PREDICTION OF THE FLOW INSIDE A MICRO GAS TURBINE COMBUSTOR

Johann Jeffrie Bin Muhamad Ridzwan
Mohammad Nazri Mohd Jaafar
Department of Aeronautics and Automotive
Faculty of Mechanical Engineering
Universiti Teknologi Malaysia
81310 UTM Skudai, Johor, Malaysia

ABSTRACT
The main purpose of this study is to predict the flow inside a micro gas turbine combustion chamber. Basically, the flame stabilizer or also known as swirler is an essential component due to its function to produce good flames and improves the efficiency of the micro gas turbine directly. The FLUENT software has been used to analyse the flow of flame stabilizer system. This software usage is the most efficient way to obtain the simulations results. From the results, an understanding of the flow inside the combustor will be achieved.

Keywords: Recirculation zone, corner recirculation zone, gas turbine, flame stabilizer.

1.0 INTRODUCTION
Swirling flow is a main flow produced by air swirled in gas turbine engine. Such flow is the combination of swirling and vortex breakdown. Swirling flow is widely used to stabilize the flame in combustion chamber. Its aerodynamic characteristics obtained through the merging of the swirl movement and free vortex phenomenon that collide in jet and turbulent flow. This swirl turbulent system could be divided into three groups and they are jet swirl turbulent with low swirl, high jet swirl with internal recirculation and jet turbulence in circulation zone. Each and every case exists due to the difference in density between jet flowing into the combustion chamber and jet flowing out into the atmosphere from the combustion chamber.

When air is tangentially introduced into the combustion chamber, it is forced to change its path, which contributes to the formation of swirling flow. The balance in force could be demonstrated by the movement of static pressure in the combustion chamber and can be calculated by measuring the distribution of the tangential
velocity. Low pressure in the core center of the swirling flow is still retrieving the jet flow in the combustion chamber and thus, produces the not-so-good slope of axial pressure. Meanwhile at the optimum swirl angle, the swirl finds its own direction and as a result, swirl vortex is formed.

The recirculation region in free swirl flow is shown in Figure 1. Due to assumption that the flow is axially symmetrical, thus only half of the flow characteristics are discussed. The recirculation region is in the OACB curve. The point B is known as stagnation point. The flow outside of the OACB curve is the main flow, which drives the recirculation along the AB solid curve. The ultimate shear stress could happen at points near to point A, along the boundary of recirculation. The condition of zero axial velocity is represented by hidden curve AB. Every velocity component decreases in the direction of the tip. After the stagnation point, the reverse axial velocity will disappear far into tip, the peak of velocity profile will change towards the middle line as an effect of swirling decrease.

![Figure 1: Recirculation zone in swirling flow](image)

As the level of applied swirl increase, the velocity of the flow along the centerline decreases, until a level of swirl is reached at which the flow becomes stationary. As the swirl is increased further, a small bubble of internal recirculating fluid is formed. This, the vortex breakdown phenomenon, heralds the formation of large-scale recirculation zone that helps in stabilizing the flame. It has been suggested [3, 4] that the large torroidal recirculation zone plays a major role in the flame stabilization process by acting as a store for heat and chemically active species and, since it constitutes a well-mixed region, it serves to transport heat and mass to the fresh combustible mixture of air and fuel.
The level of swirl or swirl strength can be represented in term of swirl number. Determining the swirl number is of great important in burner design since it contributes to the correct setting for the swirl blades. Past researchers have studied the effect of varying the blade angle, which in turn vary the swirl number, on combustion performance. Drake and Hubbard [5] studied the effect of swirl on completeness of combustion and discovered that there was an optimum swirl blade setting. Claypole and Syred [6] investigated the effect of swirl strength on the formation of NOx. They varied the swirl number from 0.63 to 3.04 using natural gas (mainly methane). At swirl number of 3.04, much of the NOx in the exhaust gases was recirculated into the flame front. The total emissions of NOx were reduced, however, at the expense of reduced combustion efficiency.

Other earlier researchers who studied on the effect of varying the swirl strength were mainly interested on the flow pattern and temperature profiles resulted from varying the swirl strength. They were emphasizing the effect of swirl on the generation of torroidal central recirculation zones and flame geometry rather than the effect of swirl strength on emissions formation. Mestre [7] on the other hand, compared the effect of swirling and non-swirling system on combustion. He demonstrated that the existence of swirl help improves combustion efficiency, decreases all pollutants and increases flame temperature. He also observed that during the present of swirl, a shorter blue flame was observed indicating good mixing while non-swirling system showed a longer yellow flame indicating that there is still some fuel left unvaporized.

Studies have been done to determine the major effect of the swirling flow application to the jet flow. Swirling flow can stabilize the combustion process and improve the mixing of air and fuel in the flow [8]. Air swirler is used to form a tangent flow and swirling flow inside the combustor. Swirl flow affects the flame length, size, density, and flame stabilization. Swirl number, $S_N$ is a number without dimension that represents the swirl momentum flux divided with the axial momentum flux multiplied by the nozzle radius.

For a weak swirl that is $S_N < 0.4$, it will increase the width of the jet flow and jet flow growth at the intake. The damaged to the jet stream will increased following the increase of swirl number [8]. The slope of the axial pressure is not big enough to produce an internal recirculation. Swirl also affects the rate of the convergence flow and the decreased rate of the flow velocity. Determination from different test condition
shows that there is a wall at the right angle of the flow and the orifices, expansion velocity of the jet stream, do not increase continuously with the swirl number. Jet stream will expand and the entire region at the jet is a reversed flow. Not much of the flame, in a weak swirl apply nowadays because of its instability.

![Diagram of swirling flow](image)

Figure 2: Axial velocity profile and swirl inside a strong swirl [2]

For a strong swirl $S_n > 0.6$, radial and axial pressure slope are formed at the end of the nozzle and causing a recirculation in the axial direction [3]. It is known as a toroidal center recirculation flow zone. This flow functioned to stabilize the flame where high turbulence conditions occur.

### 2.0 COMPUTATIONAL FLUID DYNAMICS

The computational fluid dynamics (CFD) is carried out by using one of the most popular CFD simulation software – Fluent 6.2 which is available in the Aeronautical Laboratory, Universiti Teknologi Malaysia. Besides Fluent 6.2, other software involves in CFD simulation is Gambit. In this study, CFD is used to obtain the flow characteristics inside the combustion chamber.

Computational Fluid Dynamics (CFD) has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics, from aerospace propulsion to weather prediction. CFD is commonly accepted as referring to the broad topic encompassing the numerical solution, by computational methods, of the governing equations which describe fluid flow, the set of the Navier-Stokes equations,
continuity and any additional conservation equations, for example energy or species concentrations. As a developing science, CFD has received extensive attention throughout the international community since the advent of the digital computer. The attraction of the subject is twofold. Firstly, the desire to be able to model physical fluid phenomena that cannot be easily simulated or measured with a physical experiment, for example weather systems or hypersonic aerospace vehicles. Secondly, the desire to be able to investigate physical fluid systems more cost effectively and more rapidly than with experimental procedures.

CFD is the art of replacing the integrals or the partial differential derivatives in these equations with discretised algebraic forms which in turn are solved to obtain numbers for the flow field values at discrete points in time or space. Physical aspects of any fluid flow are governed by three fundamental principles:

i. Mass is conserved
ii. Momentum is conserved
iii. Energy is conserved

The corresponding equations can be obtained in integral or differential forms for a domain. The equations in integral form are [9]:

\[
\int \int \int \frac{\partial \rho}{\partial t} \, dV + \int \int \rho \vec{v} \cdot \vec{n} \, dS = 0
\]  

(1)

\[
\int \int \int \frac{\partial (\rho \vec{u})}{\partial t} \, dV + \int \int (\rho \vec{i} + \rho \vec{u} \vec{V}) \cdot \vec{n} \, dS = \int \int \int \nabla \cdot \vec{q}_{\text{Body}} \, dV + F_{\text{Viscous}}
\]  

(2)

\[
\int \int \int \frac{\partial e}{\partial t} \, dV + \int \int \rho \vec{V} \cdot \vec{n} \, dS = \dot{Q} - \dot{W}
\]  

(3)

Basically there are three main procedures involved in CFD simulation – pre-processing, solver and post-processing. The content of each procedure is shown in Figure 3.
Grid generation of the solid model must be conducted before it can be transferred to Fluent for further calculation and post-processing. The grid generation mentioned will be conducted in Gambit. The solid model drawing will be imported to Gambit in ACIS format.
Figure 4: The view of swirler with combustion chamber after meshing with in GAMBIT.

Figure 5: The face of 45° swirler with wireframe view (contains 365195 elements for the whole geometry)
3.0 RESULTS AND ANALYSIS

Figures 6, 7 and 8 show the velocity vector patterns in the center plane of the axisymmetric combustor for three different degrees of swirler which represents a swirler with vane angles of 30°, 45° and 60°. As the variations in the flow patterns are observed close to the inlet, the velocity vector plots are shown up to 1/3 of the length of the combustor in the axial direction.

Figure 6 shows the flow pattern in a dump combustor with a straight vane swirler of 30° at the inlet, having a vane angle that result in inlet swirl number of 0.67. The figure reveals the formation of a corner recirculation zone immediately downstream of the inlet. The corner recirculation is caused by flow separation from the side wall of the combustor as the fluid stream enters, while, the central recirculation is the result of the adverse pressure gradient at the axis created by the swirl. The flowing stream of fresh fluid passes through the two recirculation bubbles with a high velocity till it spreads over the entire cross-section downstream of the recirculation zones.

Figure 7 shows the flow pattern in a combustor with a 45° swirler. As can be seen, the recirculation in the centre of the combustor is larger than the recirculation for the 30° swirler (Figure 6). The central recirculation bubble grows in size both in axial
and radial directions and the maximum width of the recirculation occurs closer to the inlet. It reduces the corner recirculation zone, and the size is smaller than the 30° swirler.
Figure 8 shows the flow pattern in a combustor with a 60° swirler. The recirculation in the centre of the combustor is larger than the recirculation for the 30° or 45° swirler. The central recirculation bubble grows in size both in axial and radial directions and the maximum width of the recirculation occurs closer to the inlet. It reduces the corner recirculation zone considerably and only a small portion of corner bubble exists.

Comparison were done among the three figures. Note that the air velocity for the three models before entering the swirler are the same. Nevertheless, after the air flows through the swirler it can be seen that there are variations of the velocity between the three models. The air flow velocity through the 60° swirler will increase and will be the highest among the three models followed by 45° and 30° (refer to Figures 6, 7 and 8). The highest velocity will be followed by the highest swirler vanes angle of deflection opposing the stream (higher swirl numbers). It is the main factor on why the 60° swirler can provide the strongest air circulation among the three models. It is followed by 45° swirler and 30° swirler produced the weakest circulation among them all. Recirculation magnitude will be reduced with the increased of distances from the swirler. This formed of circulation zone is known as the primary zone that gives the highest concentration of flow rate and the strongest friction between the air molecules.

[Graph showing dynamic pressure for 30° swirler]

Figure 9: A dynamic pressure for 30° swirler
As observed from the analysis that have been done earlier, it is shown that the 60° vane angle swirler was able to produce the best and the strongest circulation and recirculation flow inside the combustion chamber but on the contrary this 60° vane angle swirler has its own disadvantages. With the increased of swirl vane angle whereby it also means the increase of swirl numbers, this factor will contribute to the production of high velocity flow in the recirculation zone. And because the high velocity in that zone, there will be significant losses of pressure (refer to Figures 9, 10 and 11). From the Figure 9, 10 and 11 you can see the 60° swirler has the highest lost of pressure followed by 45° and 30° swirler.

From the theory we know that turbine gas combustion is a continuous combustion with a high controlled pressure in a combustion chamber. Therefore lost of pressure will results in an inefficient combustion process. This will disturb the smoothness of the combustion process.

Figure 10: A dynamic pressure for 45° swirler
From all the results that have been obtained, it shows that in term of good combustor characteristics, the 45° vane angle swirler is the best among the three models. It took under consideration all the aspect to make up the best selection. This 45° swirler provides the optimum combustion performances whereby it can produce sufficient recirculation inside the combustor to enhance mixing process between fuel and air, and a small loss of pressure. The 45° swirler provides the optimum operating condition inside the combustion chamber. This will result in a better burning process and will increase the efficiency of the combustion.

4.0 CONCLUSION

The variations in side wall expansion angle and the vane angle modifies the flow pattern in the combustor and then near the inlet of the combustor. These can be concluded from the overall streamline pattern and from the mean axial and tangential velocity distributions near the inlet. The length of the central recirculation zone increases with swirl number for all geometries studied. The strength of the central recirculation increases as the swirl number increases and more mass found to circulate in the recirculation bubble. At a very high inlet swirl number, an optimum side
expansion angle is very low, the recirculation is the maximum. When the side expansion angle is very low, the recirculation strength is found to be less at all swirl numbers. The increase of swirl number will increase the central recirculation whereby the central recirculation velocity will also be increased. This will result in loss of pressure in the in the combustor particularly in the central recirculation zone. These will affect the combustion performances in the combustor. An optimum operating condition should be utilized where only sufficient recirculation zone needed with a low loss of pressure.

REFERENCES