PAPER • OPEN ACCESS

Research on Simulation and Planning of Combustion Chamber in Gas Turbine

View the article online for updates and enhancements.

Research on Simulation and Planning of Combustion Chamber in Gas Turbine

Zeyuan Liu, Rongxin Lv*, Jiankai Zuo, Jiatong Chen, Xiangquan Yin, Xinyu Zhao

Shenyang Aerospace University, Shenyang, 110136

*lvrongxin@sau.edu.cn, lrxin@msn.com

Abstract—A gas turbine is an internal combustion power machine that drives the impeller to rotate at a high speed with a continuous flow of gas as the working medium, and converts the energy of the fuel into useful work. Nowadays, all developed countries attach great importance to the development of gas turbines, and the combustion chamber of the gas turbine is an important part of it. If the combustion chamber is properly optimized, the combustion efficiency will be greatly improved. The change of the number of blades of the gas turbine will have a great influence on the combustion effect inside the combustion chamber. In this paper, the gas turbine blade values are modified to study its influence on the combustion chamber flow field characteristics and methane mixing characteristics. Finally, when other conditions are constant, the larger the value of the blade, the more conducive to the formation of the return zone of the combustion chamber and the mixing of gas, and the better the combustion effect in the combustion chamber. This study also provides a reference for the design of gas turbine combustor.

1. INTRODUCTION

Gas turbine technology is a high-tech integration of multiple technologies. It is also a strategic emerging industry related to national energy and national defense security. It has become one of the important symbols of a country's technological level, military strength, and comprehensive national strength. It has an irreplaceable strategic position and role in the fields of electric power, energy mining and transmission, ships and national defense land and distributed energy systems. Developed countries attach great importance to the development of gas turbines, and the combustion chamber is one of the three core components of gas turbines. The characteristics of the flow field distribution and mixing characteristics inside the combustion chamber are the key factors to measure the pollutant emission and operating efficiency of gas turbines. In order to effectively improve combustion efficiency and reduce pollution emissions, countries all over the world have conducted extensive research on the simulation of gas combustion in the combustion chamber.

Scholars at home and abroad have proposed many simulation methods. In the [1], based on the combustion chamber model and the sub-damping modal model, the author introduced a program to obtain the combustion chamber vibration model from the gas turbine, with the purpose of laying the foundation for the analysis of acoustic phenomena in the future. In the [2], the author conducted a comprehensive 3D numerical simulation of the combustion chamber of the V94.2 Siemens gas turbine, which considered the details of the diffusion mode, and concluded that the transformation of the booster chamber has a great influence on the uniformity of the airflow distribution, so the conclusion has a great influence on the maximum flame temperature of each burner. Babak Kashir [3] et al. used

numerical simulation as a research method and found that when the swirl number is small, the tail of the premixed flame has radial convergence; when the swirl number increases, the premixed flame presents an inverted cone shape. The tail of the flame gradually opened. Cheon Hyeon Cho [4] used CFD to simulate the change of NOx emissions in a double cone combustor, and the results showed that NOx emissions are closely related to the uniformity of fuel and air mixing.

P.Weigand [5] took the methane combustion process as the research object, studied the swirling diffusion flame of methane fuel in the combustion chamber, and found that the flow field structure does not change with the equivalent ratio. At the same time, there is a high-speed internal recirculation zone and an external low-speed recirculation zone in the combustion zone. And Stopper Ulrich [6] used optical instruments to analyze the flame and flow field characteristics of the micro gas turbine combustor. The experimental results show that the flame structure in the combustor changes significantly with the Reynolds number, and the pressure oscillation in the combustor and the swirl of the nozzle Shedding is close. In addition, references are also given in [7-10] in terms of gas turbine simulation, structure, performance and application, as well as the introduction of machine learning methods such as neural networks.

This paper mainly studies the effect of swirl number on the flow field distribution and fuel mixing characteristics in the swirl premixed combustion. The research objects are two-stage cyclone and gas turbine combustor. Firstly, the swirler and combustion chamber models are established and the corresponding fluid domains are extracted, and the swirl number is changed by modifying the number of swirler blades. Then this paper uses different swirl numbers to mesh the model, and finally import the meshed model into Fluent for numerical simulation. According to the simulation results of the combustion chamber model with different swirl numbers, we observed its flow field distribution map, fuel concentration distribution map and recirculation zone distribution. By comparing the changes in the flow field and fuel distribution under different swirl numbers, the swirl number is summarized the effect on both.

2. MODEL ESTABLISHMENT

In this paper, according to the size of the gas turbine combustion chamber in actual production, various parameters of the combustion chamber are determined, and the model is established on the platform of UG NX 12.0 software. This article uses a two-stage cyclone, divided into duty class and main combustion class. The combustion chamber is a single-tube combustion chamber, and the swirler is placed at the head of the combustion chamber. In the working process, the gas and air enter the swirler from the fuel nozzle and the air nozzle respectively, pass through the swirler's main combustion stage blades and duty stage blades, and enter the combustion chamber in a rotating jet manner and are fully mixed. The specific combustion chamber model diagram is shown in Figure 1.
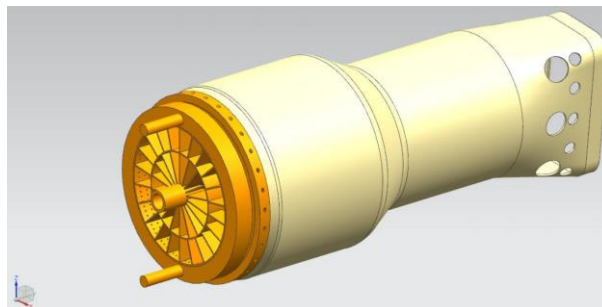

Figure 1.   Combustion chamber model.

This paper establishes multiple combustion chamber UG models according to the number of blades, and calculates the corresponding swirl number according to the calculation formula of the swirl number, which is convenient for the later analysis of related parameters. When assuming that the installation angles of the two-stage swirler main combustion stage blades are all 60°, the structural parameters of each model and the corresponding swirl numbers are shown in Table 1.

TABLE I. MODEL STRUCTURE PARAMETERS CORRESPONDING TO DIFFERENT SWIRL NUMBERS

Combustion chamber model number	Number of blades in main combustion stage of two-stage swirler	Swirl number
1	18	3.993
2	22	4.092
3	30	4.195

According to the parameters in Table 1, three sets of combustion chamber models are established. At the same time, the Boolean operation function in UG NX 12.0 is used to extract the fluid domain of the combustion chamber and swirler. In the extraction process, in order to facilitate subsequent experiments, the asymmetric structure at the tail of the combustion chamber was simplified to a cylinder, and some complicated boundaries were appropriately simplified, and then later calculations were performed. The extracted fluid domain model diagram and cross-sectional diagram are shown in Figure 2.

After completing the drawing of the UG model, this article performs meshing in the mesh module of ANSYS Workbench. In order to improve the accuracy of the Fluent numerical simulation results, the number of grids in the model needs to be large enough, and when the number of grids is too large, it will cause computational complexity. Therefore, it is necessary to constantly adjust the various parameters of the grid, not only to ensure that there is a sufficient number of grids, but also to minimize the computational cost.
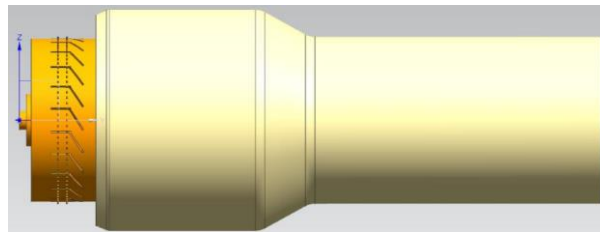

Figure 2.   Combustion chamber fluid domain model diagram.

In the process of drawing the grid, first set the boundary layer of the model. The model boundary used in this paper mainly includes main fuel nozzle 1, main fuel nozzle 2, duty fuel nozzle 1, duty fuel nozzle 2, air inlet, gas outlet and wall surface. After setting the boundary layer, set the meshing parameters. This study uses a tetrahedral mesh, the mesh size is set to 8mm, and the Relevance Center and Relevance parameters are adjusted at the same time. The final number of grids drawn is 4,153,226. The grid model of the combustion chamber is shown in Figure 3.
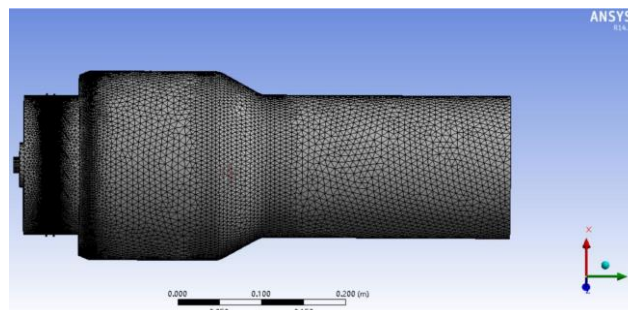

Figure 3.   Meshing model diagram.

This paper uses Fluent in ANSYS Workbench to numerically simulate the mesh model. In order to ensure that the imported mesh model can be effectively simulated, the mesh needs to be checked. After checking, set the parameters of the solver. First set the solver type Pressure-Based, the space condition

is Axisymmetric, and the time condition is Steady. Then activate the energy equation, select the standard k turbulence model and the methane-air composition model. At the same time, the material properties are set to *nitrogen-oxide (no)* model. Then set the boundary conditions of the model. The inlet boundary conditions of the model used in this paper are all velocity boundary conditions, and the corresponding inlet and outlet temperature and gas composition are set. The methane concentration of 4 fuel nozzles is 92.18%, and the composition of the air inlet is set according to the real air composition. The specific boundary condition parameter settings are shown in Table 2.

After setting the above parameters, set all the components in the Under-Relaxation Factors and Energy in the solution parameters to 0.95. Since this article studies the gas flow field and gas mixing characteristics, there is no need to set up chemical reactions. Finally, the flow field is initialized, the number of iterations is set, and the simulation results are obtained.

TABLE II. BOUNDARY CONDITIONS OF INLET AND OUTLET OF COMBUSTION CHAMBER

| Boundary name	Parameter name	Parameter value
Duty class fuel nozzle_1	Velocity Magnitude	32.35m/s
	Temperature	446K
	Turbulence Length Scale	0.003m
	Species Mass Fraction	$CH_4 = 0.9218$
Duty class fuel nozzle_2	Velocity Magnitude	10.99m/s
	Temperature	446K
	Turbulence Length Scale	0.003m
	Species Mass Fraction	$CH_4 = 0.9218$
Main combustion stage fuel nozzle_3,4	Velocity Magnitude	71.3m/s
	Temperature	446K
	Turbulence Length Scale	0.003m
	Species Mass Fraction	$CH_4 = 0.9218$
Fuel export	Temperature	933K
	Backflow Turbulence Length Scale	0.003m
	Species Mass Fraction	$O_2 = 0.05$ $H_2O = 0.1$ $CO_2 = 0.1$

3. RESULTS OF SIMULATION EXPERIMENTS

The experiment is mainly divided into two situations: the influence of the number of main combustion stage blades on the flow field and the influence of the number of main combustion stage blades on the methane concentration.

The number of blades in the main combustion stage will affect the number of swirls. A reasonable design of the number of blades plays an important role in improving the combustion effect of the combustion chamber and reducing the manufacturing cost of the combustion chamber. Therefore, this section changes the swirl number by changing the number of main combustion stage blades, and then studies the effect of swirl number on the flow field inside the combustion chamber. Figure 4 shows the gas streamlines in the combustion chamber obtained by Fluent numerical simulation under the condition of three different blade numbers.

It can be seen from Figure 4 that the flow field distribution under different numbers of blades is similar to the flow field distribution under different blade installation angles, and there is a backflow zone at the outlet of the swirler. At the same time, the high-speed gas flow is concentrated between the recirculation zone and the swirler outlet, and at the back of the combustion chamber. Comparing the gas flow field with different numbers of blades, it can be found that the larger the number of blades, the larger the corresponding return zone, and the more uniform the return zone distribution. It can be

known from the literature that the reflux zone of the combustion chamber plays a key role in the effect of gas mixing. The recirculation zone can make the temperature distribution of the gas in the combustion process more uniform, and the mixing with air is more uniform, the more stable the recirculation zone, and the larger the distribution area, the more conducive to the mixing of gas and air. In summary, the greater the number of blades, the more beneficial the mixing of methane and air.
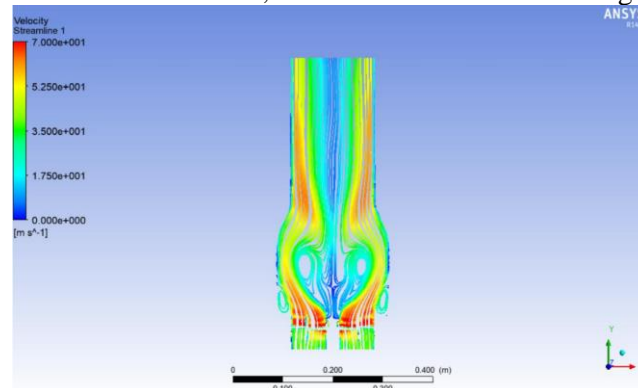
(a) The Gas velocity distribution when the number of blades is 18.
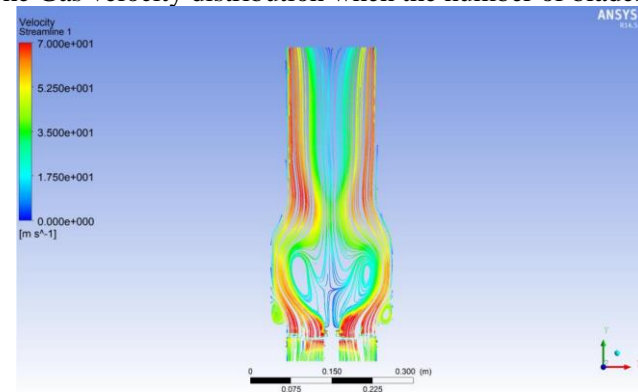
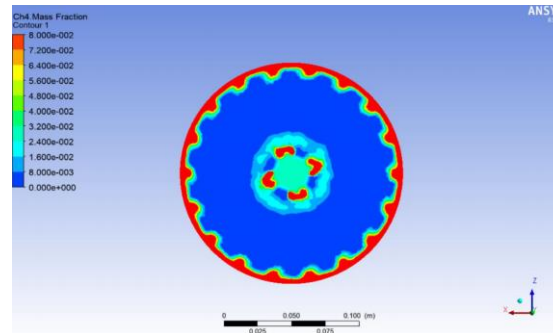(b) The Gas velocity distribution when the number of blades is 22.
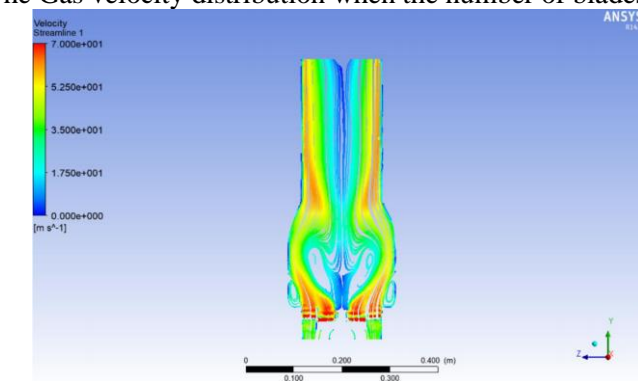
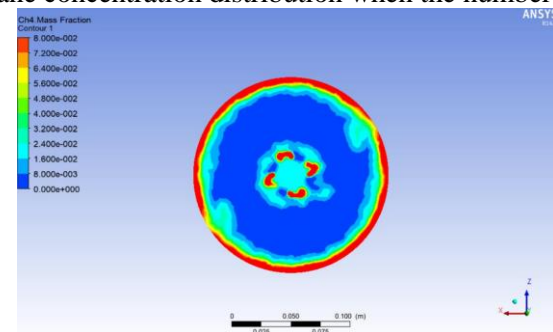(c) The Gas velocity distribution when the number of blades is 30.

Figure 4.   Numerical simulation results of gas flow velocity with different blade numbers when the blade installation angle is 60o.

The next situation mainly studies the effect of the number of blades in the main combustion chamber on the distribution of methane concentration inside the combustion chamber. Under the premise of a fixed blade installation angle, the number of blades in the main combustion stage has a greater effect on the distribution of methane concentration. Impact. Therefore, this section changes the swirl number by changing the number of blades in the main combustion chamber of the combustion chamber, and then studies the effect of swirl number on the distribution of methane concentration. Figure 5 is a contour
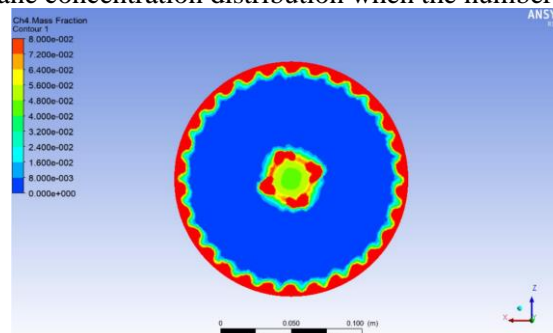
map of the methane gas concentration distribution at the exit section of the combustor obtained by Fluent numerical simulation under three different numbers of main combustion stage blades.

(a) The Methane concentration distribution when the number of blades is 18.

(b) The Methane concentration distribution when the number of blades is 22.

(c) The Methane concentration distribution when the number of blades is30.

Figure 5.   Numerical simulation results of methane gas concentration under different number of blades when the blade installation angle is 60o.

From the three simulation results in Figure 5, it can be seen that the methane concentration distribution in the outlet section of the cyclone under different blade numbers has the same characteristics. The high concentration of methane gas is mostly distributed at the four points at the center of the section and the outlet of the cyclone. The wall surfaces correspond to the passage between the fuel inlet 1 and the combustion chamber, the fuel nozzle 3 and the fuel nozzle 4 respectively. At the same time, careful observation shows that the methane concentration around the fuel nozzle 1 at the center of the cross section is also higher than that of the air inlet, and the highest methane gas concentration in this part of the three blade numbers corresponds to when the number of blades is equal to 30.

4. CONCLUSION

In this paper, by changing the size of the number of blades of a gas turbine, the characteristics of the gas flow field and the mixing characteristics of methane in the combustion chamber under different numbers of blades are studied. In this paper, three groups of combustor UG models are established and

the fluid calculation domain is extracted. We draw the extracted fluid domain and import it into Fluent, set the corresponding initial parameters and boundary conditions, and perform numerical simulations. Finally, the following conclusions were drawn: Under the same other conditions, the larger the number of blades, the larger the distribution area of the return zone of the corresponding combustion chamber gas flow field, the more stable the return zone, and the more conducive to the mixing of gas and air. As for the methane gas distribution at the exit section of the cyclone, the larger the number of blades, the more uniform the methane gas distribution. Therefore, the larger the number of blades, the more conducive to the formation of the return zone in the flow field of the combustion chamber and the mixing effect of methane gas.

REFERENCES
[1] I. M. Alonso, E. E. Rodriguez Vazquez, L. Alvaro Montoya Santiyanes, H. J. Zuniga Osorio and H. G. Cuatzin, "Vibrational model for a gas turbine combustion chamber gotten from experimental tests," 2016 IEEE International Engineering Summit, II Cumbre Internacional de las Ingenierias (IE-Summit), Boca del Rio, 2016, pp. 1-4.
[2] A. Raja, A.A. Shirazpour and M. Peiman, "Modification for Combustion Chamber of V94.2 SIEMENS Gas Turbine in Diffusion Mode," 2019 International Power System Conference, Tehran, Iran, 2019, pp. 195-201.
[3] Babak Kashir, Sadegh Tabejamaat, Nafiseh Jalalatian. A num erical study on combustion characteristics of blended methane-hydrogen bluff-body stabilized swirl diffusion flames. International Journal of Hydrogen Energy, 2015, 40(18): 6243-6258.
[4] Cheon Hyeon Cho, Gwang Min Back, et al. Anumerical approach to reduction of NOx emission from swirl premix burner in a gas turbine combustor. Applied Thermal Engineering, 2013, 59(1-2): 454-463.
[5] P.Weigand, W.Meier, X.R.Duan, et al. Investigations of swirl flames in a gas turbine model combustor:I.Flow field, structures, temperature, and species distributions. Combustion and Flame, 2006, 144(2):205-224.
[6] Ulrich Stopper,Wolfgang Meier, Rajesh Sadanandan, et al. Experimental study of industrial gas turbine flames including quanification of pressure influence on flow field, fuel/air premixing and flame shape. Combustion and Flame, 2013, 160(10): 2103-2118.
[7] Zuo J., Zhang C., Chen J., Wu Y. Application of Wavelet Neural Network Prediction Model Based on Particle Swarm Optimization. Frontier Computing.FC 2019. Lecture Notes in Electrical Engineering, vol 551. Springer.
[8] T. A. Kuznetsova."Automatic Control System of Low-Emission Combustion Chamber Based on Neural Network Emission Model," 2018 International Russian Automation Conference, Sochi, 2018, pp. 1-6.
[9] J. Zuo, C. Zhang, J. Chen, Y. Wu, Z. Liu and Z. Li, "Artificial Intelligence Prediction and Decision Evaluation Model Based on Deep Learning," 2019 International Conference on Electronic Engineering and Informatics (EEI), Nanjing, China, 2019, pp. 444-448.
[10] S. I. Serbin, I. B. Matveev and G. B. Mostipanenko, "Investigations of the Working Process in a "Lean-Burn" Gas Turbine Combustor With Plasma Assistance," in IEEE Transactions on Plasma Science, vol. 39, no. 12, pp. 3331-3335, Dec. 2011.