Performance Investigation on an Ultra-compact Interstage Turbine Burner with Trapped-vortex Slot Inlet

Hongtao Zhang *, Guangqi Luo, Lei Guan, Jianchen Zeng
Airforce Engineering University, Xi’an, China

*Corresponding author e-mail: nuaa202zht@163.com

Abstract. Ultra-Compact Combustor (UCC), which is one of mainstream design concepts of Interstage Turbine Burner (ITB), has the advantages of compact structure and high combustion efficiency. A design concept of an UCC with trapped-vortex slot inlet was proposed and numerical simulation of the stability, emissions, internal flow velocity and temperature distribution was carried out. The results indicated that the UCC with trapped-vortex slot inlet could enhance the mixing of combustion mixture and the mainstream airflow, improve the combustion efficiency, outlet temperature and the uniformity of outlet temperature field.

1. Introduction
Aviation gas turbine engine generally use after-burners which has the advantages of improved engine thrust, but with low combustion efficiency and higher fuel consumption rate. Increasing turbine inlet temperature will become the main method to increase thermal efficiency due to temperature ratio [1]. But the high temperature brings material, cooling and other aspects of the problem for the purpose of enhancing the performance of the engine. ITB realized the improvement of engine cycle work contribution to burning again through the channel between high and low pressure turbine blades [2,3].

There are two main design options for ITB at present. One is that a combustion chamber is arranged in the high and low pressure turbine transition section of the channel, the other one is that UCC scheme, which adopts the addition of a circumferential cavity through around the low pressure turbine vane, for combustion of fuel mixed with air [4-6]. Mawid M A promoted the mixing of gas and mainstream in the circumferential cavity by setting a radial vane cavity (RVC) on the UCC guide vanes [7,8]. Because of the conventional UCC performance decreasing with the increase of the main channel size, a rectangular structure based on the principle of cavity vortex combustion is proposed [9].

The secondary flow is the source of high speed swirling flow in the circumferential cavity, which determines the centrifugal acceleration of the oil droplets, and has an important influence on the combustion performance of the UCC. Therefore, it is necessary to study inlet way of the secondary flow. Compared with the inlet way from hole inlet, slot inlet can make the secondary flow obtain a wider axial distribution when entering the circumferential cavity, so as to promote mixing fuel-air in the circumferential cavity, and improve the combustion efficiency. The trapped-vortex combustion has the advantages of high combustion efficiency, low fuel-depleted boundary and low total pressure loss. On the basis of the inlet way of slot, a model of an UCC with trapped-vortex slot inlet is proposed in this paper, which has the trapped-vortex combustion and g-loading combustion advantages. By means
of numerical simulation, the combustion performance and internal flow characteristics of an UCC with trapped-vortex slot inlet are analyzed.

2. Definition and verification of the mode of UCC

2.1. Definition of the mode of UCC
The verification model of UCC is established and used as the benchmark model (Model-0) [10]. In view of the fact that the benchmark model is a central symmetric structure and purpose of saving computing resources and speeding up the calculation, the 1/6 model is only simulated in this paper, as shown in Figure 1. In order to describe the flow field, a blade with one side of a radial slot is used as a guide vane.

Three typical test cases are set up by UCC benchmark model [10]. Specific parameters are shown in Table 1. The turbulent chemical reactions in the baseline model are numerically simulated under the three conditions with CFD. The stability and emission parameters are obtained and compared with the experimental data of Reference 10. The accuracy of the calculation method is verified.

Table 1. Three typical test cases of UCC

| Condition | \( Main(\text{kg/min}) \) | \( Cavity(\text{kg/min}) \) | \( P(\text{kPa}) \) | \( dP/P(\%) \) | \( T(\text{K}) \) | \( \Phi_{\text{overall}} \) | \( \Phi_{\text{cav}} \) |
|-----------|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|
| LMLP      | 2.200           | 0.484           | 284.1           | 4.7             | 528             | 0.294           | 1.620           |
| LMMP      | 2.245           | 0.469           | 340.6           | 3.0             | 541             | 0.289           | 1.677           |
| HMHP      | 4.914           | 0.960           | 408.2           | 8.2             | 550             | 0.147           | 0.899           |

2.2. Computational Grid
The computational domain is meshed by ANSYS, ICEM and CFD and the mesh type is structured grid, with a total number of 1 million 250 thousand cells, as shown in Figure 2. At the same time, the wall region is encrypted and the first layer mesh is controlled between 30–100 \( y^* \), which satisfies the condition of non-equilibrium wall function.

Figure 1. The configuration of Model-0

Figure 2. Computational grid of Model-0
2.3. Boundary Condition
The convective boundary condition is adopted at the top wall of the circumferential cavity. The convection heat transfer coefficient is set at 8.1 W/(m² · K) and the ambient temperature is set to 500K. The combined boundary conditions are applied to both sides of the circumferential cavity and the entrance and exit walls. The convection heat transfer coefficients are set at 8.1 W/(m² · K), 10.2 W/(m² · K) and 12.1 W/(m² · K) respectively. The ambient temperature is set at 300K and the emissivity of radiation heat transfer is set at 0.85. The other walls are subjected to adiabatic wall condition without sliding. In order to approximate the real distribution of oil droplets better, five cone sprays are set up to simulate the fuel injection. The parameters such as diameter, half cone angle and flow rate of oil droplets are shown in Table 2 and the injection speed is set at 30.5m/s.

Table 2. Five groups of conical spray parameter

| Spray group | Half cone angle(°) | Droplet diameter(μm) | Percentage of flow(%) |
|-------------|--------------------|----------------------|-----------------------|
| 1           | 40.00              | 40.00                | 10                    |
| 2           | 30.00              | 42.50                | 25                    |
| 3           | 35.00              | 45.00                | 30                    |
| 4           | 32.00              | 47.50                | 25                    |
| 5           | 38.00              | 50.00                | 10                    |

2.4. Verification of the Calculation Result
The comparison between the numerical simulation results of the model, the experimental results and calculated results of the Reference 10 in the three test cases is shown in Table 3. It can be seen that the total pressure loss, carbon dioxide molar concentration and combustion efficiency calculated in this paper are greater than the experimental results. The molar concentration of carbon monoxide, oxygen and nitrogen oxides, and total outlet temperature are lower than the experimental data. However, the error between the parameters and the experimental data is kept within the acceptable range, and it can be well consistent with the law of each parameter changing with the working condition under the test condition.

Table 3. Comparisons between numerical simulation results of the model and the experimental and calculated results

| Condition | Type | dP/P(%) | CO (ppm) | CO₂(%) | O₂(%) | NOₓ(ppm) | T_exit(K) | ηb(%) |
|-----------|------|---------|----------|--------|-------|----------|-----------|-------|
| LMLP      | A*   | 5.00    | 1089     | 3.20   | 16.40 | 37.30    | 1254.0    | 97.50 |
|           | B**  | 3.76    | 1377     | 3.69   | 14.99 | 38.76    | 1166.5    | 94.90 |
|           | C*** | 5.73    | 990      | 3.97   | 14.66 | 13.40    | 1204.5    | 98.81 |
| LMPM      | A*   | 3.30    | 1264     | 3.70   | 15.70 | 48.20    | 1256.0    | 97.70 |
|           | B**  | 2.74    | 1423     | 3.74   | 14.91 | 54.86    | 1167.1    | 95.95 |
|           | C*** | 4.40    | 1222     | 4.00   | 14.65 | 13.60    | 1230.2    | 98.57 |
| HMHP      | A*   | 8.30    | 478.0    | 1.60   | 18.70 | 15.60    | 931.0     | 98.00 |
|           | B**  | 6.39    | 287.2    | 1.78   | 18.16 | 15.93    | 871.5     | 96.49 |
|           | C*** | 8.90    | 304.0    | 2.04   | 17.80 | 3.80     | 915.3     | 99.30 |

A* Experiment in Reference
B** Model in Reference
C*** Model established in this article
3. Effects of Trapped-vortex Slot Inlet on the Performance of UCC

Figure 3 shows a geometry model of an UCC with trapped-vortex slot inlet (Model-1). On the basis of Model-0, the inlet way of secondary flow is changed from whole inlet to slot inlet. The normal angle of the slot inlet is set to 37 degrees to keep the inlet flow angle of secondary flow unchanged and the width of the slot inlet is set at 0.96 mm to keep the inlet flow speed constant. A circumferential inlet slot is arranged at the back of the circumferential cavity to promote the formation of the stationary vortex in the second half of the circumferential cavity. The other structures are the same as those of Model-0.

Figure 3. The configuration of Model-1

3.1. Stability and Emission

Table 4 shows the comparison of pressure loss, combustion efficiency, outlet temperature, component concentration and pollutant emission between Model-1 and Model-0 in three conditions.

| Condition | Model | \( \frac{dP}{P} \) (%) | CO₂ (%) | O₂ (%) | CO (ppm) | UHC (ppm) | NOx (ppm) | \( T_{exit} \) (K) | \( \eta_b \) (%) |
|-----------|-------|----------------|--------|-------|---------|----------|----------|-------------|-----------|
| LMLP      | Model-0 | 5.73 | 3.97 | 14.66 | 990.3 | 110.80 | 13.4 | 1204.5 | 98.24 |
|           | Model-1 | 6.22 | 4.04 | 14.62 | 3.2 | 0.04 | 7.0 | 1242.0 | 99.99 |
| LMMP      | Model-0 | 4.40 | 4.00 | 14.65 | 1222.0 | 30.80 | 13.6 | 1230.2 | 98.57 |
|           | Model-1 | 5.35 | 4.01 | 14.66 | 16.2 | 0.04 | 11.6 | 1245.2 | 99.98 |
| HMHP      | Model-0 | 8.90 | 2.04 | 17.78 | 304.1 | 8.60 | 3.8 | 915.3 | 99.53 |
|           | Model-1 | 10.94 | 2.04 | 17.78 | 23.4 | 0.52 | 7.0 | 929.3 | 99.95 |

By comparing the parameters of Model-1 and Model-0, it can be seen that the CO and UHC molar concentrations of the Model-1 exit section are significantly lower than those of Model-0 but higher than it in total pressure loss. The reduction of combustible components CO and UHC means that combustion reactions in Model-1 are more complete than Model-0. Therefore, the combustion efficiency of Model-1 is obviously improved compared with that of Model-0 and remains above 99.95%. At the same time, the average temperature of the outlet section has also increased to a certain extent, the growth extent is above 14K. This is mainly due to introducing trapped-vortex in the circumferential cavity could enhance the mixing of combustion mixture with the mainstream air, promote combustion reactions in the combustor, thereby greatly reducing the concentration of combustible components and improving combustion efficiency. While enhancing the extent of mixing also brings a problem that increased the total pressure loss.

3.2. Velocity Field

The velocity distribution nephogram of Model-1 and Model-0 under LMLP conditions are shown in Figure 4. Both Model-1 and Model-0 have high velocity zones in the downstream channel near the...
circumferential cavity exit, but the radial and circumferential distrabution range of high-speed region of Model-1 is most extensive than Model-0, especially at the side of the blade basin. It illustrates that introducing the trapped-vortex in the circumferential cavity could drive radial transport of combustion products from the circumferential cavity into the main airflow and enhance fuel-air mixing, which is beneficial to improve the outlet temperature field of the combustor. Because the momentum exchange of micro fluid group, the high-speed region of Model-0 will disappear rapidly. Although the high speed region of Model-1 is still shrinking, but the outlet section has not disappeared completely. At the same time, the velocity stratification of Model-1 in the downstream channel of the circumferential cavity is earlier than that of Model-0. The distribution of lower velocity region in the middle part of the channel is larger.

![Figure 4. Velocity distribution nephogram of Model-1 and Model-0 under LMLP condition](image)

3.3. Temperature Field
Temperature distribution nephogram of Model-1 and Model-0 under LMLP condition are shown in Figure 5. In Model-1, there is no high temperature region similar to Model-0 in the cavity-in-cavity, the distribution area of the high temperature region in the circumferential cavity is also greatly reduced, and the temperature in the latter half of the circumferential cavity is much higher than that in the first half. This is mainly due to the trapped-vortex of second half that enhancing mixed-combustion of the mainstream and the combustion products of first half. It is not only improving flow temperature of second half in the circumferential cavity, but also improving temperature distribution of the cavity-in-cavity and circumferential cavity. In the main-stream channel of downstream in the circumferential cavity, Model-1 only distributes a small high temperature area on both sides of the blade, decreases with the flow of the main-stream and disappears completely at the outlet section. Compared with Model-0 which has always a high temperature area at the top of the blade channel, Model-1 effectively improves the working environment of the outlet wall. At the same time, the lower temperature range distribution at the bottom of the main-stream channel of Model-1 is smaller than that of Model-0 and disappears rapidly with the flow of the main-stream. This is mainly due to radial transport capacity of the circumferential cavity combustion mixture was improved, high temperature combustion products can be further drive into the mainstream. Through the energy exchange, the temperature of the main-stream is lowered with the main-stream temperature raising, thus improving the overall temperature level of the downstream of the main-stream. In the outlet section, the top and bottom of Model-0 are also distributed high temperature zone and low temperature zone, but the temperature distribution of Model-1 is very uniform.
Figure 5. Temperature distribution nephogram of Model-1 and Model-0 under LMLP condition

The outlet radial temperature varying with the relative height of the guide vanes of Model-1 and Model-0 under three conditions are shown in Figure 6. The radial mean temperature of the bottom region of Model-1 is higher than that of Model-0, while the radial average temperature in the top region is lower than Model-0, and the range of temperature change is much lower than that of Model-0. At the same time, the radial mean temperature of Model-1 increases with the vane height and presents a parabola distribution at first and then decreases. The maximum of radial mean temperatures are at 70%, 60% and 55% height of vane, being closer to the ideal inlet temperature distribution of the turbine rotor. Therefore, compared with Model-0, the radial temperature distribution of the outlet section in Model-1 is greatly improved.

Figure 6. Exit radial temperature varying with the relative height

Table 5 shows the temperature distribution coefficient of outlet section of Model-1 and Model-0 under three kinds of working conditions. The outlet temperature distribution (OTDF) and radial temperature distribution (RTDF) of Model-1 are obviously smaller than that of Model-0 under three conditions. This is because of the presence of trapped-vortex in the circumferential cavity enhancing mixed-combustion of the combustion mixture and main stream, which improves the ability of driving high temperature combustion products from the circumferential cavity into the main airflow. The high temperature combustion products can exchange momentum and energy better with low temperature mainstream airflow in the downstream channel, so that the high temperature area and low temperature area of outlet section basically disappeared, improves the uniformity of the outlet temperature field of the combustor.
Table 5. Temperature distribution coefficients of outlet section of Model-1 and Model-0 under three conditions

| Condition | Model  | OTDF | RTDF |
|-----------|--------|------|------|
| LMLP      | Model-0| 1.34 | 0.64 |
|           | Model-1| 0.53 | 0.08 |
| LMMP      | Model-0| 1.01 | 0.22 |
|           | Model-1| 0.61 | 0.09 |
| HMHP      | Model-0| 1.90 | 0.53 |
|           | Model-1| 0.61 | 0.05 |

4. Conclusion

A geometric model of UCC with trapped-vortex slot inlet is established for numerical simulation about three-dimensional turbulent combustion reacting flow in the combustor. This article discusses the influence of inlet way of the secondary flow on the performance of UCC and the results are shown below:

(1) The model established accurately reflects the flow field parameters of UCC with good applicability and reference, which can be used for further investigations of UCC.

(2) Compared to the conventional UCC, an UCC with trapped-vortex slot inlet due to the inlet way of secondary flow by hole inlet to slot inlet and introducing trapped-vortex inside the circumferential cavity, enhanced the mixing of combustion mixture and mainstream airflow, could effectively improve the combustion efficiency and outlet temperature of the combustor, and improve the uniformity of the outlet temperature field, while the total pressure loss increased to a certain extent.

(3) The above-mentioned research conclusions provided valuable guidance and advice to the structure improvement of UCC under different condition.

References

[1] Shen Wei-dao, Jiang Zhi-Min, Tong Jun-geng, Engineering thermodynamics, Higher Education Press, (2000) 280-295.
[2] Sirignano W A, Liu F, Performance increases for gas–turbine engines through combustion inside the turbine, Journal of Propulsion and Power, 1999, 15 (1): 111-118.
[3] Shang Shou-tang, Cheng Ming, Liu Dian-chun, Research status and development trend of turbine interstage combustion chamber, Aeronautical Science and technology, 2011(4): 79-82.
[4] Lee A S, Singh R, Probert S D, Performance of two combustor turbofan engine under design and off design conditions, AIAA 2008-4901, 2008.
[5] Lee A S, Singh R, Probert S D, Two-Combustor Engine's Transient Performance, AIAA 2009-4837, 2009.
[6] Zelina J, Ehret J, Hancock R D, Ultra-compact combustion technology using high swirl for enhanced burning rate, AIAA 2002-3725, 2002.
[7] Mawid M A, Park T W, Thornburg H, et al, Numerical analysis of Inter-Turbine Burner (ITB) concepts for improved gas turbine engine performance, AIAA 2005-1162, 2005.
[8] M. A. Mawid, Thornburg H, Sekar B, Performance of an Inter-Turbine Burner (ITB) concept with three-different vane cavity shapes, AIAA 2006-4740, 2006.
[9] Sekar B, Thornburg H, Briones A M, Effect of Trapped Vortex Combustion with Radial Vane Cavity Arrangements on Predicted Inter-Turbine Burner Performance, AIAA 2009-4603, 2009.
[10] David S. Moenter, Design and numerical simulation of two-dimensional ultra-compact combustor model sections for experimental observation of cavity-vane flow interactions, Ohio: Air University, 2006.