Study on the influence of hot streak and swirl on the film cooling performance of the leading edge of the guide vane

Chen Xiao¹, YI Wei-lin¹

¹Beijing Institute of Technology, Beijing

Abstract. In order to study the impacts of the non-uniform conditions of the combustor exit on the flow loss of the turbine guide blade and the cooling performance of the leading edge film, a hot streak and swirl distribution generation program has been developed to implement the parameterization specification of the non-uniform flow state. In addition, the calculation of this paper is based on GE-E3, the numerical simulation analysis is conducted for the swirl/hot streak of the positive blade on the leading edge of the film cooling guide blade. The results showed that the presence of hot streak increased the non-uniformity of vane heat load and had no effect on vane wall pressure distribution. The presence of swirl changes the pressure distribution on the vane wall, affects the migration form of hot streak in the passage, and reduces the cooling effect of the film cooling on the vane. The temperature at the outlet section of the guide vane is uniform.

Keywords: hot streak; swirl; turbine blade; leading edge film cooling; entropy distribution

1. Introduction

With the continuous improvement of the performance requirements of aero-engine like the thrust-to-weight ratio and thermodynamic efficiency, the turbine operating temperature is getting increasingly higher. The high temperature far exceeds the metal material tolerance range so that the turbine life is dramatically shortened. Due to the limited scientific development of turbine blade materials, film cooling technology [1] has been widely studied and applied as an effective cooling method. The leading edge of the turbine blade is completely exposed to high temperature fluids and is the most critical part. Giridhara [2] and Liu [3] respectively performed experimental and numerical simulations on the film cooling of the leading edge of the blade. Liu [3] studied the semi-cylindrical leading edge model with three vent film cooling holes on the leading edge, and analyzed its flow and heat transfer. Malkell [4] also used the cylindrical leading edge as the research object for numerical simulation calculations. It was found that due to the flow separation of the cooling flow, the film cooling efficiency was reduced at a higher blowing ratio. In the same amount of cooling flow, the film holes of the three rows are more efficient than the film holes of the single row. Li[5] et al. conducted a numerical simulation study on the cooling effect of fan-shaped film holes. Chen[6] et al. studied the interaction between the film cooling flow on the inner end wall of the turbine runner and the main secondary flow structure. The exploration of film cooling has been studied by many scholars. Most of the research is carried out under uniform conditions or on flat plates. However, the actual turbine blade surface will be affected by the structure of the combustion chamber, and the import conditions will be very complicated. To improve the fidelity of the simulations, realistic non-uniform inlet conditions must be included.

The complex combustor structure and working mode result in non-uniform exit temperature, and the local temperature is much higher than the ambient temperature. This local high temperature zone is
called hot streak. Many scholars have conducted experimental measurements on the total temperature of combustor exit flow, such as Qureshi [7], Barringer [8] and others. Paolo [9] studied the change of the hot streak generated by the burner in the vane, and found that the temperature of the hot streak in the static vane has obvious attenuation, and the hot streak changes significantly to the leading edge. Sean [10] studied the effect of hot streak on the vane impact with film cooling by experimental methods, and found that the intensity of hot streak was reduced by 70% under the effect of film cooling. An [11] obtained numerical simulations that the presence of hot streak increased the unevenness of the thermal load of the stationary leaves, and the influence of hot streak at different circumferential positions on the thermal load of the turbine was given. Dong [12] performed hot streak analysis on the blades with film cooling, and found that the hot streak not only caused a large thermal shock to some areas of the blade, but also reduced the effect of film cooling.

In order to reduce NOx emissions, the combustion chamber tends to adopt Lean-Burn technology, and it is necessary to introduce a swirler to ensure flame combustion stability [13], so that the turbine inlet is further affected by the swirling flow while the hot streak is present. The influence of complex flow angle distribution on the turbine flow field and temperature field is difficult to ignore. Turrell [14] confirmed that there is a highly rotating vortex core at the exit of the combustion chamber through observing the trajectory of the film cooling jet. Tommaso Bacci [15] et al. simulated the existence of swirling flow at the leading edge of the vane through an experimental method, and the result pointed out that the swirl flow has a significant effect on the pressure distribution of the leading edge of the vane. Through the analysis of the blade adiabatic efficiency, it is also obtained The effect of swirl on film cooling cannot be ignored. Zhao [16] used two directions of rotation and two circumferential clock positions to simulate the swirl, and the effect of the inlet vortex on the pressure distribution was concentrated on the suction side. Stavros Pyliouras [17] studied the existence of swirl at the exit of the combustion chamber to reduce the efficiency of the turbine. Qureshi [18, 19] added a swirl generator at the turbine inlet to simulate the swirl flow existing at the turbine inlet, and it was found that the blade load changed, and the hot streak caused a significant radial migration under the action of the swirl. The aerodynamic performance of the turbine has a large impact.

Most of the above literatures study the effects of hot streak or swirls on heat transfer and flow in the turbine alone, and less consideration is given to leading edge film cooling. The swirl flow affects the main flow near the leading edge of the vane of the high-pressure turbine, as well as the cold flowing out of the leading edge of the vane, which will have a great impact on film cooling. The first stage turbine vanes are facing the high temperature and high pressure gas discharged from the combustor, and the cooling performance of the film is not clear enough. In this paper, the E3 turbine is taken as the research object, and the aerodynamic and heat transfer effects generated by the combination of hot streak and eddy current are studied by numerical simulation.

2. Research objects and numerical methods

2.1. Computation mode

The experiment report for core machine high pressure turbine provided by GE Company is adopted for the computation model in this paper [20]. Only the vane cascade is considered, and the number of vanes is 46. The numerical calculation model is shown in Figure 1. The position is measured according to the experimental results in [21]. P40 is the assumed combustor-turbine interface, located 40% of the axial chord length in front of the leading edge of the vane. P1 represents the vane exit data measurement plane. Referring to the experimental cooling structure [22], three rows of cylindrical cooling holes row1, row2, and row3 are set at the leading edge of the blade, as shown in Figure 2, and the relevant parameters are shown in Table 1.
Figure 1. Computational domain of the E3 guide vane

Figure 2. E3 guide blade mesh and film cooling hole.

Table 1. Size and location of film cooling hole

| Diameter (mm) | No. of holes on each Row | Span angle (degree) | Distance to the geometrical midline position of the leading edge (%) |
|---------------|--------------------------|---------------------|---------------------------------------------------------------------|
| Row 1         | 0.019                    | 12                  | 30                                                                  | 3                                                   |
| Row 2         | 0.019                    | 12                  | 30                                                                  | 0                                                   |
| Row 3         | 0.019                    | 12                  | 30                                                                  | -3                                                  |

2.2. Numerical method

The calculation uses the Numeca AutoGrid5 module to generate structured grids. A total of 2.6 million, 3.8 million, and 5.2 million sets of grids were generated. The three types of grids were calculated separately. The calculation results were compared with those measured in the experimental report [21]. The distribution of isentropic Mach number distribution under cooling conditions is compared. The distribution of the isentropic Mach number distribution in the vane surface under the three kinds of grids is shown in Figure 3. The values on the pressure surface are significantly different from the experimental values. The distribution of film holes on the pressure surface is relatively dense in the experiment. In this article, we will not consider the distribution of air film holes on the pressure surface for the time being. Considering the calculation speed, it is considered that the calculation results under the 3.8 million grid are acceptable. Therefore, the number of grids used in the numerical simulation in this paper is 3.8 million.

Figure 3. Mach number distribution at 50% span under different mesh.
This paper used Ansys CFX for the computation. The ideal air gas is selected for the flow medium, the SST model is selected for the turbulence model, the boundary conditions specify the total inlet pressure, the total temperature, the inflow direction, the outlet specifies a given static pressure, the film hole specifies the inflow flow, the inflow temperature, and the vane adopts the adiabatic and no slip shift boundary. The boundary conditions for the computation are shown in Table 2.

| Boundary conditions                  | values   |
|-------------------------------------|----------|
| Turbine inlet average temperature (K)| 708.5[K] |
| Total inlet pressure (Pa)            | 343228 [Pa] |
| Imported turbulence intensity (%)    | 5%       |
| Outlet average static pressure (Pa)  | 208865 [Pa] |
| Cooling massflow/inlet massflow (%)  | 1.7%     |
| Cold air jet temperature (K)         | 378.56 [K] |

2.3. Parameterization method for hot streak boundary condition
Before simulating the effects of hot streak, the actual distribution pattern of the combustion chamber outlet temperature is required to be considered to specify the appropriate turbine inlet hot streak conditions. Previous researchers have proposed a number of parameterization simulation strategies. In this paper, the two-dimensional Gaussian distribution function listed in Reference [22] is used to calculate the hot streak distribution of the turbine inlet, and the actual formula used to calculate the hot streak distribution of the turbine inlet is shown in Equation (1).

\[
T_f = T_0 + \sum_{i=1}^{3} A \exp \left\{ - \frac{(m-m_i)^2}{2\sigma_m^2} - \frac{(n-n_i)^2}{2\sigma_n^2} \right\}
\]

The rectangular surface of the turbine inlet fan surface is simplified. The circumferential halved node is M=50, and the averaging node in the elevation direction is N=71. The coordinates represent the position of the current node and represent the core position of the first hot streak. The accumulation of the three Gaussian distributions is to ensure the periodicity of the temperature distribution in the circumferential direction. Therefore, the following relationship exists

\[
\begin{align*}
\left\{ m_1+M = m_2 = m_3-M \\
n_1 = n_2 = n_3
\end{align*}
\]

In formula (1), \(T_0\) is the base value before the Gaussian distribution is superimposed, \(A\) represents the fluctuation amplitude of the Gaussian distribution (\(A = 0.4\)), and controls the degree of contraction of the Gaussian distribution in the elevation and circumferential directions \((\sigma_m=15, \sigma_n=18)\), \(n_2=40\), respectively. In order to ensure that the average flow rate of the turbine inlet temperature does not change after the hot streak is introduced, the basic value \(T_0\) of the Gaussian distribution is the difference between the total turbine inlet temperature of 1277.6R and the arithmetic mean of the right index portion of the formula (1).

2.4. Parameterization method for swirl boundary condition
In this paper, the non-uniform inlet airflow angle is set on the inlet boundary surface by setting the components in the axial, radial and tangential directions in the cylindrical coordinate system of the Ansys CFX software.

The configuration of the swirl flow refers to the Reference [23]. The position of the vortex core always overlaps with the core position of the hot streak. On this basis, the tangential velocity of the fluid on the combustor-turbine intersurface can be obtained by solving the Equation (3).
The parameter in the equation is used to control the direction of the vortex. The parameter represents the size of the vortex core diameter, which was set to 16.8 mm in this study, approximately 36.3% of the length of the arc in the entrance surface. For the setting of the vortex intensity $\Gamma$, the maximum circumferential airflow angle should be referred to the actual combustion chamber exit conditions. Based on the test measurement data on the angular distribution of the turbine inlet airflow provided in the published references, the actual combustion chamber exit condition is about ±50°. Based on this, the vortex intensity $\Gamma$ is set to 2.857$ms^{-2}$, and the maximum circumferential airflow angle obtained is 41.8°.

The selected hot streak-swirl timing position is more representative of the aligning vane leading edge. The schematic diagram of hot streak and swirl was shown in Figure 4, which shows the total temperature distribution and the projection of the velocity vector on the inlet.

In order to study the effects of uniform inlet conditions, hot streak in the inlet and swirl on the film cooling and vane, the final example settings are shown in Table 3.

|                | With or without hot streak | Swirl           |
|----------------|---------------------------|-----------------|
| Case 1         | No                        | None            |
| Case 2         | Yes                       | None            |
| Case 3         | Yes                       | Reverse swirl   |
| Case 4         | Yes                       | Positive swirl  |

![Schematic diagram of hot streak and swirl.](image)

**Figure 4.** Schematic diagram of hot streak and swirl.

3. Results

3.1. Effects of hot streak

Figure 5 shows the vane temperature distribution for uniform inlet (Case1) and inlet with hot streak (Case2). It can be seen from the figure that the film cooling of the vane wall is different under the condition of Case1 and Case2, and the unevenness of the temperature distribution of the wall surface of Case2 is significantly greater than the unevenness of the wall temperature distribution of the blade under the uniform inlet condition of Case1. Case1 and Case2 have a low temperature zone at the leading edge of the high span, because the cooling holes have a radial span angle in jetting direction, causing the cooling airflow to concentrate at the high span. The wall temperature distribution of Case1 is generally uniform, and the maximum temperature is about 700K. In the case of Case2 with hot streaks, the temperature distribution in the leading edge region is unevenness. A high temperature region appears at the leading edge of 20%-50%, and this high temperature region always extends to the suction surface. The high temperature region appears here due to the location of the hot streak at
the entrance. The presence of hot streaks significantly boosts the unevenness of the thermal load of the vane.

![Figure 5. Contour of vane temperature distribution](image)

The pressure distribution on the vane wall of Case1 and Case2 was shown in Figure 6. It can be seen that there is no difference in the pressure distribution under the two inlet conditions, which is consistent with the conclusion of An et al. [10]. Figure 7 shows the pressure change at the position of chord length. There is no difference in the pressure in the channel under the two inlet conditions. That is, the hot streak does not affect the pressure change in the vane wall surface and the channel, so the flow situation of the main flow and cooling flow is the same. Through the above analysis, it can be concluded that the uneven thermal load of the vane caused by the presence of hot streak in the turbine inlet needs to be paid more attention.

![Figure 6. Contour of vane pressure distribution](image)

![Figure 7. Contour of passage pressure distribution](image)

3.2. Effects of swirl

3.2.1. Effects of swirl on temperature

Under the influence of hot streak, the temperature distribution on the vane wall surface is uneven, and the influence of swirl at the turbine inlet will further affect the cooling effect of film cooling on the wall surface. Fig. 8 shows the temperature distribution on the vane under swirl flow. In Figure 8-a, the contour on vane surface shows the temperature of the vane under the effect of positive swirl. There is a high temperature zone in the high span at the leading edge, and the temperature is up to 797K. Fig. 8-b shows the temperature distribution of the vane surface under the action of reverse swirl, which is more uniform than that of the leading edge of positive swirl. Under the action of positive and reverse swirl, high temperature zone appeared on the suction surface. Under the action of positive swirl, high
temperature zone concentrated in the middle span zone, and under the reverse swirl, the distribution of cooling jet flow was opposite. In general, the temperature distribution on the wall surface of the vane under reverse swirling flow is more uniform than that under forward swirling flow.

![Figure 8. Contour of vane wall temperature distribution](image)

3.2.2. Effects of swirl on pressure
The hot streak obtained from 2.1 has no effect on the static pressure of the vane wall. The change of the wall temperature is related to the flow structure of the main stream and the cooling jet. Figure 9 shows the pressure distribution of the vane wall under positive and reverse swirling, and the swirl has a significant effect on the vane surface pressure near the leading edge. Figure 9 shows the pressure distribution on the vane wall surface under the positive and reverse swirl. It can be seen that the swirl has a significant effect on the vane pressure near the leading edge. Under the action of positive swirl, the pressure stagnation line in the low-span area of the leading edge of the vane is shifted to the pressure side, and the pressure stagnation line in the high-span area is shifted to the suction side.

Under the action of reverse swirl, the change of pressure stagnation line the leading edge is opposite to that under positive swirl. This change is also related to the flow mode of the cooling jet. Under the action of swirl flow, a tangential velocity is generated, which results in a change in the pressure gradient in this region and a decrease in static pressure, so the change of the pressure stagnation line will occur.

![Figure 9. Contour of vane wall pressure distribution](image)

3.2.3. Effects of swirl on the shape of hot streak
Considering the influence of the swirl of the turbine inlet on the hot streak under the condition of film cooling in the leading edge, the position of the hot streak and the swirl is positively on the leading edge of the vane. The hot streak are blended at the leading edge with the cooling flow, and the range of hot streak spreads. The leading edge of the vane divides the hot streak into two parts, and the main flow with swirling is also divided into two parts by the vane. The two parts of the massflow still maintain the original direction of rotation, and the radial velocity component near the wall on both sides causes the flow to move along the span. Figure 10-a) shows the change of the hot streak in the chord length without swirling. The cooling jet is ejected from the film hole, which can form a gas film between the wall and the mainflow, thus preventing the vane temperature from being too high. The hot streak is divided into two parts by the vane, one part is on the pressure side, and the other part is on the suction side. After mainflow is mixed with the cooling gas sprayed from the film hole, the temperature of the hot streak is obviously lowered, and the highest temperature at the outlet is 706K. High
temperature on both sides gathered together at the exit of the trailing edge, and the shape is approximately circular.

The change of hot streak along the chord length in Case3 and Case4 was shown in Figure 10-b) and Figure 10-c), respectively. On the 10% chord length section, there is a high temperature the pressure side of Case3 in 60%-80% span of the pressure side and the 40%-60% span of the suction side. Similarly, the high temperature main flow and the wall surface of the pressure surface in Case4 are separated by cooling film, and there is a high temperature zone near the wall surface of the suction surface 20%-50% span. At the 50% chord length section, the passage temperatures of Case3 and Case4 were lowered, and the temperature at the pressure side and the suction side near the wall did not change significantly. Under the action of the two direction swirls, the range of the hot streak in the radial direction becomes large. Under the action of positive swirling, the high temperature zone of the low span of the exit is “elongated” in the circumferential direction, and the high temperature zone of the mid-high span is “compressed”. Under the action of reverse swirling, the high temperature zone of the mid-high span the exit is “elongated”, and the high temperature zone of the mid-low span is “compressed”.

![Figure 10. Contours of shape of hot streak along the chord](image)

3.2.4. Effects of swirl on heat transfer

In order to study the heat transfer characteristics of the vane wall, the convective heat transfer coefficient $h$ is defined as follows

$$h = \frac{q}{T_{aw} - T_w}$$

(4)

$T_w$ is the temperature of the constant temperature wall;

$q$ indicates the heat flux of the vane wall at a constant temperature 566.8K;

$T_{aw}$ is the adiabatic wall temperature.
Figure 11 shows the convective heat transfer coefficient distribution of the vane wall of Case2, Case3 and Case4. It is a strong heat exchange zone at the leading edge of the film hole, where the cooling flow is strongly blended with the main flow, so that the convective heat transfer coefficient of the zone is large, and the heat transfer coefficient is gradually reduced to constant away from the area of the hole row. The convective heat transfer coefficient of Case2 and Case3 is uniform compared of Case4 in leading edge. In the three Cases, the heat transfer ratio of the pressure surface is lower than that of the suction surface. Although the cooling flow and the mainstream mixing are severe, the wall surface curvature of the pressure surface is large and the pressure difference in the channel is affected. There is a tendency to stay away from the pressure surface, so the coefficient of convective heat transfer coefficient is low at 0% - 50% of the chord length on the pressure side. Under the action of the positive swirl, the range of the high heat transfer zone at the proximal wall of Case3 is larger than that of the suction side of the Case2, and under the reverse swirl, the high heat transfer zone at the lower end wall disappears. In Case 4, the convective heat transfer coefficient in the 30%-40% span area of the leading edge becomes larger, and the convective heat transfer coefficient near the cooling film hole in the mid-high span is also larger than that in Case3 and Case4.

3.2.5. Effects of swirl on exit of vane

After introducing the hot streak and the swirl condition, the temperature and airflow angle of the vane exit will change, that is, the total temperature and airflow angle of the vane inlet are directly related to the turbine inlet temperature and the airflow direction. The total temperature distribution of the vane outlet is given in Figure 12. In Case2 without swirl, there is still hot streak at the exit of the vane, and the total temperature of the hot streak is 785K. In the presence of the swirling conditions of Case3 and Case4, the shape of the hot streak at the exit of the vane is destroyed. Under the influence of the radial velocity, the range of the hot streak in the span becomes larger, so that the distribution of total outlet temperature in the direction of developing height is more uniform.

During the migration process, the high temperature fluid is affected by the vane splitting, the leading edge film cooling jet and the swirling flow. The jet angle of the vane exit changes, under the positive swirling flow. The jet angle varies widely, the amplitude is [162°, 173°], and the jet angle changes under reverse swirl is [160°, 167°].
4. Conclusions
This paper develops a program that simulates hot streaks and swirls, which can easily adjust the hot streak temperature and swirl and its relative guide vane position. Taking the GE-E3 turbine guide vane as the research object, the hot streak and the swirl flow are set to the leading edge of the vane, and the influence of the hot streak and the swirl on the leading edge with the film cooling guide vane is analyzed by CFX. The following conclusions:
1) By comparing and analyzing the influence of hot streak on the film cooling in the leading edge, it is found that the hot streak mainly causes the temperature distribution of the leading edge to be uneven, which makes the heat load unevenness of the vane increase. The presence of hot streaks has little effect on the mainstream flow and the cooling flow, so more attention should be paid to thermodynamics.
2) The presence of swirl increases the unevenness of the thermal load on the surface of the vane. In general, the effect of the reverse swirl on its non-uniformity is smaller than that under the positive swirl, and under the action of swirl, the cooling flow is not evenly distributed on the pressure side and the suction side, resulting in the deviation of the pressure stagnation line at the leading edge of the guide blade.
3) The swirling process changes the migration process of the hot streak. Since the mainstream flow has a velocity in the radial direction, the effect of the hot streak on the wall surface is changed along the direction of the heightening, and the range of the hot streak in the channel becomes larger. At the trailing edge, the range of the high temperature zone becomes larger, and the change of the circumferential direction is affected by the swirling direction.
4) At the exit, the temperature range of the hot streak in the span becomes larger, increasing the unevenness of the vane outlet. The flow angle of the high, medium and low span also changes under the positive and reverse swirl.

References
[1] Bunker, Ronald S. Evolution of Turbine Cooling[J]. ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition, Charlotte, North Carolina, USA, Monday 26 June 2017.
[2] Yepuri G B, Puttarangasetty A B T, Rajendran R, et al. Experimental Investigation of Adiabatic Film Cooling Effectiveness and Heat Transfer Coefficients over a Gas Turbine Blade Leading Edge Configuration[J]. 2014.
[3] Liu Z F, Wang Z D, Feng Z P. Study on the effect of inlet swirl on hot streak migration and rotor heat load in gas turbine[J]. Journal of Engineering Thermophysics, 2016.
[4] Maikell J, Bogard D, Piggush J, et al. Experimental Simulation of a Film Cooled Turbine Blade Leading Edge Including Thermal Barrier Coating Effects[J]. Journal of Turbomachinery, 2011, 133(1):011014.
[5] Xu Li Yan-jing Qu Liu-li Song. A Numerical Investigation on Cooling Effectiveness for Advanced Fan-shaped Film Cooling[J]. Chinese Journal of Turbomachinery. 2019,03:72-77+5
[6] Chen Ziyu; Su Xinxong; Yuan Xin. Interaction Mechanism Between Coolant and Mainflow in Turbine Endwall Region[J]. Compressor Blower & Fan Technology. 2020,03:46-51
[7] Qureshi I, Smith A D, Chana K S , et al. Effect of Temperature Nonuniformity on Heat Transfer in an Unshrouded Transonic HP Turbine: An Experimental and Computational Investigation[J]. Journal of Turbomachinery, 2012, 134(1):011005.
[8] Barringer M D, Thole K A, Polanka M D, et al. Migration of Combustor Exit Profiles Through High Pressure Turbine Vanes[J]. Journal of Turbomachinery, 2009, 131(2).
[9] Paolo G, Giacomo P. Hot Streak Evolution in an Axial HP Turbine Stage[J]. International Journal of Turbomachinery, Propulsion and Power, 2017, 2(2):6-.
[10] Jenkins S, Varadarajan K, Bogard D G. The Effects of High Mainstream Turbulence and Turbine Vane Film Cooling on the Dispersion of a Simulated Hot Streak[J]. Journal of Turbomachinery, 2004, 126(1):203.
[11] An B T, Liu J J, Jiang H D. Numerical Investigation on Unsteady Effects of Hot Streak on Flow and Heat Transfer in a Turbine Stage[J]. Journal of Turbomachinery, 2009, 131(3):031015.
[12] Dong P, Wang Q, Guo Z, et al. Conjugate Calculation of Gas Turbine Vanes Cooled with Leading Edge Films[J]. Chinese Journal of Aeronautics, 2009, 22(2):145-152.
[13] Tsukagoshi K, Arimura H, Tanaka K, et al. Development of Air Cooled Combustor for Mitsubishi G Class Gas Turbine[C]. ASME Turbo Expo 2010: Power for Land, Sea, and Air. 2010.
[14] Turrell M D, Stopford P J. CFD Simulation of the Flow Within and Downstream of a High Swirl Lean Premixed Gas Turbine Combustor[J]. Proceedings of ASME Turbo Expo 2004, Vienna , 2004.
[15] Tommaso B, Riccardo B, Alessio P, et al. Adiabatic Effectiveness on High-Pressure Turbine Nozzle Guide Vanes under Realistic Swirling Conditions[J]. Journal of Turbomachinery, 2018, 141(1).
[16] Liu Z, Ye L, Wang C, et al. Numerical simulation on impingement and film composite cooling of blade leading edge model for gas turbine[J]. Applied Thermal Engineering, 2014, 73(2):1432-1443.
[17] Pyltiouras S, Schiffer H P, Janke E, et al. Effects of Non-Uniform Combustor Exit Flow on Turbine Aerodynamics[C]. ASME Turbo Expo: Turbine Technical Conference & Exposition. 2012:11.
[18] Qureshi I, Smith A D, Povey T. HP Vane Aerodynamics and Heat Transfer in the Presence of Aggressive Inlet Swirl[J]. Journal of Turbomachinery, 2013, 135(2):21040-21041.
[19] Qureshi I, Beretta A, Chana K, et al. Effect of Aggressive Inlet Swirl on Heat Transfer and Aerodynamics in an Unshrouded Transonic HP Turbine[J]. Journal of Turbomachinery, 2012, 134(6):061023.
[20] Timko L P. Energy Efficient Engine high pressure turbine component test performance report[R]. CLEVELAND, OHIO: NASA, 1984.
[21] Cha C M, Hong, S, et al. Experimental and Numerical Investigation of Combustor-Turbine Interaction Using an Isothermal, Nonreacting Tracer [J]. Eng Gas Turb Power, 2012,134(8): 081501.
[22] Prenter R, Ameri A, Bons J P. Computational Simulation of Deposition in a Cooled High-Pressure Turbine Stage With Hot Streaks[C] Asme Turbo Expo: Turbomachinery Technical Conference & Exposition. 2016.
[23] Schneider M, Schiffer H P, Lehmann K. Parameterised Model of 2D Combustor Exit Flow Conditions for High-Pressure Turbine Simulations[J]. International Journal of Turbomachinery, Propulsion and Power, 2017, 2(4).