Influence of rotation speed on the steady cooling effects in the turbine disc cavity

Shuxian Chen\textsuperscript{1,\ast}, Zhenyu Liang\textsuperscript{2}, Wei Gu\textsuperscript{1}, Weiqi Yuan\textsuperscript{1}

\textsuperscript{1}Aviation Engineering College, Civil Aviation Flight University of China, Guanghan 618307, China
\textsuperscript{2}College of Computer Science and Technology, Civil Aviation Flight University of China, Guanghan 618307, China

\ast Corresponding author e-mail: bellesavana@163.com

Abstract. The flow and heat transfer characteristics in the counter-rotating turbine disc cavity is numerically simulated using the \textit{rng} $k$-$\varepsilon$ turbulence model, and the influence of rotation speed on the steady flow and the cooling effects is further investigated. The results indicate that the influence of rotation speed in low speed range on the upstream disc surface heat transfer depends on the relative driving action of rotating radial force and intake inertial force on the flow near the upstream disc. With respect to the downstream disc, the convective heat transfer coefficient on the disc surface in the lower radius region decreases while the one in the higher radius region increases with the increase of rotation speed.

1. Introduction
Nowadays, the development of aero-engine aims at greater thrust-weight ratio, lower fuel consumption and higher reliability. These three factors are closely related to engine design and research technology. Increasing the pressure ratio of the compressor and improving the total temperature before turbine are two principal ways to improve the engine thrust and the engine efficiency [1]. Continuously increasing will deteriorate the working environment of the turbine. The high-temperature components need to withstand greater heat load and dynamic load, such as the high temperature load and the vibration load transferred from the turbine blades the turbine disc. According to statistics, more than 60\% of aeroengine faults appear in high-temperature components, and this number continues to grow. One of the major causes of this phenomenon is the insufficient cooling of high-temperature components. Therefore, the research on the air flow and cooling effects in the turbine disc cavity is of great importance for engineering practice in the turbomachinery industry.

In 1921, Von Karmen [2] is the first one to explore the laminar flow on a free rotating disc using the boundary layer equation. The study of the rotor-stator disc began in the 1960s. Daily and Nece [3] proposed four basic flow states in closed rotor-stator disc flow through the experimental study. In the aspect of co-rotating disks system, Humphrey et al. [4] and Herrero et al. [5] studied flow structure between a pair of co-rotating disks by performing two- and three-dimensional computations. The flow in the counter rotating disk cavities has also received great attention. Poncet et al. [6] used computational fluid dynamics (CFD) to investigate the turbulent Von Kârmán flow generated by two counter rotating smooth flat (viscous stirring) or bladed (inertial stirring) disks enclosed by a cylinder. In this paper, the
flow and heat transfer characteristics in the counter-rotating turbine disc cavity is numerically simulated using the Rng k-ε turbulence model, and the influence of rotating speed on the steady flow and the cooling effects is investigated.

2. Physical Model

![Figure 1. The schematic diagram of physical model](image)

In this paper, the geometrical structure of the counter-rotating disc cavity is a simplified model of the turbine disc cavity. The radius of each disc is $r_0=200\text{mm}$, and the cavity gap is $G=68\text{mm}$. The cooling air entered the cavity from the center (the radius of the center inlet is $r^* = 25\text{mm}$) of the upstream disk (disk 1), and left through a small axial clearance, $s=2\text{mm}$, between the rotating shroud and the downstream disk (disk 2) at $r = r_0$. The back faces of the two disks and the enclosure shroud surface are assumed to be adiabatic. The circular outer surfaces of the disks are heated by two 800W heaters to simulate the heat transfer from the blade to the turbine disc through the tenon during the operation of the counter-rotating turbine engine.

3. Mathematical Model

3.1. Control Equation

The flow and heat transfer problems discussed in this paper are assumed to be steady-state, axisymmetric and incompressible, the governing equations of flow and heat transfer in the case of ignoring the gravity are:

\[
\frac{\partial v_r}{\partial r} + \frac{v_r^2}{r} + v_z \frac{\partial v_z}{\partial z} = 0
\]  \hspace{1cm} (1)

\[
v_r \frac{\partial v_r}{\partial r} + \frac{v_r v_z}{r} + v_z \frac{\partial v_z}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial r} + \nu \left( \frac{\partial^2 v_r}{\partial r^2} + \frac{1}{r} \frac{\partial v_r}{\partial r} \frac{1}{r^2} + \frac{\partial^2 v_z}{\partial z^2} \right)
\]  \hspace{1cm} (2)

\[
v_r \frac{\partial v_r}{\partial r} + \frac{v_r v_z}{r} + v_z \frac{\partial v_z}{\partial z} = \nu \left( \frac{\partial^2 v_r}{\partial r^2} + \frac{1}{r} \frac{\partial v_r}{\partial r} \frac{1}{r^2} + \frac{\partial^2 v_z}{\partial z^2} \right)
\]  \hspace{1cm} (3)

\[
v_r \frac{\partial v_r}{\partial r} + \frac{v_r v_z}{r} + v_z \frac{\partial v_z}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \left( \frac{\partial^2 v_r}{\partial r^2} + \frac{1}{r} \frac{\partial v_r}{\partial r} \frac{1}{r^2} + \frac{\partial^2 v_z}{\partial z^2} \right)
\]  \hspace{1cm} (4)

\[
dc_r(\frac{\partial T}{\partial r} + v_z \frac{\partial T}{\partial z}) = \lambda \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial T}{\partial r} \right) + \frac{\partial^2 T}{\partial z^2} \right]
\]  \hspace{1cm} (5)
Where \( v_r, v_\phi, v_z \) respectively denote the velocity component of the direction \( r, \phi, z \), \( v \) is kinetic viscosity of cooling airflow, and \( T \) is the airflow temperature, \( p \) is the pressure, \( \rho \) is the density, \( T \) is the temperature, \( \lambda \) is the heat conductivity.

3.2. Turbulence Model
The rng \( k-\varepsilon \) turbulence model was employed to describe the turbulent flow and heat transfer in the counter-rotating disk cavities.

4. Mathematical Model

4.1. Heat Transfer Coefficient of the Upstream Disc and Downstream Disc

![Figure 2](image)

**Figure 2.** The heat transfer coefficient on disc surface

Fig. 2 is the radial distribution of \( h \) of the disc surface in the contra-rotating disc cavity with inlet flow rate \( \dot{m} = 250 \text{kg/h} \) and the same rotation speed \( \Omega = 1000 \text{r/min} \). It can be seen that the value of \( h \) of downstream disc increases firstly with the increase of radius and then decreases gradually after the position of 100mm in radial direction, and then rapidly increases at the outer rim of the disc.

The rotational inertial force of the flow near the upstream disc increases with the increase of radius, and the radial velocity increases. Therefore, \( h \) of the upstream disc increases gently from 25mm to 150mm in the radial position. At 150mm to 170mm, the radial velocity significantly increases in the opposite direction because of the radially inward flow of the counterclockwise vortex cells from the downstream disc, and the \( h \) also increases significantly. On the outer rim of the upstream disc, owing to the boundary layer separation of the flow near the corner, the value of \( h \) suddenly drops.

4.2. Effect of Rotation Speed on Heat Coefficient of the Upstream Disc

![Figure 3](image)

**Figure 3.** The heat transfer coefficient of upstream disc for different rotation rate

Fig. 3 shows the heat transfer coefficient of the upstream disc surface for the five rotation speeds respectively. Under the above characteristics of radial velocity variation, with the increase of rotating speed from 500rpm to 1000rpm, the local heat transfer coefficient \( h \) of the upstream disc increases in the 0.09m-0.14m radius range and 0.16-0.19m radius range, and decreases in 0.025m-0.09m radius range and 0.14m-0.16m radius range. The overall value of \( h \) of the upstream disc surface increases.
When the rotation speed increases to 1500rpm, in the radius range of 0m-0.15m, the heat transfer effect increases due to the increased rotation force of the fluid on the surface of the upstream disc. However, in the high radius range of 0.15m and above, the cooling effect is weaker. Continue to increase the speed, the radial velocity of the flow near upstream disc surface increases, the cooling effect becomes better. Therefore, in the low rotation speed range, the influence of rotating speed on the value of $h$ of the upstream disc depends on the relative effect of the driving action of radial force caused by the rotating and intake inertial force near the upstream disc surface. In the high rotating speed range, the heat transfer coefficient increases with the rotational speed increasing.

### 4.3. Effect of Rotation Speed on Heat Coefficient of the Downstream Disc

Fig. 4 shows the heat transfer coefficient of the surface of the downstream disc at the five rotation speeds respectively. In the lower radius region, impact cooling in the low radius region plays a major role in the cooling at the downstream disc surface. With increase of rotation speed, the clockwise reflux near the upstream disc weakens the impact cooling effect. Therefore, the $h$ of the downstream disc surface decreases with increasing rotation speed at this region. In the higher radius region, convection cooling plays a major role. The rotation speed increases, the boundary layer becomes thinner and the $h$ increases, as showed in Fig. 4.

## 5. Conclusion

![Figure 4. The heat transfer coefficient of downstream disc for different rotation rate](image)

For the upstream disc, the influence of rotation speed at low speed range on the disc surface heat transfer depends on the relative strength of rotation radial force and intake inertial force on the fluid near the upstream disc. The upstream disc surface heat transfer coefficient increases, the disc surface temperature decreases with increasing the rotation speed in the high speed range.

For the downstream disc, the convective heat transfer coefficient in the low radius region decreases while the one in the high radius region increases with increasing rotational speed. And both the temperature and the radial temperature difference along the downstream disc decrease with increasing rotation speed.

### Acknowledgments

This work was supported by the National Natural Science Foundation of China (Grant No. 51306201), the Science and Technology Planning Project of Sichuan Province of China (Application No. 2019YJ0722), the Science Research Foundation of Civil Aviation Flight University of China (No.J2019-033), and the Jiangsu Province Key Laboratory of Aerospace Power Systems (Grant No. APS-2013-04).

### References

[1] Y. Z. Cao, Z. Tao, G. Q. Xu, Aeroengine heat transfer. Beijing: Beihang University Press, 2005.
[2] T.V. Karman, Laminar and turbulent reibung, Z. Angew. Math. Mech. 1921, 1: 233-252.
[3] J. W. Daily, R. E. Nece, Chamber dimension effects on induced flow and frictional resistance of enclosed rotating discs, J. Basic Eng. 1960, 82: 217-232.
[4] J.A.C. Humphrey, C.A. Schuler and D.R. Webster, Unsteady laminar flow between a pair of disks corotating in a fixed cylindrical enclosure, Phys. Fluids. 1995, 7(6): 1225-1240.

[5] J. Herrero, F. Giralt, and J.A.C. Humphrey, Influence of the geometry on the structure of the flow between a pair of corotating disks, Phys. Fluids. 1999, 11(1): 88-96.

[6] Poncet, S., Schiestel, R., and Monchaux, R., Turbulence modeling of the von Kármán flow: viscous and inertial stirrings, Int. J. Heat Fluid Flow. 2008, 29(1): 62-74.