Influence of non-uniform inlet velocity and non-uniform temperature distribution on turbine blade

Xiuhe Zhang[1]
School of Aeronautics and Astronautics, Shanghai Jiao Tong University
Shanghai 200240, China
E-mail:zxh117413910040@sjtu.edu.cn
Fang Chen[2](🖂)
School of Aeronautics and Astronautics, Shanghai Jiao Tong University
Shanghai 200240, China
E-mail: fangchen@sjtu.edu.cn

Abstract The turbine blade bears higher and higher temperature of upstream flow with the increasing requirements for the engine thrust. The heat resistance of turbine blades is a severe problem for the life of turbine blades. Therefore, it is essential to study the flow field and heat transfer of the blade to find better ways to improve the heat resistance of the turbine and increase the thrust of the engine. To better the efficiency of the turbine blades, researchers at home and abroad have conducted a lot of research on the heating of turbine blades under different working conditions and the aerodynamics of the flow field. In this paper, the numerical investigation which is studying the different radius ratio of swirl and hot streak using CFD 3D RANs solver with the k-ω turbulent model to get the distribution of non-uniform temperature and swirl on blades. The circumferential and radial temperature gradients at the exit of the combustion chamber increase the disturbance of the inlet flow field and increase the circumferential and swirl for the flow, i.e., there is a non-uniform velocity distribution in the non-uniform temperature and inlet flow. The computation model utilizes standard blade C3X, which sets μmone boundary layers and adopted Pave mesh. The results show that for turbine blades when there are swirls, and hot streak affect together, the fluid on the suction side is biased toward the hub, and the liquid on the pressure side is deflected toward the shroud. The larger the radius ratio of the swirl and the hot streaks(SHR), the more complex the flow field temperature around the blade is, and a local high-temperature region is formed near the hub of the blade. The larger the radius ratio of the swirl and the hot streak, the smaller the eddy viscosity in the flow field and the maximum eddy viscosity appears at the axial position of 0.2.

Keywords: turbine blade; swirl; hot streak; the radius ration of the swirl and the hot streaks; C3X; eddy viscosity

---

1.Introduction

The requirements for the thrust of aero-engine are higher and higher; at the same time, the temperature of the exit flow of the combustor is getting higher. The working temperature of blades reached at 2000K [1]. This concludes the working environment of blades more drastic and affects the life of the blades and engine. Besides, characteristics of velocity also have an impact on the interaction between blades and fluid. The velocity of the exit of the combustor can be disturbed with the vibration of the engine, which makes the inlet velocity of HP turbine blades is non-uniform. Moreover, the inlet temperature of the aero-engine high pressure (HP) turbine is usually unsteady or distorted; these temperature variations arise from combustor framework characteristic and combustor surface cooling [2]. The main feature of the non-uniform temperature of unsteady flow is hot streak. The temperature of hot streaks is typically twice as much as the free stream temperature [2]. It is well known that the heat resistance of blades is significant to the lifetime of the engine.

In the past decades, turbine temperature distortion has been of particular interest. An et al. [2,4] revealed that the fluctuations of heat load significantly increases in consideration of hot streak at the inlet. Kim et al. [4] found that the unsteady interaction of stator and rotor results in the periodic variation of the heat transfer on the blade tip and casing and the non-uniform inlet temperature makes the heat flux fluctuation. Wang et al. discovered that the core of a hot-streak which in the uniform flow migrates to the tip under the influence of a positive swirl and the efficiency of turbine decrease with the hot-streak migration [5]. Jenny et al. researched that the hot streaks cause localized hot spots on the blade surfaces in a high-pressure turbine and change the distribution of heat loads [6].

The studies by Jenkins and Bogard [7] indicated that the attenuation rate of hot streak depends on the location of the hot streak relative to the vane due to the isolation of the hot streak core by the vane wall. Shang and Epstein [8] found that temperature distortion significantly increases both the non-uniformity of heat load on blade surface and the total heat load by as much as 10-30% (mainly on the pressure surface).

On the other hand, there are many investigations of the unsteady velocity on the inlet of the turbine blades. Carullo et al. [9] found that the high free-stream turbulence augmented the heat transfer on both the pressure and suction sides of the blade as compared with the low free-stream turbulence case. Salvadori et al. [10] showed that the circumferential velocity non-uniformity and slip velocity vector govern the wake behavior. Satta et al. [11] found that upstream wakes not only induced a wake narrowing next to blade trailing edge, but also a smaller wake spreading.

In general, the flow of the exit on the combustor is unsteady, which may exist non-uniform temperature and non-uniform velocity simultaneously. Although there are many investigations about the effects of non-uniform temperature and velocity in the inlet flow on the turbine blades, respectively, there are a few investigations on the impacts of both non-uniform temperature and velocity. Wang et al. studied the effect of the coupling of the swirl and hot spots at the turbine inlet on the motion of the fluid in the entire flow field and the thermal load distribution at the top of the blade [12]. Huang et al. used large eddy simulation (LES) technique and horizontally set small flame reservoir method to numerically study the influence of inlet vortex on flow development and combustion dynamics in a lean premixed swirling stable combustion chamber [13]. Khanal, B. et al. studied the coupling of swirl and hot spot to find that the direction of the swirl can strongly influence the heat transfer characteristics of the blade and its dependence on the timing position, and the heat transfer of the blade in the case of combined vortex and hot stripe cannot Predict [14] by each overlay.

In this paper, based on this method, this paper studies the thermal load and unstable pressure distribution of turbine blades after turbine blade transient numerical simulation under the condition of coupled hot streaks and swirl in turbine inlet boundary conditions. Through research, the influence of the radius ratio coefficient of the swirl and hot streak on the heat transfer of the turbine blade in the flow field and the movement of the fluid in the flow field determines the optimal radius ratio coefficient of the swirl and hot streaks. The results show that for turbine blades when there are swirls, and hot streak affect together, the fluid on the suction side is biased toward the hub, and the fluid on the pressure side is deflected toward the shroud. The larger the radius ratio of the swirl and the hot streaks, the more complex the flow field temperature around the blade is, and a local high-temperature region is formed near the hub of the blade. The larger the radius ratio of the swirl and the hot streak, the smaller the eddy viscosity in the flow field and the maximum eddy viscosity appears at the axial position of 0.2. This paper finds that there is a relatively stable environment for the heat transfer between the flow field and the turbine blade when the radius ratio of the swirl to the hot streaks is between 1.9 and 2.2.

2.Method and Computational Setup

2.1 Geometry and mesh

The geometry investigated in this work is NASA's C3X [15]. The geometry is shown at fig.1, which also display that the whole work computational is translational periodic. To improve the speed of calculation and the accuracy of the results, the model grid uses a structured network. The advantage of structured grids over unstructured grids is that the generated grids are of higher quality, and the convergence of model calculations is better and closer to the actual model, the surface mesh distribution is shown at fig.2. Since the number and quality of the grid have a significant influence on the accuracy of the calculation results, it is necessary to perform grid verification before work and select the appropriate number of networks. This paper selects 0.99 million, 1.5 million, 3.02 million and 5 million models of four different amounts of meshes were calculated, and finally a fluid domain model of 3.02 million grids was selected considering the accuracy and existing calculation conditions. For the fluid model domain: the number of nodes in the model mesh is 3027960, and the number of meshes is 2934424. The boundary layer grids are set on one side of the fluid region of the blade, considering the wall shear stress. The size of the first layer grid is 10e-6, and the size ratio of the grid is 1.1. And the maximum side-to-length rate is 2562.69, considering the wall shear force on the blade. The airflow angle of the fluid relative to the blade is 59.89°, and the airflow exit angle in the flow field is 72.38°.

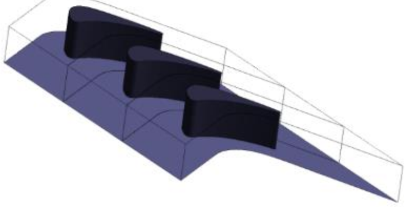


Figure 1. Computational domain and model structure

Figure 2. The distribution of surface mesh at the fluid domain

2.2 Solver and Computation method

This paper uses the commercial software ANSYS CFX 19.0 for numerical calculation. CFX adopts a finite element method based on the finite element method and piecewise structured grid.

Based on ensuring the conservation characteristics of the finite volume method, it absorbs the numerical accuracy of the finite element method. CFX contains many turbulence models. Satisfy the use of most engineering examples. CFX is a robust and fast solution that uses full implicit mesh coupling to solve both momentum equations and continuity equations. The CFD CFX was used to solve the 3D unsteady compressible Reynolds-averaged N-S equations, and the standard k-ω SST turbulence model was used to study the smooth model of turbine blades. The 3D fully implicit coupled N-S solver CFX is applied to obtain the uncertain time-accurate solution. The turbulent model and the structured grid are used to solve the 3D unsteady viscous flow. The CFX also can use-k-ε model to study 3D hard problems.

This paper is based on the Reynolds average Navier-Stokes equation (RANS). The flow field is performed in a Cartesian coordinate system. The RANS control equation is:

$$\frac{\partial\bar{\rho}}{\partial t}+\frac{\partial}{\partial x_i}\left(\bar{\rho}\tilde{u}_j\right)=0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(\bar{\rho}\tilde{u}_j\right)+\frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{u}_j\tilde{u}_i+\bar{\rho}\widetilde{u_j^{"}u_i^{"}}+\delta_{ji}\bar{p}-\overline{\tau_{ij}}\right)=0, i=1,2,3 \tag{2}$$

$$\frac{\partial}{\partial t}\left(\bar{\rho}\tilde{u}_j\right)+\frac{\partial}{\partial t}\left[\bar{\rho}\tilde{u}_j\tilde{H}+\bar{\rho}\widetilde{u_j^{"}H^{"}}+\overline{q_j}-\tilde{u}_i\overline{\tau_{ij}}-\bar{\rho}u_i^{"}\widetilde{\left(\tau_{ij}/\rho\right)^{"}}\right]=0 \tag{3}$$

2.3 Boundary Condition

In this paper, the hot spot has the following characteristics concerning the surrounding fluid: $m=T_{hs}/T_m$, $\rho_{hs}=\rho_\infty/m$, $u_{hs}=u_\infty\sqrt{m}$, $v_{hs}=v_\infty\cdot\sqrt{m}$, $M_{hs}=M_\infty$, $a_{hs}=a_\infty\sqrt{m}$. In the parameters of the hot spot at the inlet, the center position of the hot spot is 0.5 concerning the blade direction of the turbine blade, and only one hot spot of each turbine blade collides with the leading edge of the blade (Fig. 3). Show) When setting the basic boundary conditions, see Working Condition No. 34135 in [18], the outlet pressure is 215328.05 Pa, the steady-state inlet pressure is 315793.25 Pa, and the temperature is 701 K. For swirling, this paper uses a swirling rotation clockwise as the object of uneven velocity distribution (Fig. 4). By setting the swirl-hot spot radius ratio (SHR), this paper studies the relationship between the optimal speed inhomogeneity and temperature non-uniformity by studying the uneven velocity and temperature conditions of nine different SHR coefficient ratios. And their effect on the heat transfer of the flow field and turbine blades. This paper select nine different SHR : 1.270, 1.385, 1.524, 1.693, 1.905, 2.177, 2.540, 3.048, 3.81.

$$SHR=\frac{R_{swirl}}{R_{hs}} \tag{4}$$

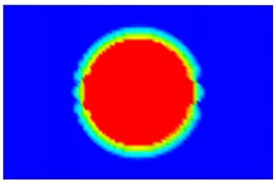


Figure3. The location of hot streaks

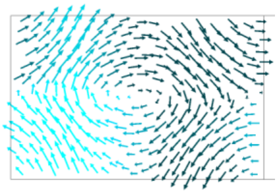


Figure4. the distribution of inlet swirl vector

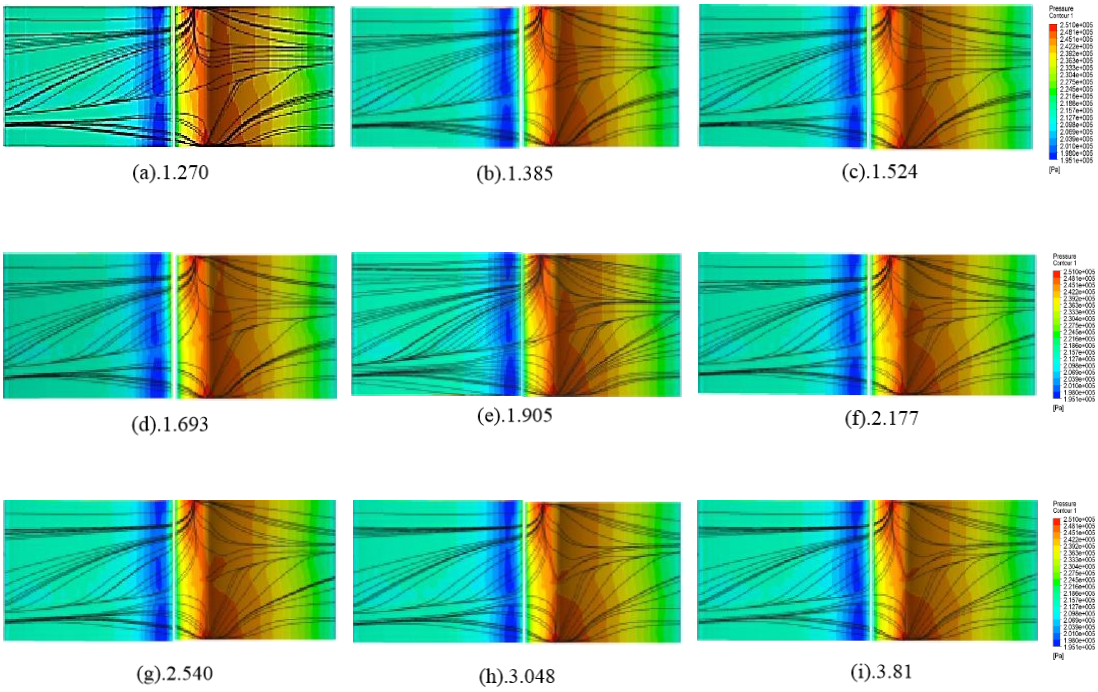
Table1. Cases data of different SHR

CASE	1	2	3	4	5	6	7	8	9
SHR	1.270	1.385	1.524	1.693	1.905	2.177	2.540	3.048	3.81

4. Result analysis

4.1 The effect on pressure and velocity

The swirling flow will have an essential influence on the movement of the hot fluid, which will change the local high-temperature region of the blade, due to non-uniformity temperature and non-uniform speed at the exit of the combustion chamber, the movement of the fluid in the turbine passage produces a variable shape. More stringent requirements are imposed on the cooling system of the blade. It can be seen (Fig. 5) that under the influence of the forward swirling flow, the flow field above the 0.4 pitch on the suction side moves in the direction of the blade hub, and the flow field near the hub migrates upward, which leads to a Forming a velocity interface, the larger the SHR, the narrower the separation zone, especially in part near the leading edge, the change of the separation zone will be more visible. On the suction side, as the SHR increases, the pressure on the surface of the blade gradually increases. When the $\mathrm{SHR} = 2.177$, the low-pressure region of the suction surface reaches a minimum. Later, as the SHR increases, the low-pressure area of the suction surface gradually increases. On the pressure side, the flow field migrates toward the shroud surface under the influence of the swirling flow, but a distinctly distinct area like the suction side is not formed on the pressure side. On the pressure side, we can find a point where the pressure and velocity are higher in the hub and shroud. When the $\mathrm{SHR} < 0.5$, there is no significant change in the change in the high-pressure region as the SHR increases. When the $\mathrm{SHR} \geq 0.5$, the transformation of the local high-pressure area of the blade surface in the pressure surface is apparent, indicating that the influence of the swirl on the hot fluid increases with the increase of the hot spot radius.



(a).1.270 (b).1.385 (c).1.524

(d).1.693 (e).1.905 (f).2.177

(g).2.540 (h).3.048 (i).3.81

Figure 5. the distribution of pressure and velocity on blade

4.2 The effect on temperature

The temperature profile of the turbine blade surface will change, creating a localized high-temperature region due to the presence of hot streaks. At the same time, the presence of the swirling flow affects the trajectory of the hot streaks to cause deviations, resulting in further changes in the local high-temperature region on the turbine blade. It is seen that the fluid temperature around the turbine blades in different blade span directions (Figure 6). The three graphs show that the temperature of the fluid at different leaflet locations is also very different: the difference in temperature variation at 0.5 pitch obviously, the temperature difference at the 0.3 pitch is second, and the difference in the 0.7 pitch is the smallest. At the 0.3 leaf spread (Fig.6(a)), the difference in temperature change tendency on the suction side with the increase in SHR is almost the same. However, the variation of the temperature change trend on the pressure side is more evident along the blade axis direction. It can be seen that the temperature change trend is not apparent when the SHR is less than 0.5, but when the SHR is more significant than 0.5, the temperature on the pressure side will have a high-temperature region at 0.5 in the axial direction, and then there will be a relatively low temperature at 0.65. The area then rises in temperature along the axial direction. It is evident that the region with the highest temperature is at the trailing edge, and the higher the SHR, the higher the maximum temperature at the trailing edge (figure 6(a)). It can be seen that when the SHR is more significant than 0.5, as shown in case 6, case7and case 8 in the figure6(b): the higher the SHR, the more obvious the change in temperature on the pressure side, at the axial distance of 0.2. A high-temperature region occurs because the vortex rotates the hot spot to the turbine blade area, while the other lower SHR conditions are that the vortex pulls the hot spot away from the blade surface to fuse with the surrounding fluid. When the SHR is more significant than 0.5, the fluid temperature at the trailing edge is higher than the fluid temperature at the leading edge, which may be higher than the thermal fringes, which cannot be mixed with the relatively cold fluid to reduce the overall heat. As shown in Figure 6(c), the temperature trends around the entire blade are similar. As the SHR increases, the temperature around the blade increases, but no high-temperature region occurs. From the figure, we can see that the improvement of SHR has a more significant influence on the lower part of the leaf exhibition, and there will be local high-temperature areas, so we need to control the value of SHR reasonably.

4.3 The effect on Eddy Viscosity

The distribution of eddy viscosity was investigated at three different locations along the pitch direction, as shown in figure8. It is found that the vortex viscosity of the three graphs has a similar distribution space: the eddy current intensity reaches a maximum at the axial 0.25 position and then gradually decreases, and there is another mutation at the axial position of 0.3 through the comparison of the three graphs. The viscosity has a relatively high lift. The eddy viscosity then continues to decrease. The eddy viscosity indicates that the motion of the fluid driven by the swirl is relatively slow, and the velocity of the fluid is relatively low. It can also be seen that the value of the eddy viscosity increases with increasing leaflet position: the eddy viscosity does not change much in the region below 0.5 pitch. In areas where the pitch is above 0.5, the viscosity of the fluid increases with the direction of the leaf height.

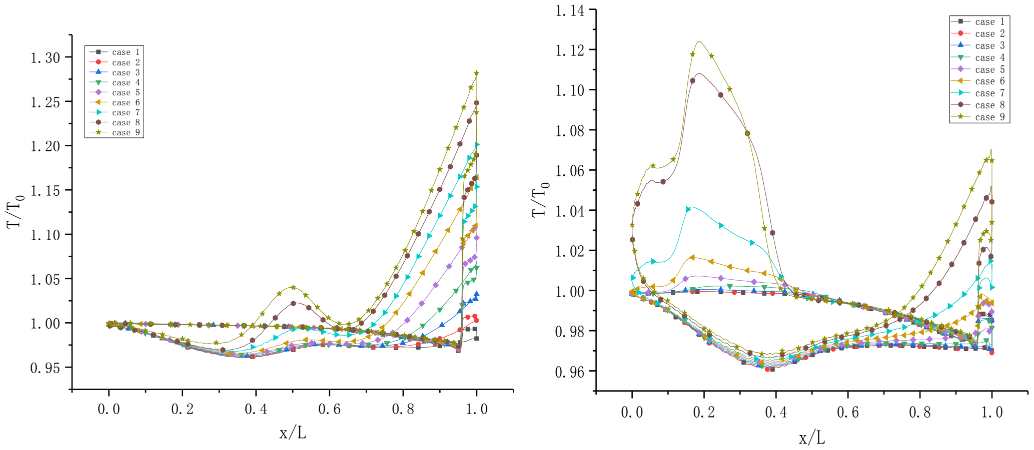
5. Conclusion

This paper study the effect of non-uniform temperature and non-uniform velocity found that different from the swirl and hot streaks radius radio, which have significant influence for the heat transfer on blade and flow field at blade passage. We can get some important conclusions which can help the designer to design the blade and film cooling system, so that improve the function of the engine.

The low-pressure region decreases at the suction surface with SHR increase by the combined effects of swirling and hot streaks, and the high-pressure area decreases at the pressure surface. On the side of the suction side, the fluid is displaced toward the hub, and there is a clear separation zone, and the size of the separation zone gradually decreases as the SHR increases. On the pressure side, the fluid is deflected toward the shroud face, with no distinct separation zones but with separate interfaces.

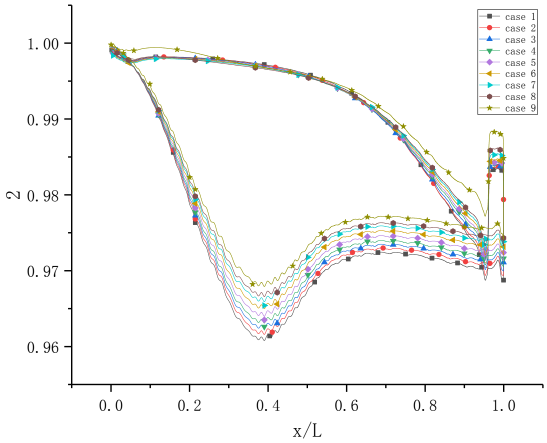
The local high-temperature region will be generated on the lower surface of the blade with the increase of SHR. The higher the SHR, the more obvious the local high-temperature position, and the more unstable the temperature distribution on the blade surface. The temperature of the blade trailing edge more than the leading edge.

Two eddy currents are created along the axial direction of the blade. The first position (ie, the area with the highest eddy viscosity) is at the axial position of 0.25 and the second relatively small position is at the axis of 0.35. These two locations are caused by the swirling flow, which has an essential influence on the load distribution of the blade and the motion of the flow field.



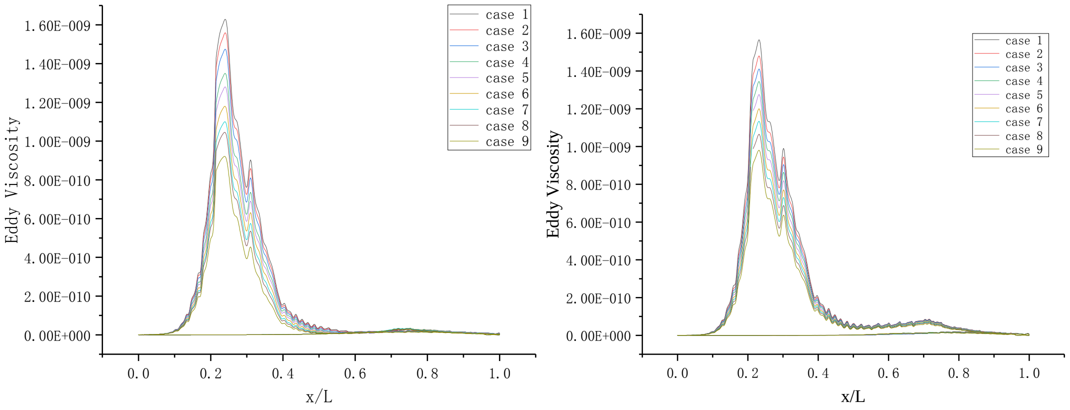
a. 0.3 Pitch

b. 0.5 Pitch



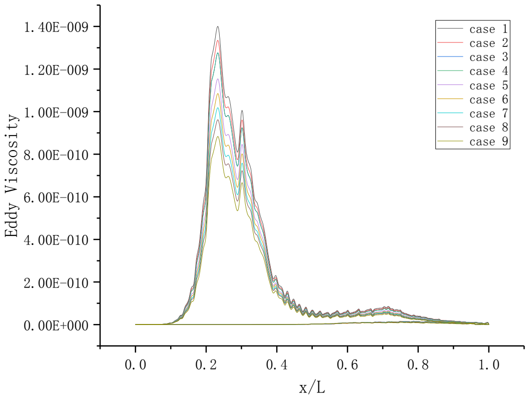
c. 0.7 Pitch

Figure 6. the distribution of temperature on the blade surface



a. 0.3 Pitch

b. 0.5 Pitch



c. 0.7 Pitch

Figure 8. the distribution of eddy viscosity on the blade surface

Acknowledgment

The authors gratefully acknowledge the financial support of the National Natural Science Foundation of China (No. 11672183 and 91641129). I am also grateful to Prof. Borovik Igor at Moscow Aviation Institute for the collaboration researches.

Reference

1. Meng Ni, Zhu H et al. Review of turbine blade cooling technology for aero-engines (in Chinese) [J]. Gas Turbine Technology, 2005, 18(4):25-33.
2. An BT, 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-TRANSACTIONS OF THE ASME, 2009, 131(3).
3. An B.T., Liu, J.-J. and Jiang, H.-D. (2008), "Numerical investigation of unsteady effects of the hot streak on flow and heat transfer in a turbine stage," Proceedings of ASME Turbo Expo, Berlin, Germany, GT2008-50415.
4. Kim S I, Rahman M H, Hassan I. Effect of turbine inlet temperature on rotor blade tip leakage flow and heat transfer [J]. International Journal of Numerical Methods for Heat & Fluid Flow, 2009, 22(1):73-93.
5. Wang J, Ge N, Sheng C. Analysis of swirling flow effects on the characteristics

of unsteady hot-streak migration [J]. Chinese Journal of Aeronautics (In English), 2016, 29(6).

6. Jenny P, Lenherr C, Kalfas A, et al. Effect of Hot Streak Migration on Unsteady Blade Row Interaction in an Axial Turbine[C]// ASME Turbo Expo 2010: Power for Land, Sea, and Air. 2010:2573-2584.

7. Jenkins, S., and Bogard, D. G., 2004, "The Effects of the Vane and Mainstream Turbulence Level on Hot Streak Attenuation," ASME Paper No. GT2004-54022.

8. Shang T, Epstein A H. Analysis of hot streak effects on turbine rotor heat load [J]. Journal of Turbomachinery, 1997, 119(3):V001T01A041.

9. Carullo J S, Nasir S, Cress R D, et al. The Effects of Free-stream Turbulence, and Turbulence Length Scale and Exit Reynolds Number on Turbine Blade Heat Transfer in a Transonic Cascade[C]// 2007:1077-1089.

10. Salvadori S, Montomoli F, Martelli F, et al. Aerothermal Study of the Unsteady Flow Field in a Transonic Gas Turbine With Inlet Temperature Distortions[J]. Journal of Turbomachinery, 2011, 133(3):031030.

11. Satta F, Ubaldi M, Zunino P. Unsteady Inflow Effects on the Wake Shed from a High-Lift LPT Blade Subjected to Boundary Layer Laminar Separation [J]. Journal of Thermal Science, 2012, 21(2):97-108.

12. Wang J, Ge N, Sheng C. Analysis of swirling flow effects on the characteristics of unsteady hot-streak migration[J]. Chinese Journal of Aeronautics, 2016, 29(6):1469-1476.

13. Huang Y, Yang V. Effect of swirl on combustion dynamics in a lean-premixed swirl-stabilized combustor[J]. Proceedings of the Combustion Institute, 2005, 30(2):1775-1782.

14. Khanal B, He L, Northall J, Adami P. Analysis of radial migration of hot-streak in swirling flow through high-pressure turbine stage. J Turbomach 2013;135(4):041005

15. Hylton L D, Nirmalan V, Sultanian B K, et al. The Effects of Leading Edge and Downstream Turbine Vane Heat Transfer[J]. Final Report General Motors Corp.indianapolis In.allison Gas Turbine Div, 1988