Analysis of the Influence of Cyclone Structure Characteristics on Combustion Chamber Performance

Wenwei Xiang¹, Jiang Liu¹, 2, *, Peifen Weng¹, 2 and Zaiguo Fu¹

¹School of Energy and Mechanical Engineering, Shanghai University of Electric Power, Shanghai 200090, China
²Shanghai Engineering Research Center of Energy-Saving in Heat Exchange Systems, Shanghai University of Electric Power, Shanghai 200090, China

*Corresponding author e-mail: jiangliuxjtu@shiep.edu.cn

Abstract. The cyclone is an important part of the combustion chamber of the gas turbine and has a direct impact on the combustion performance in the combustion chamber. Therefore, it is of great value to study the structural characteristics of the cyclone. This paper firstly analyses the structural characteristics of cyclone that have the greatest impact on the performance of the combustion chamber through the relevant data and the structural parameters of the actual combustion chamber. Then appropriately simplifies the structure of the combustion chamber, designing a F-class combustion chamber with cyclone. Numerical simulation method is compared with the actual combustion chamber data to verify whether the design of the cyclone meets the requirements, thus preliminary determining the blade installation structure and the number of blades of the combustion chamber cyclone. In the end, using the model combustion chamber as a blueprint, adjust the blade characteristics of the cyclone for different installation angles (40°, 45°, and 50°) and the number of blades (6, 8, and 10). The combustion chamber model is simulated to determine that the flow field of the combustion chamber is the best when the angle is 45° and the number of blades is 8.

1. Introduction

The cyclone is an important part of the combustion chamber. It has been widely used in a variety of combustion equipment, such as industrial furnaces, gas turbines, etc. In heavy-duty gas turbines, the cyclone effectively controls the flame stability and flame intensity in the combustion chamber [1]. Therefore, the structural characteristics of the cyclone, such as the blade installation angle, the number of blades, the cyclone channel area. It is extremely critical to the combustion performance of the combustion chamber. In recent years, gas turbines have developed rapidly in the field of efficient and clean power generation. Relevant scholars have done a lot of work to improve the performance and stability of gas turbine combustors, and also conducted a lot of experimental and numerical simulation studies on combustor cyclone.

Fu Zhongguang and Zhang Yizhong [2] studied the swirling combustion characteristics of a syngas micro-gas turbine, the swirl intensity was changed by changing the installation angle of the combustor cyclone. there is an optimal swirl angle for combustion and the flow achieves the best effect, the intensity
of the swirling flow in the duty zone has little effect on the combustion and flow in the combustion chamber.

Liu Yang, Chen Pengfei et al. [3] analysed the influence of cyclone on the temperature distribution of flue gas at the outlet of the combustor under different installation angles by three-dimensional modelling of a certain aerospace reformed gas turbine combustor and numerical calculation using fluent. It shows that the installation angle is in the range of 73.5°-83.5°, and reducing the installation angle can improve the temperature and speed of the flue gas at the outlet of the combustion chamber.

Tangirala VE et al. [4] and JOHNSON MR et al. [5] optimized the structure of the cyclone in the design, revealing the influence of different type cyclone on mixing of fuel and air, the temperature and velocity of the combustion chamber, pollutant emissions.

In order to further understand the working principle of the cyclone, optimize the design of the combustion chamber, and understand the effect of the geometric parameters of the cyclone on the combustion performance of the combustion chamber, this paper takes the combustion chamber of a heavy-duty F-class gas turbine as the research object, and uses Solidworks software for the combustion chamber and the cyclone was simplified in three-dimensional modeling. The CFD software Fluent was used to numerically simulate the combustion chamber. On the basis of obtaining the temperature field, velocity field and pollutant distribution data. The impact of different blades and the installation angle of the cyclone on the combustion performance of the combustion chamber were further studied. This study put forward the improvement plan of the cyclone structure design.

2. Physical model of combustion chamber

In this paper, a heavy F-class annular gas turbine combustion chamber is used as a prototype. 24 gas inlets are evenly arranged around the head of the prototype combustion chamber. Because the model is periodic and in order to simplify the entire numerical calculation process, during the modelling process, the structure of the combustion chamber is simplified and the inlet of the combustion chamber is intercepted that is 1/24 of the whole, so the speed of numerical calculation is increased without affecting calculation accuracy. Figure 1 shows the model combustion chamber and the cyclone. In the appearance of the cyclone, a cyclone is arranged in the outer circumferential direction at the head of the combustion chamber. Cyclone is divided into two parts: a duty fuel inlet and a duty fuel inlet. The cyclone ensure that a recirculation zone can be formed inside the combustion chamber to generate vortices to ensure uniform mixing of fuel and air, efficient and stable of combustion [6].

![Figure 1. The appearance of the model combustion chamber and cyclone](image)

The flow field structure and combustion characteristics in the combustion chamber are largely determined by the air intake method of the swirling combustion chamber. In the chamber, the temperature distribution, the combustion efficiency and generation of pollutants are inseparable from the number of cyclone blades and the installation angle. In this paper, the blade settings of cyclone are shown in Table 1, and the cyclone model is shown in Figure 2. First, select the three groups 1, 2, and 3 with the number of blades of 6, 8, 10, and the installation angle of 45° for simulation calculation and obtain the number of the cyclone that is most suitable for the flow field distribution and combustion characteristics in the combustion chamber. Then the installation angle is changed (40°, 45°, 50°) to study
the effect of installation angle on the combustion chamber. Finally, the structural parameters of the cyclone are analyzed based on the simulation results of the five groups. The influence law of flow field distribution, combustion characteristics and pollutant emission provide reference for the design of annular combustion chamber.

Table 1. Combustion chamber cyclone blade settings

| Cyclone number | 1 | 2 | 3 | 4 | 5 |
|----------------|---|---|---|---|---|
| Number of blades (pieces) | 6 | 8 | 10 | 8 | 8 |
| Blade installation angle (°) | 45 | 45 | 45 | 40 | 50 |

Figure 2. Cyclone model

In this paper, the entire fluid area of the model combustion chamber is used as the calculation area. When meshing the model, the inlet of the fuel on duty, the premixed fuel, and the outlet of the combustion chamber are encrypted. The structured coarse grid is used for processing when the surface has little effect on the combustion. The entire computing domain contains structured and unstructured grids. Part of the grid is shown in Figure 3. The grid spacing is 1 mm, and the total number of model grids is about 3 million, it meets the calculation accuracy requirements.

Figure 3. Combustion chamber grid

3. The method of numerical simulation calculation

3.1. Mathematical model

The physical and chemical processes in the combustion chamber are extremely complex and variable, including turbulent fluid flow, gas and air mixing, turbulent combustion chemical reactions, heat and mass transfer, and convection heat transfer. The combustion in the combustion chamber follows the law of conservation of mass, the law of conservation of momentum, the law of energy conversion and conservation, the law of composition conversion and balance, etc. The basic equations for gas phase control can be expressed generically as:
The dependent variable - $\varphi$ can be expressed as speed m/s, momentum kg*m/s, temperature, enthalpy kJ/kg, mass fraction, etc. $\Gamma_\varphi$, $S_\varphi$—the exchange coefficient and the source phase of $\varphi$.

3.2. Calculation model and boundary conditions

The interior of the combustion chamber is a strong swirl with shear flow, so the turbulence calculation uses the realizable k- $\varepsilon$ model, which is an improved k- $\varepsilon$ model, which adds rotation and curvature-related items to the turbulence viscosity calculation. Therefore, the effect of dealing with flow problems such as rotation and separation is significant [7], which can ensure the accuracy of the numerical calculation results. The SIMPLE algorithm is used for the coupling of speed and pressure, and the discreteness of the governing equations adopts the second-order precision upwind interpolation format.

The import and export parameter settings of the model combustion chamber are shown in Table 2.

| Name          | Main fuel quantity | On duty fuel quantity | Main combustion grade air flow | On duty air flow | Inlet temperature | Outlet pressure |
|---------------|--------------------|-----------------------|-------------------------------|-----------------|-------------------|-----------------|
| Uint Numerical value | kg/s               | kg/s                  | kg/s                          | kg/s            | K                 | MPa             |
| 0.6           | 0.038              | 26.35                 | 2.291                         | 697.24          | 1.8               |                 |

Side wall: Periodic symmetry boundary.

The combustion model uses a finite rate-vortex dissipation model, which can effectively limit the reaction rate of the inlet section and makes the model suitable for the simulation calculation of premixed combustion [8]. SIMPLE algorithm is used for the coupling calculation of pressure and velocity. The actual combustion process in the combustion chamber involves complex components and elementary reactions. In order to reduce the complexity of the model, this paper uses a simplified methane-air two-step reaction for gas combustion [9].

4. The effect of the blade number on the combustion chamber

4.1. Effect of the number of cyclone blades on the velocity field in the combustion chamber

The effects of three different cyclone blade numbers (blade installation angle 45°, blade number 6, 8, 10) on the velocity field in the combustion chamber were calculated through simulation. It can be seen from Figure 6 that under the three different blade numbers, the gas enters the combustion chamber through the main fuel inlet and the duty fuel inlet and generates respectively corresponding return areas near the center of the combustion chamber. A low-speed region with a relatively large range is formed in the recirculation zone which helps the fuel to stay and can fully burn the fuel. The velocity streamline diagram also shows that the three flow fields of the combustion chamber still maintain good. Two swirling vortices with opposite rotation directions are formed in the central area of the combustion chamber. The flue gas is brought back to the head of the combustion chamber to ignite the cold gas, which helps to stabilize the flame and prevent flameout during operation. It can also be seen that the maximum speed is different for different numbers of blades. When the number of blades is 6, the maximum speed is about 240m/s, and the outlet speed is greater than 220m/s, which is caused by insufficient backflow strength. Its exit speed is too high. The maximum speed is 190 m/s when the number of blades is 8 and 10, and the exit speed is around 170m/s. According to the actual gas turbine operation, the average speed of the annular gas turbine combustor outlet is currently 140-180m/s, so the outlet velocity of the combustion chamber is obviously too large when the number of blades is 6. When the number of blades is 8 and 10, the outlet speed distribution meets the actual requirements.
It can also be seen from Fig. 4 that the flow fields generated under different blade numbers are different. As the number of blades increases, the swirl vortex in the central area of the combustion chamber develops in the direction of expansion, and the position of the vortex gradually moves toward the gas inlet. The size and position of the swirl vortex will directly affect the combustion characteristics and flow field distribution in the combustion chamber. When the number of blades is 6, the swirl vortex is close to the outlet, which is not conducive to the mixing of the incoming flow and the flue gas. When the number of blades is 10, the swirling vortex close to the gas inlet that is likely to cause flashback, which poses a safety hazard. For the position of the swirl vortex, it is more appropriate when the number of blades is 8. Its position is in the appropriate area of the combustion chamber, which can stabilize the combustion and avoid the risk of flashback. At this point, the number of blades 8 is significantly better than the number of blades 6 and 10.

4.2. Effect of the number of cyclone blades on the temperature distribution in the combustion chamber

Figure 5 shows the temperature distribution of the $Z = 0$ section inside the combustion chamber under different blade numbers. It can be seen that the fuel-air mixtures in the three figures are ignited after entering the combustion chamber, and a high temperature zone is also formed in the combustion chamber. In comparison, when the number of blades is 6, its maximum temperature is relatively low. This is due to its high flow speed and the return zone closer to the combustion chamber outlet. The gas is discharged from the outlet too quickly and cannot be fully mixed and burned. The central temperature in the combustion chamber is relatively low.

**Figure 4.** Flow field distribution of $Z = 0$ cross-section with different blade number.
Figure 5. Temperature distribution of Z = 0 section under different blade number of blades.

When the number of blades is 6, the fuel cannot be fully burned, so its wall temperature is significantly lower than that of 8 and 10 blades. In contrast, when the number of blades is 8 and 10, because their swirl vortex is close to the position of the gas inlet, and the vortex continues to expand, so that the burned gas is quickly and evenly mixed with the incoming gas to make the fuel burn more fully. Because the model in this article is no wall cooling hole, mixing hole, the key recirculation area, central reflow area and other key areas lack the influence of external cooling airflow, the temperature distribution quickly reaches a uniform state-1500K.

From the above comprehensive analysis, it can be seen that the increase of the number of swirl blades will fully mix the high temperature gas, cold air and fuel, which also shows that the increase number of blades is conducive to improving the combustion temperature and fuel combustion efficiency. However, the increase in the number of blades also leads to a higher temperature at the gas inlet, which is likely to form a hidden safety risk of tempering. At this point, when the number of blades is 8, it is the most suitable.

4.3. Effect of different blade numbers on NOx production in combustion chamber

Through the simulation, the generation rate of NOx in the combustion chamber at the Z = 0 section under different blade numbers is shown in Fig. 6. As can be seen from Fig. 6 and the temperature distribution diagram in the previous section, when the number of cyclone blades increases, the temperature inside the combustion chamber changes greatly, the generation rate of thermal NOx in the combustion chamber changes significantly. It can be seen that the overall temperature distribution of the combustion chamber and the generation trends of nitrogen oxides are not much different. The generation rate of thermal NOx near the wall of the combustion chamber is smaller in the whole. The maximum generation rate of thermal NOx is upstream of the corner recirculation zone and the central recirculation zone, which are also high temperature zones. This phenomenon can also reduce the rate and concentration of nitrogen oxides in this area by increasing the cooling air in the later optimal design of the combustion chamber.
Figure 6. Generation rate graph of thermal NOx with Z = 0 cross section under different cyclone blade numbers.

From the comparison of the three figures in Figures 6 shows that the lower local temperature and the smaller reflux zone range are conducive to controlling the generation of nitrogen oxides during the operation. In summary, it can be concluded that temperature is an important factor affecting the rate and concentration of pollutants in the combustion chamber. The higher the temperature, the higher the rate of nitrogen oxide generation and the higher the concentration of nitrogen oxide emissions. Therefore, controlling the local high temperature zone and average temperature will be one of the key means to control the generation of nitrogen oxides.

5. The effect of blade installation angle on combustion chamber

Through the study of the influence of different blade numbers on the combustion chamber flow field and combustion characteristics, it is found that when the number of blades is 6, 8, and 10, each has its own advantages and disadvantages. When the number of blades is 8 and 10, the effect on the combustion chamber is not very different, they have excellent in the return area, temperature distribution and velocity flow field. but the number of blades 8 is more conducive to the operation of the combustion chamber than the number of blades is 10, blades 8 has a more stable high temperature area, lower of nitrogen oxide emissions. When the number of blades is 6, its pollutant emission control is the most effective among the three types of blades, but it is inadequate compared with the number of blades of 8 and 10 in terms of combustion stability, efficiency, and high temperature areas. In general, 8 pieces of blade is the most suitable for the operation of the combustion chamber. Therefore, 8 blades are selected as the basis, changed the blade installation angle (40°, 45°, 50°) to study the effect of installation angle on the flow field and combustion characteristics in the combustion chamber.

5.1. Effect of different blade installation angles on the velocity distribution of Z = 0 section of combustion chamber

Figure 7 shows the distribution diagram of the Z=0 cross-sectional flow field in the combustion chamber when the installation angle of the cyclone blades is 40°, 45° and 50°. From the figure, it can be seen that the trend of velocity distribution is similar at each installation angle. The fuel gas mixture enters the combustion chamber through the main fuel inlet and the duty fuel inlet and produces a good recirculation area near the center of the combustion chamber. There are relatively large low-speed areas in the recirculation area. This area helps fuel stay. The velocity streamline diagrams also show that, the flow fields of the combustion chamber with different installation angles of the cyclone are still good. Two
vortexes with opposite rotation directions are formed in the central area of the combustion chamber. The outlet velocity is in the area of 170m/s, which is in line with the actual combustion chamber operation.

**Figure 7.** Flow field distribution of Z = 0 section under different cyclone blade installation angles.

In comparison, when the installation angle is 45°, the flow field distribution in the combustion chamber is relatively uniform, and the two vortexes with opposite rotation directions are more distinct, which helps the stability of combustion in the combustion chamber.

5.2. Effect of different blade installation angles on the temperature distribution of Z = 0 section in the combustion chamber

It can be seen from Figure 8 that under the three different installation angles, the temperature distribution of the Z = 0 cross section in the combustion chamber is almost the same. At the same time, due to the absence of wall cooling holes and mixing holes in the combustion chamber used in the simulation, the temperature distribution of combustion area quickly reached a uniform state, and the temperatures are all around 1500k.

**Figure 8.** Temperature distribution of Z = 0 section in combustion chamber under different blade installation angles.
5.3. Effects of different blade installation angles on NOx production rate of Z = 0 section of combustion chamber

Figure 9 shows the nitrogen oxide generation rate of Z = 0 section when the installation angle is 40°, 45° and 50° respectively. It can be seen from the figure that the NOx generation rate at different installation angles is not much different. But when the installation angle is 45°, the rate of NOx production is maximum. According to the higher the temperature, the faster the NOx generation rate can be obtained, so the center temperature is the highest when the installation angle is 45° and combustion efficiency is the highest at this angle. From an overall point of view, the corner recirculation zone and the central recirculation zone of the combustion chamber are still the key areas for thermal NOx generation, and changing the cyclone installation angle will not easily change the characteristics of NOx in these two areas.

![Blade Installation Angles Diagram]

Figure 9. NOx generation rate diagram of Z = 0 section under different blade installation angles.

6. Conclusion

By changing the structural parameters of the cyclone, the paper simulates the effect of the cyclone on the combustion chamber under different blade numbers (6 pieces, 8 pieces, 10 pieces) and different blade installation angles (40°, 45°, 50°). It can be seen from the analysis results that when the number of cyclone blades is in the range of 6-10, the return area in the combustion chamber continuously moves to the inlet of the combustion chamber, and the swirling vortex expands toward the side walls, although this allows the fuel and the air is mixed more fully, which can enhance the temperature of the central area and improve the combustion efficiency, but too close to the entrance of the combustion chamber can easily lead to flashback, which needs to be avoided.

At the same time, the generation of NOx is closely connected with the temperature. The higher the temperature, the more NOx is generated. The NOx generation area is mainly concentrated in the recirculation area and the angular flow area. This phenomenon can also be reduced in the later combustion chamber optimization design by increasing the area cooling air and other methods to reduce the temperature of the area, thereby reducing the nitrogen oxide generation rate and concentration. However, the installation angle of the cyclone blades in the range of 40-50° has little effect on the flow field in the combustion chamber. Compared with the installation angle, the angle 45° can improve the efficiency of the combustion chamber.

In summary, in the structural parameters of the number of blades 6, 8, 10 and the installation angle of 40°, 45°, and 50°, when the number of blades is 8, the installation angle is 45°; the performance of flow field distribution and combustion characteristics is the best.
Acknowledgments
This work was financially supported by Capacity Building Projects in Local Universities of Science and Technology Commission of Shanghai Municipality (No.19020500900).

References
[1] LINCK M B, GUPTA A K, Twin-fluid atomization and novel liftedswirl-stabilized spray flames, [J]. Journal of Propulsion and Power, 2009, 25 (2) : 344-357.
[2] Fu Zhong Guang, Zhang Yi Zhong, Lu Ke, Numerical study on swirl combustion characteristics of micro gas turbine combustor burning syngas, [J]. Thermal Power Generation, 2015, 44(03) : 15-20.
[3] Liu Yang, Chen Peng fei, Shi Jun Rui, Jia Li Na, Effect of Installation Angle of Gas Turbine Cyclone on Outlet Temperature Quality of Combustor, [J]. Journal of Engineering for Thermal Energy and Power, 2017, 32 (S1) : 41-45+128-129
[4] Tangirala V E, Tolpadi A K, Danis A M, et al, Parametric modeling approach to gas turbine combustion design, [C]. Proceedings of ASME Turbo Expo 2000. Munich, Germany, 2000.
[5] JOHNSON M R, LITTLEJOHN D, NAZEE R W A, et al, A comparison of the flow fields and emissions of high-swirl injectors and low-swirl injectors for lean premixed gas turbines, [J]. Proceedings of Combustion Institute, 2005, 30:2867-2874.
[6] BAEK G M, SOHN C H, Study of numerical modeling of swirl-premix burner for simulation of gas turbine combustion,[J]. Transactions of the Korean Society of Mechanical Engineers B, 2013, 37(2): 118.
[7] FU Zhong Guang, Wang Rui Xin, Shi Li, Song Jia Sheng, Effect of cyclone channel width on swirling premixed combustion:numerical simulation, [J]. Thermal Power Generation, 2018, 47 (02):63-70+77.
[8] Wang Hai lian, Large eddy simulation of combustion instability of lean burn premixed swirl, [D]. Dalian: Dalian University of Technology, 2014: 38.
[9] Zhang Xin Gang, Study on self-excited instability of premixed combustion in combustion chamber of gas turbine, [D]. Nanjing: Southeast University, 2007: 59.