Numerical simulation on velocity and temperature of a slotted swirl combustor

Haozhen Li, Jiming Lin, Yong Zhang
1. College of Mechanical Engineering and Automation, Huaqiao University, Xiamen, Fujian Province, 361021, China
2. Key Laboratory of Process Monitoring and System Optimization for Mechanical and Electrical Equipment (Huaqiao University), Xiamen, Fujian Province, 361021, China
*Corresponding author’s e-mail: linjiming@hqu.edu.cn

Abstract: Cyclone combustion chamber has the advantages of compact structure, short flame length, good combustion stability, high combustion efficiency and low emissions of pollutants, etc. As one of the key components of the cyclone combustion chamber, the cyclone has a direct impact on the working performance of the combustion chamber. In this paper, we propose a slotted swirler and describe its structure in detail. We simulate the cold state flow field and the hot state temperature field in the combustion chamber of a cyclone with open seam using the CFD method, taking into account the effect of open seam on the air flow field and temperature distribution after the cyclone at different angles. The simulation shows that the temperature distribution in the slotted swirl combustor exhibited a center-focused flame of higher temperature, compared with the non-slotted swirler, the slotted swirler exhibits certain advantages in terms of drag reduction, combustion stability and combustion efficiency, which has a certain guiding effect on the improvement of the working performance of the combustion chamber.

1. Introduction
Swirl combustion has been widely implemented in civil, industrial and military combustion facilities, including gas turbine engines, furnaces, aero-engines and fuel cells [1]. In the combustion chamber, the cyclone is a key component of the combustion chamber head, so the exploration of the cyclone is particularly important [2]. In addition to having a flame stabilizing effect, the cyclone also has the function of enhancing the degree of atomization and improving the combustion efficiency [3]. The cyclone goes from single stage, to double and triple stage, and the cyclone blade goes from straight to curved [4].

Previous studies of single-stage cyclones have mostly considered the influence of changes in the number of cyclones, structurally [5]. Some scholars have studied the return zone from the changes of blade type, number and mounting angle. Among these combustions, the swirl combustions are of particular interest due to their numerous merits [6-8]. In the aspect of experimental study on swirl combustor structure, Beer and Chigier systematically investigated the characteristics of single-stage swirl combustors [9]. A tangential swirler was proposed by Litvinov et al. to study the characteristics of the processing vortex core [10]. Eldrainy proposed a swirler which is capable of varying the swirl number while remaining the same Reynolds number [11]. In the aspect of numerical simulation of swirl combustor, K. Khanaf et al. studied the influence of swirl velocity and burner wall temperature on NOx formation, and the chemical equilibrium system considered included 16 reactions and 20 substances. The results
show that increasing the swirling flow has a monotonous reduction effect on BOTH CO and unburned hydrocarbons, and NOx increases first and then decreases with the increase of swirling number\textsuperscript{12}. Parag Rajpara\textit{e}t al. used the large eddy simulation turbulence model (LES), the turbulence chemical model with assumed shape probability density function (PDF) and the discrete ordinate (DO) radiation model to study the heat distribution and emission characteristics of the combustion Chambers of conventional and reverse flow gas turbines\textsuperscript{13}.

In the swirl combustion chamber, flame stability is achieved by the formation of a negative pressure zone downstream of the cyclone, which is known as the central recirculation zone\textsuperscript{14}. The shape, size and flow characteristics of the central recirculation zone have an important influence on the operation of the cyclone combustion chamber and the structure of the cyclone has a significant influence on the recirculation zone and the flow pattern in the combustion chamber, thus the structure of the swirler plays an important role in determining the combustion performance and the formation of exhaust gases\textsuperscript{15}.

Based on the above considerations, the present work proposed a non-premixed slotted swirl combustor and investigated its performance and design flexibility. Although numerous worthy research works on swirler design have been conducted and summarized above, literature survey shows that few studies have been conducted to investigate the combustion performance of a slotted swirl combustor \textsuperscript{16}. Both cold-flow and combustion experiments were conducted to study the flow and combustion characteristics in the slotted swirl combustor. Comparison was made for the data acquired in the same combustor with non-slotted swirler. The results obtained in the work are believed to be of value for further detailed experiment and numerical simulation research on slotted swirl combustion. For further elaboration, it has been agreed that the Combustor with Slotted Swirler is abbreviated as CSS and the Traditional Swirl Combustor abbreviated as TSC.

The structure of the paper is organized as follows: a thermodynamics analysis of slotted swirl combustion is given in Section 2. Numerical specifications are given in Sections 3. The simulation results of CSC and NSC in cold and hot states are shown and discussed in Section 4. Concluding remarks and suggestions for future study are given in Section 5.

2. Blunt body stabilizes flame mechanism

For the design optimization of the burner, in addition to maintaining the high efficiency of the combustion chamber, it is important to maintain the stability of the combustion without problems such as blowout and backfire. Maintaining the stability of the flame is important for the temperature operation of the combustion chamber.

Well-established burner stabilization technologies are mainly achieved through fuel-rich/depleted combustion, enhanced heat reflux and enhanced initial combustion.

In this paper, the blunt body method of flame stabilization is used to stabilize the flame. The blunt body is widely used in the field of combustion and it is one of the common and effective flame stabilizing structures. The shapes of the obtuse body vary, and the common ones are round, semi-conical and V-shaped\textsuperscript{17}. The flame-stabilizing mechanism of this method is that when the high speed gas flow through the blunt body, under the action of the gas viscous force, the gas flow after the blunt body is taken away to cause pressure reduction, so that part of the gas flow downstream of the blunt body to flow in the opposite direction to compensate for the gas in this region, as shown in Figure 2.5. The reflux brings the hot flue gas back to the vicinity of the passivated body region, so that the reaction temperature is rapidly increased to ensure the stability of the flame.
The key mechanism of cyclonic combustion is the formation of a cyclonic zone downstream of the cyclone. The reflux zone is characterized by long residence time, high turbulence intensity and high temperature. When the return flow in the zone joins the main stream, a shear layer is formed which provides a good place for mixing, preheating and combustion of the fuel/air mixture \[18\]. The high temperature and low velocity return flow plays an important role in the ignition of the main stream. Based on these insights, it is expected that a certain amount of airflow and fuel injected directly into the recirculation zone through a trough or hole in the cyclone separator can rapidly warm up and ignite in the recirculation zone. As a result, the main stream is more likely to ignite and burn.

3. Numerical method
Fig. 2 is a schematic diagram of the eddy current combustion chamber used in this study. The structure comparison between the TSC and the CSS is shown in Fig. 3. The inner and outer diameters of the swirler are 150 mm and 436 mm respectively. Twelve 30° blades are uniformly installed around the swirler. The blade installation Angle is the included Angle between the rotary plane at the blade radius R and the chord length of the blade interface. There are twelve outer ring holes on the slit cyclone, 50 mm away from the axis, and 6 inner ring holes, 25 mm away from the axis. The circular holes are uniformly distributed along the radial direction of the cyclone.
Methane is used as combustion material in this numerical experiment. Since the purpose of the present work is to experimentally test and numerically validate the applicability of the proposed CSS, the commercial CFD software FLUENT was used for all the simulations. Realizable $k$-$\varepsilon$ equation was chosen as the turbulent model. To improve the reliability and accuracy of cyclone prediction, Shih et al. proposed a Realizable $K$ model \cite{19}, whose governing equations for turbulent kinetic energy $K$ and dissipation rate are as follows:

\begin{align}
\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho ku_i)}{\partial x_i} &= \frac{\partial}{\partial x_j} \left[ \left( u + \frac{u_i}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \varepsilon_k - \rho \varepsilon \\
\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} &= \frac{\partial}{\partial x_j} \left[ \left( u + \frac{u_i}{\sigma_k} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho \varepsilon \frac{E}{K} - \rho C \frac{\varepsilon}{k} + \frac{\varepsilon}{k + \sqrt{\varepsilon}} \tag{1}
\end{align}

Where, $C_1 = \max \left( 0.43, \frac{\eta}{\eta + 5} \right) , \ C_2 = 1.9 , \ \sigma_k = 1.0, \ E = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) , \ \sigma_\varepsilon = 1.2, \ \eta = \left( 2E_\eta \cdot E_H \right)^{1/2} \frac{k}{\varepsilon} . 

Realizable $k$-$\varepsilon$ equation model meets the constraints of Reynolds stress and can simulate the real world situation better in a rotating jet. Velocity inlet and pressure outlet boundary conditions are used, assuming that all parameters such as velocity, pressure, temperature and turbulent kinetic energy are uniformly distributed at the inlet of the combustion chamber. The inlet conditions are velocity inlet, Combustor walls were assumed to be adiabatic and a standard wall function was used to describe near wall flow physics using a semi-empirical formula; the fuel used is methane; and the outlet pressure is set at -50 Pa. More sophisticated liquid-phase models will be considered in future work. Species Transport model was used to simulate combustion process of fuel/air mixture. The SIMPLE algorithm was used to calculate the pressure–velocity coupling. Moreover, a segregated solver was chosen to compute the whole physical process, with solution residuals being smaller than $10^{-5}$ for continuity and $10^{-5}$ for energy. 

Because of the irregular shape of the geometry model, unstructured tetrahedral mesh was used with refined meshing on the front of the combustion. About 7 million grids were divided, and the grid division results were shown in Fig. 4.
4. Results and discussion

4.1 Cold-flow simulation

The cold-flow experiment in the present study is aimed to quantify the velocity distribution in the combustor for both TSC and CSS. The incoming velocity of airflow is about 8.94 m/s. Fig. 5 shows the contour of velocity magnitude of the TSC and CSS in the longitudinal direction, respectively. The velocity behind the cyclone is smaller in the CSS than in the TSC because of the airflow from the CSS.

From the cold flow structure, it can be seen that when the cyclone is opened, the backflow zone formed between the slits ensures a stable ignition of the fuel here, which in turn ignites the entire chamber. In addition, it can be found that when the slit is not opened, not all of the fuel enters the reflux zone and some of it escapes from the combustion chamber or may be ignited at the exit of the chamber, but in the CSS, the air entering through the slit pushes the fuel towards the reflux zone, which ensures the fuel stays in the main combustion zone.

Based on the simulation results, it can be obtained that the flow velocity in the downstream region of the CSS is smaller than that of the TSC. This is due to the fact that the flow through the trough partly counteracts the negative pressure of the return flow, thus reducing the flow velocity in the recirculation zone. We obtain that the recirculation zone downstream of the CSS with reduced flow rate should be able to provide better flame holding performance than the CSS. As the fluid passes through the trough, the size of the recirculation zone increases. As a result, the injected fuel has a longer residence time to mix with the air flow in the recirculation zone and then burn. Due to the increased residence time, better fuel atomization can be obtained, resulting in improved combustion efficiency.

Fig. 5. Velocity contour of slotted swirl combustor on a longitudinal cross-section

Fig. 6 shows that at the same installation angle, the return velocity of the CSS is greater than that of the TSC, and the recirculation zone is longer. The central axial flow velocity of CSS is larger than that of the TSC.
In addition, in order to better explore the combustion chamber velocity distribution, five planes were intercepted along the axis in the combustion chamber hood at the positions of 240mm, 270mm, 300mm and 330mm to obtain the axial velocity distribution in four cross-sections. From Fig.7 show that the axial velocity in the center near-wall area of the CSS has a three-peak structure, and as the distance from the flame plate gets longer, the axial velocity has a three-peak structure. When the CSS has a five-peak structure in the center near the wall, when the CSS jet velocity is large enough, a smaller reflow zone will be formed around the seam, which can reduce the risk of flame release and improve the stability of the flame.

Since turbulence intensity is related to fuel atomization, pressure loss, it is important to study its distribution in the combustion chamber cavity, especially near the reflux zone. Fig.8 shows the turbulence intensity distribution behind the cyclone. In the case of the CSS, it can be seen that the
turbulence intensity at the back of each cyclone is concentrated at the outlet along the edge of the cylinder wall of the combustion chamber and is distributed symmetrically in a "spoon" shape, in layers, and decreases along the axis. This means that the fuel comes out of the injector nozzles more fully mixed and that the increased turbulence strengthens the shearing effect of the airflow.

![Turbulence Intensity](image)

Fig. 8. Schematic diagram of turbulence intensity at the head of the TSC/CSS.

4.2. Warm-flow simulation

Fig. 9 shows that the velocity of the airflow in the combustion chamber of the CSS is less than the velocity of the airflow in the combustion chamber of the TSC at the reflux zone. Meanwhile, the area of the reflux zone is significantly larger after the slit is opened. This is because the introduction of slotted reduces the negative pressure in the reflux zone, weakens the filling effect of the main stream, reduces the turbulence intensity of the tail stream, moves back the meeting point of the main stream, causes the reflux zone to become longer, thus increasing the residence time of methane in the reflux zone and strengthening the combustion; especially, a small amount of fuel forms a duty flame in the reflux zone at high temperature and low speed that is not easy to be extinguished, which enhances the flame stability and improves the combustion efficiency. As the turbulence intensity decreases, fluid losses are reduced.

![Velocity Contour](image)

Fig. 9 Velocity contour of (A) TSC and (B) CSS on a longitudinal cross-section.

The focus of the combustion simulation in the present study is to examine the overall combustion performance of the combustor with the proposed CSS. Figs. 10 illustrates the temperature distributions.
of the swirl combustors without/with the slotted swirler, respectively. The highest temperature region occurs in the shear layer where the backflow encounters and interacts strongly with the mainstream. It is observed from the experimental and numerical results that the highest temperature region occurs in the shear layer where the backflow encounters and interacts strongly with the mainstream.

Fig. 10 shows the numerical results of the temperature profiles in the combustor along the axial direction. It can be seen that the temperature in the CSS is higher than the TSC, especially around the expansion area of the combustor where shear layer occurs, with the maximum value of 2200 K. Meanwhile, it is known that the temperature of the CSS increases significantly in the middle of the combustion chamber cavity, which implies an increase in combustion efficiency, while near the wall position, the temperature of the slotted combustion is significantly lower than the CSS and it is found that the high temperature zone after seam opening is concentrated around the position x= -3m and the wall temperature is lower than before slitting. The cyclone outlet temperature was also much lower than before slitting.

Fig. 10 Temperature contour of (A)TSC and (B) CSS on a longitudinal cross-section.

5. Conclusion
A slotted swirler was proposed and numerical simulations were conducted to study the cold-flow and combustion in the combustor. Specifically, the velocity and temperature distributions at the representative axial locations were measured to examine the flow and combustion characteristics without/with the slotted swirler. It was found that the CSS is able to form a larger recirculation zone in the downstream of the swirler. The velocity in the zone is substantially reduced compared to the case of TSC. The simulation shows that the temperature distribution in the slotted swirl combustor exhibited a center-focused flame of higher temperature, compared with the TSC, the CSS exhibits certain advantages in terms of drag reduction, combustion stability and combustion efficiency, which has a certain guiding effect on the improvement of the working performance of the combustion chamber. After the cyclone opened the slot, the reflux area was shortened, and the reflux area slowed down significantly. The turbulence intensity at the front end of the slot outlet was enhanced, and the temperature at the same location of the main combustion area increased. The present study preliminarily demonstrates that the CSS holds potentials of being an effective flame holder for gas turbine combustors. In the slotted swirl combustor cyclone, part of the airflow will entrain a small amount of fuel into the reflux zone for combustion, which helps to intensify the combustion of the mainstream fuel.
Acknowledgment
This work was financially supported by the Pilot Project of Fujian Province (2018H0021)

References
[1] Al-Abdeli YM, Masri AR. Review of laboratory swirl burners and experiments for model validation. Experimental Thermal and Fluid Science. 2015;69:178-96.
[2] Jun Song Kim, Il Won Seo, Donghae Baek, et al. Recirculating flow-induced anomalous transport in meandering open-channel flows. 2020, 141
[3] P.G. Hill, D. Zhang, A review of swirling flow and combustion in spark-ignition engines, Prog. Energy Combust. Sci. 20 (1994) 373-429.
[4] De Meester R, Naud B, Merci B. Hybrid RANS/PDF calculations of Sydney swirling flames [C]//4th European Combustion Meeting. 2009.
[5] T. Owaki, A. Umemura, Premixed swirl combustion modes emerging for a burner tube with converging entrance, Proc. Combust. Inst. 31 (2007) 1067-1074.
[6] M. Chaffarpour, B. Chehroudii, Experiments on spray combustion in a gasturbine model combustor, Combust. Sci. Technol. 92 (1-3) (1993) 173-200.
[7] C.F. Edwards, R.C. Rudoff, Structure of a swirl-stabilized spray flame by imaging, laser doppler velocimetry, and phase doppler anemometry, Proc. Combust. Inst. 23 (1) (1991) 1353-1359.
[8] Y. Huang, V. Yang, Dynamics and stability of lean-premixed swirl-stabilized combustion, Prog. Energy Combust. 35 (4) (2009) 293-364.
[9] N. Syred, J. Beer, Combustion in swirling flows: a review, Combust. Flame 23 (2) (1974) 143-201
[10] A. Petchenko, V. Bychkov, L.E. Eriksson, A. Oparin, Flame propagation along the or tex axis. Combust. Theor. Model, 10 (4) (2006) 581-601.
[11] Y. Eldrainy, K.M. Saqr, H.S. Aly, T.M. Lazim, M.N. M. Jafar, Large eddy simulation and preliminary modeling of the flow downstream a variable geometry swirler for gas turbine combustor.
[12] Kanafer K, Aithal S M. Fluid-dynamic and NOx computation in swirl burners[J].International Journal of Heat and Mass Transfer, 2011, 54(23): 5030-5038.
[13] Rajpara P, Dekhatawala A, Shah R, et al. Thermal and emission characteristics of reverse air flow CAN combustor[J]. International Journal of Thermal Sciences, 2018, 128: 175-183.
[14] S. Terhaar, T.G. Reichel, C. Schrödinger, L Rukes, C.O. Paschereit, K. Oberleithner, Vortex breakdown types and global modes in swirling combustor flows with axial injection, I. Propul. Power (2014) 1-11.
[15] S. Nemoda v. Bakic. S. Oka. G. Zivkovic. N. Grnomarkovic. Experimental and numerical investigation of gaseous fuel combustion in swirl chamber, Int. I. Heat Mass Transf, 48(2005)4623-4632.
[16] Y. Huang, V. Yang, Effect of swirl on combustion dynamics in a lean-premixed swirl-stabilized combustor, Proc. Combust. Inst. 30 (2005) 1775-1782.
[17] T.C. Lieuwen, V. McDonell, D. Santavicca, etal. ,Burner development and operability issues associated with steady flowing syngas fired combustors, Combust. Sci. Technol. 180 (6) (2008) 1169-1192.
[18] S. Terhaar, T.G. Reichel, C. Schrödinger, etal. Oberleithner, Vortex breakdown types and global modes in swirling combustor flows with axial injection, I. Propul. Power (2014) 1-11.
[19] Y. Li, R. Li, D. etal. Combustion characteristics of a slotted swirl combustor: An experimental test and numerical validation. International Communications in Heat Mass Transfer, 2015;66:140-7.