CFD Analysis of Swirling Flow Fields Inaxisymmentric Gas Turbine Combuster

Mr. G Satheesh Kumar[1]
Assistant Professor,
Dept of Mechanical Engineering
Anurag Group of Institutions (CVSR),
Hyderabad, India

Mr. Harish Mugutkar[2]
Assistant Professor,
Dept of Mechanical Engineering
Anurag Group of Institutions (CVSR),
Hyderabad, India

Mrs. D Swetha[3]
Assistant Professor,
Dept of Mechanical Engineering
Anurag Group of Institutions (CVSR),
Hyderabad, India

***Abstract*— Swirling flow generates recirculation and provides better mixing and turbulence. To study the flow behavior and its effect in combustion chamber, CFD simulation is performed and recirculation zone and shape of the flame region is found out for axisymetrical 12 vane swirler with 50° vane angle. Three dimensional velocities for swirling flows are found out by means of computational fluid dynamics (CFD), for producing Recirculating Zone to control the length & stability of flame in the Gas Turbine Combustor. The core objective of this study is to present the effect of swirler geomentry and inlet conditions on recirculation length. The flow through 12 vane 50 deg swirler is investigated by using ANSYS CFX software. Present work gives complete behaviour and knowledge about recirculation zone & effect of guide vanes.**

Keywords*— *Swirler, Combustion Chamber, Recirculation Length, Vane Angle.

NOMENCLATURE:

swirl number	S
$Axial\ flux\ of\ angular\ momentum$	$G_\emptyset$
$Flux\ of\ axial\ momentum$	G_x
radius of swirler	R
radius of hub	R_h
vane angle	θ
turbulence kinetic energy	K
turbulence eddy dissipation	ε

I. INTRODUCTION

A basic requirement of all gas turbine combustors is that flame stay lighted over a wide range of operating conditions. This is specially arduous for the air craft combustor which has to cope with the adverse conditions of low pressure and temperature. Toroidal flow reversal is essential , because it entrains and recirculates the portion of hot combustion products to mix with the incoming air and fuel and keep the flame lightened.

Swirl flows offer an interesting field of study for aerospace and mechanical engineers in general and for combustion engineers in particular since it involves complex interaction of recirculation and turbulent mixing which aid flame stabilization in combustion systems. Swirling flows, which are highly complex, have the characteristics of both rotating motion and free turbulence phenomenon encountered as in jets and wake flows. Swirling flows in both reacting and non-reacting conditions occur in a wide range of applications such as gas turbines, marine combustor, burners, chemical processing plants, rotary kilns and spray dryers. Swirling jets are used as a means of controlling flames in combustion chambers.

The presence of swirl results in setting up of radial and axial pressure gradients, which in turn influence flow fields. In case of strong swirl the adverse axial pressure gradient is sufficiently large to result in reverse flow along the axis and the setting up of an internal circulation zone.

The degree of swirl is characterized by swirl number, which is defined as the ratio of axial flux of angular momentum to the flux of axial momentum [4].

$$S = \frac{Axial\ flux\ of\ angular\ momentum}{Flux\ of\ axial\ momentum}$$

$$S = \frac{G_\emptyset}{RG_x}$$

For curved vanes, swirl number is defined as

$$S = \frac{2}{3}\frac{1-[{}^{R_h}/_{R}]^3}{1-[R_h/R]^2}\tan\theta$$

Where R is radius of swirler, R_h is radius of hub, S is swirl number, θ is vane angle.

However for vane swirlers, the tan (θ) alone may be taken as a measure of the degree of swirl1 where is the vane angle. Swirl flows can be classified into weak, medium and strong swirl based on the swirl number ‘S’. If swirl number is less than 0.3 it is usually classified as weak swirl and if it is between 0.3 and 0.6 it is called medium swirl and if the swirl number is greater than 0.6, it is called strong swirl. Huang and Tsai2 have indicated that a larger recirculation zone is induced when the swirl number is greater than 0.6. Swirling flows are highly three dimensional and it is quite complex to obtain enough details experimentally to fully understand it and to comprehend the mechanisms involved. At present many numerical solutions are quite possible for swirling flows, but Leschziner and Rodi3 suggest that any attempt to identify and rectify turbulence models defects for swirling flows is bound to be frustrated unless comprehensive experimental results of the jet are available. As well any numerical solution strongly relies upon the basic experimental benchmark data for validating the results. Hence the basic objectives of the papers are: (i) to provide a systematic and Experimental measurements of axial, radial and tangential velocity profiles downstream of the swirler in order to understand the flow field characteristics, (ii) to provide a base line experimental data for the validation of numerical results, and (iii) to understand the effect of turbulence and recirculation zone created by vane swirler.

II. COMPUTATIONAL FLUID DYNAMICS PROCEDURE

In the CFD simulations, accurate results can be obtained. CFD is a software which is used to analyze the flow field by giving the initial and boundary conditions over the fluid domain through which accurate solutions can be obtained. The following are the steps involved in carrying out the analysis using Ansys CFX solver.

1. Modeling of geometry:

This is the first step in building and analyzing the flow model. It includes geometry creation (swirler) in solid works as shown in fig.1 and combustion chamber in Ansys workbench sketcher tool as shown in fig.2 which is the completion of the geometry.

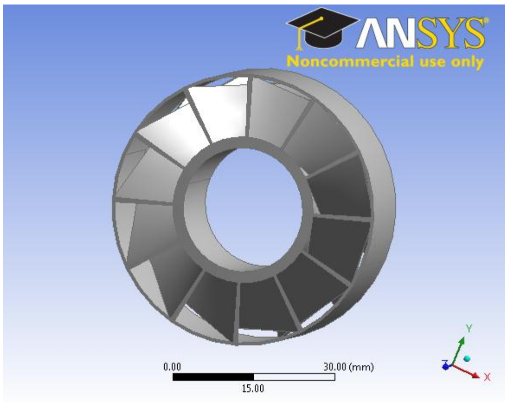


Fig 1: 12 vane swirler with 50˚ vane angle solid works model

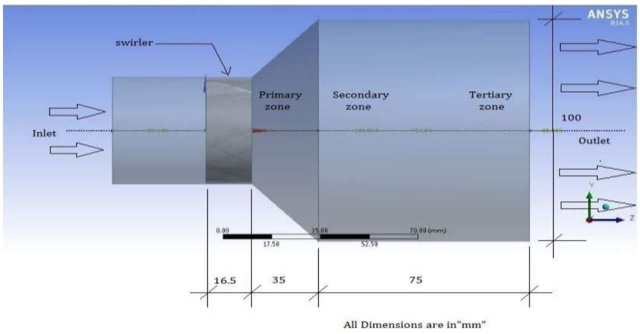
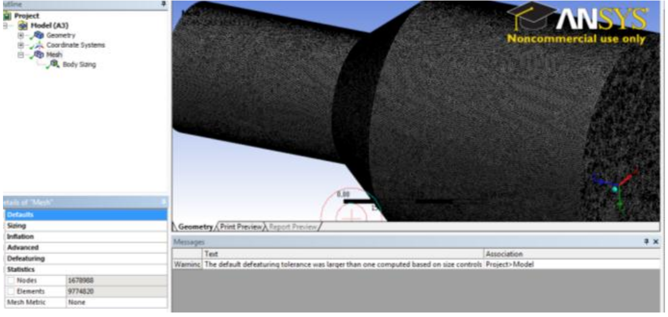


Fig.2: complete geometry of combustion chamber by using CFX

2. Meshing the flow geometry:

This is the process of converting the geometrical entities into finite element entities. The model is meshed with the 10 node tetrahedral 3-D elements. Named selections are given for inlet, outlet and wall of the meshed entity. Fine mesh is chosen to get the meshing convergence for better accurate results as shown in fig.3.



Nodes	1678988
Elements	9774820
Mesh Metric	None

	Text	Association
Warning	The default defeaturing tolerance was larger than one computed based on size controls	Project>Model

Fig. 3 Meshing of complete geometry

3. Set up:

After updating the meshed geometry, the required different inlet flow conditions (massflow rate=0.17kg/sec and 0.24 kg/sec, air temperature= 25deg) and outlet condition of relative pressure =0.25Pa are provided in setup for carrying out further analysis. Complete setup is shown in fig**.4**

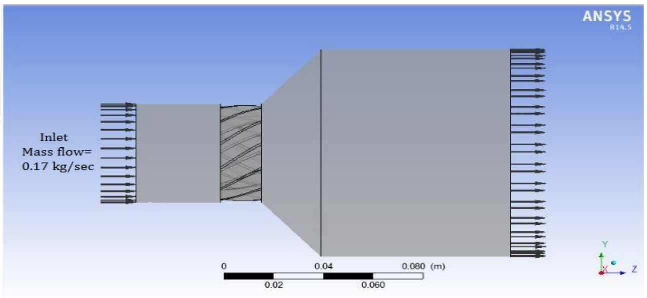


Fig.4 set up of complete geometry

Computational Details:

12 vanes swirler is designed in solid works which is then imported into Ansys CFX. Then total geometry is fine meshed for obtaining accurate results. In order to obtain the flow field details in a swirling flow the governing equations to be solved are

a) Continuity equation;
b) Three momentum equations;
c) Turbulence kinetic energy model equations.

In this case to solve the problem control-volume based technique is used in which segregated solver is used. The segregated solver uses under relaxation factors to control the update of computed variables at each iteration. If the residuals continue to increase after the first 4 or 5 iterations, then the values under relaxation factors should be reduced. SIMPLE (Semi Implicit Pressure Linked Equation) scheme is used for pressure velocity coupling. This procedure is based on cyclic series of guess and correct operations to solve the governing equations. This procedure continues until the solution converges.

k-ε is a two equation turbulence model. K is the turbulence kinetic energy and is defined as the varience of the fluctuation in velocity. ε is the turbulence eddy dissipation(rate at which the velocity fluctuation dissipate). Two equation turbulence models are widely used as they offering a good compromise between numerical effort and computational accuracy.

RESULTS & ANALYSIS:-

Pressure Distribution:

In the Primary zone, the inlet pressure at the centre, which increases as it moves into the mid-plane, At the near to the wall the pressure drops. In the entire diffuser region the pressure remain constant. In the Expansion Chamber pressure remain constant shown in the fig.5.

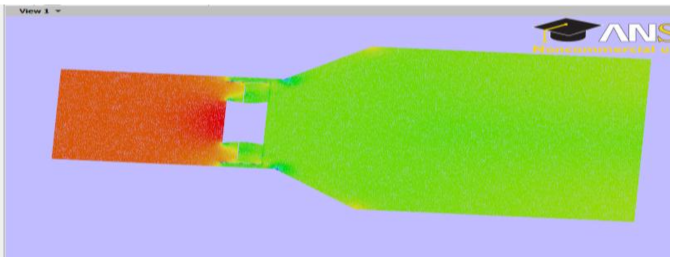


Fig.5:Pressure distribution with 12 vanes swirler in combustion chamber

Velocity Distribution:

In the Primary zone, at the center the velocity gradient is low as compare to the velocity gradient near the wall and the mid plane. At the wall the velocity is increasing in the expansion chamber. In the central chamber the velocity gradient is negative due to flow reversal and it indicates the recirculation of swirling flows. This recirculation creates better mixing of air and fuel and stabilizes the flame in the Gas Turbine combustor.

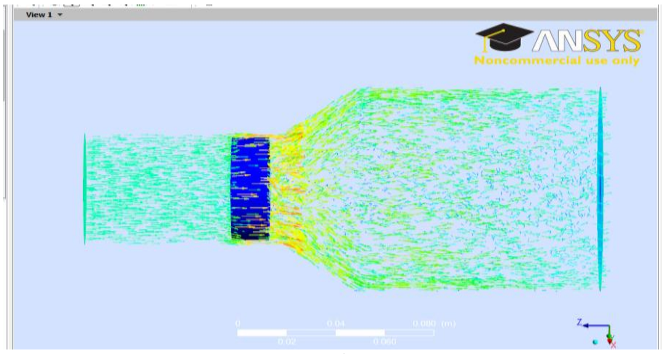


Fig.6: Velocity distribution for 12 vane swirler in CFX

VELOCITY PROFILES:

Figure 7 and 8 shows that the axial and swirl velocity profiles for 12 vanes, 50^0 vane angle for an inlet massflow of 0.17kg/sec. Radial velocities are comparatively much smaller than the axial and swirl components, hence those profiles have not been considered in the present investigations. The velocity profiles at 5 downstream stations x/d=0.01765,0.0245,0.0365,0.0458 and 0.0568 have been shown in figure 7. From the axial velocity profile at different downstream stations, it can be seen that the maximum radial distance is covered with negative velocity components at x/d=0.01765 and this distance slowly decrease along the downstream stations. The negative component of this velocity vanishes between x/d=0.0458 to 0.0568,which indicates that the flow reversal after that distance is not felt at all. The locus of the zero axial velocity components is the indicative of the flow reversal zone. From these profiles, it can also be inferred that the flow tends toward corner recirculation.

Figure 9 and 10 shows the axial and swirl velocity components for 12 vanes, 50^0 vane angle swirler with increased mass flow of 0.24kg/s. Again the measurements have been done at the same downstream stations. In this investigation with increase of mass flow rate ,the velocity increases through the swirler and hence the length of recirculation length tends to increase.

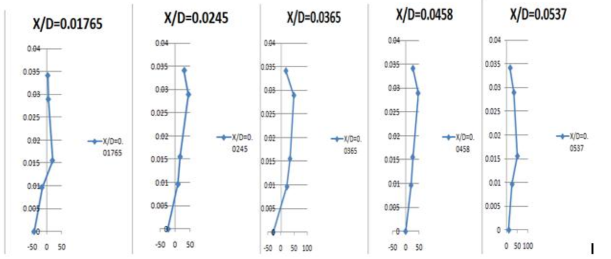


Fig.7: Axial velocities of swirling flow field in the primary zone for m=0.17 kg/sec.

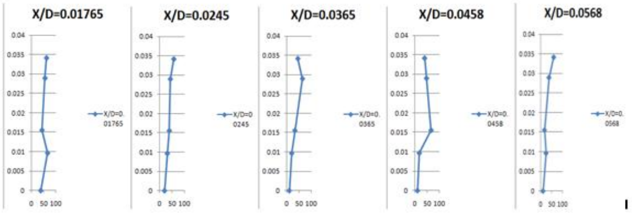


Fig.8: Swirl velocities of swirling flow field in the primary zone for m=0.17 kg/sec.

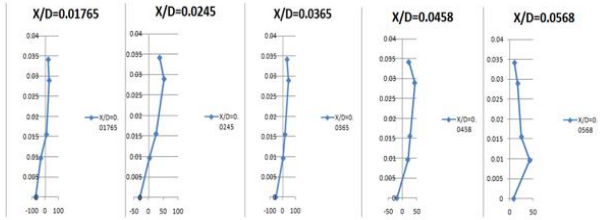


Fig.9: Axial velocities of swirling flow field in the primary zone for m=0.24 kg/sec.

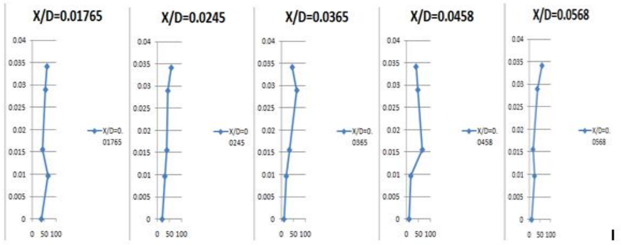


Fig.10: Swirl velocities of swirling flow field in the primary zone for m=0.24 kg/sec.

III. CONCLUSIONS

1. In CFD analysis, it has been found that swirler creates strong swirling flows and recirculation zone helps for better mixing of air and fuel for good combustion.
2. Low pressure region occurs in the central part of combustion chamber. The shape of the combustion chamber will avoid the atmospheric disturbance and detain the atmospheric entering into the chamber. In the entire Expansion Chamber, Complete recirculation zone is formed and to get good air-fuel mixing with good flame stabilization. The recirculation zone is found in the primary zone of the combustion chamber.
3. Swirler geometry and the inlet flow conditions are the most important parameters which affect the recirculation length in the swirling flow.
4. In swirling flow fields if inlet mass flow increases the recirculation length of the flow increases.

REFERENCES

[1] S.Bharat Krishna, Non Member & Prof. V.Ganesan, Member, CFD analysis of flow through combustion swirler, vol-86, April 2005 IE(I) Journal.
[2] R Thundil Karuppa Raj & V.Ganesan, Experimental study of recirculating flows induced by vane swirler, Indian Journal of Engineering & material Science, Vol.16, February 2009, pp.14-22.
[3] E.Kilik, better swirl generation by usnig curved vanes, California state university, Long Beach, California.
[4] Rhode D L, Lilley D G & McLaughlin D K, AIAA J, 21 (4) (1983) 593-600.
[5] Somer H T & Ramos J I, AIAA J, 23 (1985) 241-248.
[6] Ganesan V, Recirculation and Turbulence studies in an isothermal Model of a Gas Turbine Combustor Chamber, Ph.D Thesis, IIT Madras, Chennai, India, 1974.
[7] Mathur M L & Maccallum N R L, J Inst Fuel, 214 (1967) 238-245.
[8] Rao A N, Experimental and Theoretical Investigations of swirling flows, Ph D Thesis, IIT Madras, Chennai, India, 1974.
[9] Lilley David G, J Propuls, 2 (1986) 64-72.
[10] Koutomos P & McGuirk J, J Exp Fluids, 7 (1989) 344 –354.
[11] Ahmed, Saad A, Turbulence statistics of a confined swirling flow, Proc ASME Fluids Engineering Division, Vancouver, Canada, 1997, 3316.
[12] Anacleto P M, Fernandes E C, Heitor M V & Shtork S I, Combust Sci Technol, 175 (2003) 1369-1388.