Influence of Circumferential Uneven Inlet Conditions on the Aerodynamic Performance of Turbine Stator

Bing Ji\textsuperscript{1}, Zhiduo Wang\textsuperscript{*}, Jun Li\textsuperscript{1}

\textsuperscript{1}Aeronautics Engineering College, Air Force Engineering University, Xi’an, China

\textsuperscript{*} Corresponding author: wangzhiduo@foxmail.com

Abstract. The application of the combination of rotating detonation combustor and turbine to propulsion system is a recent research focus. This paper verifies the feasibility of this combined propulsion method through numerical simulation, and the flow law and flow loss mechanism in the turbine stator passage under complex inlet conditions are obtained. The results show that the flow field in the turbine stator has serious circumferential unevenness. The interaction between the rotating pulsating wave and the blade is the main cause of flow loss.

1. Introduction
Rotating detonation is another mode that exists in the combustion process. It has significant advantages such as high thermodynamic efficiency, fast energy conversion, self-pressure gain, and small entropy. The concept of rotating detonation combustion was first proposed by Voitsekhovskii\cite{1}. In recent years, the energy efficiency of the rotating detonation combustion and the propagation mechanism of detonation wave have been widely studied through experimental tests and numerical calculations \cite{2, 3}. However, there are relatively few studies on the combination of rotating detonation combustor and turbine. Only simple experiments and numerical simulations are done\cite{4, 5}, and there are few studies on the flow mechanism inside the turbine under complex conditions. This paper studies the influence of the rotating pulsating flow field on the flow characteristics of the turbine stator, and analyzes the flow loss mechanism.

2. Materials and methods

2.1. Calculation model
The geometric model used in the numerical calculation in this paper is the first stage of the GE-E3 engine high-pressure turbine, due to the limitation of computing resources, this paper only studies 4 channels among them. The calculation model was meshed with NUMECA AutoGrid-5 software, and the grid independence was verified before the formal calculation. The details of computational domain and grid division are shown in Figure 1, and the total number of grid nodes is 5 million.
2.2. Numerical Methods

First, the steady numerical calculation is carried out, and the result is used as the initial field of the unsteady calculation. In the steady calculation, the average total pressure and average total temperature are set at the inlet of the turbine, and the average static pressure is set at the outlet. The turbulence intensity of the inlet is 10%, and the wall is set to adiabatic and non-slip condition. The Frozen Rotor model is used for the interface processing. The turbulence model during the numerical simulation is SST $k$-$\omega$, and the transition model is Gamma-Theta.

In the unsteady calculation, based on the uneven circumferential rotation distribution characteristics of the combustion chamber outlet flow field, the total pressure and total temperature conditions at the turbine inlet are simplified to a sawtooth wave distributed in the circumferential direction and rotating at a specific linear velocity. Figure 2 shows the circumferential distribution of the turbine inlet conditions at the initial moment in the unsteady calculation. In the figure, $P_{0m}$ is 344.74kPa, and $T_{0m}$ is 709K. $k$ is 0.3, which expressed as the circumferential pulsation coefficient of the import parameter, which is defined as the ratio of the fluctuation amplitude of the import parameter to the average parameter, reflecting the magnitude of the pulsation amplitude of the flow field. The circumferential rotational linear velocity of the inlet flow field is 1500m/s.

![Figure 2. Turbine inlet boundary conditions at the initial moment](image)

3. Results and discussions

Through numerical simulation, the law of fluid movement in the turbine stator is obtained.

3.1. Research on flow characteristics

Figure 3 shows the instantaneous total pressure cloud diagram at the 50% blade height section of the turbine when the inlet rotating pulsating wave passes through a single vane channel. The four complete vanes included in the black dashed rectangle in the figure are the actual vanes in the calculation model in this paper. The calculation domains on the upper and lower sides are the results of periodic matching. At the same time, the four vanes are positive along the angle $\theta$. The directions
are respectively marked as V1-V4; the direction of movement of the inlet rotating pulsating wave has been marked with an arrow.

![Figure 3. Instantaneous total pressure distribution at 50% height of turbine](image)

At the moment shown in Figure 3(a), the circumferentially uneven flow field generated by the inlet rotating pulsating wave forms a high pressure area near the pressure surface of the stator blade V1. Then there is a large pressure gradient along the circumferential direction in the cascade channel. The high pressure area on the pressure surface side of V1 begins to migrate to the suction surface side near the vane above, forming obvious secondary flow. The migration process is shown by the circle in the figure. When the high-pressure fluid approaches the pressure surface of the upper vane, the total pressure has been significantly dissipated and attenuated.

In the following migration process, the motion law of the high-pressure fluid is the same as the change process of the high-pressure fluid on the suction side of the vane V3 shown in the figure (elliptical area). It can be seen that the total pressure of the high-pressure fluid on the suction side of the vane V3 has experienced a process of increasing and then decreasing. The increase in total pressure this time is mainly due to the acceleration of the fluid on the suction side of the blade.

After a preliminary understanding of the migration process of the rotating pulsating wave in a single stator channel, the following discusses the migration and development law of the rotating pulse wave in the entire 4-channel calculation domain.

As shown in Figure 4(c), after the sweep of the circumferential rotation pulse wave, the pressure surface side of the leading edge of the vane V2 forms a high-pressure fluid area. It can be clearly seen from the section of channel I that the flow field distributed along the circumferential direction has a pressure gradient opposite to the rotation direction of the inlet rotating pulsating wave. In the subsequent development process, the high-pressure fluid group on the section of channel I began to migrate from the pressure surface side of the stator blade V2 to the suction surface side of the stator blade V1 along the negative direction of angle $\theta$. At the moment shown in Figure 4(e), the migration and movement traces of the high-pressure fluid group can be clearly observed on the cross-section of the channel I (the area circled by the black dashed line in the figure). It can be seen from the figure that the forwards of the high-pressure fluid area that migrate in the reverse direction are basically evenly distributed in the radial direction, but the demarcation line of the tail is not as obvious as that of the forwards, and there is a low-pressure wake at the 50% leaf height. Until the main body of the high-pressure fluid area moves to the side of the suction surface attached to the next blade, the low-pressure
wake at its tail disappears completely (as shown in Figure 4(f) channel I). This is an obvious feature of the reverse migration in the high-pressure fluid area.

Figure 4. The complete migration process of rotating pulsating waves in the computational domain.

After the inlet rotating pulsating wave sweeps each stator blade, a high-pressure fluid area will be formed in the cascade channel on the leading edge of the blade, especially on the pressure surface side. Due to the uneven distribution of the pressure in the cascade channel in the circumferential direction, a pressure gradient opposite to the movement direction of the rotating pulsating wave is generated, so that the high-pressure fluid area migrates in the reverse direction in the cascade channel. Therefore, when the inlet rotating pulsating wave completes a periodic sweep in the computational domain, the reverse migration and reattachment phenomenon of the high-pressure fluid masses will appear in the four cascade channels.

3.2. Analysis of flow losses

In addition to the circumferential secondary flow in the channel and the horseshoe vortex (Figure 5) on the leading edge of the stator blade, there is also a more obvious flow separation in the cascade channel. To explain this phenomenon, Figure 6 shows the flow in a transient stator blade channel. Limit flow lines on the wall and distribution of channel flow lines. Take the formation and development process of the horseshoe vortex on the following end wall as an example. When the inlet flow reaches the front edge of the stationary blade, the fluid near the lower end wall gradually entrains downwards to form a horseshoe vortex due to the stagnation effect of the front edge wall. Subsequently, under the action of the upstream flow, the down-rolling fluid is divided and then bypasses the leading edge of the blade and develops downstream from both sides of the channel to form the pressure side branch and the suction side branch of the horseshoe vortex.
When the separation area of the horseshoe vortex on the end wall of the adjacent stationary blade is large, the corresponding horseshoe vortex suction side branch will converge with the pressure side branch of another horseshoe vortex nearby. In order to better understand the motion law of horseshoe vortex in the cascade channel, the streamline distribution on Plane S1 of the channel entrance cross section was made. The red arrow in the figure shows the convergent development process of the horseshoe vortex on the lower end wall, and the blue arrow represents the convergent development process of the horseshoe vortex on the upper wall.

The side branch of the horseshoe vortex suction surface of the stationary vane V2 moves downward along the wall surface due to the stagnation action of the blade wall surface, and starts to gradually migrate to the middle of the cascade channel under the influence of the mainstream of the channel. The horseshoe vortex pressure side branch of the stationary vane V3 is also moved to the middle of the cascade channel under the action of the lateral secondary flow on the lower end wall. After the two converge somewhere in the middle of the lower end of the channel, they are converged to move upward in the radial direction due to the restriction of the solid wall of the lower end wall.

Since the meridian channel of the stator is basically symmetrical in geometric design, the upper and lower end walls of the channel have similar flow conditions, and there is also the convergence and development of horseshoe vortices. The up-rolled converging fluid cluster from the lower end wall and the down-rolled converging fluid cluster from the upper end wall with the same strength meet in the middle of the cascade channel. Subsequently, they are separated from each other in the circumferential direction, and a certain scale of flow separation is produced in the middle of the channel.

During the entire process of the rotating pulsation wave passing through the 4-channel calculation domain, there are different degrees of flow separation in each cascade channel. In the cascade channel with strong reverse secondary flow on the end wall, the flow separation zone in the channel is closer to the suction side of the stator blade. When the degree of separation between the suction side branch of the horseshoe vortex and the pressure side branch of the adjacent horseshoe vortex is close, the flow separation zone in the channel will be located in the middle of the channel. If the separation state of the horseshoe vortex suction side branch in the channel is stronger, the flow separation will occur on
the pressure side of the channel near the stator blade, or even on the pressure side of the blade (area shown by the red ellipse).

Figure 7 shows the limit streamline and turbulent energy distribution of the suction surface of the vane. It can be seen that in the area near the upper and lower end walls of the suction surface of the vane, the turbulent flow energy is relatively large, indicating that there is a certain degree of flow loss in this area. The pressure surface side branch of the horseshoe vortex from the lower end wall of the stationary blade moves to the suction surface side of the adjacent blade under the action of the transverse secondary flow in the channel. Subsequently, the boundary layer of the main flow and the end wall of the channel is continuously drawn, thereby forming a channel vortex. The channel vortex continues to roll up as it develops downstream, resulting in greater flow loss.

![Figure 7. Limit streamlines and turbulent flow energy distribution on the suction surface of the vane.](image)

4. Conclusions
Through numerical simulation, the influence of the inlet rotating pulsating flow field on the turbine operating characteristics is studied. The law of fluid movement in the stator channel and the mechanism of flow loss are analyzed.

1) It is feasible to use a sawtooth wave to simulate the exit flow field of the rotating detonation combustor.
2) The circumferential rotating pulsating flow at the turbine inlet will cause the reverse migration of the high-pressure fluid group in the cascade channel.
3) The secondary flow in the cascade channel, horseshoe vortex on the end wall, channel vortex and flow separation are the main components of flow loss.

Acknowledgments
This paper is supported by Natural Science Basic Research Program of Shaanxi (Program No. 2020JQ-473).

References
[1] Voitsekhovskii, B.V. (1959) Statsionarnaya dyetonatsiya. J. Doklady Akademii Nauk SSSR.,129: 1254-1256.
[2] Fotia, M.L., Hoke, J., Schauer, F. (2017) Study of the Ignition Process in a Laboratory Scale Rotating Detonation Engine. J. Experimental Thermal & Fluid Science., S113985359.
[3] Frolov, S.M., Aksenov, V.S., Gusev, P.A. (2015) Experimental studies of small samples bench rocket engine with a continuously-detonation combustors. J. Goren. Vzryv (Mosk.),8: 151-163.
[4] Ji, B., Li, J., Wang, Z.D. (2020) Effect mechanism of pulsating flow on unsteady aerodynamic performance of turbine. J. Journal of Aerospace Power,. 35: 410-421.(In Chinese)
[5] Wolański, PIOTR. (2015) Application of the Continuous Rotating Detonation to Gas Turbine. J. Applied Mechanics & Materials., 782: 3-12.