DESIGN AND EXPERIMENTAL TEST OF A 2.5-STAGE HIGHLY LOADED AXIAL COMPRESSOR

Yang Chengwu
Key Laboratory of Light-duty Gas-turbine/Institute of Engineering Thermophysics, Chinese Academy of Sciences
Yangchengwu2015@163.com
Beijing, P. R. China

Zhao Shengfeng
Key Laboratory of Light-duty Gas-turbine/Institute of Engineering Thermophysics, Chinese Academy of Sciences
zhaoshengfeng@iet.cn
Beijing, P. R. China

Han Ge
Key Laboratory of Light-duty Gas-turbine/Institute of Engineering Thermophysics, Chinese Academy of Sciences
Hange898525@sina.com
Beijing, P. R. China

Lu Xingen
Key Laboratory of Light-duty Gas-turbine/Institute of Engineering Thermophysics, Chinese Academy of Sciences
xingenlu@hotmail.com
Beijing, P. R. China

Zhang Yanfeng
Key Laboratory of Light-duty Gas-turbine/Institute of Engineering Thermophysics, Chinese Academy of Sciences
zhangyf@iet.cn
Beijing, P. R. China

ABSTRACT
A 2.5-stage axial compressor, which is representative of compressors for small size aero-engines, at average work coefficient of 0.42, has been designed and tested in Institute of Engineering Thermophysics, Chinese Academy of Sciences. The aim of this work is to address the design methodologies, test techniques and the flow characteristics in highly loaded axial compressors, and to give a brief overview for a student or new engineer to the concept of compressor design and analysis. The design relied heavily on CFD techniques while minimizing conventional empirical design methods. Typical compressor design methodologies, such as root-enhanced, large reaction, low aspect ratio, multiple circular arc profile (MCA), dihedral and cantilevered stator were employed aiming at improving the compressor performances. Especially, the dihedral and cantilevered stator was studied in detail. Three-Dimensional (3-D) CFD analyses were performed to validate the design methodologies and explore the flow and loss mechanisms. Results showed that the experimental data agree well with predicted results, and the compressor achieves the design pressure ratio of 2.7 and the peak adiabatic efficiency at design speed of 87.2%.

INTRODUCTION
In order to increase the cycle efficiency, modern gas turbines are being designed with high stage pressure as a means of achieving higher overall pressure ratio. This can be achieved by either increasing the blade speed or increasing the stage loading (i.e., enthalpy) coefficient. Once the blade speed is limited by mechanical constraints, then higher pressure ratio can only be achieved by increasing stage loading. However, the associated higher adverse pressure gradient on suction surface of the profiles increases the possibility of a two-dimensional laminar boundary layer separation, tip clearance leakage flow loss, the development of three-dimensional hub corner stall and radial flow migration. All of these could increase the flow blockage, reduce the de-swirling ability of the blade and increase overall loss, thus limit compressor efficiency and stall margin [1]-[5].

The performance of axial compressors for small aero-engines decreases more remarkable as loading increases due to so called “Size Effect”, which raises challenges for designing highly loaded axial compressors. In order to study the flow and loss mechanisms in small size axial compressors at high flow turning conditions, a 2.5-stage axial compressor with average work coefficient of 0.42 has been designed and measured in Institute of Engineering Thermophysics, Chinese Academy of Sciences. The work coefficient is defined by Equation 1.

$$\Psi = \frac{\Delta H}{U_{tip}^2}$$  \hspace{1cm} (1)

In current paper, the aerodynamic design process, methodologies and principles were described at first. In
particular, the effects of dihedral and cantilevered stator were analyzed in detail. Then the 3-D CFD analyses were carried out to validate the performance and show the detailed flow field. Finally, the test rig was introduced and comparisons between the CFD and test results were performed.

AERODYNAMIC DESIGN

Design Objectives and Process
The compressor was designed to equip as a research device for axial compressors in small size aero-engines. The flow rate of T700 engine was referenced at first, but it was reduced to 4.6 kg/sec for consideration of the motor power. The average stage pressure ratio reaches 1.64 which results in the work coefficient up to 0.42. Constant shroud diameter was applied to keep low Diffusion Factor and make measurements conveniently. At design point, the rotor operates at a speed of 25000 rpm. Key aerodynamic and geometrical parameters for the compressor were summarized in Table 1.

Table 1 Design Specifications of the Compressor

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design mass flow rate (kg/sec)</td>
<td>4.6</td>
</tr>
<tr>
<td>Design rotational speed (rpm)</td>
<td>25000</td>
</tr>
<tr>
<td>Design pressure ratio</td>
<td>2.7</td>
</tr>
<tr>
<td>Design isentropic efficiency</td>
<td>0.865</td>
</tr>
<tr>
<td>Average work coefficient</td>
<td>0.42</td>
</tr>
<tr>
<td>Inlet total pressure (Pa)</td>
<td>101325</td>
</tr>
<tr>
<td>Inlet total temperature (degree)</td>
<td>15</td>
</tr>
<tr>
<td>Number of stage</td>
<td>2</td>
</tr>
</tbody>
</table>

Given the objectives and geometry constraints, it was apparent a design tool that automated geometry generation and CFD analysis should be required. Siemens NX8.0 was used as 3-D blade generation tool and the commercial software FINE/Turbo 9.0 was used as 3-D CFD code. The resultant design flow chart is shown in Figure 1.

As is known to all, the loading distribution along streamwise is very important for multi-stage axial compressors. In transonic axial compressors, high temperature at rear stages inlet makes the blade to do work more difficult than that of the front stages, so the rear stages usually are allocated less loading. In the current 2.5-stage compressor, the work coefficient was chosen to be 0.45 and 0.39 for the first and rear stage and thus the corresponding total pressure ratio is 1.72 and 1.57, respectively.

Once the stage loading has been specified, the rotor inlet flow angle is mainly determined by the reaction. Cumpsty[6] recommended an optimum reaction of 0.5 for conventional compressors. However, Dickens and Day [7] obtained other conclusions by CFD calculations for highly loaded compressors. Their work indicated that increasing the reaction relieves the stator whilst increases the aerodynamic demand on the rotor. However, the rotor tolerates the low de Haller numbers much better than the stators and the resultant losses in the rotor increases more slowly than in the stator as the de Haller number decreases. They summarized that it was necessary to increase the reaction to achieve optimum efficiency with increasing stage loading. Given the high loading of the compressor, the reaction for the first stage and rear stage was set to 0.65 and 0.6 originally. The inlet guide vane (IGV) was applied which would be used as variable guide vane in future. In the paper, the IGV was designed with fixed vanes.

As described above, the shroud size should be as large as possible to make measurements conveniently. This resulted in small flow coefficient, which would lead to low efficiency and narrow stall margin. The inlet flow coefficient was chosen to be 0.45 after weighting the advantages and disadvantages. The calculated shroud diameter is 276 mm and the hub-to-tip ratio is 0.72 which is slightly higher than the NASA Rotor 37 of 0.71. The flowpath height at the second stator exit was estimated by Mach number, which is about 0.3 in general. The meanline vector diagrams, which were calculated by the allocated loading and the flowpath height, were depicted in Figure 2.

Stage1, $\alpha_1 = 9.25$ degree $\alpha_2 = 58.62$ degree
Stage2, $\alpha_1 = 18.9$ degree $\alpha_2 = 59.45$ degree

Figure 2 Vector Diagrams on Meanline Section

The flowpath length mainly depends on the aspect ratio once the height has been determined. Low aspect ratio was applied for the designed compressor to expand the stall
margin in high loading condition. The pitch aspect ratio was selected to be 1.2 and 1.1 for the first and second rotor respectively.

In the spanwise, the tip clearance leakage loss would increase rapidly as pressure difference grows between pressure and suction sides due to increasing load. Thus, it was better to decrease the load of tip region and add the load to blade root. Total pressure ratio distribution along span for the compressor is depicted in Figure 3. The final flowpath of the compressor is shown in Figure 4.

![Figure 3 Pressure Ratio Distribution along Span at Rotors Exit](image)

**Figure 3 Pressure Ratio Distribution along Span at Rotors Exit**

**Figure 4 Flowpath Chart of the Compressor**

The coupled Euler/boundary layer code MISES [8] was used to set the blade metal angles and the solidity. The exit metal angles were chosen to keep work input distribution depicted in Figure 3. The leading edge metal angle was iteratively adjusted to give appropriate incidence (i.e., -1 degree). The solidity was primarily selected by considering the midspan section and was selected to give a predicted suction surface boundary layer with a shape factor between 2.1 and 2.3 at the trailing edge, i.e., just prior to the onset of separation. This approach effectively kept the midspan local diffusion factor constant. These values and other design parameters used in the design are summarized in Table 2. The 3-D blades and vanes were generated by using NX8.0.

### Table 2 Design Parameters of Blade Rows

<table>
<thead>
<tr>
<th>Rows</th>
<th>IGV</th>
<th>Rotor1</th>
<th>Stator1</th>
<th>Rotor2</th>
<th>Stator2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blade Number</td>
<td>31</td>
<td>36</td>
<td>59</td>
<td>48</td>
<td>69</td>
</tr>
<tr>
<td>Mean Chord Length (mm)</td>
<td>22.8</td>
<td>31.8</td>
<td>24.5</td>
<td>23.7</td>
<td>21.2</td>
</tr>
<tr>
<td>Mean Camber (degree)</td>
<td>32.5</td>
<td>47.1</td>
<td>30.8</td>
<td>54.1</td>
<td></td>
</tr>
<tr>
<td>Solidity at Tip</td>
<td>0.93</td>
<td>1.43</td>
<td>1.94</td>
<td>1.535</td>
<td>1.91</td>
</tr>
<tr>
<td>Clearance (mm)</td>
<td>0</td>
<td>0.2</td>
<td>0.15</td>
<td>0.2</td>
<td>0</td>
</tr>
</tbody>
</table>

The commercial software FINE/Turbo 9.0 was used for numerical simulation. The computational grid of the compressor model is too large to describe in short. Due to similar grid topology for each blade row, the grid topology of the first rotor was depicted in detail as instance, shown in Figure 5. Total computational model for a single passage of the rotor consisted of 540,000 grid points, approximately. There were 51 nodes in pitchwise direction and 119 nodes in streamwise direction. The type of O topology was used to improve the orthogonality of the meshes around blade surface and H topology for other blocks. 17 nodes in clearance region and 73 total nodes across the spanwise were chosen. The grid was refined in the near-wall regions and at the leading and trailing edge. The value of y+ associated with the first node off the wall was limited to 3. As a result, the total computational grid of the compressor consisted of 2,260,000 approximately.

![Figure 5 Grid for Blade-to-Blade Section View of the First Rotor](image)

**Figure 5 Grid for Blade-to-Blade Section View of the First Rotor**

3-D calculations were performed to assess the performance and the flow patterns in blade rows. It was found that large-scale flow separation appears in the Effects of Dihedral on the Highly Loaded Stators

Three-dimensional Reynolds-averaged Navier-Stokes (RANS) procedures were applied to obtain the performance and flow fields. The Spalart-Allmaras model was used for turbulence closure. The spatial treatment of the equations was performed using Jameson’s center scheme. Time integration was performed using a four steps Runge-Kutta scheme. Scalar eigenvalue-based second and fourth order difference smoothing operators were used to stabilize the time-marching scheme. To speed up convergence to steady state, local time stepping, residual smoothing and multi-grid techniques were applied.

The ideal gas was considered as working fluid. Uniform standard total pressure and temperature with axial flow direction were applied as inlet boundary conditions. Average static pressure was applied as outlet condition. Mixing plane method and frozen rotor method was used as the inter-stage interface and stage interface respectively.

**Effects of Dihedral on the Highly Loaded Stators**

3-D calculations were performed to assess the performance and the flow patterns in blade rows. It was found that large-scale flow separation appears in the
hub/suction surface and tip/suction surface corner regions for the stator vanes with conventional linear stacking lines due to high loading.

Considerable researchers [9-11] demonstrated that positive dihedral could delay onset of the corner separation/stall by unloading near the endwall and overloading around midspan. They also clarified that overloading around the midspan could increase the airfoil loss. In general, the shape of stacking line could be different from one to another due to various blade profiles, so there was a balance between loss and corner separation. The second stator with dihedral was studied at first and the shape of stacking lines was plotted in Figure 6. In the figure, the stacking line of Case 1 is almost linear. For Case 3, the dihedral angle, which was defined by Sasaki [10], is about +8 degree.

**Figure 6 Shapes of the Studied Stacking Lines**

Figure 7 shows the radial distribution of the total pressure loss and normalized flow rate. In the plots, the $CP^1$ is defined by Equation 2 and the normalized flow rate is expressed by Equation 3. As shown in the figure, the loss decreases evidently in the regions below midspan as positive angle increases while the loss increases slightly in the regions above 50% span. Mass flow increases in the regions below 30% span but decreases rapidly in the regions above. It indicates that the flow in the regions where positive dihedral was applied improves. Then why the parameters change as the positive dihedral angle increases? The Mach number distribution on trailing edge cross section depicted in Figure 8 could answer the question.

\[
CP^1 = \frac{P_{tin} - P_t}{P_{tin}} \quad (2)
\]

\[
\text{Normalized Flow Rate} = \frac{\text{Mass Flow Rate}}{\text{Mass Flow Rate at Midspan}} \quad (3)
\]

**Figure 7 Parameters Distribution across Span for the Second Stator at Varying Stacking Lines**

In the figure, there is large scale separation region in the suction/hub corner for Case 1. The separation region almost occupies the whole pitchwise below midspan. The separation region shrinks as the positive dihedral angle increases. There are still low speed cells in the suction/hub corner but the separation nearly disappears for Case 3 and velocity is more homogenous for the case. However, low momentum cells expand in the regions above midspan. In particular, low momentum cells in near shroud regions increases more evidently and tends to separate from suction surface. The comparisons indicate that positive dihedral near hub delays onset of the hub corner separation/stall by unloading near hub regions and overloading above the midspan.

The stacking line which is shaped between Case 2 and Case 3 was chosen after integrated tradeoff for the flow pattern in near hub and casing regions. The first stator was also optimized using dihedral according to the previous analyses. However, the positive dihedral angle was small (about 4 degree) in order to achieve good match with the second rotor, so there was still small corner separation near hub region which may decrease the stall margin. In consideration of this, cantilevered stator was intended to be applied for the stator for mitigating the corner separation.

**Figure 8 Contours and Isolines of Mach number on Exit Cross Section at Varying Stacking Lines**

**Effects of Hub Clearance on the Highly Loaded Stator**

Doukelis et al. [12] indicated that increase of clearance size resulted in more penalties in clearance region while it affected the flow in entire passage. Gbadebo et al. [13] concluded that the 3-D separations on the blade suction surface were largely removed by clearance flow for clearance about 0.58% of chord. Despite such efforts, interaction mechanisms of the hub clearance flow with endwall boundary layers remains unclearly, especially in high flow turning or high loading conditions.

The hub clearance size at 0%, 0.75%, 1.5%, 2.25% and 3% of the first stator hub chordlength were considered. Adiabatic efficiency curves were plotted in Figure 9. Note that near stall point was just the numerical unstable operating point in which the global residual was larger than 1×10^{-4} or the static pressure fluctuates periodically at certain stations.

It can be seen that the first stage reveals much higher efficiency for the stator with zero clearance which is expressed by 0% chordlength clearance. The peak efficiency decreases about 0.5% as clearance size varies from 0 to 0.75% chordlength but the normalized mass flow at near stall point decreases about 2%. However, the efficiency and stall margin change slightly as clearance size grows from 0.75% to 3% chordlength. It could be concluded that hub clearance
results in more loss but strengthens the flow stability. It should be pointed out that stall margin expressed by mass flow rate was extended by 2% for the stator with hub clearance compared to the stator with zero clearance. Then how do these happen?

**Figure 9 Adiabatic Efficiency Map of the First Stage at Variable Hub Clearance Sizes**

Figure 10 shows the changes of leakage mass flow associated with the clearance sizes at peak efficiency point and near stall point. The plots show that the leakage flow rate keeps linear relationship with clearance size. The flow rate also grows as the compressor operates toward heavy loading conditions, but the increment is much smaller compared to that caused by clearance size. In conclusion, the leakage mass flow rate mainly depends on clearance size.

**Figure 10 Leakage flow Passing Through Clearance Region at Varying Clearance Sizes**

Comparing Figures 9 with 10, it is clear that the loss increase and change of the stall margin could be attributed to increase of leakage mass flow as clearance size grows. It could be concluded that it was the leakage flow that affects the flow field in the stator passage and the negative effects increase as leakage flow rate grows with increasing clearance size.

In order to figure out the loss sources, total pressure loss coefficient distribution across span for the first stator at peak efficiency point was considered, as depicted in Figure 11. In the region of 0 to 15% span, the loss increases as hub clearance size grows from 0% to 3% chordlength and the difference becomes larger. In the region of 15% to 35% span, loss decreases as clearance size grows. In the region of 35% to 65% span, loss of stator with zero clearance is a little less compared with cantilevered stator and the loss increases slightly as clearance size grows. It indicates that the leakage flow improves the flow fields in 15% to 35% span region but deteriorates it in 0 to 15% span and 35% to 65% span region. However, increase of leakage flow rate has a little negligible effect on the flow in the region of 35% to 65% span. What is the flow mechanism behind the phenomenon?

**Figure 11 Total Pressure Loss Coefficient along Span for the First Stator**

In order to explore the influence of hub clearance leakage flow on the operating range or flow stability, the streamlines on suction surface and hub endwall are depicted in Figure 12. For all the cases, the stage is at the same flow rate with near stall point of the stator with zero clearance for all the cases.

**Figure 12 Streamline on Suction Surface and Hub at the Conditions of Near Stall Point for Shrouded Stator**

The pictures show that there is a large corner separation in the hub/suction corner from 30% chord to trailing edge for the stator with zero clearance and the corner separation disappears once hub clearance applied. For the cases of cantilevered stator, the streamline patterns are almost the same under the conditions, that’s why the stall margin keeps the same for the case of 0.75%, 1.5% and 3% chordlength. The separation line in middle and upper span regions locates at upstream compared to zero clearance stator but it almost locates at the same station as clearance size varies from 0.75% to 3% chordlength. All of these could be attributed to the traverse leakage flow moving from pressure surface to suction surface across the hub clearance.
In conclusion, the separation origin of the stator is pushed downstream by the clearance leakage flow passing through hub clearance region, thus the separation is more unlikely to happen on the suction surface. That was mainly responsible for the stall margin extending. In addition, the streamline patterns almost do not change with the clearance size or clearance leakage flow rate. The clearance equaling 0.75% hub chordlength was chosen as design clearance. 3-D numerical calculations were conducted after the optimizations.

3-D Prediction

Figure 13 shows the performance maps of 90%, 95% and 100% design speed and the design point. It can be seen from the plots that total pressure ratio and adiabatic efficiency was 2.75 and 0.868 respectively at design point, which achieve the design goals. In addition, the effects of fillets in manufacture were not modeled by calculations, so the calculated mass flow rate is slightly larger than design requirement. Peak efficiency locates at 90% design speed and it was about 0.88.

Figure 13 Calculated Compressor Performance Maps

The flow structure at design point was analyzed to assess rationality of the design. Figure 14 depicts Mach number distribution on the blade-to-blade sections of 2%, 50% and 98% span. Note that the IGV was omitted in the pictures for space reason. It can be seen from the 2% span section that flow in near hub regions are good and there is no obvious flow separation in all rows. Noteworthy, there are low momentum cells in middle pitch at the first stator trailing edge location, which may be caused by the hub clearance leakage flows depicted in Figure 12. There are also a few low momentum flow cells at the suction surface near the trailing edge, but the flow behaves well. The low momentum cells could be the cells depicted in Figure 8 which exists in the suction surface/hub corner.

Flow in all rows presents good at the midspan section, except that there is small scale reverse flow on suction surface near trailing edge for the first rotor. The reverse flow region was barely removed due to high Diffusion Factor, which reaches 0.68. Comparatively speaking, flow states in near tip regions are better and there are only a few low speed cells near pressure surface.

Figure 15 depicts radial distribution of pressure ratio for the first stage and compressor. The figure shows that pressure ratio decreases from hub to shroud, which is consistent with the original design input. It can be inferred that pressure ratio of the second stage presents the type of reversed “C”, which was higher in the middle region and lower near both ends.

In conclusion, the compressor achieves design goals of pressure rise and efficiency, and the flow is good.

EXPERIMENTAL TEST

Test Rig

The experimental data was acquired at the compressor test rig in Institute of Engineering Thermophysics, Chinese Academy of Sciences. The compressor is driven by an 800 kW AC motor and specified rotation speed of the motor is about 3000 rpm. The compressor and the motor are connected by a gear box with transmission ratio of 1:12. The flow rate is measured by a flow tube mounted in intake duct. The sketch of the rig is depicted in Figure 16.

The measurement stations of the compressor are given in Figure 17. In order to capture the mean values of each test station, the radial distribution probe rake were allocated circumferentially on each plane. Detailed message of the probes is summarized in Table 3. The photo of the compressor on test rig and the probes is given in Figure 18.
Table 3 Distribution of Probes

<table>
<thead>
<tr>
<th>Station</th>
<th>Probes type</th>
<th>Number</th>
<th>accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>I - I</td>
<td>5-point total pressure</td>
<td>3</td>
<td>±0.2% FS</td>
</tr>
<tr>
<td></td>
<td>3-point thermocouple</td>
<td>3</td>
<td>±0.5degree</td>
</tr>
<tr>
<td></td>
<td>Static pressure</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>L1-L1</td>
<td>static pressure</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>II - II</td>
<td>3-point total pressure</td>
<td>6</td>
<td>±0.2% FS</td>
</tr>
<tr>
<td></td>
<td>3-point thermocouple</td>
<td>6</td>
<td>±0.8degree</td>
</tr>
<tr>
<td></td>
<td>Static pressure</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>L2-L2</td>
<td>static pressure</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>L3-L3</td>
<td>static pressure</td>
<td>4</td>
<td></td>
</tr>
<tr>
<td>III - III</td>
<td>2-point total pressure</td>
<td>6</td>
<td>±0.2% FS</td>
</tr>
<tr>
<td></td>
<td>2-point thermocouple</td>
<td>6</td>
<td>±0.8degree</td>
</tr>
<tr>
<td></td>
<td>Static pressure</td>
<td>4</td>
<td></td>
</tr>
</tbody>
</table>

Performance Maps

The calculated and measured performance maps at different rotating speed is depicted in Figure 19. It can be seen from the test performance that the adiabatic efficiency is about 0.866 and total pressure ratio is about 2.73 at design point. The comprehensive stall margin, which is defined by Equation 4, is about 15.3%. The total pressure ratio and stall margin both achieve the design goals. Test efficiency is slightly lower than design requirement. In addition, peak efficiency at design speed is 0.872 or expressed by 87.2%.

\[
\text{Stall Margin} = \left( \frac{m_{\text{out}}}{m_{\text{in}}} \right) - 1
\]  

Note that, compared to calculated data, measured mass flow rate is slightly larger for all speed lines and the value is about 0.06 kg/s. The value amounts to 1.3% of the design flow rate, which is within the margin of test error.

\[
\eta = T_{\text{in}}(\pi^{k-1} - 1)/(T_{\text{ex}} - T_{\text{in}}')
\]

Figure 19 Calculated and Measured Performance at Constant Speed Lines

At part speeds, the measured efficiency is higher than the calculated efficiency at 60% and 70% design speeds and the difference decreases as speed increases. The difference disappears when the speed exceeds 80% of design speed. These may be attributed to the calculation formula of the efficiency, which is defined by Equation 5. The temperature test error is large due to the small value at low speeds, so the calculation efficiency error is also large. Moreover, the measured and calculated efficiency reaches the peak value of 0.88 at 90% design speed. The calculated and measured pressure ratio and stall margin matches well for all speed. It should be pointed out that the calculated stall flow at 95% design speed is much smaller, which may be attributed to wrong judgment to convergence or numerical error.
comprehensive stall margin was 15.3% and peak efficiency is 87.2% at design speed. All of these achieve design goals.

**Parameters Distribution**

The radial distribution of the total pressure and temperature was tested on station $\Pi - \Pi$ by moving the probes radially. Total pressure ratio and adiabatic efficiency distribution at the station are depicted in Figure 20. It can be seen from the figure that there is evident deviation between the calculated and measured results, but the largest deviation was only about 2.2%. The trend of total pressure ratio and adiabatic efficiency agree well. Note that, parameters near endwall were not measured due to limited stroke of the displacement mechanism.

![Figure 20 Calculated and measured parameters distribute across spanwise at near peak efficiency point](image)

**CONCLUSIONS**

From the research described and discussed in this paper the following conclusions were drawn:

- At design point, the pressure ratio and the adiabatic efficiency is 2.73 and 0.866 respectively. The comprehensive stall margin is 15.3% and peak efficiency is up to 0.872 at design speed. All of these achieve the design goals.

- The flow in the compressor is perfect besides some small scale low momentum cells appear on suction surface at design point.

- The dihedral has significant effects on controlling the corner separation/stall. Positive dihedral could decrease the large separation region caused by high loading and so improve the compressor performance.

- Cantilevered stator could suppress the corner separation/stall but increase loss.

**NOMENCLATURE**

<table>
<thead>
<tr>
<th>CFD</th>
<th>computational fluid dynamics</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\psi$</td>
<td>work coefficient</td>
</tr>
<tr>
<td>$H$</td>
<td>enthalpy</td>
</tr>
<tr>
<td>$U$</td>
<td>rotor wheel speed</td>
</tr>
<tr>
<td>$V$</td>
<td>absolute velocity</td>
</tr>
<tr>
<td>$W$</td>
<td>relative velocity</td>
</tr>
<tr>
<td>LE</td>
<td>leading edge</td>
</tr>
<tr>
<td>3-D</td>
<td>three-dimensional</td>
</tr>
<tr>
<td>d</td>
<td>design</td>
</tr>
<tr>
<td>t</td>
<td>total</td>
</tr>
<tr>
<td>s</td>
<td>stall</td>
</tr>
<tr>
<td>SS</td>
<td>suction surface</td>
</tr>
<tr>
<td>FS</td>
<td>full scale</td>
</tr>
<tr>
<td>$\pi$</td>
<td>total pressure ratio</td>
</tr>
<tr>
<td>in</td>
<td>inlet</td>
</tr>
<tr>
<td>ex</td>
<td>exit</td>
</tr>
</tbody>
</table>

**REFERENCES**


**ACKNOWLEDGMENTS**

The authors would like to express appreciation for the support of the National Natural Science Foundation of China [Project Number=5160060167].