CFD ANALYSIS OF GAS TURBINE COMBUSTOR PRIMARY ZONE USING DIFFERENT AXIAL SWIRLER CONFIGURATIONS

[1]PRASANNA GOUDA, [2]K. SRINIVASAN, [3]G. SIVARAMAKRISHNA, [4]K.S. SHASHISHEKAR

[1]M.Tech Project Trainee, Gas Turbine Research Establishment, DRDO, Bangalore&
Department of MechanicalEngineering, SIT,Tumkuru, Karnataka.
[2, 3] Scientists, Gas Turbine Research Establishment, DRDO, Bangalore, Karnataka.
[4]Professor, Department of MechanicalEngineering, SIT,Tumkuru, Karnataka.
E-mail: [2]srinivasan@gtre.drdo.in

Abstract— This paper presents CFD analysis results of primary zone of a gas turbine combustor having an axial swirler. Swirler aerodynamics place a vital role in the quality of flow pattern in primary zone and has a direct impact on the combustor performance. Several parametric studies have been carried out to study the effect of various swirler parameters like vane angle, vane shape (flat or curved), inlet angle etc., on the strength and size of recirculation zone existing in the primary zone. Analysis has been carried out using the commercially available CFD code ‘ANSYS Fluent’. Geometrical modeling has been carried out using Gambit. Through the study, it has been found that the geometrical parameters of swirlersignificantly affect the recirculation zone pattern in primary zone of combustor.

Keywords— Axial Swirler, Recirculation zone, Combustor, Swirl number.

I. INTRODUCTION

Most of the gas turbine combustors employ swirlers in and around fuel injectors to enhance mixing, increase residence time and stabilize the flow in primary zone by creating toroidal re-circulation zone. Even though there are several types of swirlers viz., axial, radial, discrete jet designs etc., axial swirlers are the ones most commonly used.

Lefebvre [1] carried out studies on aerodynamics of swirler, which clearly describes about different types of swirler, swirler geometry details, swirl number, recirculation zone etc. He stated that curved vane swirlers are aerodynamically more efficient compared to flat vane asthe flow turns gradually, which inhibits flow separation on the suction side of vane. This results in generation of high swirl and radial velocity component at swirler exit which helps to develop stronger recirculation zone.

P.Muttukumaret.al., [2] have analyzed flow through an axial swirler. They reported that swirler can control the combustor performance by assistingair-fuel mixing process, and producing recirculation zone which act as flame anchor and increases the residence time. Proper selection of swirler is necessary to increases the performance of combustor and reducethe NOx. They have used four different swirlers of angle of 30, 40, 50, 60 degree. They concluded that the flow field effect is significant as the swirl angle at a constant pressure drop across the swirler.
Arun Raj et al. [3] investigated the effect of degree of swirl on the flow characteristic using Particle Image Velocimetry (PIV). This experiment was performed using swirl angles 30 and 48 degrees. It was clearly shown that vane angle has a predominant effect on central recirculation zone. A high degree of swirl gives better mixing compared to medium degree swirl as in high degree of swirl, micro level mixing takes place between fuel-air. Ilker et al. [4] have studied the effect of swirl number on combustion characteristics such as velocity temperature etc, k- epsilon model was used for turbulence modelling. Different swirl numbers viz., 0, 0.1, 0.2, 0.3, 0.4, 0.5, 0.6, and 0.7 are considered for this analysis. The results show that, the combustion characteristics such astemperature, velocitydepends on swirl number. Fluid dynamic behavior of natural gas diffusion flame, axial velocity, temperature, central recirculation zone and corner recirculation zonestrongly depend on the degree of swirl. Jun Cia [5] et al. carried out their studies on different turbulent models to calculate three dimensional swirling flows with discrete jets in a confined model combustor. In this study, k-epsilon and Reynolds Stress Model are compared with experiments. They concluded that Reynolds stress model gives better performance compared to k-epsilon model. J.M.Khodadadiet.al. [6] carried out systematic study to asses the applicability of k-epsilon model and one of its variants to detect decaying swirl in developing turbulent pipe flow. From this study they stated that, k-epsilon model predicted turbulence well in pipe flow. Yehia A. Eldrainy et al. [7] studied multiple swirler configurations for gas turbine combustor. This study demonstrated the impact of the ratio of tangential to axial flow rate on Central Recirculation Zone (CRZ). They concluded that, length of CRZ is directly proportional to above mentioned ratio.

II. GEOMETRICAL DETAIL

In present analysis an annular combustor with 18 swirlers and 18 air blast atomizers equally spaced along the circumferential direction has been used. A 20^o sector combustor model has been used, as shown in Figure 1, with periodic boundary conditions. Uni-Graphics NX-7.5 software was used for creating the geometric model. Different parts of combustor like swirler, air blast atomizer, dome and flare are shown in Figure 1. The outer and the inner annuli are provided with one row each of primary and dilution holes. In addition to this, liners are cooled by 12 rows of cooling rings. Atomizer is surrounded by swirler**.** Table 1 shows the geometrical details of swirler configurations used for this study. 3-D model of the baseline swirler is shown in Figure 2.

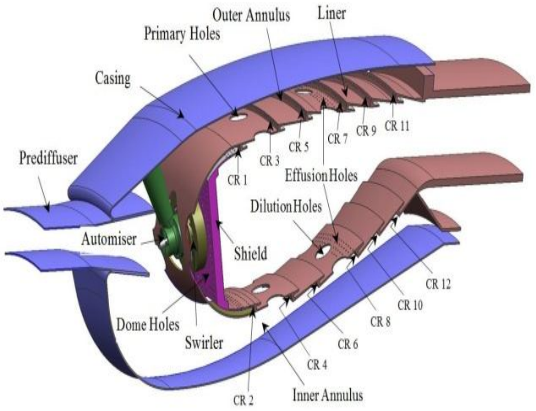


Figure 1 3-D 20 Degree Sector CAD Model of the Combustor used for Analysis.

Table 1: Geometrical Details of Swirler Configurations Analyzed.

Sl. No.	Model No.	Vane Type	Vane Angle (Degrees)	Vane Inlet Angle (Degrees)	Feature at vane exit
1	Base Model	Flat	50	38	Fully cut
2	1	Flat	50	0	Fully cut
3	2	Curved	50	0	Fully cut
4	3	Curved	50	0	No cut
5	4	Curved	55	0	No cut

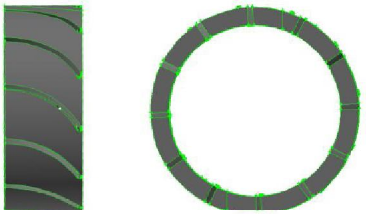


Figure 2 3-D Models of different swirlers configuration used in present study.

III. GRID GENERATION

Accuracy of CFD solution is based on grid size. In the present configuration, hexahedral and tetrahedral type of meshes have been generated using Gambit software. Figure 3 shows the computational grid for 20 degree sector model of annular combustor with swirler considered for present study. Grids of all the models have been generated using the same methodology.

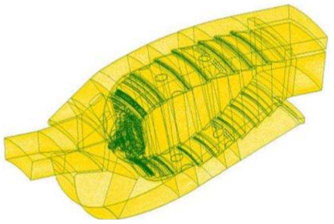


Figure 3 3-D Grid Generated

IV. GRID QUALITY

The quality of grid plays an important role in the precision and stability of the numerical computation. The attributes associated with the mesh quality considered in the present studies are aspect ratio and equi-angle skewness. The present studies have around 99% of the cells having skewness below 0.65. The worst element in the grid has a skewness of 0.98. Aspect ratio of the computational grid is greater than 0.35.

V. TURBULENCE MODEL

In this present study, the two-equation Realizable k-epsilon turbulent model has been used. The Realizable k-epsilon model differs from the standard k-epsilon model in its formulation. Benefit of the Realizable k-epsilon model is that it more accurately predicts the spreading rate of both planar and round jets. It is also likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation. [11]

VI. BOUNDARY CONDITIONS

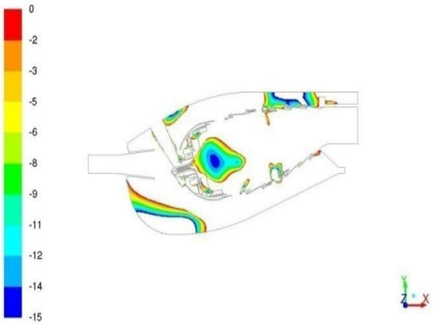
Total pressure and total temperature are considered as inlet boundary conditions and static pressure has been specified at outlet boundary. All faces bounding the flow except the side faces are defined as walls. Both the extreme faces in the circumferential direction are defined as periodic.

VII. RESULTS AND DISCUSSION

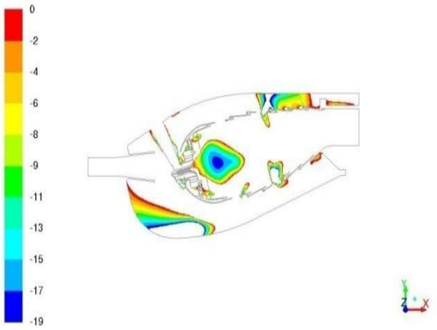
CFD analysis has been carried out with different swirler configurations to study the flow patterns in primary zone of combustion chamber. Axial velocity, radial velocity, tangential velocity, turbulent kinetic energy plays an important role in the development of recirculation zone and its strength. The results are discussed in the following sections.

(i) Reverse Axial Velocity Distribution

Axial velocity is an important parameter to study the recirculation zone inside primary zone of combustion chamber. Figure 4 shows reverse axial velocity contours at recirculation zone area for base model and Model 3. Model 3 gives high negative velocity compared to base model as shown in the figure. This is due to more air flow is getting recirculated by swirler Model 3. From this contour plot it is evident that recirculation zone is controlled by primary air jets. Compared to Model 3, the base model recirculation zone getsaffected more by primary jets due to faster decay in swirl flow radial velocity.



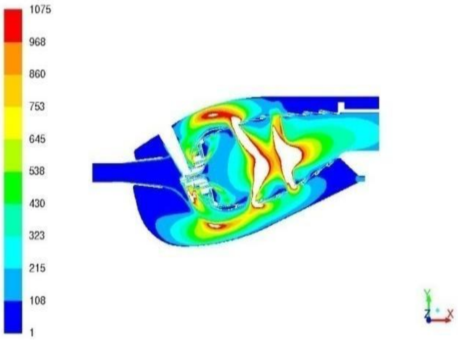
(a) Contour plot of reverse axial velocity (Base Model)



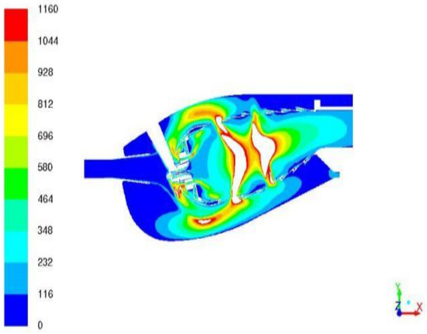
(b) Contour plot of reverse axial velocity (Model 3)
Figure 4 Contour plots of reverse axial velocity for Base Model and Model 3

(ii) Turbulent Kinetic Energy Distribution

Turbulent kinetic energy is defined as mean kinetic energy per unit mass associated with eddies in turbulent flow. Figure 5 shows the contour plots of turbulent kinetic energy. CFD results show that highest turbulent kinetic energy peak is at the exit of the swirler jet. Compare to base model, Model 3 is having higher turbulent kinetic energy due to higher velocities and velocity gradient at swirler exit and along the swirling jet. From this turbulent kinetic energy comparison, it is evident that curved vane swirler generates better shear layer compared to flat vane swirler, which is beneficial for better mixing.



(a) Contour plot of turbulent kinetic energy (m2/s2) (Base Model)



(b) Contour plot of turbulent kinetic energy (m2/s2) (Model 3)
Figure 5 Contour plot of turbulent kinetic energy (m2/s2)

(iii) Axial Velocity Along The Centerline of Combustor

Mean axial velocity along the centerline of combustor is shown in Figure 6. It shows that the mean reverse axial velocity increases from swirler exit plane and diminishes towards the stagnation point of recirculation zone. Model 2,3,4 (curved vanes) generates longer recirculation zone compared to base model. But for model 2 and 4, the maximum reverse axial velocity occurs further downstream than model 3.This may happen due to the influence of primary jets. All curved vane swirlers generates higher reverse axial velocity due to higher reverse mass flow rate. Even though curved vane swirlers perform better, they get highly influenced by primary air jetsand it can be concluded that, even though geometric swirl number is same for base model and Models 1, 2, 3, Model 3 generates a better recirculation zone due to higher flow turning angle and less blockage.

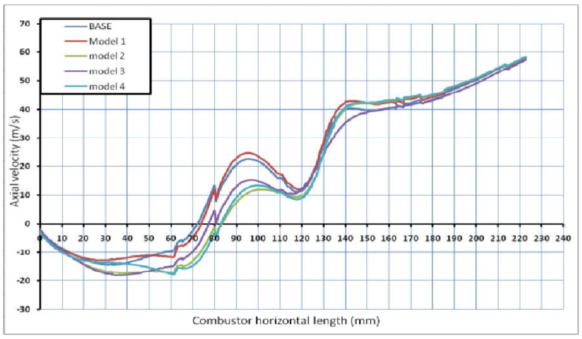


Figure 6 Variation of axial velocity along the centerline of combustor.

(iv) Axial Velocity Distribution at 10mm Downstream Of Swirler Exit

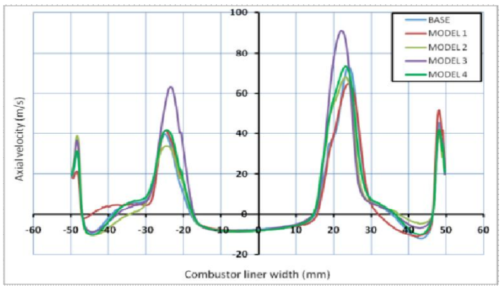


Figure 7 Axial velocity at 10mm downstream of swirler exit.

Due to curved vane effect and the associated lower blockage, Model 3 passes higher mass flow rate through it for a given pressure drop. This effect can be seen from increase in peak axial velocity for Model 3 in Figure 7. Even though Model 2 and 4 consists of curved vanes, their peak axial velocities are not comparable to Model 3. This may be due to the higher blockage in Model 2, since it is fully cut at exit and due to higher blade angle in Model 4. From Figure 7, it is evident that, for a given blade angle, curved vane swirlers pass higher mass flow rate compared to flat vane swirles.

(v) ***Tangential Velocity Distribution at 10mm Downstream Of Swirler Exit***

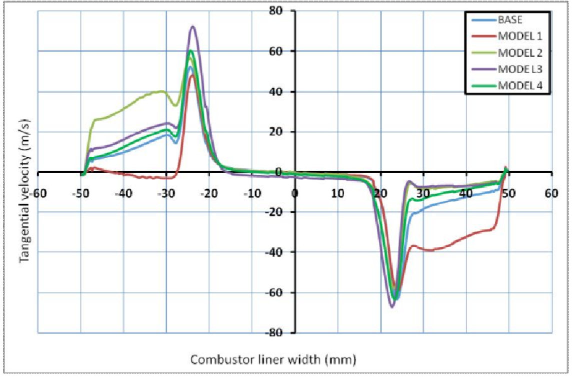


Figure 8 Tangential velocity at 10mm downstream of swirler exit

Figure 8 shows tangential velocity distribution at 10mm downstream of swirler exit. It can be seen that curved vane swirlers generate higher peak tangential velocities compared to flat vaneswirlers. From this figure, it is evident that higher swirl velocity can be imparted to incoming pure axial flow by curved vane swirlersin comparison to flat vaneswirlers. This higher swirl velocity is due to larger effective flow turning angle of curved vanes in comparison to that of flat vanes.

CONCLUSION

CFD analyses of flow in the primary zone of a gas turbine combustor have been carried out under non-reacting conditions by varying the geometrical features of swirler. The following conclusions can be drawn from this study.

1. CFD codes can be effectively used for analyzing and estimating flow field generated by vane swirlers in gas turbine combustors.
2. The flow is more efficiently turned by curved vane swirlers, which resulting in larger swirl velocities.
3. The recirculation zone is improved by changing flat vane to curved vane swirler. Also the level of turbulent kinetic energy can be increased by using curved vanes.
4. Even though geometric swirl number is same for flat and curved vane swirlers, latter generates a better recirculation zone.
5. For a given angle, curved vane swirlers pass more mass flow due to less blockage (Flow separation avoided).
6. Curved vane swirlers can be effectively used to generate better recirculation zone with improved mixing in gas turbine combustion chambers.

ACKNOWLEDGMENT

The authors would like to thank Sri. M. Z. Siddique, Outstanding Scientist and Director of Gas Turbine Research Establishment (GTRE), for permitting to publish the paper. Dr. G. S. Shiva Shankar (HOD, Mechanical Engineering Department, SIT Tumakuru) for his valuable suggestion and guidance at various stages throughout the project. And my special thanks to all members of combustion group.

REFERENCES

[1] Lefebvre, A. H., 1998, "Gas Turbine Combustion”, Second Edition, Tayler and Francis.
[2] Muttukumar, P.,S.Balakrishna, S.R.,2003"CFD Analysis of Recirculating Flows Induced by Axial Swirler", International Journal Of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol.2 Issue 6.
[3] Arun Raj, E,.Sujith, R, I,.andChakravarty, S, R., 2009, "Investigation of the Effect of Degree of Swirl on Swirl flow characteristic using PIV", 8th International symposium on Particle Image Velocitymetry-PV09,Melbourne,Australia.
[4] Ilker and Yilmaz.,2013, "Effect of Swirl Number on Combustor Characteristic in Natural Gas Diffusion Flame",Journal of Energy Resources Technology, Vol. 135/042204-1.
[5] Jun Cia, 2008, "Aerodynamics of Lean Direct Injection Combustor withMulti Swirler Arrays",Ph.D Thesis, University of Cincinnati.
[6] Khodadadi, J.M., and Vlachos, N.S.,1987, "Effect on Turbulence Model Constants on Computation of Confined Swirling Flows", Vol. 28, Technical Notes, AIAA Journal.
[7] Yehia A. Eldrainy, Hossams. Aly, and Khalid M,2009, "A Multiple Inlet Swirler for Gas Turbine Combustor", Word Academy Science, Engineering and Technology, 53, 2009, publication / 242598240.
[8] Fluent User Guide. Version 6.13.0
[9] Gambit User Guide Version 2.4.6
[10] John D Anderson, CFD: The Basics with Application.
[11] Mohammad B. S., J. Cia, and S. M. Jeng, 2010, “Gas Turbine Single Annular Combustor Sector Aerodynamics”, 48th AIAA Aerospace Science Meeting Including the New Horizons Forum and Aerospace Exposition, AIAA 2010-579.

★★★