EFFECT OF COMPLEX FLOW PROFILES AT COMBUSTOR EXIT ON TURBINE CHARACTERISTICS

Li Yongfan, Na Zhenzhe, Chen Yun, Wang Yangbing

(AECC Shenyang Engine Research Institute, Shenyang 110015, China）

Abstract:

The flow profile at the exit of the aeroengine combustor has the characteristics of strong complex swirl, circumferential and radial temperature nonuniformity and which will result in local high temperature hot region that is called hot streak. It is great significance for turbine design to investigeat the turbine flow characteristics with different circumferential positions of the hot streak locating at the combustion oulet to the turbine nozzle. Based on this background, the relevant research is carried out through numerical simulation method, and the numerical calculation method satisfying the application of engineering design is investigated comprehensively considering the calculation accuracy and calculation scale. On basis of the typical combustion outlet flow field model, the effect of hot streak at circumferential positions of the combustion outlet on turbine characteristics is obtatined. based on the investigation， it is found that the hot streak at the outlet of the combustion has little effect on the turbine performance parameters, but has great effect on the temperature field of turbine nozzle and turbine blade, which is of great significance to improve the blade reliability by avoiding local high temperature on the blade surface at specific circumferential position.Based on the technology the local high temperature phenomenon of turbine casing at combustion outlet caused by temperature field distortion of combustion is simulated, the flow characteristics at different inlet temperature field are compared and analyzed, and the numerical method is further verified with the measured wall temperature.

Keywords: aeroengine; hot streak; high pressure turbine; unsteady migration; ablation erosion

1 Introuduction

The circumferential and radial temperature nonuniformity of combustion outlet flow field is integrated by combustion stuctre, combustion organization and cooling mechanism resulting in local maximum temperature being twice the lowest temperature[1], this local high temperature region hot streak. When hot streak flows into turbine, the unsteady effect of flow field will be more and more serious, and the added secondary flow will be generated[2]. Meanwhile, due to the relative motion between turbine stator and blade, high and low temperature airflow in the blade row is caused by migrating, which resulting in blade surface overheating ,generating the huge heat load or even the ablation erosion[3,4]. Reently, with the improvement of the temperature rise of the combustion and the development of the cooling dirction of the short ring and strong jet, the inuniformity of the outlet temperature of the combustion is worse. Under the secondary flow (including cascade endwall hoseshoe vortex, horn thirst, channel vortex) in the turbine, film cooling and interference between the blade and the stator, the turbine flow field will be more complex.

Previously, foreign researchers mostly used experimental methods to carry out the effects of inlet hot streaks in gas turbine[5,6], but because of the high cost of the test, especially the combustion generator, foreign researchers began to choose numerical simulation methods to simulate turbine inlet hot streaks. In the early numerical simulation, two-dimensional calculation method was adopted[7,8], With the development of computer performance, the full three-dimensional unsteady simulation method was widely used in considering the characteristics of spatial asymmetry and flow unsteady characteristics of hot streaks, meanwhile, domestic researchers are also paying attention to the hot streak effect. Zou Zhengping team of Beijing University of Aeronautics and astronautics investigated the effect of the blade installation angle deviation on hot streak migration in turbine channel by using three-dimensional unsteady simulation[9], Feng Zhenping team of Xi'an Jiaotong University also conducted a series of thermal hot research[10,11].

Investigation of turbine temperature field and flow field when there is hot streak will be helpful to carry out the research on turbine design technology considering the effect of hot streak, that is an important direction for turbine development from traditional design to fine design.At present, the commonly turbine full three-dimensional unsteady simulation methods used mainly include the whole ring full three-dimensinal unsteady simulation and full three-dimensional unsteady simulation based on the domain scaling, but the following problems will be faced in practical engineering application,that are not suitbale for engineering applications: the total periodic grid partition of the blade rows in the turbine is really needed; the number of grids are large; the computational resources and time consuming are very large considering the effect of the air conditioning outlet flow mixing of the blades; the calculated flow field and aerodynamic performance deviate from the actual condition. In this paper an the efficient and fast method of algorithm for the calculation of the algorithm is investigated, and the technology is applied to many cases of turbine engineering, and some results are obtained.

2 Turbine simulation method considering the exit flow profiles of the combustor

Due to combustor hot streaks have the characteristics of spatial asymmetry and flow unsteadiness, the full three-dimensional unsteady simulation method is used to analyze the flow field of the turbine. However, conventional three-dimensional unsteady simulation method has limited applicability when dealing with practical engineering problems. Therefore, an efficient, accurate and full three-dimensional unsteady simulation method suitable for engineering applications is urgently needed, which is on balance of computing accuracy and computing scale.

2.1 Introduction of single-passage unsteady numerical simulation method

Due to the limitation of computing resource and time cost, the three-dimensional unsteady simulation method of the work loop is generally infeasible in practical engineering applications. Therefore, more researchers use the unsteady simulation based on blade reduction method. But there is a prerequisite for implementing this method, the turbine rotor/stator computing domain on either side of the interface should have the same circumferential size, as shown in Fig. 1, turbine which the number of rotor/stator blade can be reduced can be realized by copying the number of blade passage, the number of turbine rotor/stator blade usually inconvenience reduced, mostly prime, therefore need to scale the blade profile to adjust the number of blade by certain rules. Thus, it deviates from the real turbine, and the computing results will deviate from the actual working condition.

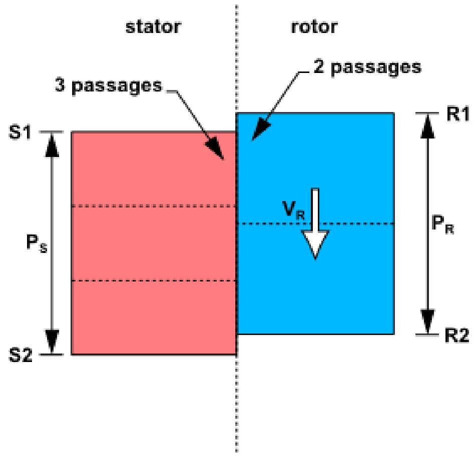


Fig.1 Simulation of domain scaling on the interface between rotor and stator

The phase-lagged periodic boundary condition proposed by Erdos et al.[12] and the time-inclined idea proposed by Giles[13] provide a new concept for the realization of single-passage three-dimensional unsteady simulation. According to such idea, ANSYS CFX proposed Transient Blade Row（TBR） modeling theory based on phase shifed periodic boundary condition to realize the three-dimensional unsteady simulation of turbomachinery with single-passage or less-passage, the following figure illustrates the common principle used in the TBRow method. The basic principle of a phase-shifted periodic condition is that the pitch-wise boundaries R1/R2 and S1/S2 are periodic to each other at different instances in time. For example the relative position of R1 and S1 at t_0 is reproduced between sides R2 and S2 at an earlier time t_0-ΔT where ΔT is defined by $(P_R\text{-}P_S)/V_R$. Here P_R and P_S are the rotor and stator component pitches, respectively, and V_R is the velocity of the rotor.

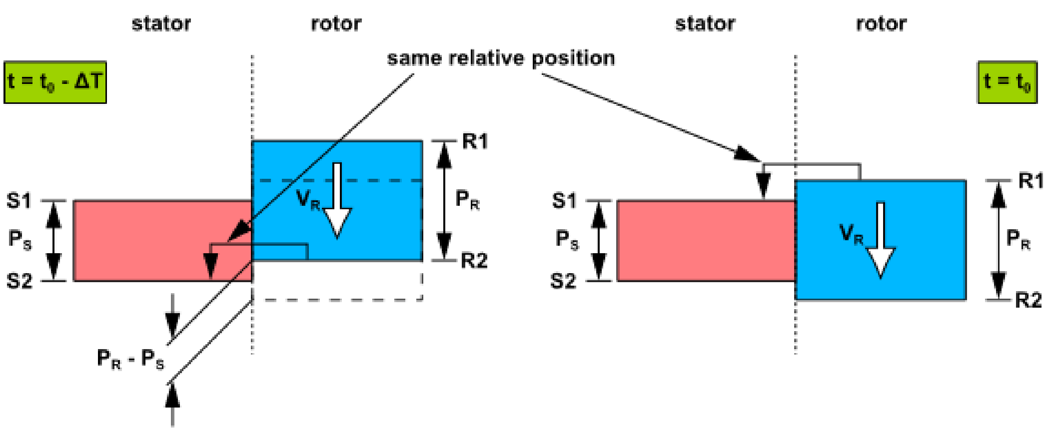


Fig.2 Phase shifed periodic boundary conditiongs

The time transformation model (TT) in TBR is usually used when there is a big difference in the number of rotor/stator blades, that means the pitch ratio of guide/rotor cascade deviates far from 0.1, but there are certain requirements for the pitch ratio of guide/rotor cascade when using this method, which is generally in the range of 0.6～1.5. For large-scale military turbofan, the number of rotor/stator blades of high pressure turbine is generally 1.5～2.0, so three dimensional unsteady numerical simulation can be carried out with single channel guide vanes and double channel guide blades.

3 Analysis of the effect of the combustor outlet flow field on turbine characteristics

3.1 Construction of a typical combustor outlet flow field model

According to OTDF at the outlet of combustor, the inlet temperature field of high pressure turbine with a single hot streak is constructed. As shown in Fig.3, the hot streak area is circular, the maximum temperature value is located at the center of the circle, and the temperature field distribution is sinusoidal. Through adjusting the center coordinates of circular hot streak, the distribution of hot streak at different positions relative to the inlet of high-pressure turbine can be obtained.

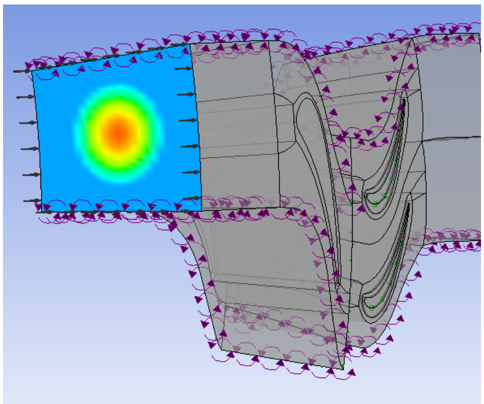


Fig.3 Temperature field model of high pressure turbine inlet with hot streak distribution

3.2 Numerical simulation and characteristic analysis under different flow fields

According to the high pressure turbine inlet temperature field modeling with hot streak, two different relative circumferential position is constructed, that the hot streak is in front the leading edge of stator blade and the middle of high pressure turbine vane . Analyzing the migration characteristics of hot streak, the comparison of surface temperature field of the blade and the influence on turbine performance by 3D unsteady numerical simulation of high pressure turbine under stage condition.

3.2.1 Analysis of turbine flow field with the hot streak front stator blade

Fig.4 shows the inlet temperature distribution with the hot streak front the leading edge of stator blade. Each stator blade corresponds to a hot streak, which is evenly distributed in the whole annular inlet field.

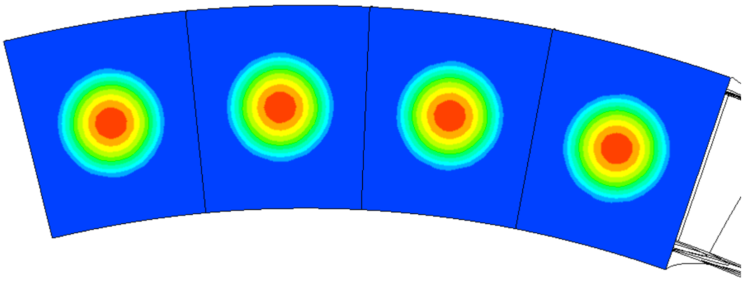


Fig.4 Hot streak front the leading edge of stator blade

Fig.5 shows the entropy nephogram of the turbine middle section when t=0, according to this entropy nephogram, the flow field unsteady calculated by TBR method has good consistency at the interface, and the upstream response is correctly transmitted to the downstream of the flow field by time delay method. In addition, the distribution of entropy is consistent with the inlet temperature in stator blade passage, that is ,the higher the temperature, the greater the entropy. Because the inlet high temperature fluid will impact different positions of the downstream rotor blade after passing through the stator blade, it will swing periodically on the leading edge of rotor blade and form a ripple wake after passing through the rotor blade.

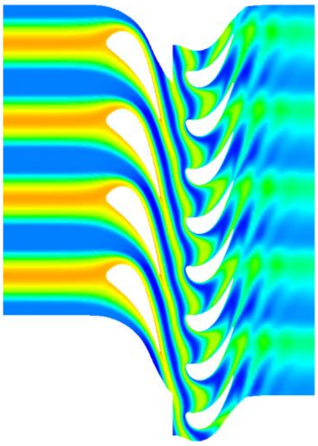


Fig.5 Entropy nephogram of turbine middle section when t=0

Fig.6 shows the time-average Mach number nephogram of turbine middle section. According to the Mach number distribution, the existence of hot streak has no significant effect on the distribution of Mach number,which is similar to the condition that without hot streak. In time-average absolute total temperature nephogram, the upstream and downstream total temperature are not continuous at the interface and there is an obvious discontinuity, which is due to the impact of the high temperature fluid on different positions of the leading edge of rotor blade within one cycle after passing through the stator blade, after time averaging, inlet total temperature of rotor blade becomes a uniform distribution.

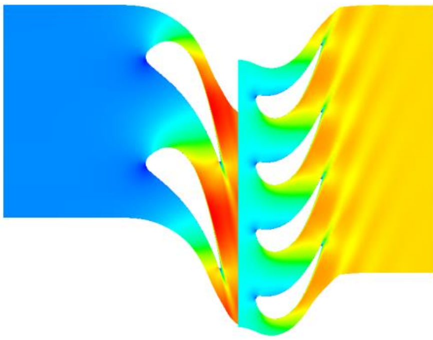
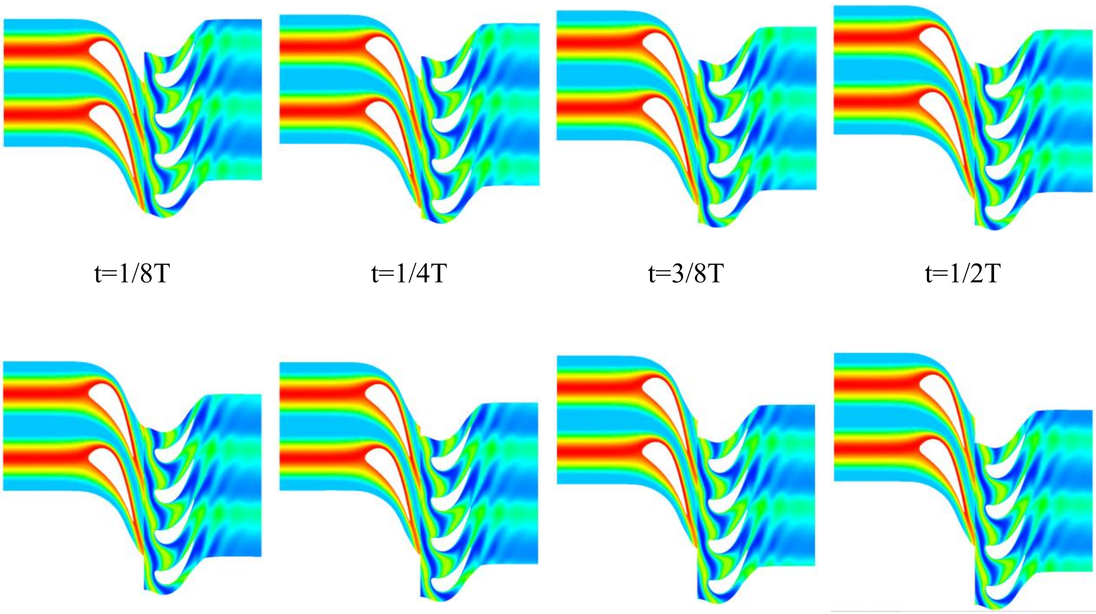


Fig.6 Time-average Mach number nephogram of turbine middle section

Fig7 shows the nephogram of the relative total temperature of the turbine blade middle section at different times in a cycle. It can be seen that the high-temperature airflow is devided into two high-temperature zones by the guide blade when passing through the guide vane passage, and flows downstream along the suction surface and pressure surface of the guide blade , finally merges behind the trailing edge of the guide blade to form a high-temperature wake flow to the rotor blade. With the rotation of the rotor blade, the high-temperature wake of the guide blade is cut by the leading edge of the rotor blade, and bends, stretched and deforms in the passage of the rotor blade, the unsteadiness is very obvious. With the expansion and cooling of the mainstream in the rotor blade passage and mixing with the mainstream , the outlet temperature of the rotor blade has obviously decreased , but due to the influence of the wake of the rotor balde, a corrugated temperature distribution is formed at the outlet of the rotor blade.



t=1/8T t=1/4T t=3/8T t=1/2T

t=5/8T t=3/4T t=7/8T t=T

Fig.7 Relative total temperature nephogram of turbine middle section in a cycle

Fig. 8 and 9 show the static temperature nephogram of rotor blade concave and convex surface at 8 consecutive times in a cycle. The hot streak at the turbine inlet migrates to the rotor blade surface after passing through the high pressure guide vane. It can be seen from the figure that the overall temperature of the rotor balde concave surface is higher than that of the convex surface, so the trend of high-temperature fluid migrating to the rotor blade pressure surface is more obvious in the turbine. Significantly, there is a also a small local high temperature area on the suction surface, which mainly exists near the edge of the blade suction surface. This is mainly due to the hot fluid split into two parts at the leading edge of the rotor, most of the hot fluid flows around the leading edge to the pressure surface of the rotor, and a small part of the hot fluid flows to the suction surface of the rotor blade.

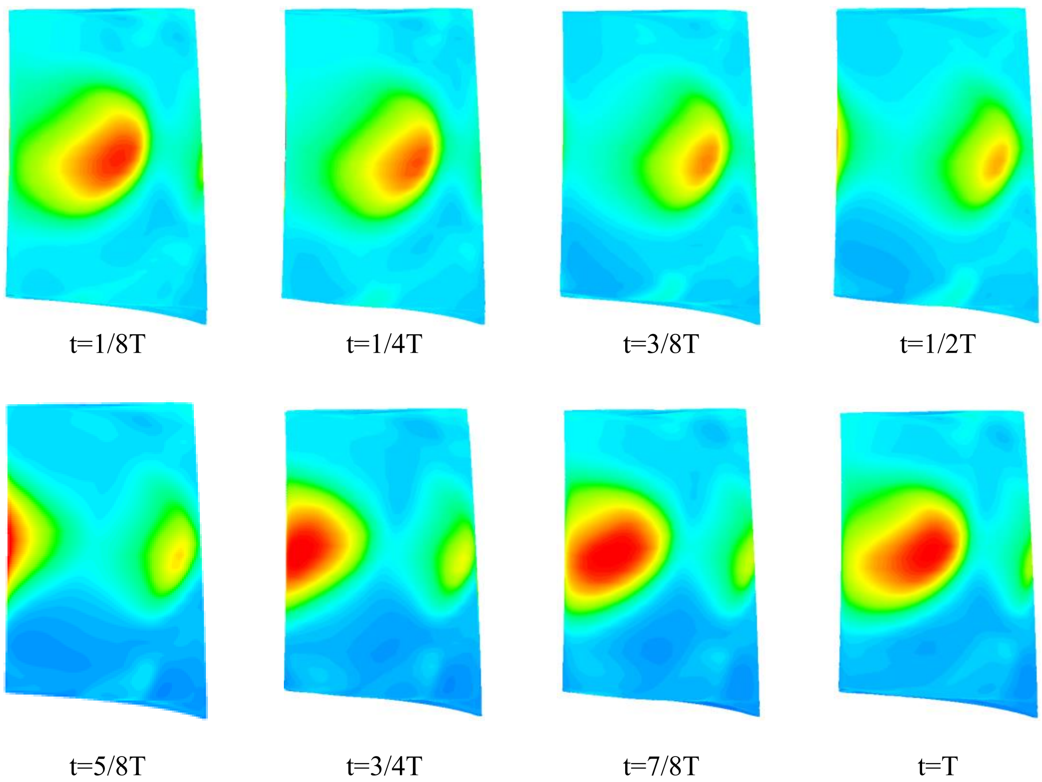
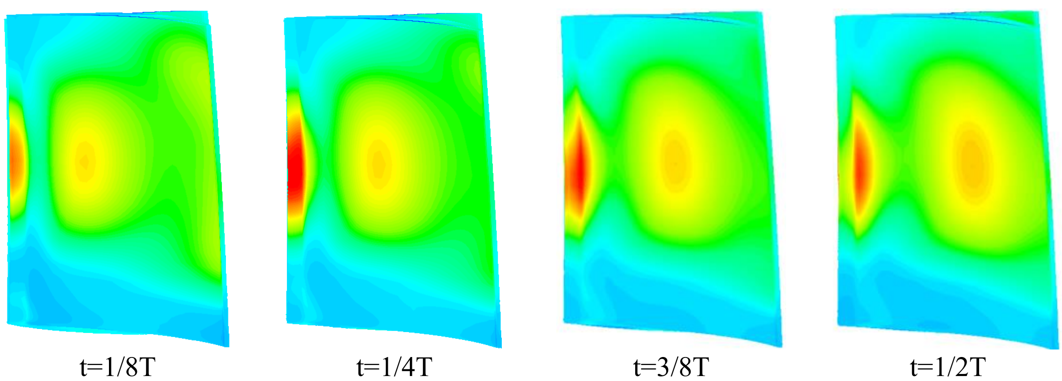
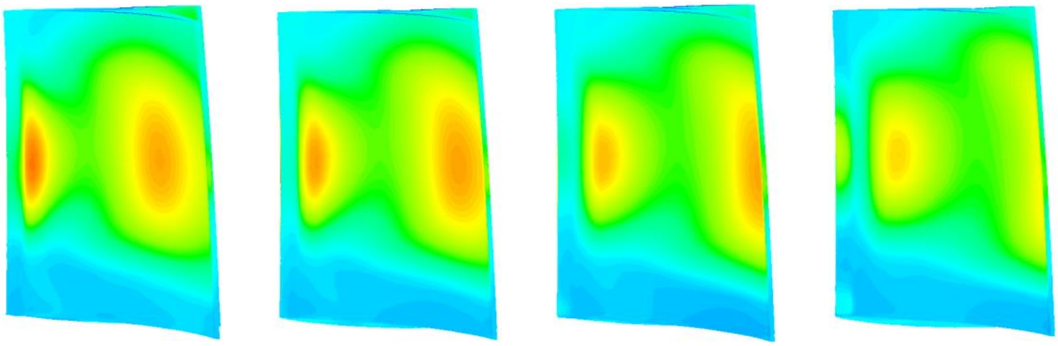


Fig.8 Static temperature distribution of suction surface in a cycle





t=5/8T t=3/4T t=7/8T t=T

Fig.9 Static temperature distribution of pressure surface in a cycle

3.2.2 Analysis of turbine flow field with the hot streak facing the slot

Fig.10 shows the entropy nephogram of the mid-section of the turbine at t=0. When the hot streak facing the centre of the slot, the hot fluid flows through the middle of the stator blade slot, while the cold fluid surrounds the stator blade.Therefore, when the hot streak is located at this position, the temperature on the surface of the stator blade is lower, which is conductive to the cooling of the stator blade.

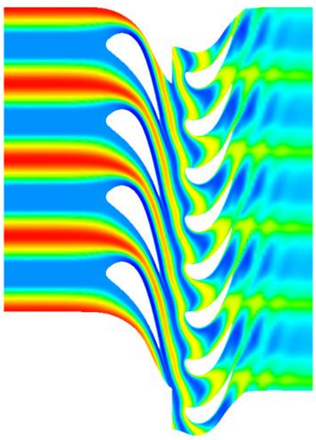


Fig.10 Entropy nephogram of the cross section of the turbine at t=0

The time-averaged Mach number nephogram of the mid-section in the turbine is shown in Fig.11. It can be seen that the change the position of the hot streak has little effect on the Mach number. Compared with the instance that the hot streak facing the leading edge of the stator blade, where there is no change in Mach number nephogram.

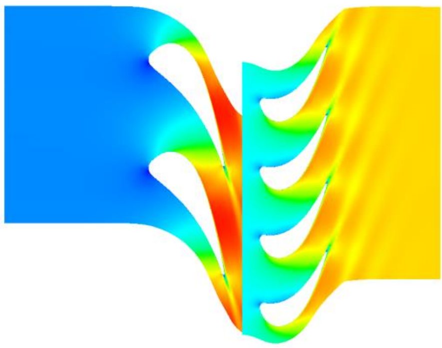
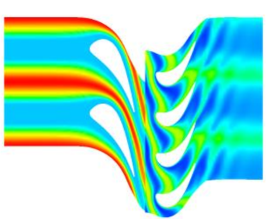
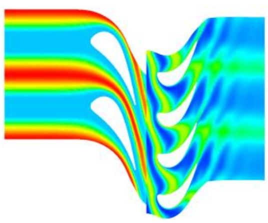
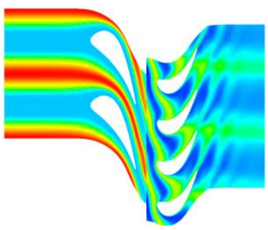
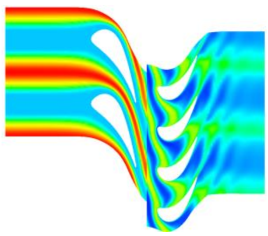
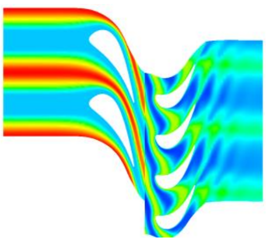
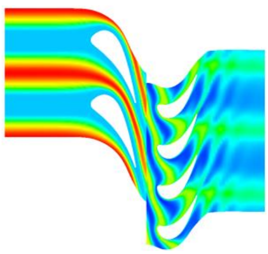
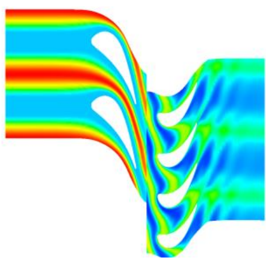
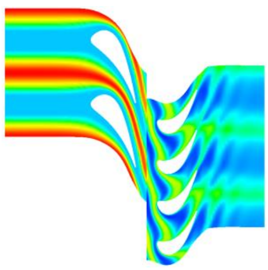


Fig.11 Time-average Mach number nephogram of the mid-section in the turbine

Fig.12 shows the relative total temperature nephogram of the mid-section of the turbine in a cycle. It can be found that the change of the hot streak will change the temperature distribution in the stator blade. The flow field structure of the hot streak in the rotor blade is similar to that of the leading edge of the stator blade.

t=1/8T t=1/4T t=3/8T t=1/2T

t=5/8T t=3/4T t=7/8T t=T

Fig.12 Relative total temperature nephogram of turbine middle section in one cycle

In order to analyze the temperature when the hot streak is located in the center of the channel, static temperature nephogram of the blade back and the blade basin in a cycle is made similar to that of Section 3.2.1(Fig.13 and 14). The temperature profile is similar to that of Section 3.2.1, this shows that when the hot streak facing the center of the channel, the hot streak still trips toward the pressure surface of the blade, so the temperature of pressure surface is higher than that of suction surface. However, the comparison with Section 3.2.1 shows that, although the migration of hot streaks is similar, but when the hot streak is facing the center of guide vane channel, the surface temperature of the blade basin is significantly higher than that of the blade basin when the hot streak is facing the leading edge of the guide vane.

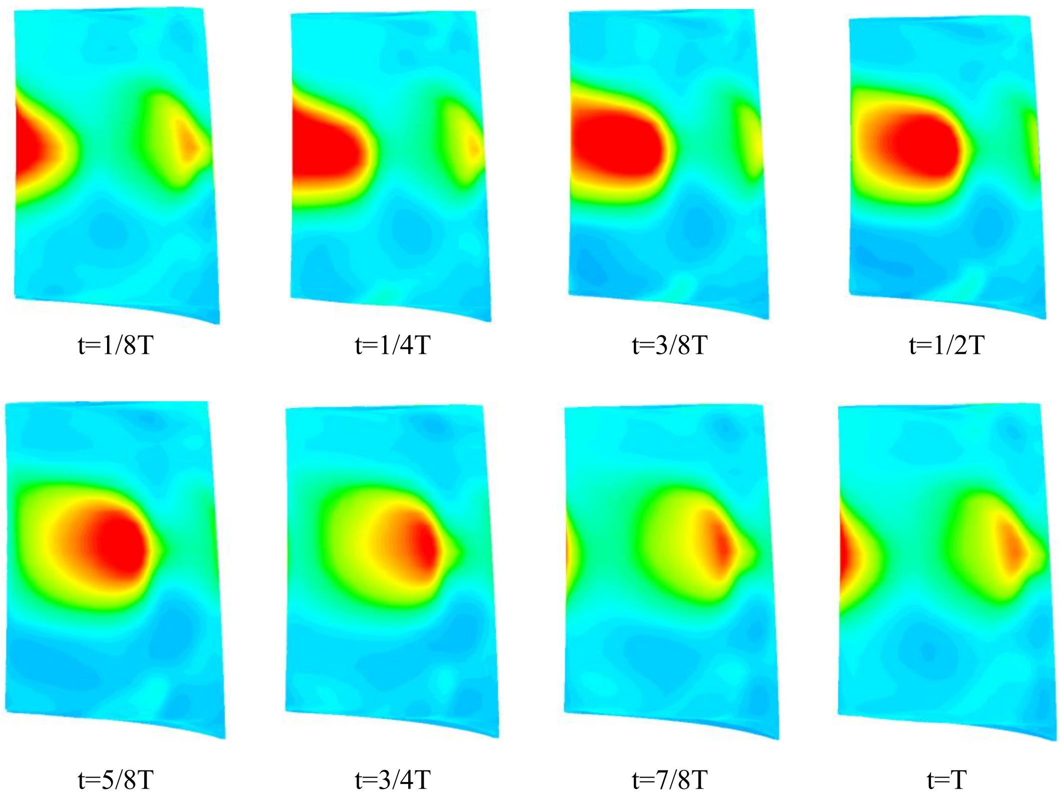


Fig.13 Static temperature profile of suction surface in a cycle

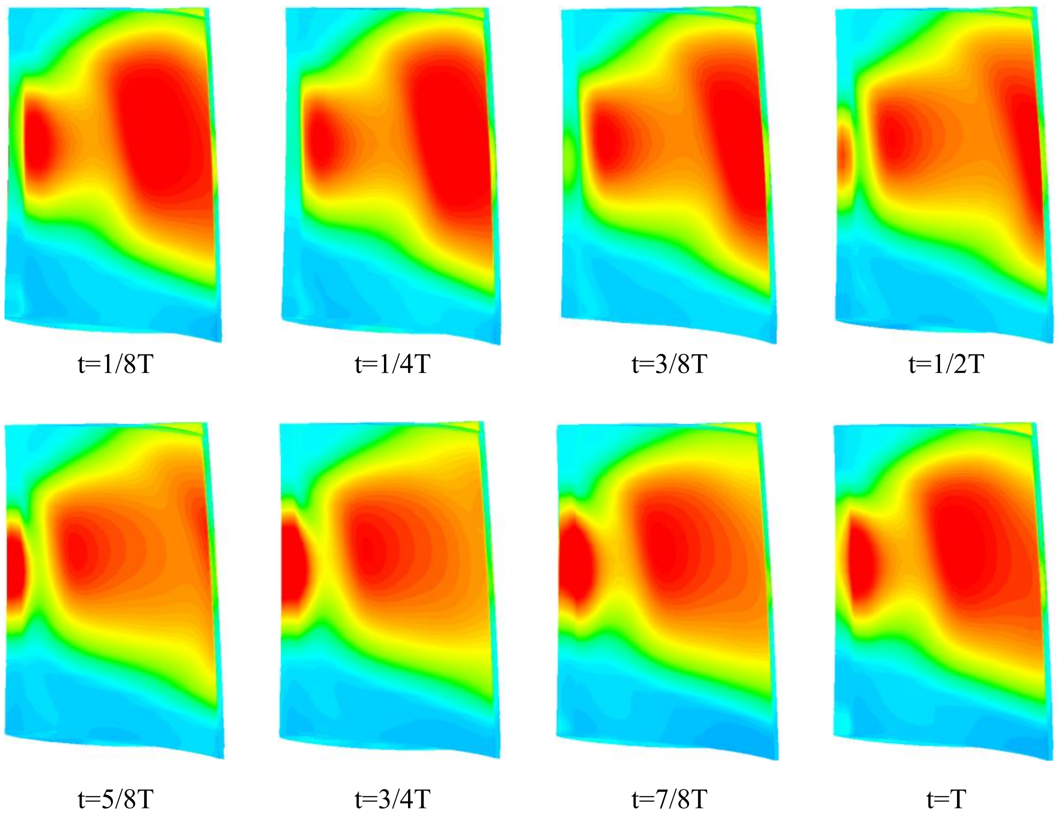


Fig.14 Static temperature profile of pressure surface in a cycle

4 Application in engineering development

4.1 Ablation fault location of turbine

It is also found that high pressure turbine ablation occurred during the test, the position of anti-surge pipe was adjusted for the test engine, which is necessary to further locate the fault mechanism by numerical simulation. The complex flow field of the combustor outlet is calculated and analyzed to provide guidance for fault location and improvement design.

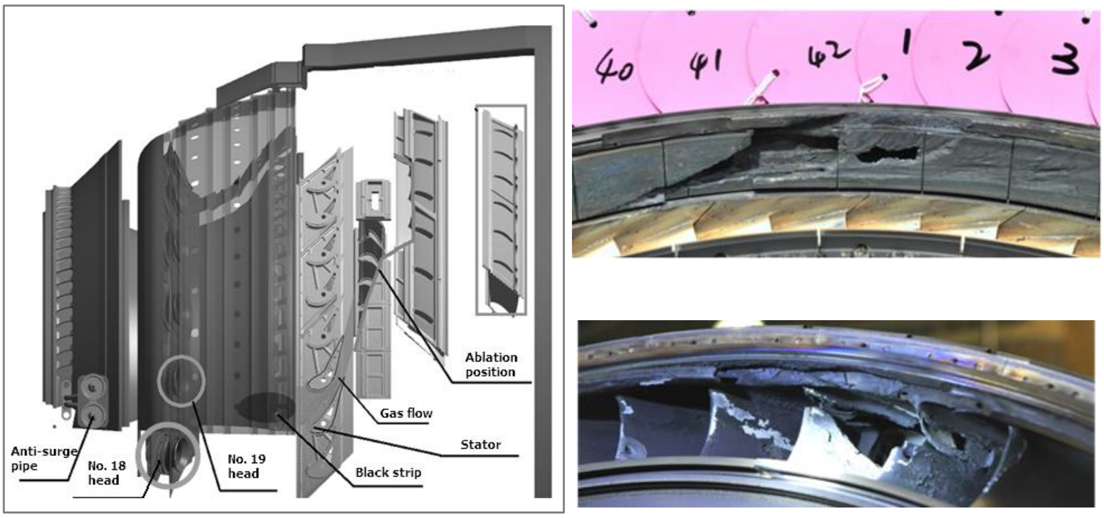


Fig.15 Local ablation of turbine

4.2 Numerical simulation analysis

a) Establishment of calculation model and explanation of model simplification

The inlet of high pressure turbine is given according to the temperature field and velocity field of combustor outlet, the mass source term is used to simulate the cold air at the blade and flange of high pressure turbine, CFX software is used to solve the steady and unsteady flow field.

The tip clearance and groove structure of high pressure turbine blade are considered. In order to simulate the flow path of hot mass from the inlet of high pressure turbine to the low pressure guide vane, the calculation model of high pressure turbine plus low pressure guide vane was adopted.

The transient interface between the rotor and stator can capture the flow field structure of the hot mass passing through the high pressure turbine blade and entering the low pressure guide vane. As shown in the Fig.16, the fault was analyzed through the above calculation, the temperature field of the main stream and related parts were qualitatively explained when the fault occurred.

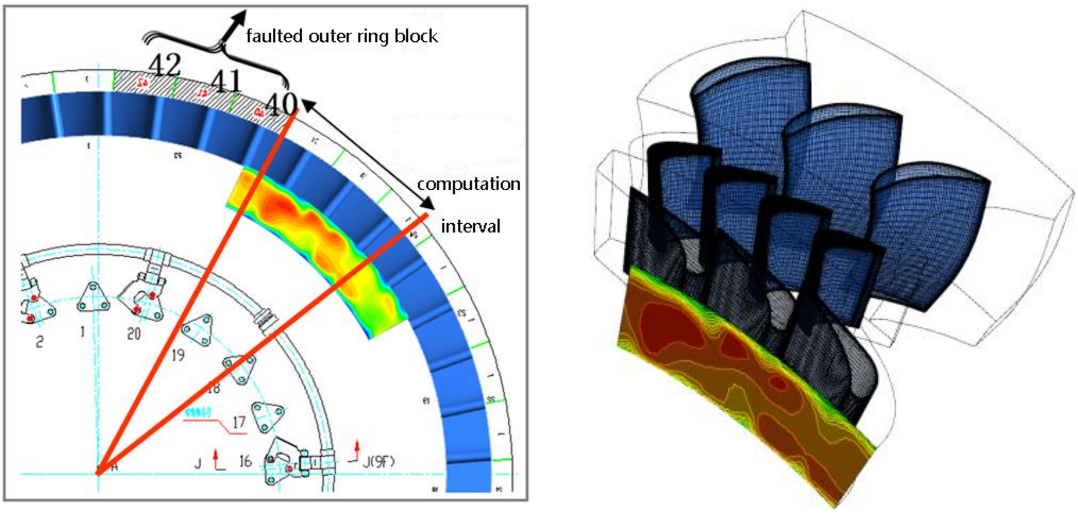


Fig.16 Computational model and boundary treatment

b) Analysis of numerical simulation results

The combustion chamber simulation results show that the existence of 3 obvious hot streaks at the exit of the No.18 and No.19 flame tube heads, corresponding to the surge preventing pressure signal extraction tubes. As shown in Fig.17, the hot streak No.1 corresponds to the high pressure turbine vanes, the hot streaks No.2 and No.3 correspond to the cascade passage close to the upper platform, and the swirl directions of these hot streaks are all different.

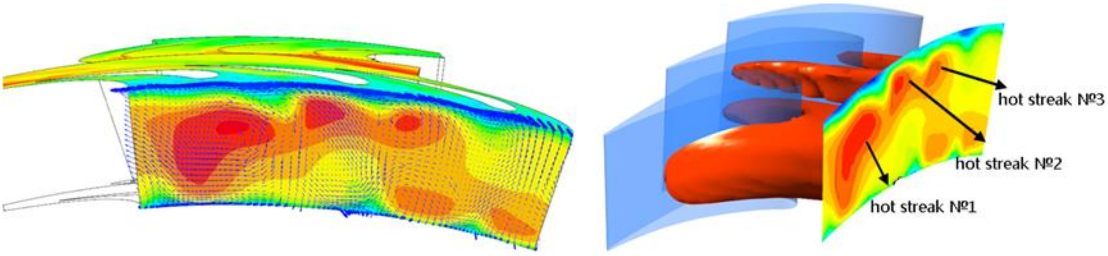


Fig.17 Hot streaks at the exit of the combustion chamber, corresponding to the No.18 and No.19 flame tube heads

Taking the temperature field and the velocity field at the exit of the combustion chamber as the inlet conditions, the stationary three-dimensional simulation of the high-pressure turbine nozzle were carried out. As shown in the Fig.18, the hot streak No.1 flows back through the middle part of the high-pressure turbine vanes and has less influence on the wall temperature of the high-pressure turbine outer ring; The hot streaks No.2 and No.3 flow through the cascade passages and move downstream under the action of the inlet swirling flow and the vortex flow in the inner cascade passage; The hot streak No.2 has a relatively large range and is more close to the upper platform, it rapidly rises up under the action of the vortex flow in the passage after entering the cascade and forms an obvious hot streak on the outer end wall, which corresponds to the position of the leading edge of the faulted high-pressure turbine outer ring.

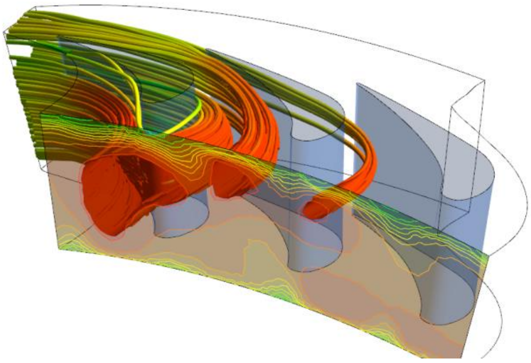
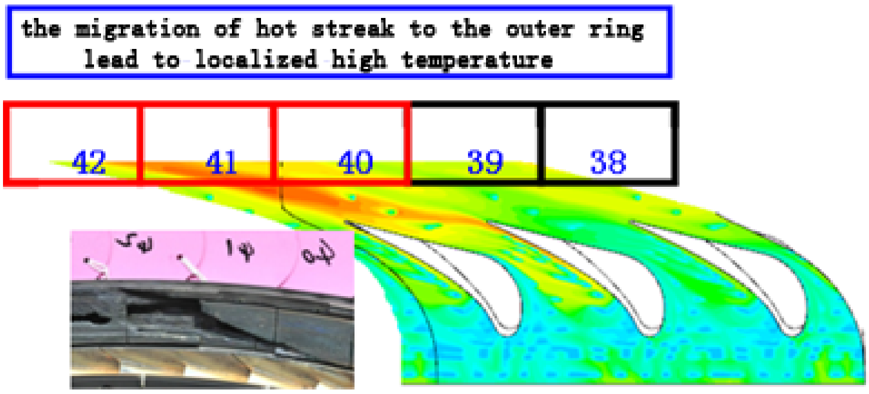


Fig.18 Flow path of hot streak entering the guide vanes



In order to further demonstrate the inter-action relationship between the outer ring ablation and LPT guide vane ablation, it conducts an unsteady calculation of the HPT and LPT with inlet hot streak. Then the migration rules of No.2 hot streak is preliminary obtained. It can be observed through the result of the calculation that the areas which the No.2 hot streak flowing along with is basically consistent with the ablation positions of LPT guide vanes, as seen in Fig.20.

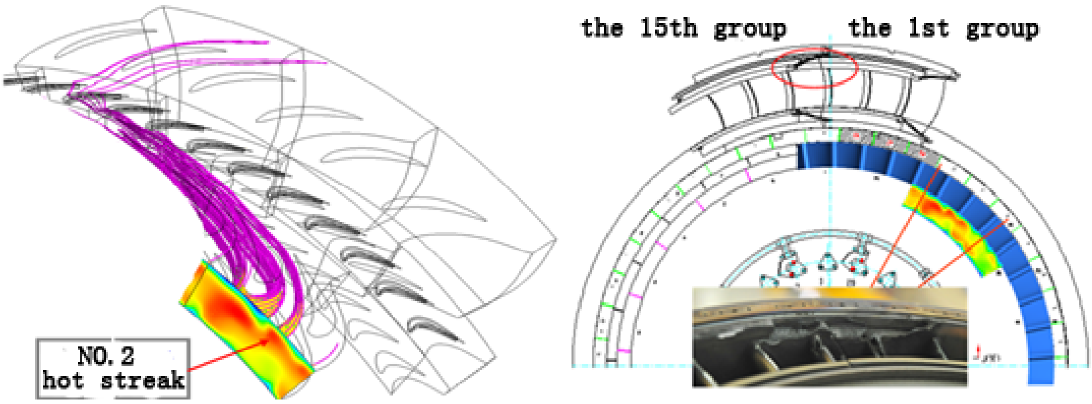


Fig.20 Hot streak entering LPT through HPT and the corresponding damage areas

4.3 Design improvement and test demonstration

It makes a calculation of the corresponding HPT section of No.18 to 19 heads after adjusting anti-surge pipe angular position. Then the variation rule of gas temperature for turbine tip endwall before and after the improvements is analyzed.

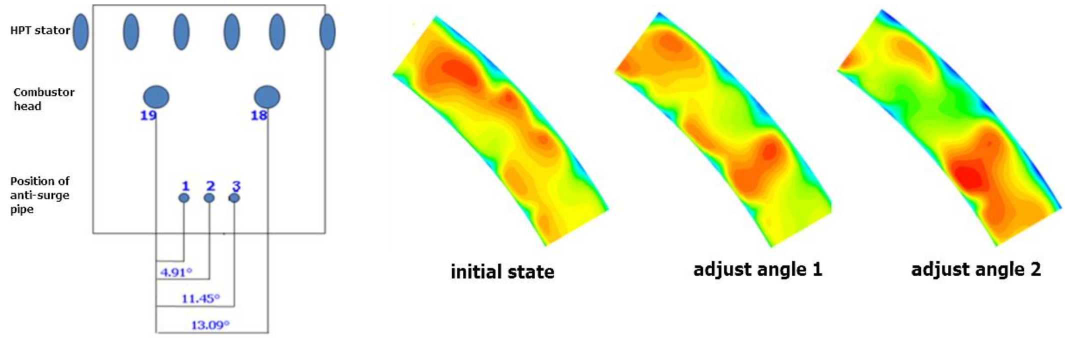


Fig.21 Turbine inlet temperature field characteristics in three relative position states

After adjusting the position of anti-surge tube, the original No.2 hot streak is eliminated, and the corresponding hot streak of No.18 head is migrated to the outer ring, so that the hot streak temperature of gas is reduced by about 190K at most.

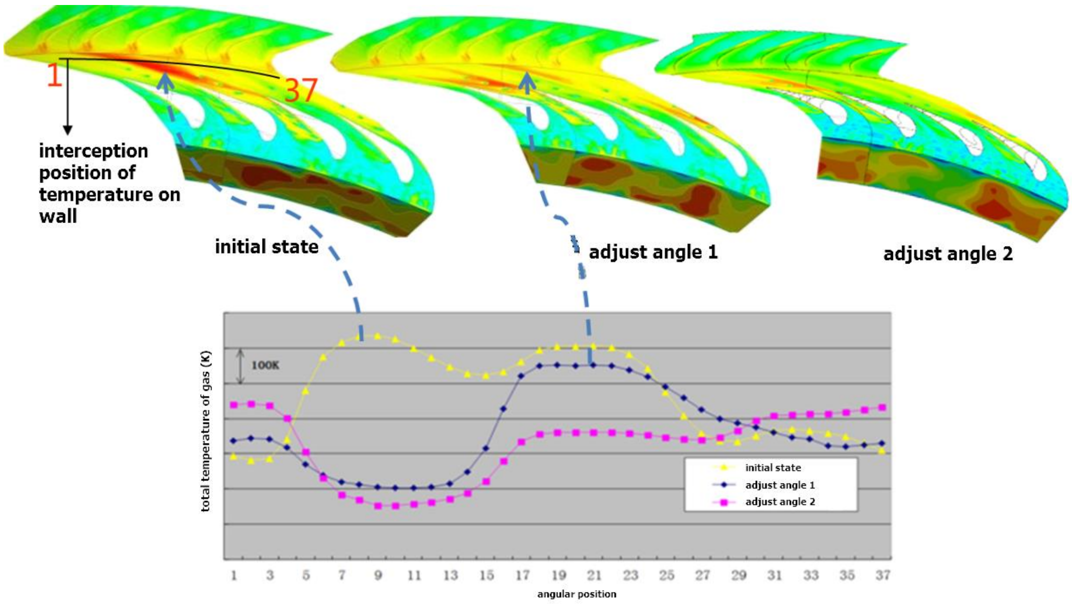


Fig.22 Comparision of the temperature distribution of the the turbine outer ring in the corresponding three temperature field states

Comparing the adjustment angle 1 and 2, it is found that the direction of fluid velocity at the hot streak has a greater fluence on the migration of the hot streak, and the hot streak area on the right side of the head corresponding to adjustment angle 2 has a smaller velocity gradient along the radial direction than the anti-surge tube angle adjustment scheme, which also inhibits the migration of the hot streak to the outer ring.

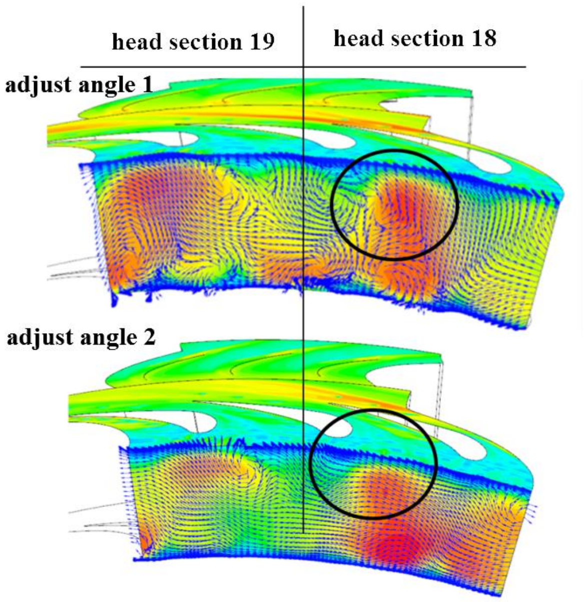


Fig.23 Characteristic analysis of inlet swirling flow for various adjust angles Concepts

To verify the numerical simulation result, the wall temperature measurement of the turbine shrould is accomplished for two conditions. The trend of wall temperature measuring results are well coincided with the theoretical calculating. The verifing test of the modified concept was accomplished without the similar question.

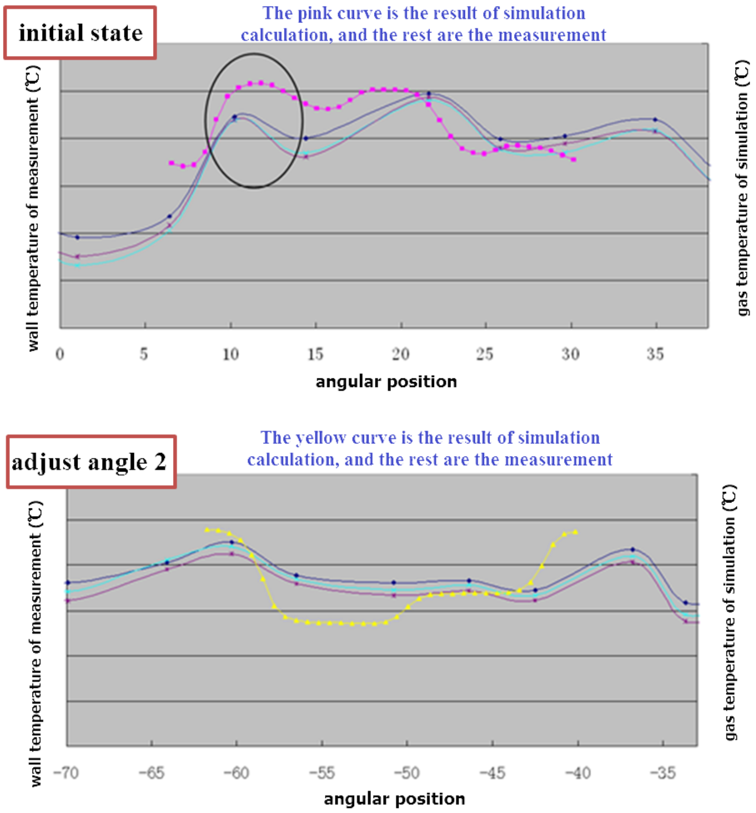


Fig.24 Comparison of wall temperature measurement of the modified concept

5 Conclusion

The research indicates that it is important to consider the flow characteristics of the combustor outlet in air-cooled turbine. The engineering application need can be accomplished Through single passage simulation. The hot streak and swirling directions have little influence on turbine performance, while great influence on the temperature profiles at stator vane and rotor blades. At the specified clocking position, the local high temperature on blade can be avoided. Based on this technology, the fault location and modified concept of the hot section components ablation have been accomplished during engineering development, and the experimental verification is successful.The design criterion of the hot section components with considering flow characteristics has been improved. The coupled simulation and integrated design for the combustor and turbine would still be the research directions.

References

1. Huang Jiahua ， Wang Huishe ， Zhao Qingjun ， et al． Development of research on turbine inlet hot streaks[J]． Aerodynamic Missile Journal，2007(10)：47-50(in Chinese)．
2. Lakshminarayana B., Horlock J G. Effects of inlet temperature gradients on turbomachinery performance[J]. ASME Journal of Engineering for Gas Turbines and Power, 1975, 97(1): 64~ 74.
3. Krouthen B, GilesM B. Numerical investigation of hot streaks in turbines. Journal of Propulsion and Power, 1990, 6(6): 769~ 776.
4. FENG Zhenping, WANG Zhiduo and LIU Zhaofang. Review on Research of Hot Streak Migration Mechanisms in Gas Turbine Stage[J]. Chin.Soc.for Elec.Eng., 2014, 34(29): 5120-5130.
5. Butler T L, Sharma O P, Joslyn H D, et al. Redistribution of an inlet temperature distortion in an axial flow turbine stage[J]. Journal of Propulsion and Power, 1989, 5(1): 64-71.
6. Shang T, Guenette G, Epstein A, et al. The influence of inlet temperature distortion on rotor heat transfer in a transonic turbine[C]. 31st Joint Propulsion Conference and Exhibit. 1995: 3042.
7. Krouthen B, Giles M B. Numerical investigation of hot streaks in turbines[J]. Journal of Propulsion and Power, 1990, 6(6): 769-776.

8. Dorney D. Numerical investigation of hot streak temperature ratio scaling effects[C]. 34th Aerospace Sciences Meeting and Exhibit. 1996: 619.
9. Li Yu，Zou Zhengping，Liu Huoxing，et al．Influence of blade-stagger departure on the migration of hot streak in turbine stage[J]． Journal of Engineering Thermophysics，2009，30(6)：944-948(in Chinese)．
10. Liu Z, Liu Z, Feng Z. Unsteady analysis on the effects of tip clearance height on hot streak migration across rotor blade tip clearance[J]. Journal of Engineering for Gas Turbines and Power, 2014, 136(8): 082605.
11. Liu Z. F., Wang Z. D. and Feng Z. P. et al. Effect of Rotor/Stator Interaction and Tip Clearance Height on Hot Streak Migration in Turbine Passage[J]. Journal of Engineering Thermophysics, 2015, 36(11): 2344-2347.
12. Erdos J I, Alzner E, McNally W. Numerical solution of periodic transonic flow through a fan stage[J]. AIAA Journal, 1977, 15(11): 1559-1568.
13. Giles M. Stator/rotor interaction in a transonic turbine. J. Propul. Power., 6:621-627, 1988.