Effect of Endwall Contouring on Flow Instability of Transonic Compressor
Yoichi Ito, Toshinori Watanabe and Takehiro Himeno

Department of Aeronautics and Astronautics
The University of Tokyo
7-3-1 Hongo, Bunkyo-ku, Tokyo 113-8656, JAPAN
Phone: +81-3-5841-6624, FAX: +81-3-5841-6622, E-mail: watanabe@aero.t.u-tokyo.ac.jp

ABSTRACT
Numerical study was carried out on a transonic compressor rotor, NASA Rotor 37, with different configurations of circumferentially contoured endwalls to understand the fundamental flow mechanisms of casing treatment effect. The result showed the wall contouring above the blade leading edge had significant stall margin improvement and insignificant efficiency and pressure ratio drops. By applying the second invariant Q, two major vortex structures were observed near the stall conditions. One was tip leakage vortex and the other was rotating instability vortex. The rotating instability vortex seemed to be induced by the interaction between the inflow and reverse flow near the endwall. In the case with wall contouring, enlarged clearance gap encouraged the tip clearance flow and relieved the interaction that caused the rotating instability vortex. Alleviation of the vortex led to the improvement of stall margin as a consequence.

MODEL COMPRESSOR
NASA Rotor 37 was adopted as the model rotor. The rotor was originally designed as an inlet rotor for a core compressor at NASA Lewis Research Center in the late 1970's. The design parameters of the rotor are summarized in Table 1. The design pressure ratio is 2.106 at the mass flow rate of 20.19 kg/s. The inlet relative Mach number is 1.13 at the hub and 1.48 at the tip, respectively, at the design speed of 454 m/s (17,188.7rpm). The rotor has 36 blades and the tip clearance is 0.400 mm. The details are reported by Reid and Moor (1978), Moor and Reid (1980).

Figure 1 (Suder and Celestina, 1994) shows the measuring stations for probes and laser anemometer systems on which static pressure, static temperature and velocity distributions were obtained.

Table 1 Design overall parameters for NASA rotor37

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of rotor blades</td>
<td>36</td>
</tr>
<tr>
<td>Tip solidity</td>
<td>1.288</td>
</tr>
<tr>
<td>Rotor inlet hub-to-tip diameter ratio</td>
<td>0.7</td>
</tr>
<tr>
<td>Rotor blade aspect ratio</td>
<td>1.19</td>
</tr>
<tr>
<td>Rotor tip relative inlet Mach number</td>
<td>1.48</td>
</tr>
<tr>
<td>Rotor hub relative inlet Mach number</td>
<td>1.13</td>
</tr>
<tr>
<td>Mass flow rate (kg/s)</td>
<td>20.93</td>
</tr>
<tr>
<td>Design wheel speed (rad/s)</td>
<td>1800</td>
</tr>
<tr>
<td>Tip speed (m/s)</td>
<td>454.136</td>
</tr>
<tr>
<td>Rotor total pressure ratio</td>
<td>2.106</td>
</tr>
<tr>
<td>Rotor adiabatic efficiency</td>
<td>0.877</td>
</tr>
<tr>
<td>Nominal Tip Clearance (mm)</td>
<td>0.356</td>
</tr>
</tbody>
</table>

Fig. 1 NASA Rotor 37 experimental and numerical measuring points (Suder and Celestina, 1994)
NUMERICAL ANALYSIS METHOD

Numerical Scheme
Steady three-dimensional flow simulations were performed with compressible Navier-Stokes equations. The three-dimensional Reynolds-averaged Navier-Stokes equations were discretized in the computational domain by a cell centered finite volume method. Time integration was accomplished by the Euler implicit method with the LU-SGS scheme. The inviscid fluxes were evaluated by the SHUS scheme with the third order MUSCL interpolations. The viscous fluxes were calculated in the central differential manner. The standard two-equation k-ε turbulence model was adopted to evaluate the eddy viscosity.

Computational Grid
The computational grid system used in the present study is shown in Fig. 2 and Fig. 3. The grid system was consisted of a structured H-grid in the main flow region and O- and H- grids in the tip clearance region. The main grid was composed of 124 cells in the streamwise direction (52 cells on the blade), 66 cells in the pitchwise direction, and 86 cells in the spanwise direction. The O- and H- grids embedded in the tip clearance region had 16 cells in the spanwise direction. The total number of cells is 718,512. The minimum grid spacing on the solid walls was 5.0x10^-6 [m], which gave y^+<1 on the walls.

Boundary Conditions and Definition of Stall Point
At the inlet boundary, total pressure and flow angle were prescribed according to the experimental data (Suder and Celestina, 1994). Total temperature was fixed at 288 K and Riemann invariant was extrapolated from the adjacent inner cells. No-slip and adiabatic conditions were imposed on the hub wall, casing and blade surfaces. At the exit boundary, the static pressure was specified. The interface between two adjacent flow passages was treated as a periodic boundary.

Identification of Vortex Structures
Vortex structure is a very important issue to understand complicated flow fields of turbo machines. However, it is difficult to identify the structures of vortices only with conventional visualization techniques based on, for instance, streamlines or pressure contours. Sawada (1995) and Jeong et al (1995) reported the method to identify a trajectory of the vortex core by using tetrahedral cells. Yamada et al (2003) showed a vortex breakdown near the stall condition with help of the normalized helicity, which shows the magnitude of streamwise vortex.

Hunt et al. (1988) defined an “eddy” as the region with positive second invariant, Q, of ∇u, with the additional condition that the pressure in the region should be lower than the ambient pressure. The second invariant Q is defined as

\[ Q = \frac{1}{2} \left( u_{ij}^2 - u_{ij} u_{ji} \right) = \frac{1}{2} \left( \| S \|^2 - \| \Omega \|^2 \right) \]  

where \( S = [r(SS')]^{1/2}, \| S \| = [r(\Omega^2)]^{1/2}, \| \Omega \| = [r(\Omega^2)]^{1/2} \), and \( S \) and \( \Omega \) are the symmetric and the antisymmetric components of ∇u, respectively. Thus, Q represents the local balance between shear strain rate and vorticity magnitude.

In this study, the second invariant Q was adopted to identify the vortex structures. The second invariant Q can indicate vortex structures in any direction. On the other hand, the normalized helicity identifies the vortex only in streamwise direction. The second invariant Q is, hence, appropriate measure in the present study because various kinds of vortex structures need to be clarified at near stall condition.

Validity of Numerical Simulation
The developed numerical code was validated through comparison of computed total pressure and temperature distributions with the corresponding experimental data obtained in the test rig of rotor-only configuration. CFD simulations in a workshop by AGARD (Dunham, 1998) were also referred for the validation.

Figure 4 shows the total pressure ratio characteristics. The flow rate was normalized by the choking mass flow in the experiment by Suder and Celestina (1994). The predicted result in the CFD gave slightly higher total pressure ratio near the stall condition compared with the experimental one. Figure 5 shows the spanwise distributions of total pressure and temperature at Stn 4 in Fig. 1 at 98% normalized mass flow. The computational prediction showed...
a little higher total temperature near the tip and hub. It was
probably because of the strict adiabatic boundary condition on the
walls in the numerical simulation. However, the simulation results
are judged to show a satisfactory agreement with the experimental
ones in comparison with other CFD simulations. (See Dunham
(1998) for details of the simulations in the workshop.)

Fig. 6 Examples of contoured endwall configurations

The results of the validation were used as the baseline solutions
for the analysis of the contoured endwall flows as well.

RESULTS AND DISCUSSIONS

Parametric Study on Endwall Configurations

Various forms of contoured endwall were tested to investigate
the effect of contouring on the flow field. Figure 6 shows the
examples of configurations and Table 2 shows the summery of the
endwall configurations.

Type B1, Type B2 and Type B3 have the same size and shape of
endwall contouring, but located at different positions as shown in
Table 2. Type B1 has the contoured part on the endwall from
leading edge position to 30% chord length position. Type B2 is
contoured from 20% to 50% chord length, and Type B3 is
contoured from 35% to 65% chord length position. The
computational results of total pressure ratio and efficiency are
shown in Fig. 7 for those three cases as well as the baseline case.
The efficiency was defined as follows:

Rotor adiabatic (temperature rise) efficiency

\[
\eta_{ad} = \left( \frac{P_4}{P_0} \right)^{\gamma-1} \gamma - 1
\]

(2)

where \( P_0 \) and \( T_0 \) are averaged stagnation pressure and temperature,
at the inlet, respectively, and \( P_4 \) and \( T_4 \) are those at the outlet,
respectively.

Though all cases with contoured wall showed deterioration in peak
efficiency and pressure ratio compared with the baseline,
Type B1 showed relatively large stall margin compared with the
other two cases. It should be noticed that the decrease in the peak
efficiency and pressure ratio in Type B1 is less than that of Type
B2 and Type B3.
Since the contouring near the blade leading edge seemed to have positive effect on the stall margin in the previous discussion, further flow simulations were performed on the contouring just above the blade leading edge noted as Type C1, C2 and C3 (see Fig. 6 and Table 2). Figure 8 shows the results of total pressure ratio and efficiency characteristics of the three cases with wall contouring. As shown in Fig. 8, noticeable improvement in the stall margin with small deficit in the peak efficiency and total pressure ratio is clearly obtained with any types of contouring compared with the baseline case. It is thought that these types of endwall contouring behave just like circumferential casing treatments. The stall margin improvement is not linear to the depth of the contouring, and thus the shallow contouring might be enough for stall margin improvement. In the following sections, Type C2 will be adopted as the typical example of wall contouring configuration to investigate the flow mechanism of the stall margin improvement based on a close look at the flow field.

**Detailed Flow Fields with/without Wall Contouring**

From the simulation results, the difference in flow characteristics was thoroughly studied between the baseline and the case with contouring Type C2. Figures 9(a) and (b) show the streamlines and the contours of second invariant $Q$ for the baseline configuration at near peak efficiency and near stall condition, respectively. The $Q$ contours are plotted inside the identified vortices based on $Q$ criteria.

The tip leakage vortex can be clearly detected from the streamlines in Fig. 9(a). The magnitude of the second invariant $Q$ in the same figure also indicates the corresponding vortex structure. At near stall condition, however, the vortex structures became more complicated as shown in Fig. 9(b).

The three-dimensional view of the constant $Q$ surface gives more comprehensive image of the vortex structures. A typical result is shown in Fig. 10. The figure indicates a contour diagram of the second invariant $Q$, showing vortical structures in the flow field near the blade tip. At the near stall condition, two large vortex structures are observed in the $Q$ diagram; one is leakage vortex marked as A in Fig. 10, and the other is endwall vortex marked as B in the same figure.

Figure 11 shows velocity vector and second invariant $Q$ diagram near the casing wall. Vortex B identified by high magnitude of $Q$ in the figure seemed to be caused by the interaction between the incoming flow and the stagnating flow near the casing. Though the flow mechanism through which the stagnant region is generated is not clear yet, the region is thought to have close relationship with the interaction between the leading edge shock wave and the tip leakage vortex as is discussed in literature (e.g. Yamada, et al. (2003), Marz, et al. (2002)). The structure of the vortex B was similar to the rotating instability vortex reported by Marz et al. (2002).
In the past, it was thought that instability arose when the tip clearance flow spilled into the adjacent blade passage from the pressure side at the leading edge. However, according to Marz et al. (2002), the tip clearance flow does not spill into the next blade passage at the leading edge, and the flow follows the pressure side of the leading edge. Consequently, the tip clearance flow, the axially reversed flow near the casing and the incoming flow interact each other near the leading edge to form a distinctive vortex structure. The low velocity region just downstream of the interaction between the leakage vortex and the shock is also thought to be one of the factors to form the rotating instability vortex in transonic flows (Mailach et al., 2001).

The compressor operates in a stable mode even when rotating instability vortex is observed. This is because the tip clearance flow does not spill over into the adjacent blade passage. However, when the flow rate is further reduced and the rotating instability vortex moves upstream of the leading edge, the flow spills into the next blade passage and the compressor flow eventually becomes unstable.

In the case with wall contouring, strong leakage flow is induced through the relatively large tip clearance gap at the contoured region. Hence some part of the incoming flow that supposed to pass along the pressure side of the blade tip flows into the tip clearance as well. Interaction between the incoming flow and the stagnating flow near the endwall is relieved due to the reduction of the incoming flow and finally the vortex B is thought to become smaller. Figure 12 shows a close-up view of the calculated velocity vectors and the second invariant \( Q \) inside the tip clearance with and without contouring. In the baseline result,
(1) The endwall contouring only above the blade leading edge region can substantially improve the stall margin.

(2) At the near stall conditions, two major vortex structures are observed in the flow field near blade tip. One of these vortices, what is called “rotating instability vortex”, seems to have a governing role of rotating stall instability.

(3) Contouring above the blade leading edge encourages the leakage flow and removes airflow that passes along the pressure side of the tip. The interaction between the incoming flow and the stagnating flow near the endwall is relieved and the rotating instability vortex is weakened. This is the reason why the significant improvement in the stall margin is possible with the help of casing treatment. The role of rotating instability vortex in rotating stall onset should be further studied.

ACKNOWLEDGMENT

The authors would like to greatly appreciate the support to the present research from Ishikawajima-Harima Heavy Industries Co., Ltd.

REFERENCES


