A Numerical Study on the Treatment Performance of Bent-grooved Casing Based on Free Deformation Method Using Computer Data Analysis Technique

Dun Ba¹, Chenguang Yuan², Juan Du¹*, Hongwu Zhang¹

¹Key Laboratory of Advanced Energy and Power, Institute of Engineering Thermophysics, Chinese Academy of Sciences, 11 Beisihuanxi Road, Beijing, China, 100190
²Beihang University, China

*Corresponding author e-mail: 53096131@qq.com

Abstract. In this study, inspired by the limit streamlines on the compressor casing. Combined with computer data technology to reduce the efficiency loss caused by the increase of casing processing.

Keywords: Bended-slot Casing Treatment, Free-form Deformation Method, Efficiency Penalty

1. Introduction
To achieve better performances, stricter limitations on weight and size have been put forward for aero-engines. Consequently, higher single-stage pressure ratio is required in advanced compressors, resulting in serious instability problems such as surge and rotating stall at off-design points. Thus, stabilizing methods need to be considered in the design process. Casing treatment (CT) has been proposed as one of the possible solutions since 1960s, leading to a great amount of studies on various geometry configurations of CT. In previous studies, stability increment is measured as the main objective of designing CT, and efficiency penalty is also evaluated to decide whether the design of CT is acceptable.

2. Compressor model
The compressor model discussed in this paper is the compact mixed-flow compressor from Leibniz University of Hannover, which is part of an electrically-powered active high-lift system for future civil aircraft. Severe instability occurs at an off-design speed of 30,000rpm in the original test, where the SM is 3.2% as shown in the performance map. In this paper, research on the performance of the compressor model will be calculated under the same rotating speed.

3. Design of bended-slot casing treatment based on FFD method
It is necessary to design a configuration of CT with negligible efficiency penalty while achieving a large stall margin improvement (SMI). According to Joshua D.et al., leakage fluid generated significant entropy, as an indication of the efficiency loss. The CT bent along the direction of the end
wall limit streamline is supposed to suck the leakage fluid without extra entropy generation, thus causing little efficiency loss. The end wall limit streamline of the compressor with solid casing at near stall (NS) condition is shown in Figure 1. In order to build the required geometry, it is convenient to conduct the FFD based on an effective CT design derived from a previous study in the research group.[2]

Figure 1. End wall limit streamline in the rotor (solid casing)

As observed in Figure 2 (a), the previous CT is axially staggered. The deformation is conducted to maintain the same CT design at the front part, where the slots are not overlapped with the rotor[3]. For the rear part, the slot is designed to be bent along the end wall limit streamline as shown in Figure 2 (b).

Figure 2. Schematic diagram of the interface between the axial-slot CT and bended-slot CT

To achieve the deformation mentioned above, the overlapped part needs to be rotated with an angle which is calculated by the leaning angle of streamline ($\theta$) as shown in Figure 2. The meridional profile of the previous CT configuration is first established by two B-splines, and then swept 60°circumferentially. The profile was then rotated along the rotor axis to form the body (pre-baseline) as shown in Figure 3 (a), and finally intersected by the casing to generate the baseline as shown in Figure 3 (b). With the intersection, the geometry of the baseline was well fitted to the casing, which avoided possible errors in the mesh generation process. In this case, the FFD method is employed on the pre-baseline at first, and then intersected by the casing to generate the final FFD design (Figure 3) for the same reason mentioned above[1].
In this study, PyGeM is used to establish and operate a rectangular-block control volume in order to modify surfaces of the baseline. PyGeM is a python library using Free Form Deformation, Radial Basis Functions and Inverse Distance Weighting to parameterize and morph complex geometries[4].

The axes of the local coordinates (X’Y’Z’) are parallel to the global axes. The control volume defined in the local coordinates is self-symmetrical to the plane YOZ. The number of control points and the length of control volume in Z’ direction are determined in order that the interface (blue shade in Figure 4) between the overlapped and the independent parts has 4 control points on its vertices. The control points with the same Z’ coordinate are connected with dash-dotted lines as shown in Figure 5, dividing the control volume into three layers. The two layers which control the overlapped part will be deformed. The parameters to initialize the volume are shown in Table 1.

![Figure 3. Pre-baseline and baseline](image)

![Figure 4. Control volume and the pre-baseline](image)

### Table 1. Initialization of the control volume

| Length of control volume in each direction (mm) | X’ | 28.000 |
| | Y’ | 9.500 |
| | Z’ | 30.940 |

| Global coordinates of O’ (mm) | X’ | -14.000 |
| | Y’ | 55.000 |
| | Z’ | -18.000 |

| Rotation angle of X’ Y’ Z’ relative to global axes (°) | X’ | 0 |
| | Y’ | 0 |
| | Z’ | 0 |
The global coordinates of the control points in the overlapped part are then calculated according to the rotation of coordinates. Finally, the modified geometry (pre-FFD design) is generated as shown in Figure 5 and intersected by the casing as shown in Figure 3 (b).

![Control volume and the pre-FFD design (after deformation)](image)

**Figure 5.** Control volume and the pre-FFD design (after deformation)

With the FFD method applied, the intended bended-slot casing treatment is generated. The influence of the CT design on the stability and efficiency of the compressor is then evaluated via CFD calculation[5].

4. **Numerical setup**

The mesh for the compressor main flow passage and CT is generated in Autogrid and IGG softwares, respectively. The flow passage consists of 1.67 million nodes and each CT consists of 0.34 million nodes with a butterfly topology grid. The minimal first grid spacing is set to 3e-6 ensuring $y^+$ less than 2 at solid walls. Further information including mesh independence study is available in Du et al. The CFD solver is ANSYS CFX, and three-dimensional Reynolds-averaged Navier-Stokes (RANS) is employed in steady simulations. The k-omega turbulence model is applied. Rotor and stator passages are connected with the mixing-plane method, whereas CTs are connected to the rotor passage by a frozen rotor mixing model. Solid walls were considered as Non-slip and adiabatic. Total pressure and inlet velocity directions are given at the inlet of the computational model. At large mass flow rate operating points, the mass flow rates is provided as the outlet boundary condition; at operating points near stall, the averaged static pressure is given at the outlet[6].

5. **Results and discussion**

5.1. **Overall performance**

The performance curves of the mixed-compressor with solid casing (SC) and casing treatment (CT) designed by FFD method are drawn in Figure 6. As shown in Figure 6, the characters $P_{ESC}$ and $N_{ESC}$ represent the peak efficiency point and the near-stall point for SC, respectively. The near-stall point means the last stable point where the computation still converges. In the following part, the aerodynamic analysis of how CT affects the compressor performance is mainly conducted at these two points. The stall margin improvement (SMI) and peak efficiency improvement (PEI) acquired by the employment of casing treatment are calculated according to equations and listed in Table 1. As shown in Tab. 4, the bended-slot CT designed by FFD achieved the SMI of 13.2% with the PEI of -0.1%, indicating that the bended-slot CT broadened the stall margin at a small cost of efficiency loss.
Figure 6. Total pressure ratio and polytropic efficiency curves as a function of mass flow rate for solid casing and casing treatment

\[
\text{SMI} = \left( \frac{\eta_{CT}}{\eta_{NSCT}} \right) \left( \frac{\eta_{SC}}{\eta_{NSSC}} \right) - 1 \times 100\% \quad (1)
\]

\[
\text{PEI} = \left( \frac{\eta_{PESC}}{\eta_{PESC} - \eta_{NSSC}} \right) \times 100\% \quad (2)
\]

5.2. Aerodynamic analysis

Figure 7 presents the streamlines colored with absolute velocity magnitude in the bended-slot casing treatment. As demonstrated in Figure 7, the flow near blade pressure side is sucked into the slots and then re-injected into the blade passage upstream of the rotor blade leading edge. Driven by the axial pressure gradient, the flow circulation formed in the slots removed the flow blockage and accelerates the flow in tip clearance region.

Figure 7. Streamlines in the bended-slot casing treatment at operating points PE_{SC} and NS_{SC}

6. Streamwise velocity contour

At point NS_{SC}, suction effect of the flow into CT alleviates blockage in the passage, which is indicated
by the streamwise velocity at rotor tip region. By suppressing the spillage of the interface between tip leakage flow and the main flow over the blade leading edge, the operating range of the investigated compressor is broadened and the total pressure ratio is raised.

7. Blockage
Suder discovered that the compressor stability is closely related to the blockage generated in the blade passage. Therefore, the blockage is calculated by Equation as below to understand how the bended-slot casing treatment reduces the flow blockage in the blade passage. The definitions of the symbols and edge criterion in Equation can be found in the study by Khalid et al. Figure 8 shows the calculated blockage at the passage outlet for SC and CT. One can see that the blockage percentage at the rotor outlet with CT is smaller than that with SC at point \( \text{NSSC} \). By reducing the blockage area, bended-slot CT broadens the stall margin of the compressor.

\[
A_b \equiv \iint (1 - \frac{\rho u_m}{\rho_{\text{edge}} u_{\text{edge}}}) \cdot dA
\]

(a) SC  
(b) CT

Figure 8. Blockage at rotor outlet at operating point \( \text{NSSC} \)

8. Stage entropy generation
The transportation of entropy in an open system with a finite control volume in a period of time is described as:

\[
dS_{CV} = \delta S_{f,m} + \delta S_{f,Q} + \delta S_g
\]

In which \( S_{CV} \) represents total entropy in a control volume, \( \delta S_{f,m} \) represents net entropy increment by mass passing through the control volume, \( \delta S_{f,Q} \) is the entropy exchange with the environment, and \( S_g \) is the entropy generated in the control volume. The thermal dynamic process in a compressor stage can be modeled as a steady open system with a finite control volume, which means a constant \( S_{CV} \) throughout time. Furthermore, \( \delta S_{f,Q} \) is also a constant due to the adiabatic condition. As a result, \( S_g \), as an indication of the total loss in the system, is only related to \( \delta S_{f,m} \) which can be calculated in CFD simulations, and the total loss is supposed to be in coordinate with the efficiency loss.

The analysis is conducted at \( \text{PESC} \) and \( \text{NSSC} \) points for both SC and CT cases. As \( \text{NSSC} \), though additional entropy generation was brought by slots, the entropy generation produced in the rotor and stator passages with CT is reduced, leading to a lower total \( S_g \) compared to the compressor with SC. This explains the higher efficiency with CT compared to SC at point \( \text{NSSC} \). At \( \text{PESC} \), the difference of \( S_g \) between CT and SC is not obvious, so does the difference of efficiency. Based on analysis above, it can be concluded that the bended-slot CT designed by FFD method broadens the SM with negligible efficiency penalty.

9. Conclusion
In this study, inspired by the limit streamlines on the compressor casing, a bended-slot casing treatment is proposed to minimize the efficiency penalty caused by the addition of casing treatment. The free-form deformation (FFD) method is used to construct the geometry of bended-slots based on a well-designed axial-slot casing treatment.

1. A bended-slot casing treatment is proposed based on the limit streamline distribution on compressor casing for the solid casing. The rear part of the slots is parallel to the end wall limit streamlines in the rotor passage in order to ensure that fluid near the casing flows into the slots smoothly, thus minimizing the shear loss between the suction flow and tip region flow. The front part of the slots is still staggered axially to make the injection flow into the blade passage parallel to the
incoming main flow. FFD method is effective in the modification of CT to provide three degrees of freedom with a few parameters. The rectangular-block control volume is easy to establish and adjust, and suitable for rotating a part of whole body by single axis.

2. The bended-slot CT designed by FFD method achieved the SMI of 13.2% with the PEI of -0.1%, indicating that the bended-slot CT broadened the stall margin at a small cost of efficiency loss. Similar to stability enhancement mechanism by CTs in other researches, for the compressor with the bended-slot CT, the flow near blade pressure side is sucked into the slots and then re-injected into the blade passage upstream of the rotor blade leading edge. Driven by the axial pressure gradient, the flow circulation formed in the slots removed the flow blockage and accelerates the flow in tip clearance region. As a result, the spillage of the interface between tip leakage flow and incoming main flow over the blade leading edge is suppressed, and the blockage area outlet is decreased, leading to a greater stall margin compared to the compressor with SC.

References
[1] Koch CC. Experimental evaluation of outer case blowing or bleeding of single stage axial flow compressor, Part VI-Final report. Washington, D.C.: NASA; 1970. Report No.: NASA CR-54592.
[2] Smith, G. D. J., and Cumpsty, N. A., “Flow Phenomena in Compressor Casing Treatment,” ASME JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER, Vol. 106, 1984, pp. 532-541.
[3] Liu Jing. Numerical simulation of mixed bend turbulence [J]; Journal of anshan university of science and technology; 04, 2006.
[4] Fang Yuanyuan, Guo Quan, Fu Zhongcheng. Numerical simulation of ejector in low-pressure atmospheric burner [J]; Journal of Beijing institute of architectural engineering; 04, 2006.
[5] Yu Haiqi, Li Huiqin, Wang Baofeng, Li Jianchao. Numerical simulation of three-dimensional flow field in the electromagnetic braking process of slab mould [J]; Journal of baotou iron & steel college; 2005, 03.
[6] Cao hongsong, Feng Shunshan, Zhang Hongfei, Li Guojie, Fan Xiao-ming. Study on numerical simulation method of the outer flow field of the parachute system [J]; Journal of testing technology; 05, 2006.