Computational study of the flow field characteristics of the radial-swirl burners for a small gas turbine

Tosaporn Soontornpasatch

Department of Mechanical and Aerospace Engineering, Faculty of Engineering, King Mongkut’s University of Technology North Bangkok, Bangkok 10800, Thailand

* Corresponding Author: tosaporn.s@eng.kmutnb.ac.th

Abstract
The aim of this study is to investigate the flow field characteristics of the radial-swirl burner which is developed for a small gas turbine. To create a stable flame during the combustion process, the burner has to create a reversal flow area or a recirculation flow. Without this zone, the combustion process cannot be created. Therefore, the characteristics of the flow field in the burner is very critical. The current study applied two types of the radial-swirl burner. One has a straight-type vane and the other has a nozzle-type vane. The flow angle of both types are varied from 40°, 50° and 60°. The mass flow rate of air is varied from 1.0 to 1.8 kg/min. The commercial computational fluid dynamic program, ANSYS FLUENT is also used. The results of the simulation are discussed in this paper.

Keywords: Radial-swirl burner, Computational Fluid Dynamics, Gas turbine.

1. Introduction
The burner is one of the most important parts in the combustor of a gas turbine. The primary function of the burner is to mix air with fuel and create a continuous and stable combustion process. To create a stable flame, the burner has to create a reversal flow area or recirculation flow. In this flow area, a high-temperature combustion gas will flow back to the burner and mix with the incoming air and fuel which create a continuous combustion process. Without the recirculation flow, the flame blowout will occur. One of the most effective methods to create recirculation flow is to use a radial-swirl burner [6]. The air flows in the radius direction into the burner and changes the inward flow angle by the flow passages. This will create the toroidal flow, which has a recirculation zone inside. In the past, there were many researchers who studied the flow characteristics in the swirl burners. A much-debated question is whether the complexity of this type of flow cannot be predicted by using this kind of analytical method. Existing research recognises the critical role played by using experiment approach. Cheng and Simone [3] studied the flow field of the swirl burner of a model gas turbine by experiment approach by using PIV method. Mohammad et al. [7] recognised the relationship between discharge coefficient of various radial air swirler at each Reynolds number by experimenting on the test rig. A considerable amount of literature has been published on the experimental work conducted by David [8]. However, the development of high-speed computer technology allows the researchers to perform the study by using a numerical model instead. There were many studies that performed by using this approach. Hasem et al.
[1] used CFD to investigate the combustion process in 100kW in swirl burner which has various inlet angle conditions. Mohd et al., [2,4] focused on the fluid dynamic and the emission performance of the double radial swirlers and a combustor aerodynamic with radial swirler by simulation approach. Vondal and Hajek [5] carried CFD to predict the flow through the swirl generator and validate the result with the measured data. The results from these works indicated that the numerical model can be used as a primary tool in the study and can give a reasonable result. Therefore, the numerical approach was also used in this study.

The specific object of this study was to study the characteristics of the flow field inside the two types of the radial-swirl burner. One has a straight-type flow passages and another has a nozzle-type. The angle of the passage varied from 40°, 50° and 60°. The mass flow rate of air was varied from 1.0 to 1.8 kg/min which is in the range that the burners are expected to be used. The size of the recirculation zone, the maximum reversal velocity and the location that maximum reversal velocity occurs are the primary variables. The commercial computational fluid dynamics code, ANSYS FLUENT 17.2, was used as a tool for this study. This study emphasized on the relation between characteristics of air flow and shape parameters of the flow passage only. The injection and combustion process are included from this study. The result from this study will be used as a basis for designing a burner for a small gas turbine. This study provides new insights into gas turbine to be used as a tool for the aerospace engineering laboratory subject in the department.

2. Radial-swirl burner

There are two types of radial-swirl burner chosen in the current study. One has a straight-type and another has nozzle-type passages and the main dimension of these burners share some similarities (Figure 1). The air flows into the burner in the radial direction and the fuel is injected from the gas port at the center of the burner. This study considers only air flow characteristics, the gas port was set to be a solid wall instead. There are seven flow passages which have 7 mm-width and 13 mm-depth at the inside surface in the burner. In the case of straight-type, the width of the passage is constant. For a nozzle-type, the width of the passage is 10.2 mm at the outside surface.

3. Computational method

The flow field was calculated by using the commercial computational fluid dynamics code, ANSYS FLUENT 17.2. To make sure that the boundary condition will not allow a false solution in the computational domain, the diameter of the control volume was set at 7.5 times. In particular, the length was set at 40 times of the burner inner diameter. As can be seen from Figure 2, the air inlet was set as a mass flow inlet and the outer surfaces of the control volume were set as outflow as shown in Figure 2.
The surface that divides the burner and the main control volume was set as a wall. The unstructured tetrahedron mesh scheme was used to create meshes in the control volume. For time-saving during the computational process, the fine meshes were applied in the area around the burner outlet and the coarse meshes were used at the area that far from the burner outlet (see Figure 3). The pressure-based solver and SIMPLE algorithm were used in the calculation and the second-order upwind scheme is applied to calculate all flow variables. The program was set to stop calculation when the residual values of all flow parameters are lower than $10^{-4}$. The temperature of the air at the burner inlet was set at 300 K. The turbulence model $k-\varepsilon$ realizable was also chosen in the current study. This turbulence model has some benefits to the standard $k-\varepsilon$ model and RNG $k-\varepsilon$ model. As such, it is suitable for the swirling flow which has a recirculation zone inside. It has been also used by many researchers regarding the study of the flow of the swirl-burners [1,4,5].

![Figure 2. The control volume that is used in this study](image1)

![Figure 3. The mesh shape inside the control volume](image2)

4. Size of the recirculation zone
The recirculation zone is a vital for swirl burner since it is mixing process between an air-fuel mixture and hot gas that flows back to the burner occurs in this area, and it has a pivotal role in the heat release rate in combustion process [4]. As illustrated in Figure 4, the size of this zone can be determined by using this graph. Figure 4 also provides shows the value of the y-component velocity of flow along the y-axis which is located at the center of the burner. The coordinate has its origin at the end wall of the burner and the direction of this coordinate can be seen in Figure 2. The size of the recirculation zone is the distance between the two stagnation points (which velocity is zero). Between this area, the y-component velocity has a negative value which indicates that the direction of flow is inward to the burner.

![Figure 4. The y-component velocity profile at along Y-axis coordinate](image3)

![Figure 5. The y-component velocity profile each number of meshes](image4)
5. Mesh dependency testing
The mesh dependency testing is performed to determine the appropriate number of meshes that will be chosen in this study. The straight-type burner which has a swirl angle at $40^\circ$ was selected for the test. The mass flow rate was set at 1.8 kg/min. The number of meshes varied from 1,153,322 to 2,953,934. The y-component velocity profiles at each number of meshes are given in Figure 5. The velocity profile drastically changed at the 1.15 and 2.31 million meshes (see also Figure 5). In the case of 1.15 and 1.39 million meshes, the recirculation zone does not develop. However, the recirculation flow occurs at 2.31 million meshes. When the number of meshes increases higher than 2.31 million, the velocity profile does not change any further (see Figure 5). Therefore, the number of meshes that has approximately 2.3 to 2.4 million was employed in this current study.

6. Results and discussion
It can be seen from the simulation data in Figure 6 that the maximum reversal flow velocity for every types of burner increases when the mass flow rate increases. In case of the straight-type burner, the maximum reversal velocity continuously increases when the angle of the inlet flow increases. For the nozzle-type burner, this value also has the same trend when the inlet angle is at $40^\circ$ and $50^\circ$. However, when the inlet angle is at $60^\circ$, the maximum velocity decreases. When compare between the two types, the straight and nozzle type at the same flow inlet angle, the nozzle-type has higher maximum reversal flow velocity except at the angle $60^\circ$ which the nozzle-type has a lower value.

![Figure 6](image1.png)
**Figure 6.** The maximum reversal flow velocity at each mass flow rate

![Figure 7](image2.png)
**Figure 7.** The location which maximum reversal velocity occurs at each

The location of the maximum reversal velocity does not change with the mass flow rate (see Figure 7) Although it changes with the value of inlet flow angle but these changes are in the narrow range between 51 mm to 55 mm from the end wall of the burner. However, for the nozzle-type at $60^\circ$, the location of maximum velocity is at 43 mm which differs very much when compares with the other cases. The relation between the size of the recirculation zone and mass flow rate shows in figure 8. The change in mass flow rate does not have any significant impact on the size of the recirculation zone. There is some change in case of $50^\circ$ nozzle-type burner at a mass flow rate between 1.0 to 1.2 kg/min. However, this happens in the narrow range only. The value of the average size of the recirculation zone for each burner at a various angle can be seen in Figure 9. Regarding the straight-type burner, the size of the recirculation zone continuously increases when the inlet flow angle increases. For the nozzle-type burner, the recirculation zone size also increases when the inlet flow angle increases from $40^\circ$ to $50^\circ$. 
However, the size decreases when the flow angle changes from 50° and 60°. When compared between straight and nozzle type, at the same angle inlet, the recirculation zone size of the nozzle-type burner is larger except at the angle 60° which it has a smaller size.

![Figure 8](image1.png)  ![Figure 9](image2.png)

**Figure 8.** The relation between mass flow rate and the size of the recirculation zone

**Figure 9.** The average size of the recirculation zone at each inlet flow angle

![Figure 10](image3.png)

**Figure 10.** Comparison flow velocity vector of the 60° straight-type (left) and nozzle-type (right) passages at mass flow rate 1.8 kg/min.

Figure 10 compares the vector flow field between the straight-type and nozzle-type burner shows in figure 10. Both of them have the inlet angle at 60° and consist of the mass flow rate at 1.8 kg/min. In regard to the straight-type, was found to cause the recirculation zone at the centreline of the burner. This pattern of the flow field also occurs with the other cases but has a difference in the size of recirculation zone and magnitude of reversal flow velocity. Whilst the nozzle-type at 60° has a difference in the flow pattern (see also Figure 10). The recirculation zone has a very small size compared to other cases. Besides, the swirling flow around the recirculation zone tends to flow outward in the radial direction or in x-axis instead of flowing upward in the y-axis direction. Another important finding was that this can cause an unwanted situation. Because when this flow pattern occurred in the real combustion process, the high temperature gas flowed along the wall that burner installed. This can cause some damage to this area by the heat that might transfers to the material. Therefore, the nozzle-type at 60° may not be suitable in the application of the combustion chamber.
7. Conclusion
This paper has argued that all type of radial-swirl burner can create recirculation. The simulation confirmed that the inlet flow angle had a direct impact on the size of the recirculation zone and maximum reversal velocity. These variables increased when the inlet flow angle increased. The nozzle-type burner created a larger size of the recirculation zone and had a higher of the reversal velocity when compared with the straight-type at inlet angle $40^\circ$ to $50^\circ$. However, at inlet angle $60^\circ$ these variables of nozzle-type were lower. The change of mass flow rate affected on the maximum reversal flow velocity but it did not have any significant impact on other parameters.

Acknowledgments
This research has been financially supported by King Mongkut’s University of Technology North Bangkok, contract no. KMUTNB-63-DRIVE-10.

References
[1] Hesham B Nick S Phil B Agustin M and Richard M 2019 Int. J. Energy Technol. 1 31-43
[2] Mohd Z Mohammad J and Tholudin L 2017 J. Teknologi 79(7-3) 33-41
[3] Cheng C and Simone H 2013 Advanced Materials Research 622-623 1119-1124
[4] Mohd N Jusoff K Mohammed O and Mohd I 2011 Int. J. the Phys. Sci. 6(13) 3091-3098
[5] Jiri V and Jiri H 2011 J. Phys. Conf. Series 318 1-6
[6] Arthur L and Dilip B 2010 Gas Turbine Combustion Alternative Fuel and Emission (Florida: CRC Press) 146
[7] Mohamad I and Mohammad J 2004 J. Mekanikal 17 99-108
[8] David G L 1977 AIAA J. 15(8) 1063-78