

Full Length Research Paper

Combustor aerodynamic using radial swirler

M. N. Mohd Jaafar¹, K. Jusoff^{2*}, Mohamed Seroleh Osman¹ and Mohd Shaiful Ashrul Ishak³

¹Department of Aeronautical Engineering, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia.

²Faculty of Forestry, Universiti Putra Malaysia, 43400 Serdang, Selangor, Malaysia.

³School of Manufacturing Engineering, Universiti Malaysia Perlis, P. O. Box 77, Pejabat Pos Besar, 01000 Kangar, Perlis, Malaysia.

Accepted 17 May, 2011

A study has been conducted to investigate the flow pattern in a gas turbine combustion chamber by simulation and experimental approaches. Flow pattern inside a combustor is important to self sustain the flame, increase mixing of air and fuel, and increase combustion intensity. Aerodynamically curved vanes allow the incoming axial flow to turn gradually. This inhibits flow separation on the suction side of the vane. Thus, more complete turning and higher swirl and radial-velocity components can be generated at the swirler exit with the added advantage of lower pressure loss. The swirl number varied from 0.49, 1.29 and 2.29 for flat vanes and only 1.57 for curved vane. The highest swirl number of 2.29 for flat vane and 1.57 for curve vane are capable of creating a clear reversal mass flow rate zone and higher swirl strength reduces the corner recirculation zone size and hence reduces the negative impact on the combustion process and the homogeneity of the wall temperature as well. Further investigation can be done for higher swirl number for both types of swirler.

Key words: Radial swirlers, recirculation zone, swirl number, flat vane, curved vane.

INTRODUCTION

In gas turbine combustor, it is necessary to maintain a stationary flame within a high velocity air stream. The flame propagates through the unburned mixture at a certain speed, accordingly the mixture have to be maintained within the similar limits for flame stabilization that is, for flame travelling at speed higher than mixture speed, the flame blowout occurs. In order to overcome this difficulty, a region of recirculation downstream the main burner or a bluff body should be established. The region creates areas of a locally low stream velocity that is equal to the flame speed which is used to hold the flame, hence to sustain the combustion.

Air swirlers are used as a flame holder by imparting swirl to the incoming air. The presence of swirl results in setting up of radial and axial pressure gradients, which in turn influence the flow fields. In the case of strong swirl, the adverse axial pressure gradient is sufficiently large to generate reverse flow along the axis and generating an

internal circulation zone (Lefebvre, 1983; Mellor, 1990; Mattingly, 1996; Ishak and Mohd Jaafar, 2009; Ganesan, 2003; Mohd Jaafar et al., 2009; Eldrainy et al., 2010). In addition, swirling flows are used to improve and control the mixing process between fuel and air streams and enhance heat release rate (Beer and Chigier, 1972; Gupta et al., 1984).

In this research, aerodynamically flat vane and curved vane radial swirlers with various vane angles were investigated. Flow pattern characteristics including velocity components and turbulent stresses, which are the main characteristics of the swirling flows, have been studied to understand the physical process by both experimental investigations and by numerical modelling using Fluent 6.3 software (Fluent 6.3 Tutorial Guide, 2006).

The main focus of this research is to investigate the dynamics of the swirling flow in gas turbine combustor using radial flow air swirlers of different vane angles and vane types. The vane angles variation will provide different swirl number which is the main parameter of the investigation.

*Corresponding author. E-mail: kjusoff@yahoo.com

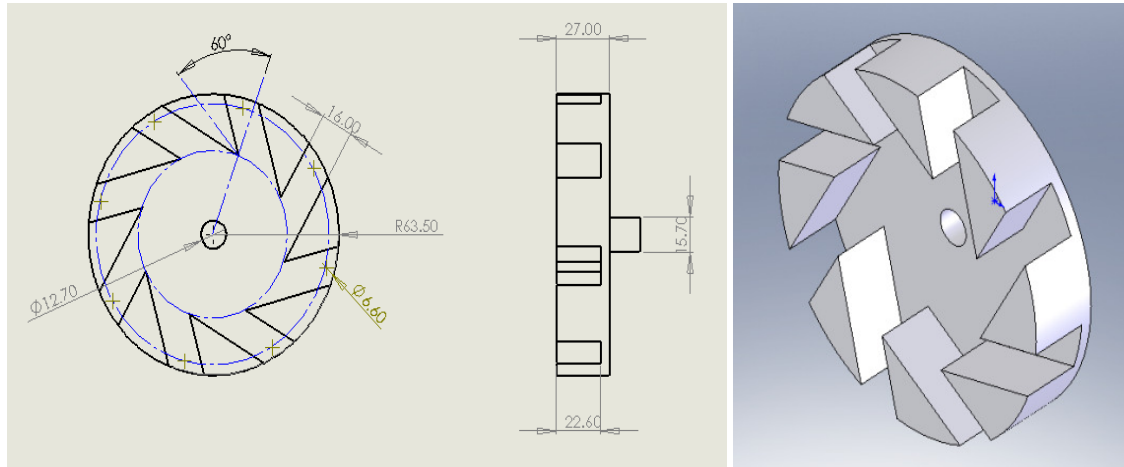


Figure 1. Geometrical details of the model swirler (all dimensions in mm).

Geometry

The curved vane swirlers produce a larger recirculation zone and a higher recirculated mass with lower pressure loss. Furthermore, strength of shear layer zones and turbulence intensities are augmented by changing flat vane to curved vane (Lefebvre, 1983). Therefore, to prove this statement, both swirlers, curved and flat vanes, are utilized in this research to show their effect on the flow pattern within the combustor.

The geometrical details of the swirlers, which were carefully machined from mild steel, are shown in Figure 1. The swirler has eight vanes, equally spaced around the circumference of the plate. The axial depth of the vane passage is 16 mm. Three different vane angles of 40°, 50° and 60° for flat vane swirlers and one curved vane swirler of 60° were used in this study.

Swirl number

The level of swirl or swirl strength can be represented in term of swirl number. Determining the swirl number is of great importance in burner design since it contributes to the correct setting for the swirl blades. Swirl flows are characterized by swirl number S which is defined as the ratio between axial flux of the angular momentum to the axial momentum. The swirl number can be written mathematically as (Beer and Chigier, 1972):

$$S = \frac{G_{\theta}}{G_x r_o} \quad (1)$$

where G_{θ} is the axial flux of the tangential momentum, G_x is the axial flux of the axial momentum, and r_o is the outer radius of the annulus. These two parameters, G_{θ} and G_x can be expressed as:

$$G_{\theta} = 2\pi \int_0^{r_o} \rho U_x U_{\theta} r^2 dr \quad (2)$$

$$G_x = 2\pi \int_0^{r_o} \rho U_x^2 r dr + 2\pi \int_0^{r_o} p r dr \quad (3)$$

where U_x , U_{θ} and p are the axial velocity, tangential velocity, and density respectively.

Al-Kabie (1989) has proposed the following non-dimensional equation to characterize the amount of rotation imparted to the axial flow:

$$C_c = \frac{C_d}{1 + \frac{C_d A_{th}}{A_{cc}}} \quad (4)$$

$$S_N = \frac{\sin \theta}{1 + \frac{1}{\tan \theta} C_c A_{th}} \quad (5)$$

where C_c is the contraction coefficient, C_d is the discharge coefficient, A_{th} is the air throat area and A_{cc} is the combustion chamber area. The technical data of the four swirlers used in this study are listed in Table 1.

METHODS

Numerical simulation

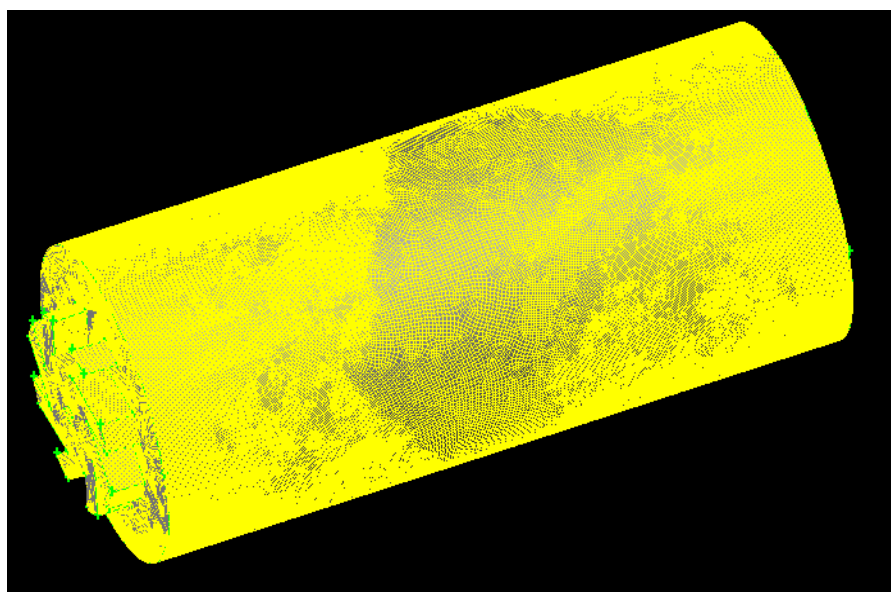
The isothermal flow inside the combustor was simulated using FLUENT 6.3 CFD codes. For the prediction of the isothermal flow fields within the combustion chambers, FLUENT solves the equations for conservation of mass and momentum in their time average form.

Geometry specification and grid set-up

The physical domain of radial swirlers was decomposed to several volumes to facilitate meshing with cooper hexahedral structured

Table 1. Technical data of the swirlers.

Types of swirler	Flat vane		Curved vane	
Vane angles (degrees)	40	50	60	60
Vane depth, L (m)		0.016		
No. of Vanes, n		8		
Outlet diameter of swirlers, d (m)		0.04		
Minimum passage width, H (m)	0.0165	0.010	0.0075	0.011
Discharge coefficient, Cd	0.69	0.65	0.64	0.64
Contraction coefficient, Cc	0.6528	0.634	0.6283	0.623
Swirl number, Sn	0.49	1.29	2.29	1.57

**Figure 2.** Combustor model mesh.

grid. The geometry meshing was done to have a variable density distribution by mean of small mesh size which was incorporated in high gradient zone and bigger size in low gradient zone. The combustor model meshing for the present work is shown in Figure 2.

Turbulence modelling

As mentioned above, FLUENT solves the equations for conservation of mass and momentum in their time averaged form for the prediction of isothermal flow fields. For the process of Reynolds decomposition and time averaging results in unknown correlation of the fluctuation velocity components, a turbulence model is required for equations closure purposes. In the present simulation, k-epsilon turbulence model was used. Turbulence is represented by the standard k-epsilon model, which provides an optimal choice and economy for many turbulent flows (Kim, 1989).

Menzies (2005) had studied the behavior of five k-epsilon variants in modelling the isothermal flow inside a gas turbine combustor and compared the results with the experimental data of Da Palma (1988) for the velocity and turbulence fields. The studied models were the standard, the RNG, the realizable, the Durbin

modified, and the nonlinear k-epsilon models. The results showed that the standard and the Durbin k-epsilon models gave the best agreement with the experimental data. This supported the finding of Jiang and Shen (1994) and Durst and Wennerberg (1991) where good agreement between k-epsilon model predictions and experimental results were reported.

Boundary conditions

The appropriate choice of boundary conditions is essential and is 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 mass flow 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 (Versteeg and Malalasekera, 1995):

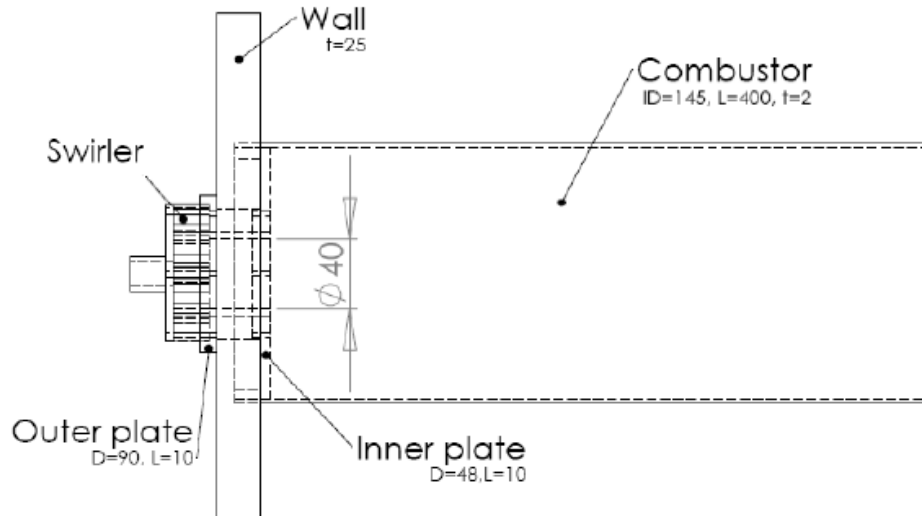


Figure 3. Combustor model geometry.

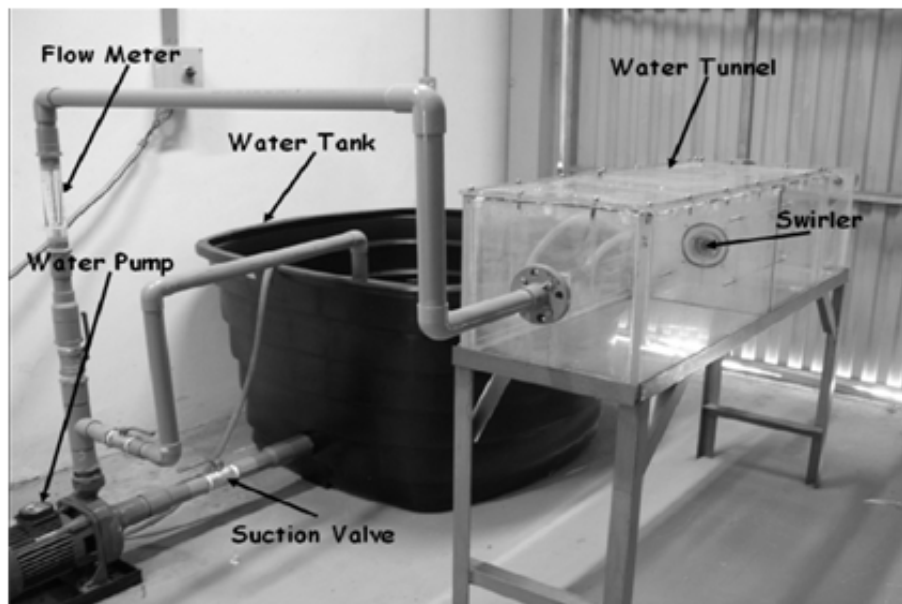


Figure 4. Water tunnel test rig layout.

$$k_{inlet} = 0.002(u^2)_{inlet} \quad (6)$$

$$\varepsilon = \frac{k_{inlet}^{1.5}}{0.3D} \quad (7)$$

Experimental

The water tunnel model test facility was used in the present study for isothermal investigation of flow patterns. The test facility is an open circuit experimental test rig arrangement (Figure 3). The main components consisted of a water storage tank, pump, flow meters,

control valve, a combustor model, and an outlet pipe. The tubular combustor model is focal point of the experiment and was placed inside the central section of the water tunnel test rig as shown in Figure 4. The whole water tunnel was filled with water in order to reduce distortion and improve visibility. The combustor model used in the water tunnel model test facility was geometrically equal to the one used for numerical simulation. It was fabricated from 3 mm thick Perspex tube with 145 mm internal diameter and 400 mm length.

For visualization of the flow pattern inside of the combustor, a piece of cotton was used as a tracer inside of the combustor. By using 10 Megapixels Nikon Coolpix L20 camera, the photographs during the experiments were captured as much as possible whilst

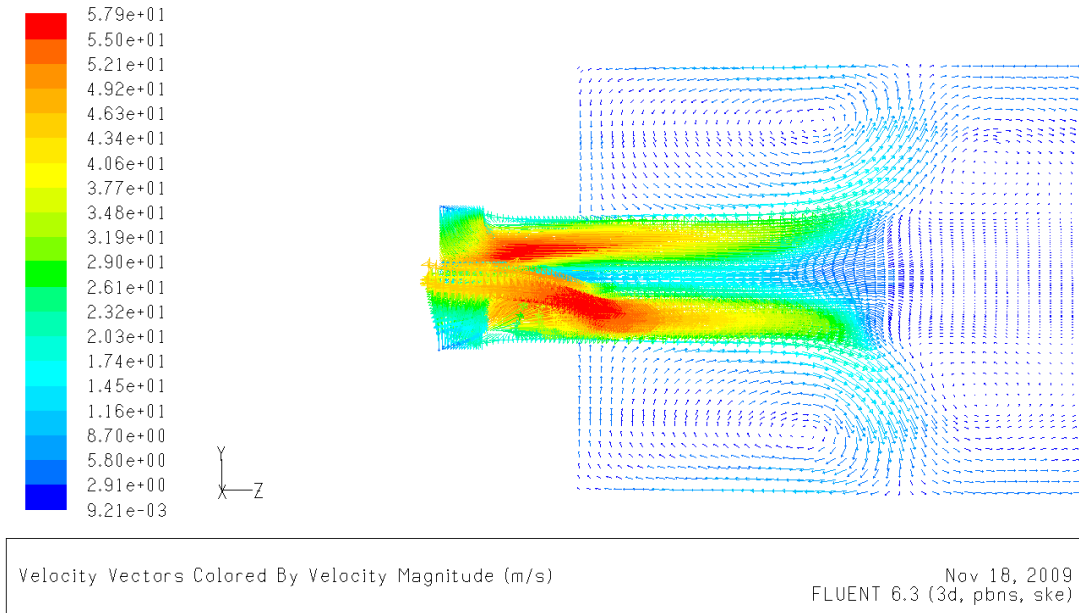


Figure 5. Velocity vectors and velocity magnitude (m/s) for 40° flat vane swirler.

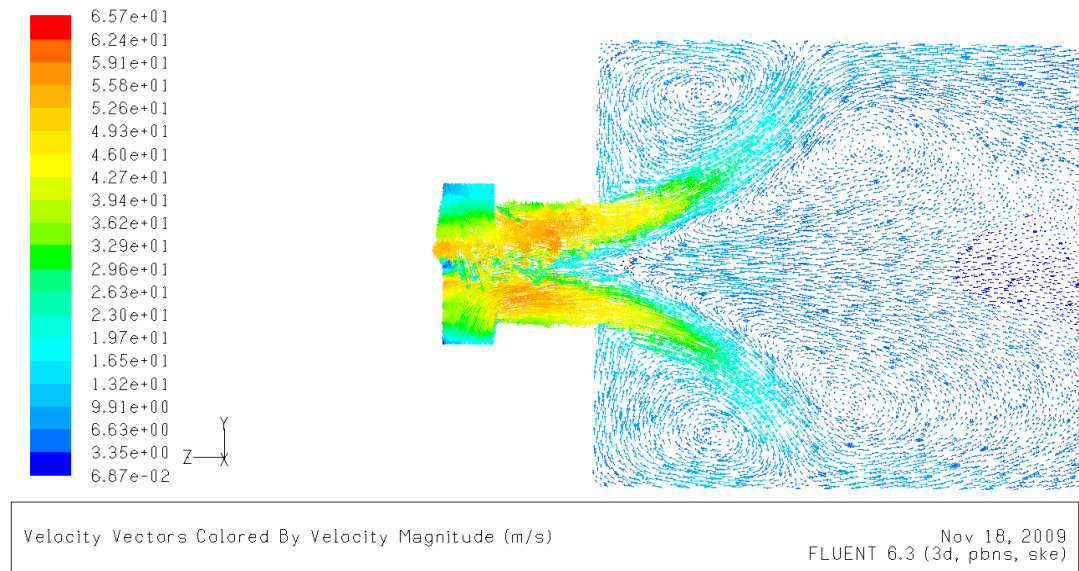


Figure 6. Velocity vectors and velocity magnitude (m/s) for 50° flat vane swirler.

the water tunnel is full and at high pressure. The high pressure situation will produce the swirl flow inside the combustor.

RESULTS AND DISCUSSION

Numerical simulation

The swirl number was numerically computed at the swirler exit plane for the four radial swirlers using the

integral shown in Equations 2 and 3. The different values of swirl number are listed in Table 1 for reference. It can be observed that these swirl numbers were in good agreement with those obtained from the investigation. Figures 5, 6, 7 and 8 show the velocity vectors and magnitudes of flow within the recirculation zone for all air swirlers. These vectors are obtained by plotting the axial velocities which ranges from maximum negative value to maximum positive value within the recirculation zone.

The flow downstream of 40° flat vane swirler shows

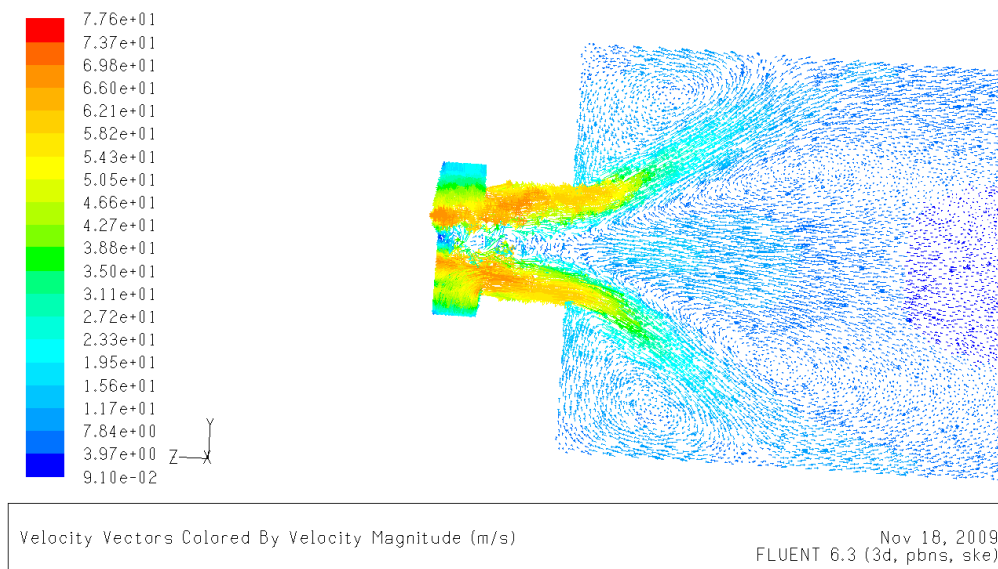


Figure 7. Velocity vectors and velocity magnitude (m/s) for 60° flat vane swirler.

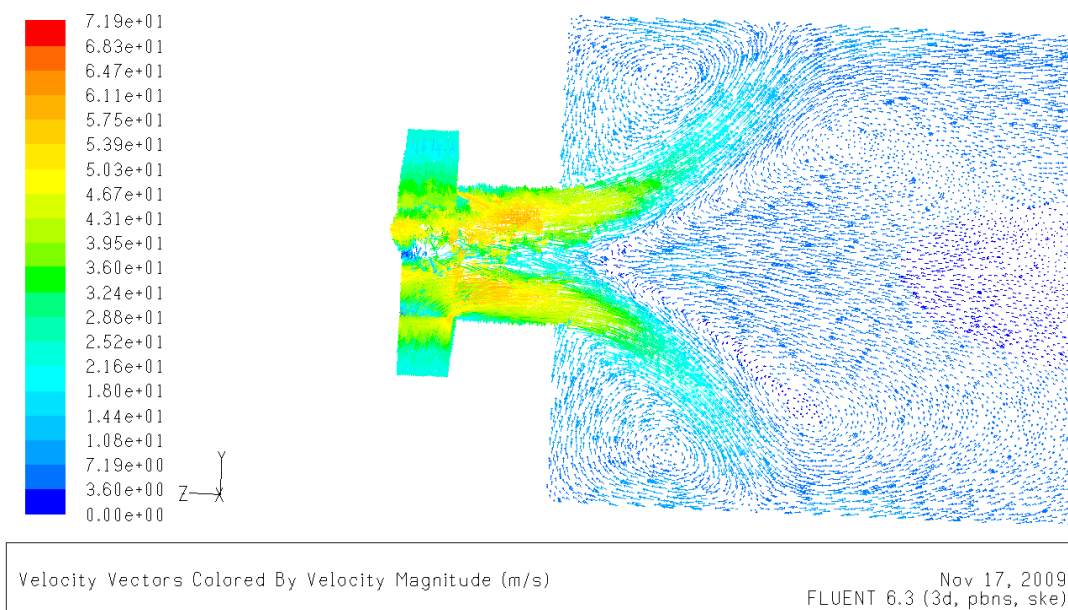


Figure 8. Velocity vectors and velocity magnitude (m/s) for 60° curved vane swirler.

maximum velocity magnitude of 57.9 m/s (Figure 5) which shows the formation of very small recirculation zone. Maximum velocity magnitude of 65.7 and 77.6 m/s were obtained for 50° and 60° flat vane swirlers respectively as shown in Figures 6 and 7, while for curved vane swirler, 71.9 m/s was obtained for 60° as seen from Figure 8.

The region of velocity magnitude is termed as central recirculation zone (CRZ), which is mainly responsible for

flame stabilization. As can be seen from Figure 5, the 40° has very small recirculation zone and shorter length with a width of 1.39 D1 and length 2.14 D1. This recirculation zone is critical as far as residence time for the reactant species and the rate of heat transfer are concerned. The above parameters, in turn, depend on the volume and surface area of the recirculation zone. Hence it will be useful to investigate the effect of the vane angle on the geometry namely the length, width, surface area and the



Figure 9. Photo of 40° flat vane swirler.

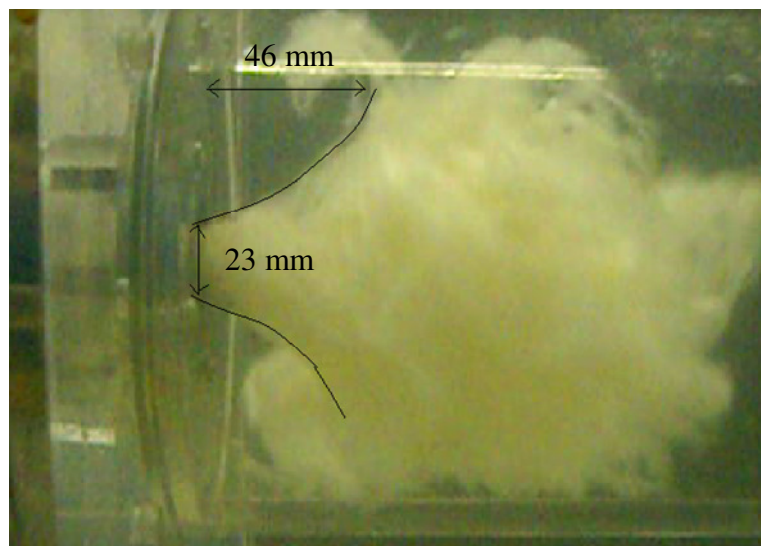


Figure 10. Photo of 60° flat vane swirler.

volume of the recirculation zone.

For the 50° swirler, Figure 6 shows that the width and the length of the recirculation zone have increased to 3.06 D1 and 4.64 D1, respectively compared to 40° swirler. The largest width of 3.33 D1 and length of 5.36 D1 were obtained for 60° flat vane swirler. Figures 5, 6, 7 and 8 also show that the size of the corner zone circulation decreases with the increase in swirl number.

Experimental

Figures 9, 10 and 11 show the photographs taken during the experimental tests for flat vane and curved vane swirlers with varying vane angles in the water tunnel test facility. Cotton was used in this experiment to help in visualizing the formation of recirculation zones. In this

experiment, only 40° and 60° vane angles were tested for both types of vanes.

For the 40° vane angle, there is none or very small swirling flow generated in the combustor and there is no recirculation zone formed. This is not in agreement with the computational simulation results. It may be due to the different density used for simulation and experiment.

The 60° vane angle produces good swirling flow and recirculation zone. For flat vane, it generates 46 mm length of CRZ and 23 mm diameter of the inlet while for the curved vane it generates 45 mm length of CRZ and 13 mm diameter of the inlet.

Comparing the results between the CFD and the experimental method, it was found that the higher the vane angle, the greater the swirling flow, and the larger the recirculation zone. Furthermore, the flat vane is better than the curved vane for both methods. Thus in this

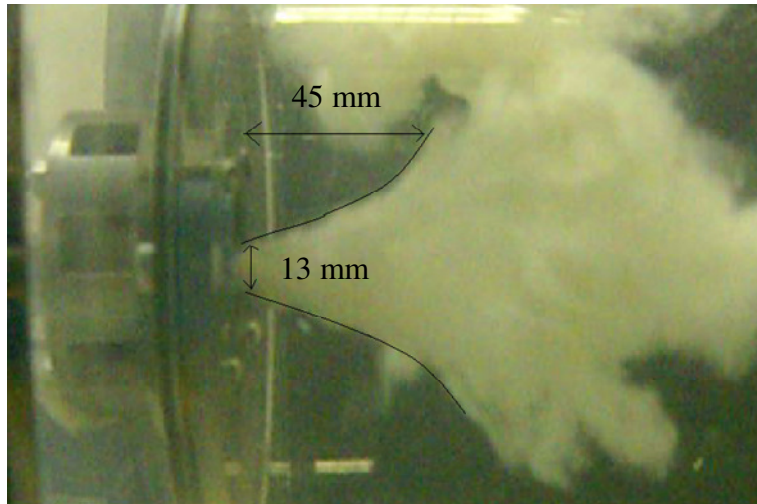


Figure 11. Photo of 60° curved vane swirler.

study, the 60° flat vane produces the best result for swirling flow and recirculation zone.

Conclusion

All four radial swirlers employed in this study show significant effects on the flow pattern within the combustor model. The 40° vane angle swirler produces very small volume of recirculation zone while 50° and 60° vane angle swirlers are able to double the zone sizes. From the parametric study carried out with varying angle of vanes, it is found that 50° swirler is the best for producing appropriate recirculation zone with reasonable pressure drop. These results are in good agreement with Mohd Jaafar et al. (2009) and Yehia et al. (2010) who studied effect of multiple inlet using axial swirlers.

With an increase in the swirler vane angle, corner recirculation zone size decreases as the tangential to axial momentum ratio increased. This dead zone has a negative impact on the combustion process and the homogeneity of the wall temperature as well.

REFERENCES

- Beer JM, Chigier NA (1972). *Combustion Aerodynamics*. Applied Science Publisher, London.
- Da Palma JMLM (1988). *Mixing in Non-Reacting Gas Turbine Combustor Flows*. PhD Thesis, University of London, UK.
- Durst F, Wennerberg D (1991). Numerical Aspects of Calculation of Confined Swirling Flows with Internal Recirculation. *Int. J. Numerical methods Fluids*, 12: 203-224.
- Fluent 6.3 Tutorial Guide, Fluent Inc., 2006.
- Ganesan V (2003). *Gas Turbines*. Tata McGraw-Hill, New Delhi.
- Gupta AK, Lilley DG, Syred N (1984). *Swirl Flows*. Energy and Engineering Science Series, Abacus Press, England.

- Ishak MAS, Mohd Jaafar MN (2009). The Effect of Radial Swirl Generator on Reducing Emissions from Bio-Fuel Burner System, *Modern Appl. Sci.*, 3(6): 45-61.
- Kim YM, Chung TJ (1989). Finite-Element Analysis of Turbulent Diffusion Flames. *AIAA J.*, 27(3): 330-339.
- Lefebvre AH (1983). *Gas Turbine Combustion*. Hemisphere Publishing Corporation, First edition.
- Mattingly JD (1996). *Elements of Gas Turbine Propulsion*. McGraw-Hill International Edition, Singapore.
- Mellor AM (1990). *Design of Modern Gas Turbine Combustor*, Academic Press.
- Menzies KR (2005). An Evolution of Turbulence Models for the Isothermal Flow in a Gas Turbine Combustion System. 6th International Symposium on Engineering Turbulence Modelling and Experiments, Sardinia, Italy.
- Mohd Jaafar MN, Eldrainy YA, Ahmad MF (2009). Investigation of Radial Swirler Effect on Flow Pattern inside Gas Turbine Combustor. *Modern Appl. Sci.*, 3(5): 21-30.
- Versteeg HK, Malalasekera W (1995). *An Introduction to Computational Fluid Dynamics, the Finite Volume Method*. Longman Group Ltd.
- Yehia AE, Hossam SA, Khalid MS, Mohd Jaafar MN (2010). A Multiple Inlet Swirler for Gas Turbine Combustors, *Int. J. Mechanical Syst. Sci. Eng.*, 2(2): 106-109.