Effects of Rotating Speed on the Flow Characteristic in Rotating Turbine Disc Cavity

Zhenyu Liang[a], Shuxian Chen [b], Wei Gu[c]

Aviation Engineering Institute, Civil Aviation Flight University of China, Guanghan 618307, China

[a]18683808030@163.com,[b]bellesavana@163.com, [c]630961375@qq.com

Abstract

The flow has direct influence on the cooling effects in rotating turbine disc cavity. The flow characteristics in the counter-rotating turbine disc cavity with the central axial air inflow were numerically simulated by using the *rng* k-ε turbulence model. The flow structure in the disc cavity and the radial velocity distribution near the two discs were researched, and the influences of the rotation speed on the flow were further explored. The results show that there are two counter recirculation vortices in counter-rotating disc cavity, and the relative size of the two vortices and the radial position of the stagnation point of the encounter depend on the relative magnitude of the air intake inertial force and the rotational force. The distribution of radial velocity is affected by the combined action of centrifugal force of rotation and inertial force of intake air. The effect of inertial force of intake air plays a key role near the downstream disc. The effect of centrifugal force disc is dominated near the upstream.

Keywords

Counter-rotating disc cavity, numerical simulation, turbulent flow, flow structure, radial velocity.

1. Introduction

The contra-rotating turbine technology is extensively used for aero-engines. On the one hand, it can reduce the number of downstream turbine guide blades and make the engine structure more compact, thereby reducing engine weight and aerodynamic loss. On the other hand, contra-rotating turbine structure can greatly reduce the gyroscopic moment of the aircraft during rotary maneuvering flight [1]. So many countries have been working on the contra-rotating turbine technology and gradually utilize this technology from the third generation of engines. Therefore, cooling in the contra-rotating turbine enclosure is particularly important. The flow structure in the counter-rotating disc cavity is complex and it has direct influence on the cooling effects. It is of great significance to study the flow characteristics of the gas in the counter-rotating disc cavity to reduce the demand for the cooling gas and improve the performance of the engine.

In the early stages of free disc exploration, the Beton [2] had a theoretical exploration of the friction torque. Then in 20th century, the era entered the rotor-stator disc phase, Daily and Nece [3] had put forward four basic flow states of the inner flow of the closed rotor-stator disc cavity on the basis of experiments, and studied the factors affecting the distribution of these regions. For the rotor-rotor disc cavity, Owen [4] probes into the sources flow of the disc cavity simplification model, analyzes the distribution of tangential velocity in the laminar and turbulent flow of the core, and then deduces the size of the source region. Gan [5] using LDV explores the flow in the contra-rotating cavity of the same speed. Xu [6], Shuiting Ding [7] studied the heat transfer characteristics of the disc cavity with different inlet positions and modes. Shuxian Chen et [8] studied the pressure distribution in the contra-rotating disc cavity and the friction torque of the disc. However, for the contra-rotating disc cavity, there is no deep research on the flow of the boundary layer and the details of the flow characteristics in the rotating disc cavity. In this paper, the numerical calculation method is used to study the flow structure characteristics of the contra-rotating disc cavity with a rotating speed ratio equal to -1, and the effect of rotational speed on the flow structure is analyzed.

2. Physical Model

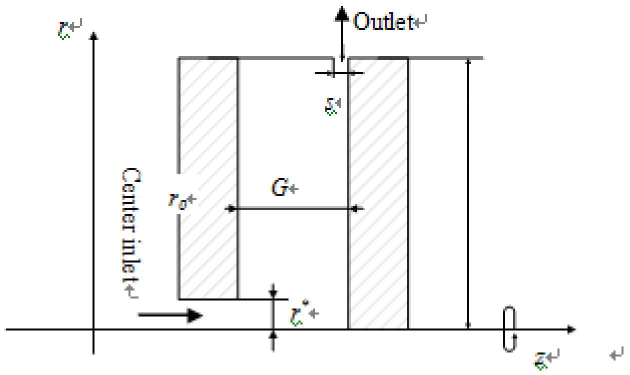

Fig.1 The sketch map of physical structure model

Cai yi et [9] simplify the structure of the contra-rotating turbine disc and the disc cavity, their experiment explores the unsteady state. The geometrical structure of the contra-rotating cavity is the same as that of the experimental structure, and the radius of the upstream and downstream disc r0=200mm, center inlet radius r*=25mm, the cavity gap G=68mm, the outlet gap between the shroud and the downstream disc s=2mm.

3. Mathematical Model

3.1 Control Equation

Due to the complex shape of the turbine disc, the flow in a real engine turbine disc cavity is a complex three-dimensional unsteady flow. In order to facilitate the study, this article makes the following assumptions: flow is steady, axisymmetric, incompressible and constant property. The flow control equations of the normal fluid in the neglected mass force are:

$$\frac{\partial v_r}{\partial r}+\frac{v_r}{r}+\frac{\partial v_z}{\partial z}=0 \tag{1}$$

$$v_r\frac{\partial v_r}{\partial r}-\frac{v_\phi^2}{r}+v_z\frac{\partial v_r}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial r}+\nu\left(\frac{\partial^2 v_r}{\partial r^2}+\frac{1}{r}\frac{\partial v_r}{\partial r}-\frac{v_r}{r^2}+\frac{\partial^2 v_r}{\partial z^2}\right) \tag{2}$$

$$v_r\frac{\partial v_\phi}{\partial r}+\frac{v_r v_\phi}{r}+v_z\frac{\partial v_\phi}{\partial z}=\nu\left(\frac{\partial^2 v_\phi}{\partial r^2}+\frac{1}{r}\frac{\partial v_\phi}{\partial r}-\frac{v_\phi}{r^2}+\frac{\partial^2 v_\phi}{\partial z^2}\right) \tag{3}$$

$$v_r\frac{\partial v_z}{\partial r}+v_z\frac{\partial v_z}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\nu\left(\frac{\partial^2 v_z}{\partial r^2}+\frac{1}{r}\frac{\partial v_z}{\partial r}+\frac{\partial^2 v_z}{\partial z^2}\right) \tag{4}$$

Where vr, vϕ, vz respectively denote the speed component of the direction r, ϕ, z, ν is kinetic viscosity of cooling airflow.

3.2 Turbulence Model

Different turbulence models have different ranges of application. So different turbulence model needs to be selected according to the specific condition of the flow. Xuejie Shi [10] selected the standard k-ε turbulence model and the rng k-ε turbulence model to perform numerical calculations on the experimental conditions in the literature [9]. Through the mutual confirmation between numerical simulation and experimental data, it is found that the analog of the latter is closer to the experimental data. Therefore, the numeric calculation of the article adopts the rng k-ε turbulence model.

3.3 Boundary Conditions

1) The cooling gas temperature of inlet is 295K;

2) The contra-rotating disc cavity outside temperature is 295K;

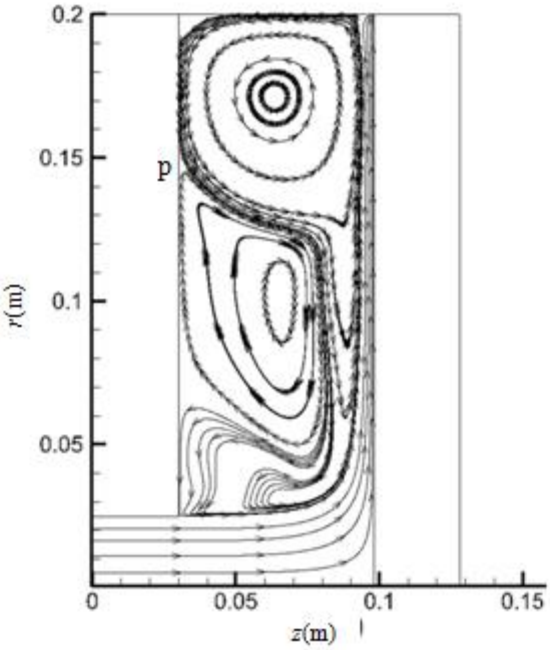

Fig.2 Streamline of cooling air flow

3) Two disc back is adibatic.

4. Calculation results and analysis

4.1 Flow Structure in the Counter Rotating Cavity

Fig. 2 is the cooling air flow chart in the same speed contra-rotating disc cavity with inlet flow rate ṁ=250kg/h, rotation speed Ω=1000r/min. The jet from the air inlet impingement downstream disc, and under the action of the centrifugal force, the radial coriolis force and the viscous force that caused by the two discs of the counter rotating turbine disc cavity with same rotational speed, the air flow outwards along the surface of the downstream disc, and reflux along the shroud at the periphery, and then flows radially inwardly along the upstream disc and meets with the airflow flowing radially outwardly of the upstream disc at the stagnation point P ,and both of them flow to the center of the disc cavity together to form a clockwise reflux vortex and a anticlockwise reflux vortex.

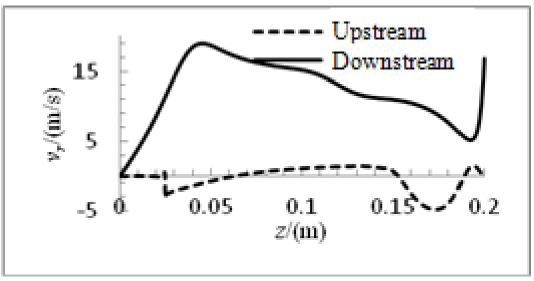

Fig.3 Radial velocity on near disc surface

4.2 Radial Velocity Distribution near Disc Surface

Fig. 3 shows radial distributions of vr near the rotating disc surfaces while $\dot{m}$ = 250kg/h, Ω=1000r/min. The radial velocity near downstream disc surface increases quickly under the action of the mainstream inertial force, then gradually decreases and then increase near the exit. The radial velocity near upstream disc surface increase near the entrance edge, followed by a counter increasing due to the anticlockwise vortex from downstream disk surface. Obviously, the radial velocity near downstream disc surface is mainly affected by the mainstream inertia force, and the radial velocity near upstream disc surface is mainly affected by the force of rotation and the effect of backflow block.

4.3 Effect of Rotational Speed on the Flow Structure in a Rotating Disc Cavity

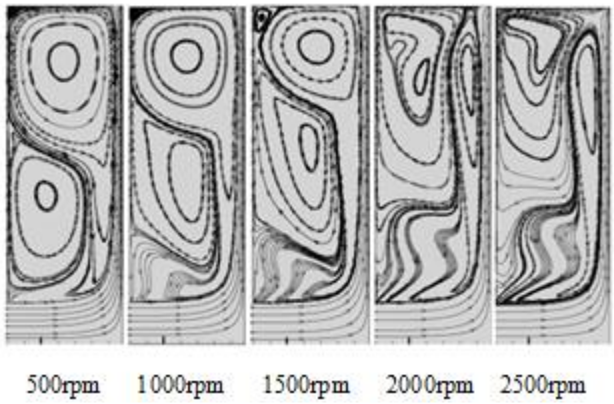

Fig.4 Streamline of different rotation rate

Fig. 4 shows the streamline diagram in counter rotating turbine disc cavity Ω=500-2500r/min, G=68mm, $\dot{m}$=250kg/h. When Ω=500r/min, the inertial force of the intake air is superior to the rotational force, and the counter-clockwise flow vortex occupies the high space in the disc cavity. As the speed increases, centrifugal force and coriolis force are enhanced, and the effect of intake inertia is weakened. The reflux from the downstream disc along the peripheral plate then to the upstream disc is gradually weakened. The radially outward flow along the upstream disc is enhanced under centrifugal force, and the anticlockwise vortex cells with high radial position were gradually compressed, and the clockwise vortex cells with low radial position expanded to the high radial position. The point P where the two opposite vortex encounter move to the outer edge of the disc with the increase of rotation speed.When the rotating speed is 1500r/min, clockwise convection vortex has been extended to the whole radius range, because the increasing strengthening of centrifugal force. With the transfer of the stagnation point, the reflux core of the clockwise vortex gradually moves to the high radius.

4.4 Effect of Rotational Speed on the Radial Velocity near Disc Surface

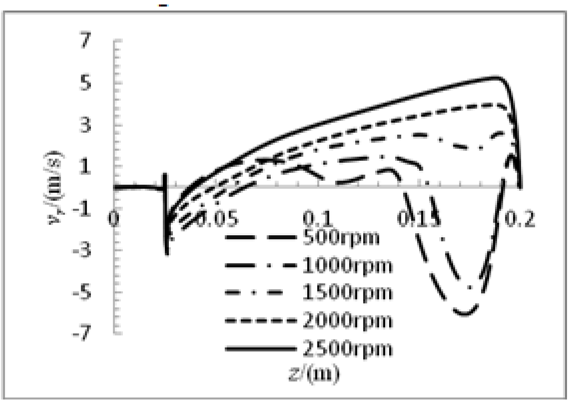

Fig.5 Radial velocity near upstream disc surface

Fig. 5 and Fig. 6 show the radial distribution of radial velocity near the upstream dick and downstream disc respectively for the different rotation speed. As can be known from Fig. 5, with the rotation speed increasing from 500 rpm to 1000 rpm, the radially inward velocity near the upstream disc decreases in 0.025m-0.09m radius range. The radially inward velocity near the upstream disc velocity increases in the rotating disc 0.09m-0.14m radius range, because the increase of the rotational force on the surface of the upstream disc. At 0.14m-0.18m radius range, the radially inward velocity decreases. The flow velocity radially inward increases within the radius range of 0.18m-0.19m. With rotation speed increasing to 1500rpm, in the radius range of 0m-0.15m, due to the increased rotational force of the fluid on the surface of the upstream disc, the radial flow velocity increases. However, in the high radius range of r>0.15m, the anticlockwise vortex cell is weaker at the periphery region, the velocity decreases. For the relatively high rotational speed of 2000rpm and 2500rpm, centrifugal force occupies a major role, and the radial velocity increases continuously with the increase of radius. In all of the above conditions, the radial velocity drops sharply due to the retarding effect of the shroud.

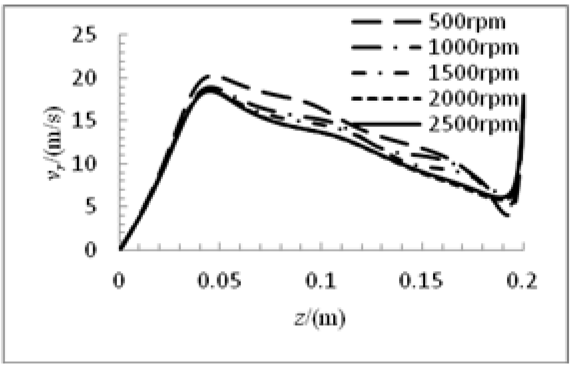

Fig.6 Radial velocity near downstream disc surface

Fig. 6 show that, for flow near the downstream, intake air inertia force plays a major role. So the radial velocity increases slightly with the rotation speed increasing.

5. Conclusion

1) Under the action of rotational force, inertial force of intake air and viscous force, there are two reverse reflux vortices in the counter rotating disc cavity. The relative sizes of the two vortex cells and the position of the stagnation point of the encounter is dependent on the relative magnitude of the rotational force and the inertial force.

2) Radial flow velocity near disc surface varies complicaly, and stagnation point position of the two opposing vortex cells encounter is the conversion point position of the radial flow velocity value.

Acknowledgements

This work was supported by the National Natural Science Foundation of China (grant number 51306201); the Joint Funds of the National Natural Science Foundation of China (grant number U1233202); and the Jiangsu Province Key Laboratory of Aerospace Power Systems (grant number APS-2013-04); the Research Innovation Program for College Graduates of Civil Aviation Flight University of China(grant No.X2016-30).

References

[1] L. R., Qiao W.Yang. Aerodynamic design and analysis of 1+1/2 counter-rotating turbine [J]. Machinery Design and Manufacture, 2006, 8: 17-19. (In Chinese)

[2] E. R. Benton. On the flow due to a rotating disc [J]. J. Fluid Mech, 1966, 24: 781-800.

[3] J. W. Daily, R E.Nece. Chamber dimension effects on induced flow and frictional resistance of enclosed rotating discs [J]. J. Basic Eng, 1960, 82: 217-232.

[4] J. M Owen. J. R Pincombe, R. H. Rogers. Source-sink flow inside a rotating cylindrical cavity [J]. J. Fluid Mech, 1985, 155: 233-265.

[5] X. Gan, M. Kilic, J. M.Owen. Flow between contra rotating discs [J]. ASME J. Turbomach, 1995, 117(2): 298-305.
[6] G.Q Xu, S.T. Ding, Q.X. Guang. Study on local heat transfer characteristics of central air intake rotating disc with the outer rim heat [J]. Journal of Aerospace Power, 1995, 10(3):229-232. (In Chinese)
[7] S. D., Z. Tao, G.Q. Xu. Study on average heat transfer characteristics of rotating cavity with outer edge axial air intake [J]. Journal of Aerospace Power, 1998, 13(3): 281-284. (In Chinese)
[8] S.X,Chen, J.Z.,Zhang X.M. Tan. Flow characteristics inside counter-rotating disc cavity [J]. Journal of Aerospace Power, 2013, 28(1): 136-141. (In Chinese)
[9] Y. Cai, G.Q. Xu, Z. Tao. Study on heat transfer characteristics of counter-rotating disc with unsteady experimental [J]. Journal of Aerospace Power, 2004, 19(3):346-350. (In Chinese)
[10] X.J. Shi , S.X. Chen, Z.Y. Liang. The influence of each parameter on the frictional moment of the disc inside counter-rotating turbine cavity [J]. Journal of XIAN Aeronautical University, 2016, 34(3):12-16. (In Chinese)