32nd Congress of the International Council of the Aeronautical Sciences
September 6-10, 2021
Pudong Shangri-La, Shanghai, China

THE FLOW AND HEAT TRANSFER OF ANALYSIS TO ANNULAR CAVITY IN ENGINE CASE

Pan Guangyan[1], Yu Xiao[2], li Yi[3], Zhao Yuxin[4]

[1234]AECC SHENYANG ENGINE RESEACH INSTITUE

Abstract

With the continuous improvement of aero-engine performance, the demand for temperature resistance of turbine casing is more intense .Therefore, the flow and heat transfer research of turbine casing becomes an important work in engine design. Numerical simulation method is usually used to analyze the flow and heat transfer to the casing. However, due to the imperfect summary of the flow and heat transfer to this typical structure, when the structure is improved slightly or the flow environment is changed, the model of the flow and heat transfer should be rebuilt. Finally, it costs more and the cycle is longer. In this paper, the heat transfer to the annular cavity of the engine casing is researched through three-dimensional numerical hydrodynamics method and one dimensional heat transfer calculation method. The flow characters and flow distribution of annular cavity, radiation calculation model and convection heat transfer law are studied as well. And the method to the research was verified through heat transfer test data of engine test run. Finite element numerical simulation and sector model can be used in this research and accurate flow heat transfer law can be obtained. When the engine is running at high state, the outer and the inner ring have high temperature difference. This can be difference from engine run in the low state. The radiation is the main way of heat exchange. And the higher temperature difference gets, the more active the air flows. The difference between the measured temperature and the calculation temperature is only 1.3% with the method from this paper. In addition, this paper shows the change of heat transfer law of the classical structure in different state of the engine, as well as the general applicable heat transfer calculation method of this casing structure.

Keywords: engine, case, annular cavity, characters of flow heat transfer law

1. Introduction

The bearing casing and inter-stage casing are widely used in various aero-engines and gas turbines. On the one hands, the casing assembly supports the engine turbine, on the other hands, it blocks gas with high temperature and bearing with low temperature. In addition, the flow passage inside the turbine casing is an important part of the aero-engine air system, which is an effective passage for cooling air. These effects determine that the gearbox assembly has the characters of large temperature difference and complex cooling structure.

In engine design, it is necessary and important to obtain the accurate temperature distribution of the casing in working condition. Through the multi-state temperatures of the gear-box the deformations of the gearbox in various states are obtained, and the cold and hot clearance of the whole engine is further analyzed.so as to achieve the goal of adjusting the engine performance.

Turbine casing is a hot end component, and its structure is very complex, with many passages. Such as flat plate, groove, porous and so on[1]. There are obvious differences between the existing conventional heat transfer model and the specific thermo physical model of engine casing. The accuracy of heat transfer boundary greatly affects the accuracy of the casing ensure the accuracy of heat transfer calculation boundary through numerical simulation and experimental test. And further obtain the accurate temperature distribution of the casing assembly.

Because the annular cavity structure is the main structure of the casing assembly, the important part of the flow heat transfer analysis of the casing assembly is to study the flow and heat transfer characteristics of the annular cavity. The flow and heat transfer in annular cavity is affected by various factors. Its essence is the organic combination of fluid dynamics, heat exchange theory and engine component. Purpose of this paper is to establish the heat transfer analysis method of the annular cavity. The flow and heat transfer law of the engine casing is revealed. The accurate temperature results of the casing components are obtained as well.

2. Analysis of technical links

Gearbox assembly is one of the important components of the engine. The typical engine structure is shown in figure 1.In the red frame, there are different casing assemblies of the engine. The engine is generally in the form of integral ring, including the outer ring of the casing frame, the support plate and the inner ring of the casing frame. For the engine in high temperature environment, there are usually heat insulation structures, such as outer channel parts, rectifier support plate and inner channel parts.

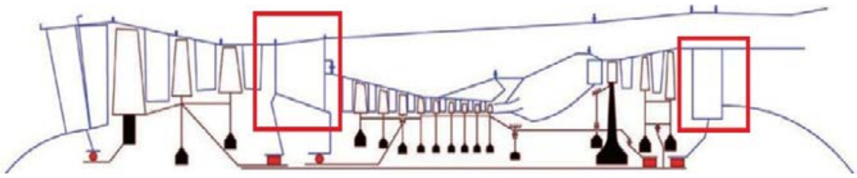

Figure 1 –Structure diagrammatic of engine

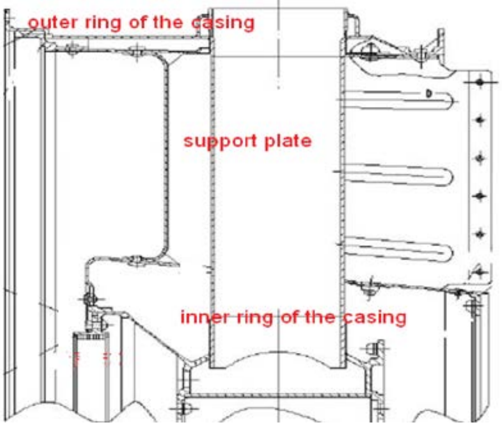

Figure 2–Engine casing diagrammatic

The air flow channel includes some different channels in the casing assembly. Such as the channel in the support plate, the channel composed by the bearing frame and the sealing ring. The air flow around the casing assembly includes environment air brushing the casing outer, inner air when the engine runs, cooling air outside the sealing ring, and the cooling air in the casing.

Generally speaking, the flow and heat transfer of turbine casing cavity can be summarized as the heat exchange of the annular and slotted channels, small holes, and round tubes[2]. According to its structural characteristics of different positions, the flow and heat transfer coefficients of the inner cavity are calculated by the corresponding and existing classical experimental correlations. Most of them use laminar or turbulent flow in the tube slot or in the tube layer, flow and heat transfer in thermal analysis. No matter which model is used. The important concerns are the geometry and flow characters of the flow chamber. We try to predict its flow characteristics to select the appropriate experimental correlation for heat transfer analysis.

However, in the heat transfer test of the engine.it is found that the air flow rate in the cooling chamber of the casing changes when the engine runs, but the data of wall temperature from the test shows that the cooling ability of the air flow in the casing is obviously stronger than that in the low state .this phenomenon shows that the traditional empirical analysis method needs to modified the heat transfer model in the actual working state of the engine.

To solve this problem, based on one- dimensional heat transfer calculation method, three-dimensional numerical hydrodynamics simulation is carried out to study the flow and transfer law.

3. The model of heat exchange

3.1 The heat exchange essence of the annular cavity

In this paper, the complex flow in the inner cavity to the inter-stage casing is taken as the research object. The flow is as shown in the figure. The air flows into the casing assembly and the front chamber of the casing through the inner tube of the support plate. the air flow path into the casing assembly is as follows: Firstly, the air flow first enters the air cavity formed by the inner wall surface and the bearing block, through the small hole on the tube, then flows into the outer cavity of the casing inner ring through the holes of the casing inner ring, and finally discharges through the annular gap structure after passing through the inner ring.

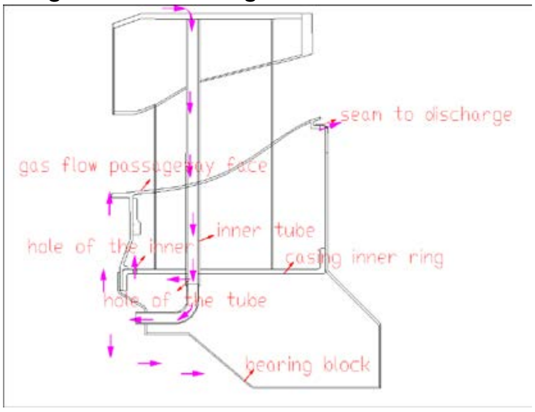

Figure 3–Components about the cooling air

According to the periodic symmetry of the annular cavity, the fluid model of the annular cavity is simplified as a fan-shaped structure. In order to simplify the calculation the outlet annular seam to discharge the cooling air is equivalent to a simple hole. The air inlet 1and 2 flow through a pipe and A1 pipe. And the main flow enters the front chamber through 4 outlet. The other part enters the outlet bearing cavity c, joins the inner ring b through channels d and d1, and flows out through outlet 3.

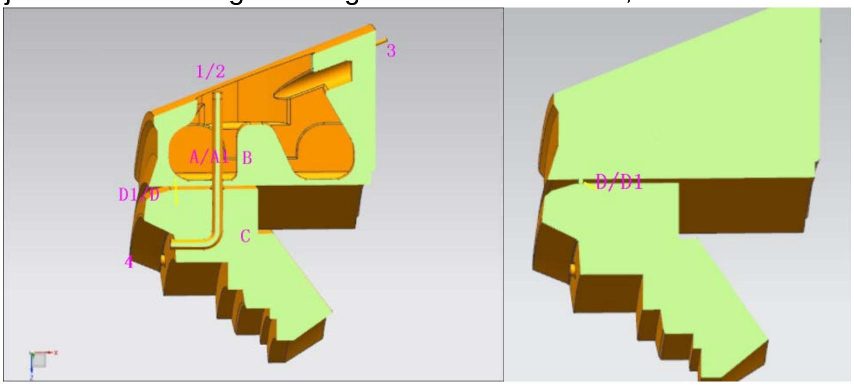

Figure 4–Cavity and components about the air flow

The main way of the heat transfer includes convective heat transfer, radiation and heat conduction: the convective heat transfer of the cooling air to the casing surface; radiation from high temperature casing to the low temperature casing inner; heat conduction from the other assemblies contacting the casing.

3.2 Heat exchange model

3.2.1 The heat transfer model of the annular cavity

The flow rate of inlet I and 2 is 0.02kg/s, and the static temperature is 601k/m, in order to consider the influence of the temperature difference inside and outside the annular cavity on the flow and heat transfer, the temperature near the main channel of the engine is about 800k,and the other wall temperatures rang from 320k to 560K,as shown in figure 3.

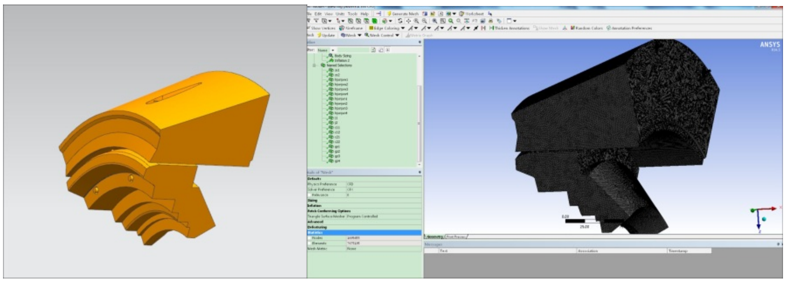

Figure 5–Infinite model of the CFD(1)

In this paper, the ideal gas model is used as the calculation working fluid. The equations composed of continuity equation; momentum equation; energy equation and turbulence equation are solved numerically. Standard K-module and scale wall function method are used to consider turbulent flow[3].

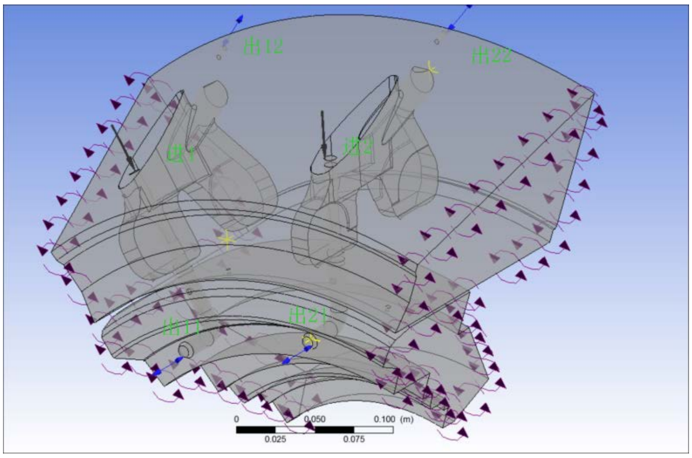

Figure 6–Infinite model of the CFD(2)

From the flow velocity FIgue1 of CFD finite element numerical simulation results .it can be seen that the flow velocity is range of 0～250m/s, most of the air flows out from tube a and tube A1, and the flow velocity is about100 relatively high flow velocity ,after the confluence of chamber c. less air is discharged through D and D1.the flow velocity is below 100.the flow velocity of air in chamber B is higher than that I chamber. And more vortices are formed is larger than that of cavity .at the same time, it can be seen from the figure that the position where the larger vortices are form is not in the section of a pipe and A1 pipe of the flow pipe. And the position where the larger vortices and flow velocity are formed is on the back wall and lower wall of B cavity.

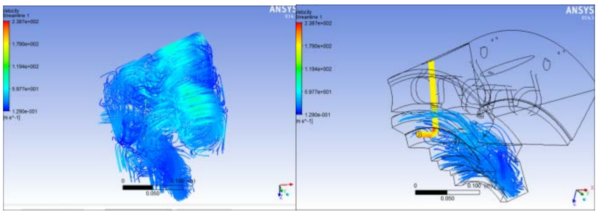

Figure 7–Distribution of the velocity

The average heat transfer coefficient distribution of the wall is shown in the figure.it can be seen from the figure that the larger flow capacity brings stronger heat transfer than that of cavity C. and the heat transfer coefficient of cavity B is generally higher than that of cavity B is higher .this phenomenon is consistent to the results of flow analysis.

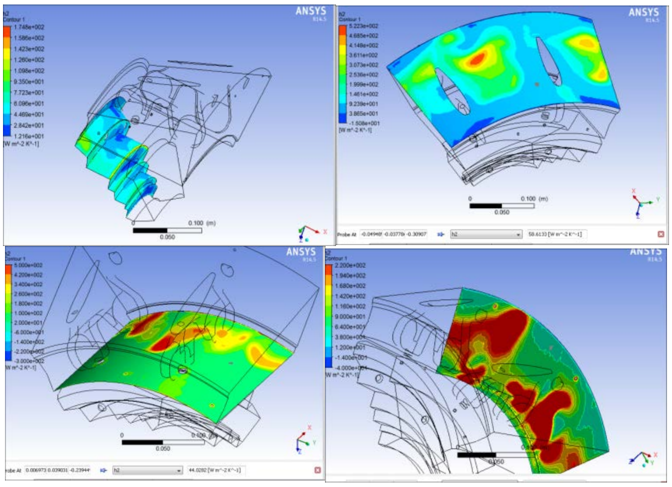

Figure 8–Distribution of the heat transfer coefficient

3.2.2 radiation heat transfer physical model of the annular cavity

In the high state of the engine run, the temperature difference between inner and the outer ring is much larger than that in the low state. Based on the principle of radiation heat generation, the space between runner and inner ring is the main location of radiation heat exchange in annular cavity.so the radiation heat physical model of annular cavity can be assumed as the heat radiation model in the right figure.

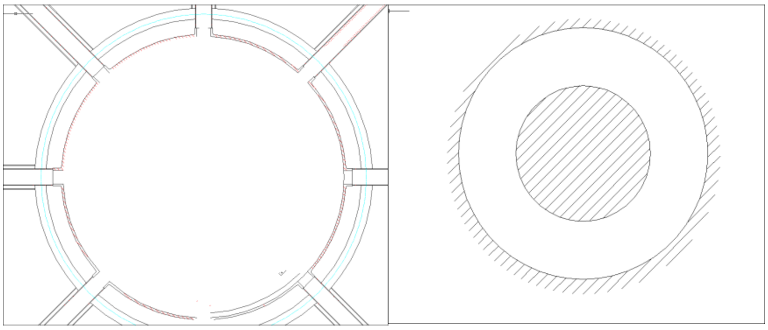

Figure 9–Radiation model

Among them, one and two surfaces are closed systems composed of diffuse gray surfaces, which are two cylindrical surfaces respectively, and surface 1 is convex surface or approximate plane. The radiation of the system can be summarized:

$$\frac{A_2 \times 5.67 \times \left[\left({T_{w1}}/{100} \right)^4 - \left({T_{w2}}/{100} \right)^4 \right]}{{1}/{\varepsilon_2} + {A_2}/{A_1} \left({1}/{\varepsilon_1} - 1 \right)} = E_{1,2} \quad (1)$$

Table 1–Meaning of the parameter

Parameter	Meaning
A1	Area of the inner
A2	Area of the outer
e1	Emissivity of the inner face
e2	Emissivity of the outer face
Tw1	temperature of the inner face
Tw2	temperature of the outer face

3.3 Model Validation

The heat transfer coefficient of the flow and the radiation heat of the cavity wall are obtained the through the heat transfer model of the annular cavity, and the temperature distribution of the inner and outer about the casing is further obtained, the rationality of the heat transfer model is verified by the test data of engine flow and heat transfer.

In order to ensure the rationality of the analysis object of the heat transfer model, the analysis object of the casing assembly includes the casing assembly, the sealing ring connected with the casing and the bearing assembly .reasonable optimization of the calculation model, and ignoring the bolt, pin, air nozzle and other local three-dimensional features, complete the modeling of the whole complex model, network to the middle.

A three-dimension finite element model is used to calculate the temperature of the casing. and the results are shown in the figure.it can be seen from the figure that the test wall temperature is consistent with the test results.it can be seen that the thermo physical model of the annular cavity established in this paper can simulate heat exchange model in the working environment the component and it can be used for further research.

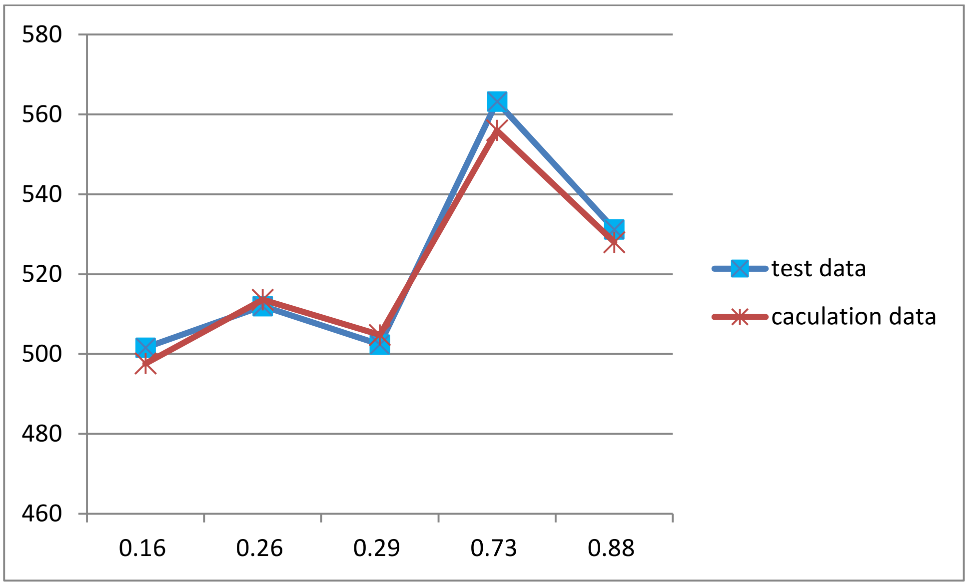

Figure 10-Temperature Comparison of the test data and the calculation (unit:K)

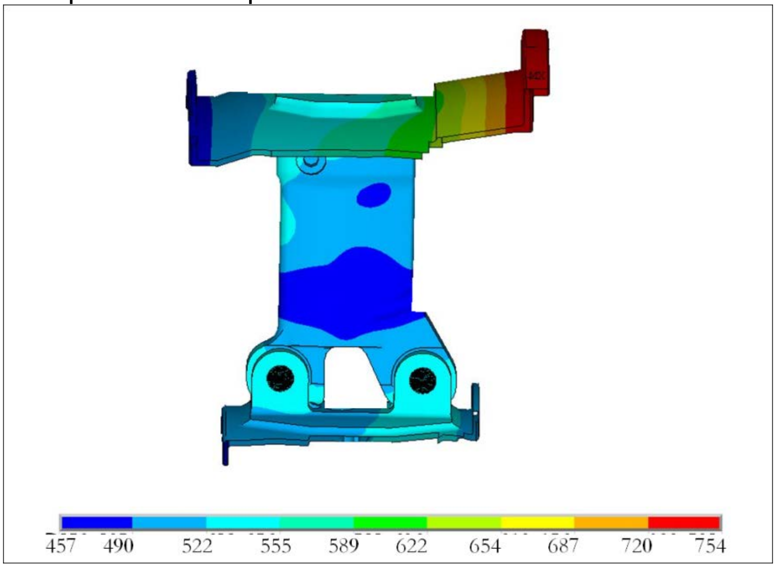

Figure 11–Temperature result of the casing (unit:K)

4. Law of the Flow and Heat Transfer

4.1 Flow Pattern

Table 2 shows the flow distribution of inlet and outlet calculated by CFD. It can be seen from the table that 92% front cavity of the casing, and 8% of the air flow discharge through the rear chamber of the casing which is the source of the air flow cooling the chamber.

Table 2–Flow distribution to the inlet and the outlet from the CFD results

		Results from the CFD	Flow distribution
Inlet	1	0.01758	-
	2	0.0207	
Outlet	4	0.0200063	51%
	4	0.0163389	41%
	3（1）	0.00137493	3.5%
	3（2）	0.00164579	4.2%

The following figure shows the velocity and Mach number distribution of section A and D of tube. It can be seen from the figure that velocity in cavity B is generally higher than that in cavity C, but the velocity is lower than that in mainstream a. the section with high velocity is on the lower wall and back wall of cavity B, and it is predicted that the flow and heat transfer ability of this is strong.

The flow velocity from the air inlet is high, which has a great impact on the wall. It flows into the large space of the inner cavity from the small hole, and the casing cavity is complex, and the formation of vortex like flow in the cavity will further cause the complexity of local flow heat transfer, which does not belong to the ordinary one-dimensional empirical heat transfer model, and the flow velocity at the vortex position is large which will produce a strong heat transfer capacity.

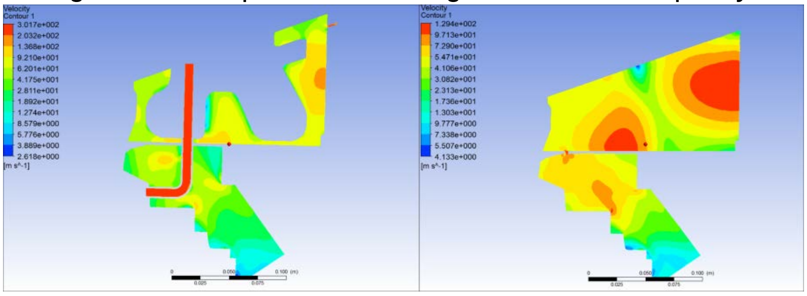

Figure 12–Velocity distribution

4.2 Heat transfer law

It can be seen from the figure that the interaction between the air flow and the wall at the impact position is very strong resulting in a large heat transfer coefficient .However, outside the impact area, the interaction decreases and the heat transfer coefficient is relatively small.

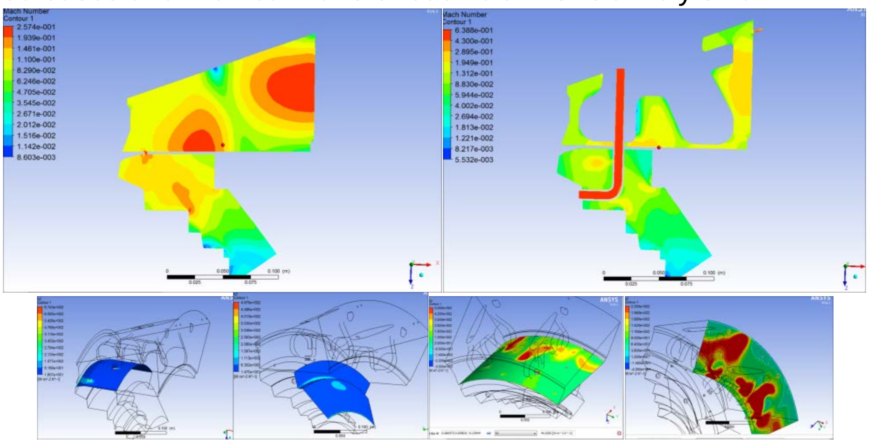

Figure 13–Comparison of mach number distribution and wall heat transfer coefficient

Generally speaking ,the air flow in the annular cavity of stator belongs to natural convection heat transfer in limited space.the following correlation is recommended for the caculation of convection heat transfer in the annnular cavity [4].

$$\overline{Nu} = 0.212(G_r P_r)^{1/4}, G_r = 1\times10^4 - 4.6\times10^5$$
$$\overline{Nu} = 0.061(G_r P_r)^{1/3}, G_r \rangle 4.6\times10^5 \quad (2)$$

The average heat transfer coefficient calculated by the empirical formula is about 15 W/(m^2 • K) ,which is different from the absolute average heat transfer coefficient calculated be CFD in the range of 30 W/(m^2 • K)～160 W/(m^2 • K).the reason is that the larger temperature difference and radiation of the fluid wall aggravate the flow ability of the air flow and bring stronger cooling effect.it it similar to the forced and natural mixed flow convective heat transfer model in horizontal tube .

$$Nu^n{}_M = Nu^n{}_F \pm Nu^n{}_N \quad (3)$$

Where is the heat transfer coefficient of mixed convection[5], and is the sum of Nusselt numbers obtained according to the criteria of forced convection and natural convection under given conditions. The table lists the surface radiant heat and convective heat transfer in the annular cavity.it can be seen that the radiant heat accounts for more than half of the heat exchange when the engine is working at high state, and becomes an important way of heat transfer for the engine components .when the radiant heat flux of the inner and outer walls is high, the air flow in the cavity is intensified, and the cooling and heat transfer capacity of the air flow is increased.

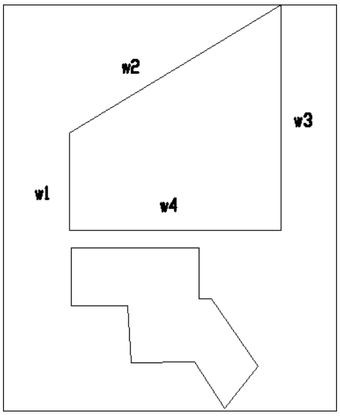

Figure 14–Location

Table 3–Heat transfer coefficient of annular cavity and comparison with empirical equation

Place	Heat flux density from the radiation	Heat flux density from the convention heat	Average heat transfer coefficient from the CFD result (W/(m^2 • K))	Empirical equation from the classic heat transfer model (W/(m^2 • K))
W1	7645	621	69	16
W2		12460	138	
W3		1467	163	

5. Conclusion

The heat transfer to the annular cavity of the engine casing is researched through the three-dimensional fluid dynamical simulation and one-dimensional heat transfer model. And the correctness of the simulation method is verified by the engine flow and heat transfer experiment. At the same time, through numerical analysis the flow and heat transfer law of the inner surface of the casing annular cavity structure is mastered:

1. In order to study the flow and heat transfer law of engine casing annular cavity, the infinite element numerical simulation method can be used to establish the fan section model. The experimental results show that this method has high accuracy, and the local heat transfer law can be further analyzed by numerical simulation.

2. In the high state of the engine, the temperature difference is larger than that in low state .The radiation between the outer and the inner ring becomes the main heat exchange in the annular cavity. The radiation model of the engine casing annular cavity can be considered to the radiation model of a geometric closed system with a cylinder nested on the gray surface.
3. Under different working conditions of engine, the characteristics of air flow and heat transfer laws are different. In the low state the flow in the annular cavity is weak. And most of them are natural convection. When the engine is in high state, the temperature difference inside and outside the casing annular cavity is large, the fluidity of the air flow in the cavity is enhanced. And the flow and heat transfer ability of the cooling air flow is also improved generally. It is mixed convection heat transfer. When the convective heat transfer coefficient Nu is studied, it is the mix Nusselt number obtained by the forced convection and natural convection criteria under given conditions.
4. Calculation method can be got through the analysis to the heat transfer law about this classical structure. And from this the temperature of the casing is got. The temperature difference between the calculation and the measured is only 1.3%.It can be considered highly credible.

References

[1] Zimmermann H. Some aerodynamic aspects of engine secondary air systems. *Journal of Engineering for Gas Turbines and Power*, 2002

[2] Cao Yuzhang. *Heat transfer of aero engine, Beijing University of Aeronautics and Astronautics Press*, 2005

[3] Rastogi A K, Rodi W. *Calculation of general three-dimensional turbulent boundary layers*. AIAA J, 1978

[4] Tao Wenshuan. *Numerical heat transfer, Xi'an Jiaotong University Press*, 2001

[5] Yan Shiming. *Heat transfer, Xi'an Jiaotong University Press*, 2001

1. Contact Author Email Address

Writer's e-mail: 29585755@qq.com
Writer's tel:13674254679

2. Copyright Statement