Effect of Velocity Variation at High Swirl on Axial Flow Development inside a Can Combustor

Mohamad Shaiful Ishak\textsuperscript{a,b*}, Mohammad Nazri Mohd. Jaafar\textsuperscript{b}

\textsuperscript{a}School of Manufacturing Engineering, Universiti Malaysia Perlis, P.O Box 77, Pejabat Pos Besar, 01000 Kangar, Perlis, Malaysia
\textsuperscript{b}Department of Aeronautics, Automotive & Ocean Engineering, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor Malaysia

*Corresponding author: mshaiful@unimap.edu.my

Article history
Received 2 April 2014
Received in revised form 24 June 2014
Accepted 24 August 2014

Abstract
The main purpose of this paper is to study the internal flow effect of varying the inlet velocities inside a combustor. The flow field inside the combustor is controlled by the liner shape and size, wall side holes shape, size and arrangement (primary, secondary and dilution holes), and primary air swirler configuration. Air swirler adds sufficient swirling to the inlet flow to generate central recirculation region (CRZ) which is necessary for flame stability and fuel air mixing enhancement. Therefore, designing an appropriate air swirler is a challenge to produce stable, efficient and low emission combustion with low pressure losses. Four various injection velocities from 30m/s to 60m/s with radial vanes angle of 50 degree were used in this analysis to show velocity effect on the internal flow field. The flow behavior was investigated numerically using CFD solver Ansys 14.0. This study has provided the characteristic insight into the flow pattern inside the combustion chamber. Results show that the swirling action is augmented with the increase in the injection velocity, which leads to increase in core reverse flow, thus enhancing mixing of fuel and air in the combustion chamber.

Keywords: Swirler; swirl number; combustor; turbulence; CFD simulation

© 2014 Penerbit UTM Press. All rights reserved.

\textbf{1.0 INTRODUCTION}

Swirling jets are used for the stabilisation and control of a flame to achieve a high intensity of combustion. The common method of generating swirl is by using angle vanes in the passages of air. The characteristic of the swirling jet depends on the swirler vane angle \cite{1}. Various investigation on the effects of swirl on the flame stability for swirl flame in the unconfined space have shown increasing fuel/air mixing as the degree of swirl increased \cite{2,3}. The size and strength of the central recirculation zone (CRZ) also increased with an increase in swirl intensity. At low flow rates or swirl number, a long yellow and highly luminous flames in produced indicating a poor mixing \cite{3}. However, when the swirl number is increased, the CRZ increases in size, initially in width until restricted by diameter of combustor and then begin to increase in length \cite{3}. Measurements of the flame length and stabilization distance carried out in the series of butane/air and propane/air flames with swirl have shown that both decrease markedly with increasing degree of swirl \cite{4}. Increasing of swirl number improves the flame stability due to the presence of the recirculation zone \cite{5}. Increasing the swirl number increase the angle of jets thereby increasing the total available surface area per unit volume of the jet for mixing with the surrounding fluid in free jet and enclosure jet diameter ratio \cite{6}. It has been shown that flames with low swirl have instability problems, because of the absence of the recirculation zone \cite{7}. As shown schematically in Figure 1, jet flow of high degree of swirl often results in significant lateral as well as longitudinal pressure gradients. Compared to its non-swirling counterpart, the jet is much wider, slower and with a central toroidal recirculation zone. In combustion, the presence of the recirculation zone plays important role in flame stabilization by providing a hot flow of recirculated combustion products and a reduced velocity region where flame speed and flow velocity can be matched. Swirls also act to shorten the flame length and this is advantageous for having more compact burner design \cite{8}.

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{figure1.png}
\caption{Jet flow of high degree of swirl (S > 0.6) resulting in significant lateral as well as longitudinal pressure gradients. Compared to its non-swirling counterpart, the jet is much wider, slower and there exist a central toroidal recirculation zone \cite{8}}
\end{figure}

The geometric swirl number (Sn) has been formulated by Al-Kabie \cite{9} and given as:

\begin{eqnarray}
S_n = \frac{2}{\pi} \tan^{-1} \left( \frac{2 \pi R_0}{L} \right)
\end{eqnarray}
The swirl number should, if possible, be determined from measured values of velocity and static pressure profiles. However, this is frequently not possible due to the lack of detailed experimental results. Therefore, it has been shown that the swirl number may be satisfactorily calculated from geometry of most swirl generator [10].

The main focus of this research is to investigate the effect of inlet velocity to the swirling flow inside the combustor. Flow pattern characteristics include velocity components and turbulent stresses, which are the main characteristics of the swirling flows, have been studied to understand the physical process both by numerical modeling CFD software Ansys 14.0 [11].

\[ S_w = \frac{\sin \theta}{1 + \frac{1}{2 \tan \theta} C_c A_w} \]  

(1)

where

- \( A_w \) is the swirler exit area
- \( A_\theta \) is the swirler minimum throat area
- \( C_c \) is the swirler contraction coefficient

Value for \( C_c \), the swirler contraction coefficient, \( C_\theta \), the swirler discharge coefficient and hence the swirl number was obtained using the following Equation (2) and Equation (3). The discharge coefficient in term of swirler pressure drop and air mass flow rates can be obtained as;

\[ C_\theta = \frac{\dot{m}}{A_w \sqrt{2 \rho \Delta P}} \]  

(2)

Where

- \( \dot{m} \) is the volumetric air flow rates
- \( \Delta P \) is the pressure drop

An expression for contraction coefficient in term discharge coefficient, throat area and swirler exit area can be obtained as follow;

\[ C_c = \frac{C_\theta}{1 + \left( \frac{C_\theta A_w}{A_\theta} \right)} \]  

(3)

The swirl number should, if possible, be determined from measured values of velocity and static pressure profiles. However, this is frequently not possible due to the lack of detailed experimental results. Therefore, it has been shown that the swirl number may be satisfactorily calculated from geometry of most swirl generator [10].

The appropriate choice of boundary conditions is essential and a critical part in modelling a flow accurately. Typical boundary conditions for FLUENT simulation are the inlet, the wall and the outlet boundaries. At the inlet of the computational region, the inlet boundary condition is defined as velocity inlet while the exit boundary is defined as outflow. Some assumptions for boundary conditions that were not directly measured had to be made as follows:

i. Velocity components and turbulence quantities at the inlet were constant,

ii. Turbulence at inlet is calculated from the following equations [17]:

\[ k_{\text{inlet}} = 0.002(u^2)_{\text{inlet}} \]  

(4)

\[ \varepsilon_{\text{inlet}} = \frac{k_{\text{inlet}}^{1.5}}{0.3D} \]  

(5)

where \( u \) is axial inlet flow velocity and \( D \) is hydraulic diameter.
In order to achieve better mixing between fuel and air in liquid fuel burner, turbulence flow must be generated to promote mixing. Turbulence energy is created from the pressure energy dissipated downstream of the flame stabilizer. In the radial swirler, turbulence can be generated by increasing the aerodynamic blockage or by increasing the pressure drop across the swirler. The discharge coefficient for radial swirler were obtained by passing an air flow through the radial swirler and flame tube while monitoring the static pressure loss upstream of the radial swirler relative to the atmospheric pressure. The results were plotted as a function of Reynolds number and presented in Figure 5. From Figure 5, it can be seen generally that all discharge coefficients were approximately constant with variation in Reynolds number. Thus the value of discharge coefficient may be concluded to be independent of Reynolds number. In the case of 50° vane angle swirler, the $C_D$ is around 0.65.
4.0 RESULTS AND DISCUSSION

All the axial flow characteristics are presented in Figures 6 to 9. To validate the CFD model, experiments were conducted to measure the axial flow velocity at the axis of the chamber for 50 m/s inlet velocity. Figure 6 shows the results of CFD analysis compared to experimental results as part of the validation of the CFD model.

A past researcher shows that, 50° swirl gives the best result in terms of swirl zone and size of recirculation areas [18, 19, 20]. From the results, further work was conducted to see the effects of variations in axial velocity on the centre core flow in the combustion chamber. The flow was varied from 30 m/s to 60 m/s (equivalent to Reynolds Number from 0.6 to 1.2 x 10^6). The results showing the variations in the transient core flow at 25 s after the air was injected into the chamber are depicted in Figures 7. The figures show the reverse axial velocities in the central axial section of the chamber. The white portion of the figures means the axial flow is positive.

The results show similar flow pattern for all injection velocities, or at all Reynolds numbers, but as the injection velocities increased the reverse flow velocities in the core increases from 15 m/s for the 30 m/s injection (Figure 8a) to 30 m/s for 60 m/s injection (Figure 8d). But the core size does not change. Taking the cross section at L/D= 0.1, the core size is r/R=0.05 for all injection velocities. The differences are mainly due to higher injection velocities, where the core reverse flow velocities also increases. These can be seen in the reverse axial velocities at different cross sections as shown in Figures 8a to 8d. Another significant result is that the core axial flow velocities at cross sections nearer to the injection point increases with injection speeds. But after the distance of L/D=0.4 the central core axial velocity does not change significantly. This can be seen clearly in Figure 7, and from Figures 8a to 8b.

The axial flow velocities along the central chamber axis were then plotted until 200 mm (L/D=0.71) from the injection point. In Figure 9, shows that near the injection point, the reverse flow velocities are directly related to the injection velocities. The higher the injection velocities, the faster are the reverse flow velocities. This is accompanied by the reduction in size of the core reverse flow volume. This is understandable since the swirl angle is constant (so is the swirl number), such that the higher injection velocities would produce higher reverse core flow, thus enhancing mixing of fuel in the chamber.
CONCLUSION

This paper presented results of CFD study on the axial flow in a circular chamber with different inlet velocity vary from 30 m/s to 60 m/s (equivalent to Reynolds Number from 0.6 to 1.2 x 10^6) of 50 degree radial vane angle swirler. The initial results were compared with actual measurements to validate the CFD model. The validated model was then used to study further variations of the flow in the chamber. The study indicated that at all injection velocity, similar flow pattern was found, but as the injection velocities increased the reverse flow velocities in the core increase but the core size does not change. The differences are mainly due to higher injection velocities, where the core reverse flow velocities also increases. Another significant result is that the core axial flow velocities at cross sections nearer to the injection point increases with injection speeds. But after the distance of L/D=0.4 the central core axial velocity do not change significantly. From the results of various injection velocities, it shows that the differences are mainly due to higher injection velocities, where the core reverse flow velocities also increases.

Acknowledgement

The authors would like to thank the Ministry of Higher Education of Malaysia & Research Management Center (project number: 01G60) for awarding a research grant to undertake this project. The authors would also like to thank the Faculty of Mechanical Engineering, Universiti Teknologi Malaysia for providing the research facilities and space to undertake this work.

References