World Academy of Science, Engineering and Technology
International Journal of Aerospace and Mechanical Engineering
Vol:5, No:2, 2011

Cold Flow Investigation of Primary Zone Characteristics in Combustor Utilizing Axial Air Swirler

Yehia A. Eldrainy, Mohammad Nazri Mohd. Jaafar, Tholudin Mat Lazim

***Abstract*—**This paper presents a cold flow simulation study of a small gas turbine combustor performed using laboratory scale test rig. The main objective of this investigation is to obtain physical insight of the main vortex, responsible for the efficient mixing of fuel and air. Such models are necessary for predictions and optimization of real gas turbine combustors. Air swirler can control the combustor performance by assisting in the fuel-air mixing process and by producing recirculation region which can act as flame holders and influences residence time. Thus, proper selection of a swirler is needed to enhance combustor performance and to reduce NOx emissions. Three different axial air swirlers were used based on their vane angles i.e., 30°, 45°, and 60°. Three-dimensional, viscous, turbulent, isothermal flow characteristics of the combustor model operating at room temperature were simulated via Reynolds-Averaged Navier-Stokes (RANS) code. The model geometry has been created using solid model, and the meshing has been done using GAMBIT preprocessing package. Finally, the solution and analysis were carried out in a FLUENT solver. This serves to demonstrate the capability of the code for design and analysis of real combustor. The effects of swirlers and mass flow rate were examined. Details of the complex flow structure such as vortices and recirculation zones were obtained by the simulation model. The computational model predicts a major recirculation zone in the central region immediately downstream of the fuel nozzle and a second recirculation zone in the upstream corner of the combustion chamber. It is also shown that swirler angles changes have significant effects on the combustor flowfield as well as pressure losses.

***Keywords*—**cold flow; numerical simulation; combustor; turbulence; axial swirler

I. INTRODUCTION

PRODUCING strong swirl in aero and industrial gas turbines combustor, is essential to the combustion air in order to assist in flame stabilization of the high intensity combustion and low emission combustion. The degree of the swirl applied to the flow affect the flame size, shape and stability as well as the combustion intensity. Swirl flows are characterized by swirl number Sn which is defined as the ratio between axial flux of the angular momentum to the axial momentum [1], [2], [3].

Y.A.Eldrainy is currently doing his PhD in Universiti Teknologi Malaysia. (e-mail: yeldrainy@yahoo.com).

M.N. Mohd Jaafar is with the Department of Aeronautical Engineering, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia. (PHONE: 607-5534755, Fax: 607-5566159; e-mail: nazri@fkm.utm.my).

T. Mat Lazim is also with the Department of Aeronautical Engineering, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia. e-mail: tholudin@fkm.utm.my).

$$S_N = \frac{\int_0^R (wr)\rho u 2\pi r dr}{R\int_0^R u\rho u 2\pi r dr} \tag{1}$$

For an axial swirler having a uniform swirl angle, the swirl number is related to the swirl angle θ, inner radius r□ and outer radius r□ as given by [3]:

$$S_N = \frac{2}{3}\left[\frac{1-(r_i/r_o)^3}{1-(r_i/r_o)^2}\right]\tan\theta \tag{2}$$

For high degree of swirl, SN of higher than 0.6, strong radial and axial pressure gradients are generated near the swirler exit plane. The relation between pressure gradient $\frac{\partial p}{\partial r}$ and tangential velocity w can be written in the following form [4]:

$$\frac{\partial p}{\partial r} = \frac{\rho w^2}{r} \tag{3}$$

As the result of these pressure gradients, an axial recirculation zone are formed in the form of a central trapezoidal recirculation zone, CTRZ, which is not observed at low swirl number (SN <.4). CTRZ plays a dominant role in flame stabilization by providing hot recirculated gases to the combustion zone and reduced velocity matching the low flame velocity. The mean axial Mach number leaving the primary zone must be of the order 0.02 to 0.05 [5]. Most swirlers are designed to produce swirl number between 0.4 and 0.6. This is to ensure adequate recirculation for anchoring the flame in order to secure stability without excessive pressure loss [1].

In this research, three different axial air swirlers with different vane angles of 30°, 45° and 60°, corresponding to swirl number 0.43, 0.74, and 1.28 respectively, were analyzed numerically at different boundary conditions to show the effect of the swirler configuration on the turbulence production, recirculation zone and also pressure loss.

Due the complexity of the flow within the gas turbine combustor, CFD is used as a regular tool to enable better understanding of the aerodynamic and process associated with combustion inside the gas turbine combustors. Nowadays CFD is employed in design and development of more efficient and low emission combustors. Numerical simulation of the flow field within the gas turbine combustor mainly utilizes Reynolds Average Navier Stokes (RANS) methods accompanied with closure turbulent models. In this paper the turbulent Reynolds stresses and other turbulent flow quantities are predicted with the standard k-□ model in calculating the isothermal flow in a model gas turbine combustor because of its robustness, economy, and reasonable accuracy for a wide range of turbulent flows [6]. The numerical approach is based on the finite volume technique with structured and unstructured grid arrangement.

II. GOVERNING EQUATION

In this study, incompressible steady state isothermal flow is assumed. Based on these assumptions, the Reynolds-averaged Navier-Stokes equations are:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{4}$$

$$\overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \tag{5}$$

where $\overline{u}$ is the mean velocity component in x_i direction, ρ is the density, p is the mean pressure, and ν is the kinematic viscosity. the overbar denotes time averaging, and the prime denotes fluctuating component.

III. TURBULENCE MODELS

The main feature of the turbulent flow is the random fluctuations of fluid velocity over space and time. These random fluctuations occur over very small distances in space and time compared to the overall domain. In order for a turbulent flow to be modeled accurately, the computational mesh spacing must be smaller than the smallest element of turbulent motion (eddy). However, it should covers the entire control volume. The computations will have to be unsteady using a time-step smaller than that of the fastest eddy. This leads to an impossibly large number of grid nodes. The calculations required are beyond the capabilities of current computing technology.

Since engineers are not usually interested in the fluctuating components of flow a statistical approach is taken. This is achieved by averaging over a time scale that is large compared with the turbulent motion. The results from these equations describe the flow in terms of the mean velocities and pressures. However, these equations contain unknowns representing the Reynolds stress and transport of mean momentum. These additional equations (turbulence modeling) are needed to be solved for all the parameters. The number of these additional equations increases with the complexity of the model. Basically, turbulent models incorporated with the governing equations are used to determine the Reynolds Stresses.

The k-ε model focuses on the mechanisms that affect the turbulent kinetic energy. The standard k-ε model in FLUENT falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Jones and Launder [7]. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism. As the strengths and weaknesses of the standard k-e model have become known, improvements have been made to the model to improve its performance.

Generally k-ε models solve two additional transport equations - one for the turbulent kinetic energy and the other is for the turbulent dissipation rate. These models uses Boussinesq hypothesis which proposes that the transport of momentum by turbulence is a diffusive process and thus the Reynolds stresses ($-\overline{u_i' u_j'}$) can be modelled using turbulent (eddy) viscosity which is analogous to molecular viscosity.

$$-\overline{u_i' u_j'} = 2\upsilon_t\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) - \frac{2}{3}k\,\delta_{ij} \tag{6}$$

IV. COMPUTATIONAL METHOD

The swirling flow enters the domain from axial swirler (r□ = 9.5 mm, r□=23.9) with different blade angles of 30°, 45° and 60° from the direction of inlet air flow. The flow expands into the combustion volume under an angle of 90 degrees i.e. the diffuser is sudden expansion type. The downstream part (combustion chamber) of the flow domain is a cylinder with a radius of 72.5 mm and a length of 400 mm.

The model geometry had been created using solid model, and grids (Figure 1) of all the computational regions considered in this study were set up with preprocessing package GAMBIT version 2.2. Grid generation and density play an important role in the prediction of solution accuracy [8]. The grid was non-uniform, with high density in zones of great interest i.e. zones of high gradients and low density in zones of less interest, so that minimal computational effort was required whilst gaining sufficient accuracy.

In order to obtain a grid-independent solution, the grid should be refined until the solution no longer varies with additional grid refinement. The solution of the CFD models has been achieved using FLUENT 6.2 [9]. At the inlet of the computational region, the boundary condition is mass flow inlet and at the exit is pressure outlet (gauge pressure at model exit is 0.0). These boundary conditions can be expressed as follows: inlet, outlet and wall boundary conditions. Some assumptions about boundary conditions that were not directly measured had to be made. These were:

• Velocity components and turbulence quantities at the inlet were constant

• Turbulence at inlet is calculated from the following equations [10]:

$$k_{inlet} = 0.002\,(u^2)_{inlet} \tag{7}$$

$$\varepsilon = \frac{k_{inlet}^{1.5}}{0.3\,D} \tag{8}$$

Where u axial inlet flow velocity and *D* is hydraulic diameter.

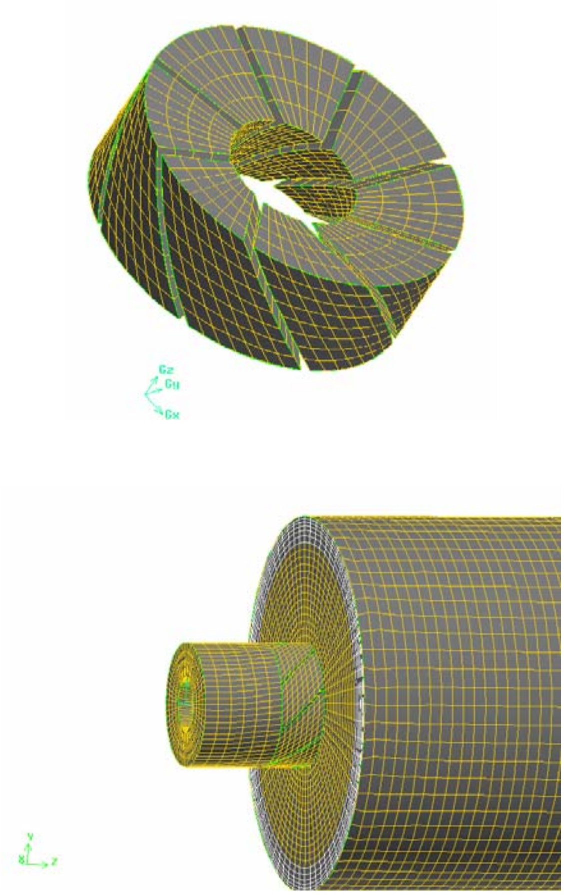


Fig. 1 Computational grid of combustor

V. Results And Discussion

In this study, computations have been made to predict the capabilities of the code developed to simulate the flow field in non-premixed sudden expansion (dump) combustor. To study the effect of the inlet velocity, different swirler angles and number of the swirler blade on the flow field of the combustor on cold flow conditions, it is assumed that swirler root and tip diameter are the same, the swirler thickness are the same for all cases under study.

A. Mesh Refinement Study

In order to obtain a grid-independent solution, the grid should be refined until the solution no longer varies with additional grid. Grid refinement and density play an important role in the prediction of solution accuracy. The effect of Mesh refinement on the solution was investigated for mass flow for 0.16 kg/s and the meshes used are 175269, 284812 and 59243 respectively. Figure 2 shows the solution of the axial velocity along the centerline of the combustor. It can be seen that the solutions converge as the mesh refined. The difference in solutions between all mesh is very small all along the pipe centerline. It can be concluded these meshes provide a sufficiently converged solutions.

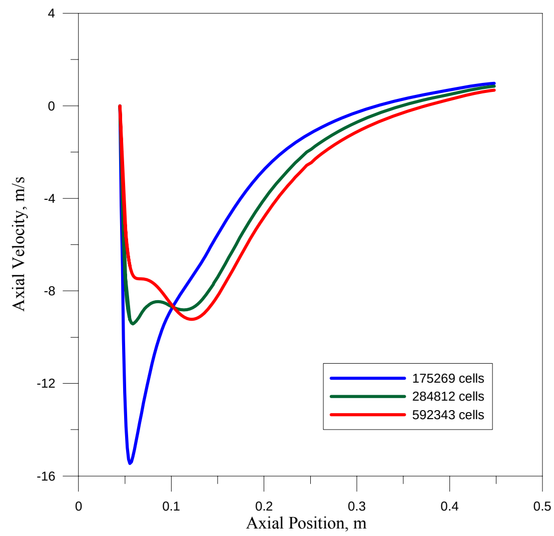


Fig. 2 Axial velocity distributions at centerline for different meshes number

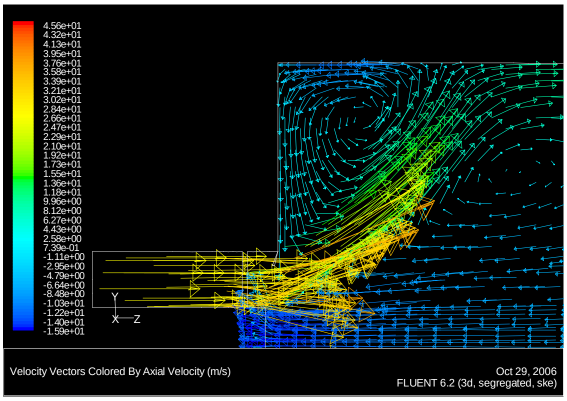
B. Recirculation zone

One of the important features of the incoming flow into a combustor is the creation of the recirculation zones. The resulting flow field domain possesses a central recirculation zone around the centerline of combustor and downstream of the air swirler, in addition to a corner recirculation zone near the upper corner provoked by the sudden enlargement of the cross sectional area of the combustor. The two zones can be identified clearly in Figures 3 (a) and (b). Figure 4 shows that the axial velocity decays in the downstream direction. After stagnation point the reverse axial velocities disappear and the peak of the axial velocity profile shifts toward the centerline as the effect of the swirl diminishes. The recirculation zone in swirling flow is closely related to the inlet flow Reynolds number ReD, swirl number SN, and affected by the upstream and downstream boundary conditions. So the effects of these factors on the swirling flow field will be investigated in the following sections.

C. Inlet Flow Rate (ReD)

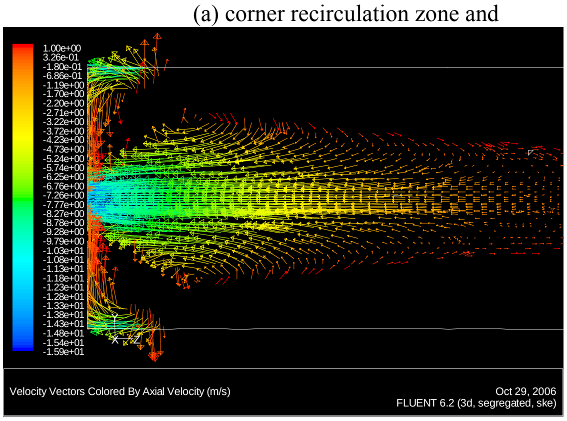
Flow rate ranged from 0.025 kg/s to 0.105 kg/s (ReD ranged from 26672 to 106690). Reynolds number can be calculated from mass flow rate as:

$$\mathrm{Re}_D = \frac{\rho U D}{\mu} = \frac{4\dot{m}}{\pi \mu D_h} \tag{9}$$



(a)

(a) corner recirculation zone and



(b)

(b) central recirculation zone (CRZ)

Fig. 3 Velocity vector within the combustor

Swirler configuration with 8 vanes, r□ =23.9, r□ = 9.5 and 45° vane angle, is used to demonstrate the effect of mass flow rate (Reynolds number) on the flow filed within the combustor. It can be observed from the negative velocity contours shown in Figure 4 that there are vortices in the top and bottom corners enclosed by the jet coming out from the swirler exit. These corner vortices are shed from the dump plane. In addition a larger vortex (CRZ) is located around the combustor center line. Both vortices reversed flow into the swirler exit direction.

Figures 5, 6 and 7 show the center-line mean axial velocity, mean tangential and turbulence intensity profile. Since the inlet mass flow rate increase, turbulence intensity, axial velocity value and tangential velocity value increase within the central recirculation zone, as a result, the turbulence strength and recirculated gas quantity increase.

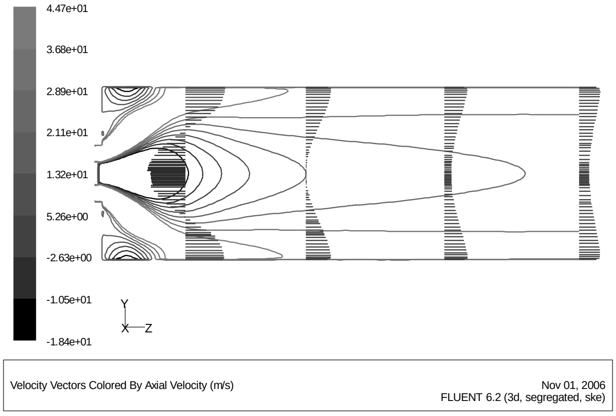


Fig. 4 Profile of axial velocity within the combustor

Consequently, it can be concluded that the flame stability and mixing quality can be improved by increasing inlet Reynolds number. However on the other side, the jet velocity also increases with inlet mass flow consequently this may lead to flame blow off at excessive rate of inlet mass flow. Besides, the combustor pressure loss increases quadratically with increase of ReD as shown in Figure 8.

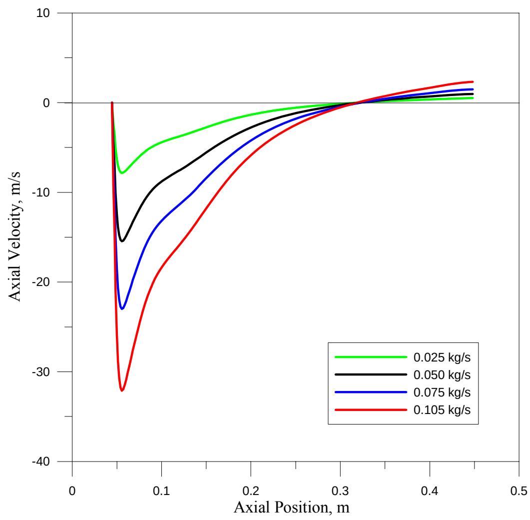


Fig. 5 Mean axial velocity distributions along the combustor centerline

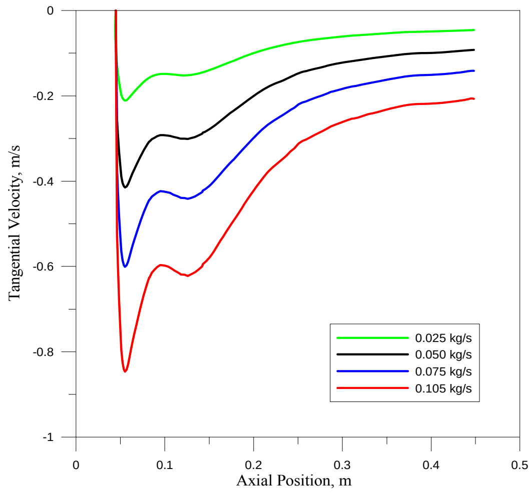


Fig. 6 Mean tangential velocity distributions along the combustor centerline

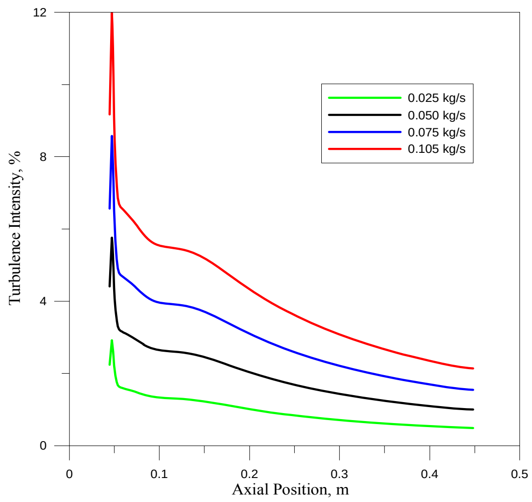


Fig. 7 Turbulence intensity distributions along the combustor centerline

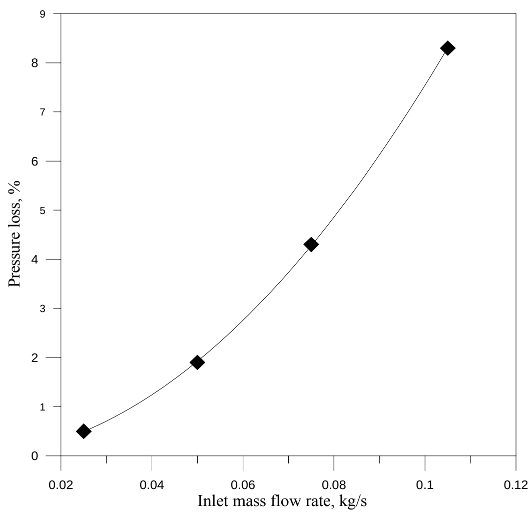


Fig. 8 Pressure loss (%) versus inlet mass flow rate

D. Swirler Vane Number

Figures 9 and 10 illustrate the effect of varying the number of swirler vanes on the CRZ for swirler configuration of 45° blade angle with inlet mass flow 0.05 kg/s.

Three swirlers with 8, 10 and 12 vanes were examined. Even though the overall shape of outer border of the central recirculation zone does not changed much, the 10 and 12-vanes swirlers produce higher turbulence intensity in the recirculation zone. The results obtained for both 10 and 12 vanes swirlers are very close.

E. Swirler Vane Angle

It is shown in Figure 11 that the location of stagnation point was moved towards downstream with the increase in the swirler vane angle. Corresponding to this downwards movement of the stagnation point the strong rate-of-strain region on the centerline, which is formed by the shear layer between reversed flow and the recovered downwards flow, also moved towards downstream. This indicates that the flame, which will be mainly anchored around the high strain rate region, will be pushed downstream with the increment of vane angle and more susceptible to blow off. In addition, when the vane angle increases, the maximum point of turbulence intensity moves upstream as the result it affects the air fuel mixing processes.

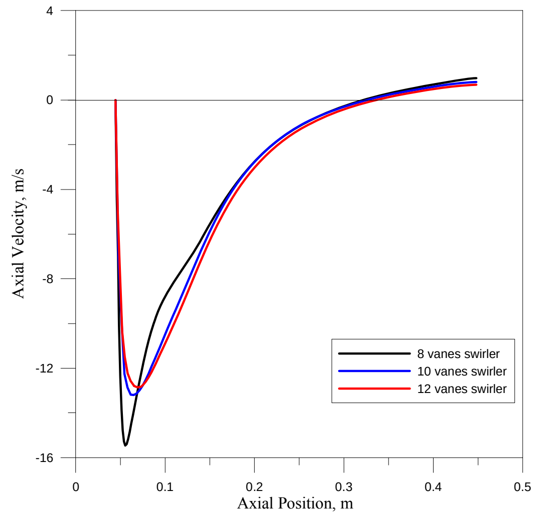


Fig. 9 Mean axial velocity distributions along the centerline for different swirler vanes number

This is due to highest turbulence region move towards the injector opening hole as the angle increases (Figure 12). Although increasing of the swirler vane angle enhances the recirculation zone, as well as it significantly increases the pressure losses (Figure 13).

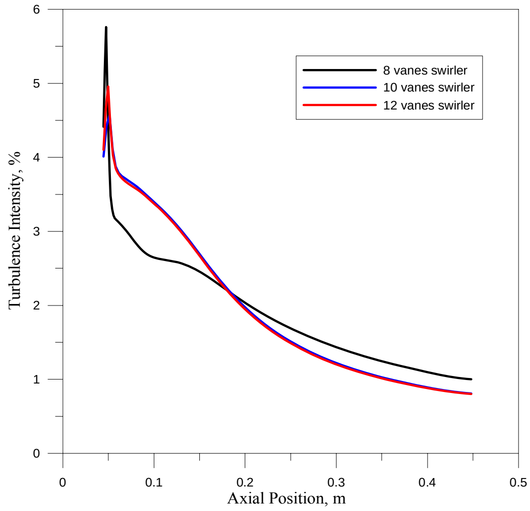


Fig. 10 Turbulence intensity distributions along the centerline for different swirler vanes number

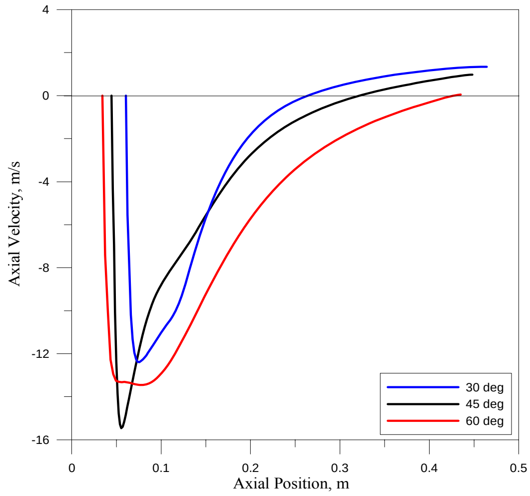


Fig. 11 Mean axial velocity distributions along the centerline for different swirler vanes angle

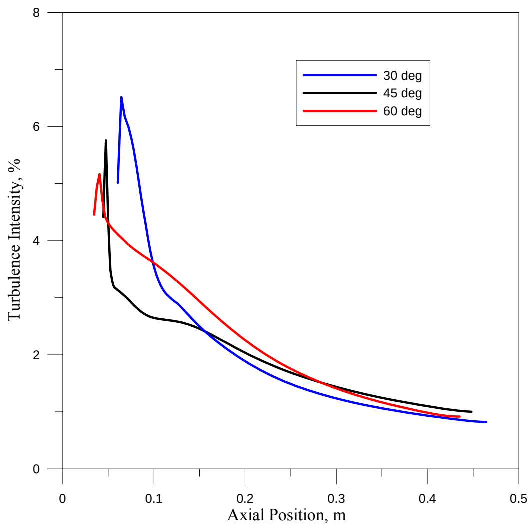


Fig. 12 Turbulence intensity distributions along the centerline for different swirler vanes angle

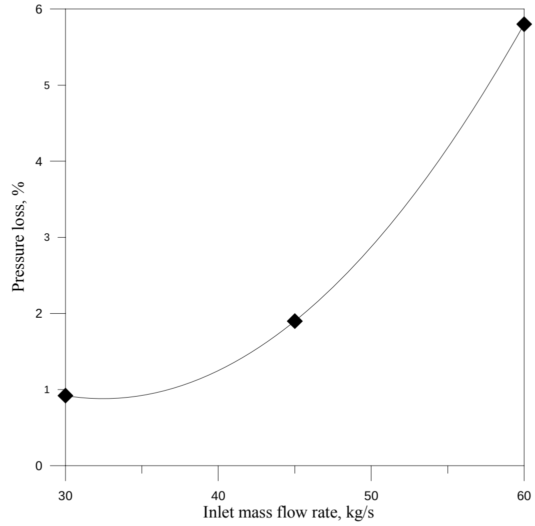


Fig. 13 Pressure loss % versus swirler vane angle

VI. Conclusion

Air swirler is one of the most effective ways to induce flow recirculation inside the primary zone. This type of recirculation provides better mixing, In addition swirling flow is used to control the stability and intensity of the combustion and the size and shape of the flame region which is dependent on the size and shape of the recirculation zone.

The factors governing the size of the recirculation zone are mass flow rate (engine load), swirler vanes angle (SN) and number. On the other hand, the swirl stabilized diffusion combustion results in higher NOx due to its higher flame temperature. Therefore, it can be concluded that:

- Systematic study of swirling flow is an efficient and effective approach for investigating complex flows and correlating combustion dynamics to fluids dynamics. The knowledge of velocity flow field and large vortical flow structures obtained from isothermal flow is helpful in understanding phenomena in combustion cases [11] [12].
- Swirler configurations are shown to be effective for control of flow field, and hence it can control emissions and combustion dynamics but each has certain limitations.

Acknowledgment

The authors would like to thank the Ministry of Science, Technology and Innovation (project number: 03-01-06-KHAS01) for awarding a research grant to undertake this project. The authors would also like to thank Faculty of Mechanical Engineering, Universiti Teknologi Malaysia for providing the research facilities to undertake this work.

References

[1] Lefebvre A. H. (1983), "Gas turbine combustion" McGraw-Hill series
[2] Correa, S.M., “A Review of NOX Formation under Gas-Turbine Combustion Conditions” Combustion Science and Technology, vol. 87, pp.329-362, 1993
[3] M. Mellor, “Design of Modern gas Turbine Combustors”, Academic Press, 1990.

[4] Philip P. Walsh and Paul Fletcher (2005), "Gas turbine Performance" Blackwell Science
[5] Y. Wang, V. Yang, and R.A. Yetter "Numerical Study on Swirling Flow in an Cylindrical Chamber" 42nd AIAA Aerospace Sciences Meeting, 2004, Reno, Nevada
[6] Karvinen and H. Ahlstedt "Comparison of Turbulence Models in Case of Jet in Cross Flow Using Commercial CFD Code" Engineering Turbulence Modeling and Experiments 6, 2005, Elsevier Ltd.
[7] Launder, B.N. and Spalding, D. B. Lectures in Mathematical Models of Turbulence. Academic Press, London, England, 1972
[8] Lin, S. and Griffiths, A. J. "CFD simulation of cooling air flows within shrouded finned heater/cooler systems used on barrel extruders" Proceedings of the Joint ASME/JSME Fluid Engineering Annual Conference, USA, 1995.
[9] FLUENT 6.2 User's Guide, Fluent Inc. 2005
[10] Versteeg, H.K. and Malalasekera, W., "An Introduction to Computational Fluid Dynamics, the Finite Volume Method", Longman Group Ltd., 1995.
[11] Eldrainy YA, Aly HS, Saqr KM, Mohd Jaafar M.N. Augmentation of Turbulence and Mixing in Gas Turbine Combustors by Introducing Unsteady Effects. International Conference on Mechanical, Industrial, and Manufacturing Technologies (MIMT 2010) 2010.
[12] Yehia A. Eldrainy, Hossam S. Aly, Khalid M. Saqr, and Mohammad Nazri Mohd Jaafar, "A Multiple Inlet Swirler for Gas Turbine Combustors", International Journal of Mechanical Systems Science and Engineering 2:2 2010.