaller than the design mass flow.

There is an over prediction for the computational results in Fig.7 compared with the test results. This phenomenon is also found in a high total pressure ratio impeller (Eisenlohr, 2002). In Eisenlohr’s opinion, perhaps the turbulence model cannot solve this problem satisfactorily.

**Fig.13 Test performance**

**CONCLUSIONS**

The present work reports a design and test of a total pressure ratio 8.1 single stage centrifugal compressor. The impeller is elaborately designed to control the shock waves. Cascade diffuser, wedge diffuser and modified wedge diffuser are designed to equip with the impeller. The flow inside the machine is analysed numerically. The results show that by optimizing the impeller blade shape and shroud contour, the shock wave at impeller leading edge near casing is reasonably controlled. A high efficiency impeller is achieved. Stage with modified wedge diffuser has higher performance and wider operating range than the stages with both cascade diffuser and wedge diffuser. The strong shock and flow separation in cascade diffuser and wedge diffuser have negative impact on stage performance. The reverse flow at wedge diffuser suction side can decrease the surge margin.
of the compressor. According to the test data of compressor stage with modified wedge diffuser, the design point total pressure ratio and adiabatic efficiency are 8.137 and 80.5%, respectively.

NOMENCLATURE

- \( m \) Mass flow
- \( r \) Radial
- \( t \) Thickness
- \( V \) Velocity
- \( \alpha \) Absolute flow angle
- \( \beta \) Blade angle
- \( \pi \) Total pressure ratio
- \( \text{max} \) Maximum
- \( \text{LE} \) Leading edge
- \( \text{TE} \) Trailing edge
- \( \text{PS} \) Pressure side
- \( \text{SS} \) Suction side

ACKNOWLEDGMENTS

The work was financially supported by the National Natural Science Foundation of China (Project No: 51476166). This support is greatly appreciated.

REFERENCES


