Analysis of jet suppressing root leakage in plane cascade

Meng Tang1 Xin Fu1 Ying Y Xu1
Nanjing University of Aeronautics and Astronautics, Nanjing, CHN
E_mail: fu_xin@nuaa.edu.cn

Abstract: The three-dimensional numerical simulation of the sealing structure of compressor cascade with jet structure is carried out to study the flow structure of the tooth cavity under the influence of jet. Based on this, the influence of jet on the flow performance in the tooth cavity was analyzed from four aspects of vorticity distribution, turbulent kinetic energy distribution, Mach number distribution and static pressure distribution. The results show that the interaction of the jet flow and the leakage flow makes the internal flow structure of the tooth cavity complicated and changeable. The two fluids are connected and mixed with each other and keep separating and converging, which makes the flow loss increase sharply and the leakage coefficient is greatly reduced. The jet flow drastically changes vorticity distribution and static pressure distribution in the tooth cavity. In the jet-affected section, the jet-affected range is approximately 2 times the circumferential width of the jet port.

1. Introduction
With the development of aero-engine technology, the pressure ratio of high-pressure compressors continues to increase. This means a further increase in clearance leakage, which will affect the performance of the entire engine. Sealing labyrinth is a non-contact sealing structure whose sealing efficiency is mainly determined by the radial clearance between rotor parts and stator parts and the number of labyrinth seals. Because of its simple structure, high sealing efficiency, suitable for harsh working environment and other characteristics, it is widely used in aircraft engine. After adopting new technology and new structure, it has broader application prospects.

Stoff[1] first used the finite difference method and the standard turbulence model to numerically simulate the incompressible flow field inside the sealing labyrinth. The results show that the numerical calculation can predict the flow phenomenon in the cavity to a certain extent. Rhode[2] studied the effects of throttling gap width, step height, and tip diameter of sealing labyrinth on the leakage of the sealing labyrinth on the basis of studying the sealing performance of the sealing labyrinth. In addition, he also studied the leakage characteristics of the special cavity shape of the stepped sealing labyrinth, and found out the cavity geometry with the greatest leakage resistance. The results of research by
Romulo et al.\textsuperscript{[3\textemdash}4\textsuperscript{]} show that the flow coefficient can be reduced by reducing the tip width of sealing labyrinth, rounding the cavity or adding grooves on the surface of stator parts, and the sealing requirements can be basically satisfied by the number of sealing teeth reaching three. Research by Doukelis et al.\textsuperscript{[5]} showed that the vortex structure of the leakage flow is more sensitive to the gap size when the hub rotates. Dento et al.\textsuperscript{[6]} studied the influence of inlet and outlet geometry of stator cavity on stator leakage flow characteristics and analyzed the mechanism of leakage flow affecting the main flow.

Studies by Meng Dejun\textsuperscript{[8]} and others show that the seal clearance of sealing labyrinth is linearly proportional to the leakage, but the influence of the sealing clearance on the total pressure loss is non-linear. Luo Yan et al.\textsuperscript{[9]} Carried out an experimental test on a two-stage low-speed axial-flow compressor and found that the flow loss inside the compressor was mainly caused by the leakage flow at the root of the blade and the separation of the angular region at the blade tip. The leakage flow at the root of the blade and the separation flow at the corner of the blade are constantly mixed with the surrounding mainstream gas, causing total pressure loss. Wang Pengfei\textsuperscript{[10]}, Wang Suofang\textsuperscript{[11\textemdash}12\textsuperscript{]}, Ji Guojian\textsuperscript{[13]}, Wang Daijun\textsuperscript{[14]} etc. respectively conducted relevant researches on the effects of rotation speed and pressure ratio on the sealing characteristics of the sealing labyrinth. The study of ma yaru et al.\textsuperscript{[15]} shows that the anti-friction structure design of the tooth-tip groove can not only adapt to the radial vibration and axial movement of the rotor, but also reduce the amount of sealing leakage and improve the sealing efficiency of the sealing labyrinth. Cao Yonghua\textsuperscript{[16]} and others conducted a two-dimensional reverse jet test and found that when the jet angle was 45° and the jet position was in the middle of the first tooth cavity, the leakage coefficient was reduced by 11.5% compared with that without the jet.

2. Computational models and numerical methods

2.1. Introduction to computational models

A simplified compressor cascade sealing labyrinth model (figure 1) which is composed of three labyrinth seals, platform bushing, wheel hub and reverse jet port is adopted in this study. The three-tooth sealing structure is located between the platform bushing and the wheel hub, and the reverse jet port is located in the axial middle of the first tooth cavity. Due to the periodicity of the flow structure, a simplified model of 10° is taken here for simulation. The sealing labyrinth model is divided into a rotational upstream segment, a jet-affected segment and a rotational downstream segment along the circumferential direction (Figure 2). The jet-affected segment is the area where the jet flow has a significant effect on the structure and performance of leakage flow in the tooth cavity. The axial width of the jet port is defined as \( l \), the circumferential width of the jet port is defined as \( d \), and the height of the jet port (H) is uniformly set as 1mm. The parameters of sealing labyrinth are shown in Figure 3.

The leakage clearance (\( e \)) is 0.39mm, the thickness of the tooth tip (\( d \)) is 0.2mm, the depth of the tooth cavity (h) is 4.3mm, and the width of the tooth cavity (D) is 4mm. The total length of the leakage flow channel is 18 times the width of the tooth cavity, the inlet of the channel (a) is 7 times the width of the tooth cavity, and the outlet of the channel (b) is 8 times the width of the tooth cavity.
2.2. Computational grids and boundary

In this paper, the grids of the computational domain are manually divided into blocks using ANSYS ICEM CFD software. The computational domain is divided into four parts: upstream cavity, downstream cavity, tooth cavity and jet port. As shown in Figure 4, the grids are all structured grids. Grids are encrypted along the wheel hub and the platform bushing and the height of the first layer of the grids between the platform bushing and the wheel hub is taken to be 0.008mm. The number of near-wall surfaces y + is controlled by adjusting the number of grid nodes and the extension ratio of the grid to meet the accuracy requirements of the calculation.

The grid-independent analysis is performed here. It can be seen from Figure 5 showing the changes of channel flow under six different numbers of grids that when the number of grids is around 2 million, the flow in the channel remains around 1.996×10-2 kg/s. As the number of grids continue to increase, the flow remains almost unchanged. Therefore, the number of computational grids in this study is set at about 2 million.

Calculation boundary conditions: the total inlet pressure is set to standard atmospheric pressure of 101325 pa, the total inlet pressure of jet is set to 102000 Pa, the total temperature is set to 300 k, the solid wall is set to adiabatic and non-slip boundary, the periodic boundary is given along the circumference, and the speed and outlet pressure are set according to different working conditions.
2.3. Calculation accuracy verification

In order to verify the accuracy of the calculation model and compare the aerodynamic losses caused by the sealing structure, this study sets the rotating speed at 0r/min, the total temperature at 300K, the total inlet pressure of sealing labyrinth at 101325Pa and the total inlet pressure of the jet at 102000Pa. The curve of the leakage coefficient of sealing labyrinth under different pressure ratios is obtained and compares with the experimental results under the same experimental conditions in literature [16]. It can be seen from Figure 6 that the 3D numerical simulation results are slightly lower than the experimental results, but the overall agreement is good. Therefore, the numerical simulation results are reliable.

3. Calculation results and analysis

3.1. Fluid flow structure in the tooth cavity of labyrinth seals under the influence of jet

In this paper, the wheel speed is set as 3000r/min and the inlet and outlet pressure ratios are both set as 1.5. In order to analyze the flow structure of the fluid in the tooth cavity under the influence of the jet, the axial width of the jet port is set as 0.3mm, the circumferential width is set as 1.0° and the incidence Angle of the jet (a) is set as 45°.

Figure 7 is the fluid flow diagram in tooth cavity 1. It can be found from the Figure (a) showing the fluid flow structure in the tooth cavity 1 that the leakage flow and the jet flow are violently mixed near the jet port, which results in a very complicated fluid flow structure in the tooth cavity.

Figure (b) shows the flow of jet fluid after it enters the tooth cavity, where the black streamline represents the breathable airflow. It can be seen from the figure that the flow direction of the black fluid is deflected toward the middle of the tooth cavity near the jet port under the influence of the jet flow. After the jet flow enters the tooth cavity, a part of the fluid (cyan streamline) forms a small clockwise jet vortex downstream of the jet port whose circumferential width is the same as the circumferential width of the jet port and the other part of the fluid (purplish red streamline) is impacted by the fluid in the breathable area of the upper wall. Under the influence of the black fluid, the
purplish red fluid gradually changes its flow direction and wraps the cyan vortex in it. Finally, both flow to the next tooth cavity with the black fluid.

Figure (c) shows the flow structure of the leakage fluid entering the tooth cavity near the jet port. It can be seen that the green fluid forms a large-size vortex together with the jet flow in the tooth cavity. The large-size vortex develops towards the rotation direction of labyrinth and eventually forms a small-sized vortex opposite to the direction of its own vortex rotation at the bottom of the tooth cavity. After entering the tooth cavity, the pink fluid also forms a large-size vortex which points to the opposite rotation direction of the labyrinth under the influence of the jet flow. Finally, it flows directly into the next tooth cavity near the jet port.

Figure (d) shows the development of the main vortex line in the first tooth cavity. The development of the blue fluid in the tooth cavity is extremely irregular. In the rotational upstream segment, the closer the blue fluid is to the jet-affected segment, the smaller its size will be. After passing through the jet-affected segment, there is a certain accumulation of blue fluid. Then the size of the vortex gradually increases and flows into the next cycle. In the rotational upstream segment, the light blue fluid develops in the form of small-sized vortices in a counterclockwise direction. When the light blue fluid flows near the jet port, it is rolled up by the pink fluid to change the flow direction. Then the two fluid flow together toward rotational upstream segment. However, when the pink fluid enters the next tooth cavity, it is mixed with the blue fluid and eventually flows toward rotational downstream segment in the form of large-size vortex.
Figure 7. The fluid flow diagram in tooth cavity 1

Figure 8. The fluid streamline diagram in the tooth cavity 2

Figure 8 is the fluid streamline diagram in the tooth cavity 2. It can be seen from Figure (a) which shows the overall flow structure of the fluid in the tooth cavity 2 that the fluid flow structure in the tooth cavity 2 is relatively simple and regular compared to the flow structure in the tooth cavity 1.

Figures (b) and (c) show the development of the main vortex fluid in the tooth cavity 2. By comparing the flow structure of the fluid in cavity 2 with that in cavity 1, it can be found that the flow structure of a part of the main vortex fluid in cavity 2 is not damaged. The blue fluid in the Figure (b) is a small-size vortex in the rotational upstream segment but still develops as a large-size vortex in the entire tooth cavity. The green fluid in the Figure (b) develops in the form of a main vortex in the rotational upstream segment and changes in the flow direction near the jet-affected segment. Finally, it flows downstream of the rotation in the form of a small-sized vortex in a counterclockwise direction. It can be seen from the Figure (c) that a part of the main vortex fluid represented by light pink streamlines is mixed with the jet flow near the jet port and directly enters the upstream cavity through the next tooth tip. We can also find that the orange fluid in the rotational upstream segment develops in the form of a small-size vortex and the orange fluid in the vicinity of the jet-affected segment is lifted up by the jet flow. Finally, the orange fluid forms large-scale vortex in the tooth cavity.

Figure (d) shows the development of the small-size vortex represented by a pink fluid at the bottom.
of tooth cavity 2. The small-size vortex develops irregularly along the circumferential direction and a part of the small-size vortex is sucked by the jet flow in the jet-affected segment and mixed with the jet flow to enter the upstream cavity through the third tooth.

In general, the jet flow does not make the flow structure in the tooth cavity develop into regular main vortex and a small-size vortex that rotates in the opposite direction. On the contrary, the interaction between jet flow and leakage flow makes the flow structure in the tooth cavity complex and changeable. Although the leakage flow still forms a main vortex and a reverse small-size vortex in the tooth cavity, the two fluids mix with each other and permeate each other in the jet-affected section, which causes the fluids to constantly change direction, separate and merge. In the end, the flow loss of the fluid increases sharply, thereby achieving the overall sealing effect and greatly reducing the leakage coefficient.

3.2. Analysis of the influence of jet on the flow performance in the tooth cavity

In order to study the influence of the jet flow structure on the performance of the sealing labyrinth, the changes of vorticity and static pressure in the tooth cavity caused by the jet flow are analyzed.

3.2.1. Analysis of vorticity in the tooth cavity

Figure 9 is the cloud diagram of vorticity distribution in the tooth cavity under the influence of jet flow. It can be seen from Figure (a) which is the axial distribution of vorticity at the jet position (5°) that the low vortex fluid in the breathable area develops close to the surface of the platform bushing from the first tooth tip to the location of the jet port. When it comes to the jet port, the low vortex fluid affected by jet vortex deflects to the middle of the tooth cavity, which makes a part of the fluid in the breathable area enter the tooth cavity to participate in the dissipation of the vortex. It can also be found in the figure that the jet flow accumulates between the jet port and the second tooth tip to generate a region of high vortex which can greatly block the leakage flow into the next tooth cavity. There is an obvious high vortex area caused by the fluid forming a small-size vortex in the tooth cavity opposite to the rotation direction of the main vortex in the convex platform area to the left of tooth cavity 2. This is consistent with the flow structure in tooth cavity 2 shown in figure 8.

Figure (b) shows the vorticity distribution at seven different positions along the circumferential direction: 0°, 2.5°, 4°, 5°, 6°, 8°, and 10°. The high vortex area formed by the jet flow downstream of the jet port is very small at 4° and 6° circumferential positions, which indicates that the range of the jet flow affecting the breathable fluid is about twice the circumferential width of the jet port. From the position of the jet port to the rotation surfaces on both sides, the area of the high vorticity region inside the tooth cavity 1 gradually decreases and the high vorticity region at the convex table on the left side of the tooth cavity gradually becomes obvious. This indicates that, outside the influence range of the jet, a small-scale vortex is formed at the bottom of the tooth cavity again. The vorticity in cavity 2 does not change significantly along the circumferential distribution, but on the 4° and 6° cross sections, a clear high vortex area appears at the center of the backflow region of tooth cavity 2. This high vorticity region is not connected to the main vortex region, mainly because the light pink fluid and orange fluid in Figure 8 (c) are separated from the main vortex on the boundary of jet-affected segment and directly enter the next tooth cavity.
3.2.2. Analysis of turbulent kinetic energy in the tooth cavity

It can be found from the Figure 10 (a) showing the axial distribution cloud diagram of the turbulent kinetic energy at the jet position (5°) that the turbulent kinetic energy close to the platform bushing is small and the gradient of the turbulent kinetic energy in the middle of the tooth cavity is small. The turbulent kinetic energy is very low in most areas of the entire tooth cavity, but at the tip of the tooth, the turbulent kinetic energy changes violently due to the large disturbance of the fluid here. At the left side of the jet port, the turbulence intensity increases due to the mixing and collision of the leakage flow and the jet fluid. Therefore, the turbulent kinetic energy increases sharply. With the influence of labyrinth configuration, the turbulent kinetic energy reaches the maximum value at the second tooth tip. The turbulent kinetic energy on the left side of the tooth cavity 1 increases, forming a local high turbulent kinetic energy region, which is consistent with the flow structure of the tooth cavity described in FIG. 7, where a local small-scale vortex is formed.

Figure 10 (b) shows the distribution of the turbulent kinetic energy of the tooth cavity at different circumferential positions. It can be seen from the figure that the distribution of the turbulent kinetic energy contour at the jet position in the tooth cavity 1 is the most complicated, and the gradient of the turbulent kinetic energy at the tip of the second tooth is the largest. From the position of jet flow along
the circumference to the two sides, the turbulent kinetic energy changes gently in the cavity 1, and the region of low turbulent kinetic energy gradually expands to the bottom of the cavity 1. As can be seen from the cloud diagram of the sections of $0^\circ$, $2^\circ$, $8^\circ$ and $10^\circ$, the contour line $k=400J$ of turbulent kinetic energy in the tooth cavity 2 splits the low-turbulent kinetic energy region into two from the convex stage of the Sealing labyrinth, and forms a crescent-shaped low-turbulent kinetic energy region at the bottom of the tooth cavity. It shows that the influence of the jet fluid on the energy of the leakage flow is mainly in the area near the jet port in the tooth cavity 1, while the main influence area in the tooth cavity 2 is far away from the jet port.

**Figure(a)**. The axial distribution of the turbulent kinetic energy at the jet position ($5^\circ$)

**Figure(b)**. The distribution of the turbulent kinetic energy of the tooth cavity at different circumferential positions

**Figure10**. The cloud diagram of the turbulent kinetic energy distribution in the tooth cavity under the influence of jet flow

3.2.3. Analysis of the mach number distribution in the tooth cavity

Figure 11 is an axial distribution cloud diagram of the Mach number of the tooth cavity at the jet position ($5^\circ$). It can be seen that the high Mach region in the upper part of the tooth cavity 1 is blown into two parts by the jet fluid, one part is on the left side of the first tooth tip, and the other part is on the left of the jet position mouth and deflected into the tooth cavity at the same time. On the left side of the jet port close to the platform bushing, the Mach number is reduced to the minimum Mach,
where the fluid forms a vortex. There is a low Mach region at the bottom of tooth cavity 2 that is not connected with the low Mach region at the main vortex core, which is similar to the streamline distribution in tooth cavity 2 in Figure 8.

![Mach number distribution](image)

*Figure 11.* The axial distribution of the Mach number of the tooth cavity at the jet position (5°)

3.2.4. *Analysis of static pressure distribution in the tooth cavity*

Figure 12 shows the static pressure distribution of the tooth cavity under the influence of the jet. It can be found from the Figure 12 (a) showing the axial distribution of static pressure in the tooth cavity at the jet position (5°) that the static pressure of the fluid is significantly reduced after the fluid passes through the tooth tip throttling. At the jet port, the jet flow and the breathable flow collide with each other, resulting in the decrease of kinetic energy and the increase of static pressure, so a small area of high pressure appears in this area. In the downstream of the jet port, the static pressure decreases due to the dissipation of the vortex. The existence of the vortex generated by the jet flow causes the breathable flow to deflect into the tooth cavity and hit the left wall of the tooth cavity at a high speed, which causes the kinetic energy of the fluid to drop sharply and the static pressure to increase. Therefore, it can be found from the figure that a small area of high pressure appears on the left wall of the tooth cavity 1.

Figure 12 (b) shows the static pressure distribution at seven different positions along the circumferential direction: 0°, 2°, 4°, 5°, 6°, 8°, and 10°. It can be seen that the static pressure distribution in the tooth cavity 2 changes little along the circumference. Comparing the static pressure distribution cloud diagram in tooth cavity 1, it can be found that the pressure gradient formed by the jet fluid at the jet port is not obvious on the 4° section and the 6° section. At the same time, the low-pressure region downstream of the jet port is also weak on these two sections, indicating that the jet fluid has a small effect on the static pressure here. This is similar to the vorticity distribution under the influence of the jet, which shows that the boundary of the influence of the jet fluid on the performance of the cavity fluid is indeed in the range of 4° and 6°.
4. Conclusions

In this paper, through the numerical simulation of the three-tooth sealing structure with jet structure, the flow structure of sealing labyrinth and the flow performance of sealing labyrinth under the influence of jet are recognized and analyzed. The conclusions are as follows:

(1) Within the jet-affected section, the jet fluid and the leaked fluid interact with each other, which makes the flow structure in the tooth cavity complicated and changeable, resulting in the failure to form a small-scale vortex at the bottom of the tooth cavity 1. Downstream of the jet, the jet fluid forms a vortex to achieve the purpose of blocking the leakage flow.

(2) A small-size vortex appears in the tooth cavity 2, but near the jet section, a part of the small-scale vortex fluid entrained by the main vortex fluid flows together with the main vortex fluid. The other part climbs upward along the wall of the tooth cavity. Finally, it is sucked by the breathable fluid and flows directly into the upstream cavity through the third tooth.

(3) The jet flow makes the vorticity distribution, turbulent kinetic energy distribution, Mach number distribution and static pressure distribution of the cavity change dramatically and the range influenced by the jet flow is approximately 2 times the circumferential width of the jet port.

(4) Within the jet-affected section, the low vortex of the breathable area is deflected into the
tooth cavity to participate in the dissipation of vortex.

Reference

[1] Stoff H 1980 Incompressible Flow in a Labyrinth Seals Journal of Fluid Mechanics J (Translated of ASME) 110:P817-829
[2] Romulo B.Freitas 2012 Computational Evaluation of Labyrinth Seal Congurations J AIAA-2012-04 76
[3] Rhode D L and Younger J S 1997 Flow Visualization and Leakage Measurements of Stepped Labyrinth Seals: Part 2 Sloping Surfaces J (Transactions of the ASME) 119:884
[4] Rhode D L 2002 Flow Visualization and Leakage Measurement of Stepped Labyrinth Seal Part 1-Annular Groove J
[5] Doukelis A, Mathioudakis K and Papailiou K 1998 The Effect of Tip Clearance Gap Size and Wall Rotation on the Performance of a High-Speed Annular Compressor Cascade C Asme International Gas Turbine & Aeroengine Congress & Exhibition
[6] John D.Denton and Eric M.Crutis 2008 The influence of shroud and cavity geometry on turbine performance:An experimental and computational study Part1:Shroud geometry J Journal of Turbomachinery 130:041001-1
[7] SHEN Hong, ZHEN Tian-hui, and CHEN Yu-jie 2011 Improvement of Aero-Engine Sealing Technology J Gas Turbine Experiment and Research 24(4): P51-55
[8] MENG De-jun, XING Lei and LI Jian 2013 Influence of Shrouded Stator Cavity Flow on Compressor Stator Performance J Aeroengine 39(6):P31-35.
[9] Luo Ju, Hu Jun, Yan Wei, Zhang Chenkai,and Yin Chao 2013 Measurement of Flow Field in Stator Passages of a Compressor with Hub Gap J China Mechanical Engineering 24(18):P2458-2468
[10] WANG Peng-fei, LIU Yu-fang, GUO Wen, LI Jiang-hai and LIU Bo 2007 Influence of High Rotational Speeds on the Labyrinth Sealing Char acteristics J Gas Turbine Experiment and Research 120(2):P45-48
[11] WANG Suo-fang, LU Hai-feng and XIA Pan-yong 2004 Numerical Analyses and Experimental Investigations of Seal Labyrinth Characteristics J Journal of Nanjing Univestics of Aeronautics&Astronautics 36(6):P732-735
[12] WANG Suo fang and LU hai-feng 2005 Numerical Analysis of Flow and Experimental Investigation of Labyrinth Sealing Characteristics at Rotational and Static States J Journal of Aerospace Power 20(3):P444-449
[13] Ji Guojian, Ji Honghu and Cao Guangzhou 2008 Effect of Rotating Speed on Leakage Characteristics of Labyrinth-Honeycomb Seal with Small Clearance [J]. Journal of Nanjing University of Aeronautics & Astronautics 40(2):142-145
[14] WANG Dai-jun, SU Yun-liang and LUO Xiang 2007 Experimental Investigation on Leakage Flow Char acteristics in Rotating Labyrinth Seals J Gas Turbine Experiment and Research 20(2)
[15] MA Ya-ru, HUO Wen-hao, LIU Jing , JIANG Li-jun, FANG Zhi and LI Zhi-gang 2019 Investigations on the Leakage Perfoemance of Wear Failure and Anti-rubbing Structure Design for a Labyrinth Seal J Journal of Turbomachinery (05)P64-71
[16] CAO Yong-hua 2017 A Study Based on Using Jet to Suppress Leakage Flow in the Labyrinth Seal D Nanjing University of Aeronautics & Astronautics

[17] Zhang Yibin 2015 Experiment Research of Compressor Labyrinth Seal and Ensemble Cupling Numerical Analysis D Nanjing University of Aeronautics & Astronautics

[18] CHEN Zhen-qi, PENG Tao, JIANG Bin and ZHENG Qun 2018 Effect of Axial Clearance on Radial Flow in Axial Compressor Near to Stall Condition J Journal of Turbomachinery (04) P16-23

[19] Jinjing Sun, Yangwei Liu, Lipeng Lu and Xavier OTTAVY 2016 Control of corner Separation By Optimized Slot Configuration in a Liner Compressor Cascade J Journal of Turbomachinery (06) P9-18

[20] FU Xin, CAO Yong-hua, ZHANG Yi-bin and ZHU Chao 2017 Study of Leakage Flow Boundary Model-based Sealing Labyrinth Analysis Method J ACTA ARMAMENTARII (04) P824-832

[21] ZHOU Bin-bin, YANG Bo and GU Chuan-gang 2012 Numerical Simulation of the Influence of Changeable Axial Gap on Compressor Clocking Effect J Journal of Turbomachinery (03) P9-15

[22] CHEN Zhe and LIU Ying-zheng 2012 Numerical Investigation on the Effect of Bleeding to the Flow-field of the Downstream Row in the Axial Compressor J Journal of Turbomachinery (03) P22-25