CFD Analysis on Different Swirl Angle on NOx Generation of Gas Turbine Burner by Using ANSYS Software

Prakhar Dubey[1], Santosh Kumar Paswan[2], Sachin Baraskar[3]
Assistant Professor, Mechanical Engineering Dept., SSSUTMS, Sehore, Madhya Pradesh, India[1]
M.Tech Scholar, Mechanical Engineering Dept., SSSUTMS, Sehore, Madhya Pradesh, India[2]
H.O.D. Mechanical Engineering Dept., SSSUTMS, Sehore, Madhya Pradesh, India[3]

Abstract:. Swirl vanes are used for imparting swirling motion to the flow. These swirl vanes with its vanes induce the air circulation in order to create a turbulence at the primary zone of the combustion chamber. A doublet swirl vanes influence air flow in both axial and radial direction. The CFD analysis has been performed in a doublet swirl vanes under a subsonic speed. The objective of this paper is to increase the vortices around the swirl vanes by changing the area, angle of swirl vanes and increasing the swirl number. Further, the primary zone length and dimension has been observed to find the extend of vortex core and turbulence intensity growth. Several parametric studies have been made to find the best simulation results among various turbulence models like k-ε and SST Reynolds stress model. Major parametric comparisons like pressure, velocity, and TKE changes have been studied. Finally, through the analysis results, it has been evident that by changing the swirl vane angle and swirl number the recirculation zone's length and vortices has been increased and improves the mixing characteristic.

Keywords: Swirl, CFD, TKE, Reynolds stress.

1. Introduction

The jet engine combustion chamber is the part of a jet propulsion system which mainly determines the efficiency and reliability of operation of the engine. The enhanced aerodynamic mixing process taking place inside the combustion chamber due to swirler are complex. The construction of the combustion chamber of a jet propulsion engine is an extremely difficult task due to the nature of smaller diameter and in length. The better progress of solution to this problem requires an understanding of the mixing processes taking place in a combustion chamber. The combustion chamber has to work within the conventional system of a jet propulsion system. This superimposes upon the combustion chamber design efficiency and upon the system of the process of combustion in it, a succession of superior mixing characteristics. As to achieve this, a combustion chamber should be designed with a minimum dimension in a such a way to achieve greater efficiency in mixing the products with compressed air and finely atomized fuel.

The swirler used in the combustion chamber, prior to primary zone enhances the turbulence which eventually results in good combustion rate. In order to revamp with minimum dimensions with good mixing characteristic, the concept of doublet swirler with angle effect is studied and analysed in this paper with using CFD at various turbulence models.

2. Literature survey

There are several authors who have studied about the swirl vanes in the combustion chamber. But the multiple inlet swirler or the Doublet has been discussed by Prashant Singh et .al.21. He mainly focused on the study of multiple inlet swirler in which the flow is divided into multiple streams. Hence the performance can be improved efficiently. Multiple inlet swirlers results in complete combustion thus maintaining emissions and pollution. Further, Multiple inlet swirlers divides the flow into two streams– one stream is guided through the conventional path while the other stream is guided through multi tangential vanes. By referring various concepts related to multiple inlet swirler, he found that for same inlet mass flow, the swirl number can be varied in fixed swirl configuration. A. Gusseabet .al.1 performed the reactive flow analysis of gas turbine combustion system in his paper. Also, the comparison between 3D / 2D RSM and

SST Model is done. The results obtained shows that 3D URANS RSM model provides more confined vortex cone, high velocity, overestimates the negative circumferential velocity by approximately 5% and also captures the intensity of flow better than the 3D URANS SST model. The main difference between 3D URANS RSM and SST k-ω is the circumferential velocity profile. Khairul Fikri Tamrinaet .al.10 explained that the swirl flows are generally turbulent in nature. CFD analysis is carried in the 2D axisymmetric channel. The results produced in both second order and third order equations are almost similar. Both two equation k-ε model and five equation Reynolds Stress model are used. The physical model used consists of burner geometry divided into two parts – Swirl generator and Combustion chamber. Swirl generator produces two coaxial jets – axial and annular swirling jet. The main difference between k-ε model and RSM model differs in the ability to predict turbulence level. P.Muthukumar et.al.21 studied the flow through axial swirler. It is important that the design of swirler is required to enhance the mixing characteristic and to minimize the NOx emissions. In this swirl vanes of four different angles 300, 400, 500 and 600 were analyzed and the results were compared. The swirl vanes were analyzed in two different turbulence models of Reynolds Stress Model (RSM) and k-ε. Straight vanes are easy to manufacture but Curved vanes produce a better aerodynamic performance. The analysis results show that as Reynolds number increases axial velocity increases, turbulence intensity increases and inlet Kinetic energy also increases.

3. Methodology

The basic geometry design of the Doublet swirler has been designed using tool CATIA. Further, the geometry is meshed using the tool ANSYS-ICEM CFD and Aerodynamic analyze is carried out in the simulation software called ANSYS-CFX. Finally, the results are extracted from post-processing tool.

4. Geometry details

The doublet swirler consists of a central hub of diameter 40 mm with a total length of 104 mm. A total of eight axial vanes surrounds this hub inscribed in a circle of diameter 102 mm. The axial vanes start at a distance of 80 mm from one end of the hub. Each axial vane starts from the origin and extends to a length of 51 mm and thickness of 4 mm. The depth of each axial vane is 20 mm. These axial vanes are covered by a cylindrical tube of outer diameter of 104 mm and thickness of 1mm. This outer cylindrical tube extends to a length of 80 mm above the axial vanes a circular layer of outer diameter 160 mm and inner diameter 102 mm is placed whose depth is about 2 mm. The tangential

vanes of the doublet swirler are placed above this circular layer. The number of tangential vanes present is also eight. Each tangential vane is inclined to an angle of about 75^0. The length of a tangential vane is about 45 mm and thickness is 4 mm. The depth of each tangential vane is about 20 mm above the tangential vanes a circular layer is placed with the same dimensions as the previous layer. Similarly, the Doublet is twisted with an angle of 54°. The twisted model is also designed with same parts and same dimensions. But both the axial vanes and tangential vanes are twisted by inclining the trailing edge of the vanes to an angle of about 54^0.
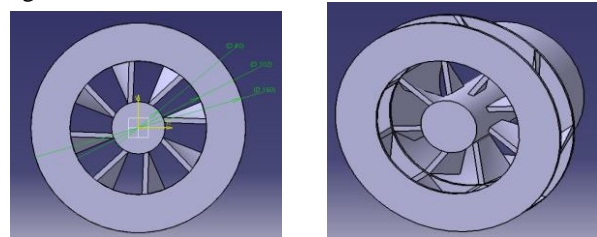

Figure 1. Doublet Swirler 54^0 twisted (Front view and 3D view)

4.1 Domain Details

The doublets are simulated inside a domain of particular parameters. The domain for swirler consists of three parts. They are inlet tube, pre-chamber and combustion chamber. The inlet tube surrounds the axial vanes of the doublet swirler. The diameter of the inlet tube is about 104.1 mm and length of about 150 mm. Secondly, the pre-chamber starts after the axial vanes with a circle of diameter 160 mm and extends to about 50 mm length ending in a circle of diameter 250 mm. The combustion chamber continues after the pre-chamber and extends to a length of 750 mm.

4.2 Grid Details

ANSYS Meshing is a discretization tool. Grid independence study has been carried out and meshing has been made to the doublet with nodes and element as 460662 and 2614729 for 90°, 461169 and 2616519 for 54° respectively. The mesh quality parameter skewness and orthogonality are kept at optimal level as per standards.

5. Turbulence Models

The four different turbulence models have been analyzed for the Standard Doublet and the Twisted Doublet. They are the k-ε standard, k-ε RNG, BSL Reynolds stress and SSG Reynolds stress. The k-ε model is studied to predict free-shear layer flows with considerably small pressure gradients.

6. Boundary Conditions

The analysis of both the models is carried out in CFX solver. Boundary conditions for inlet, outlet and wall is set as follows: Inlet is set with an average combustion chamber velocity of 60m/s. Wall is operated as a No slip wall. The outlet is adjusted to Average static pressure and zero Pascal pressure. Double precision solvers are used with High order resolution.

7. Results and discussion

The doublet swirler with standardized and twisted configurations is designed and simulated in ANSYS, post-processing has been made in order to compare the best performance among the two swirler design. The significant factor of a swirl vane is Turbulence that is analyzed and compared to four different turbulent models. These models show the Formation of the Recirculation zone around the combustion chamber. It is evident that the BSL RSM Turbulence model produces the highest number of turbulent recirculation zone. Such streamlines are simulated with standardized and twisted doublet. Further, the turbulent kinetic energy, velocity, pressure parameters are correlated and the results are sketched in the CFX graph in order to know the finest turbulent model.

7.1 Effect of Streamline with respect to various turbulent models to 90^0 Doublet swirler

In this model, two recirculation zones are formed along either side of the combustion chamber at length of about 50 mm and 325 mm from the swirler and both are almost symmetrical in figure 2.
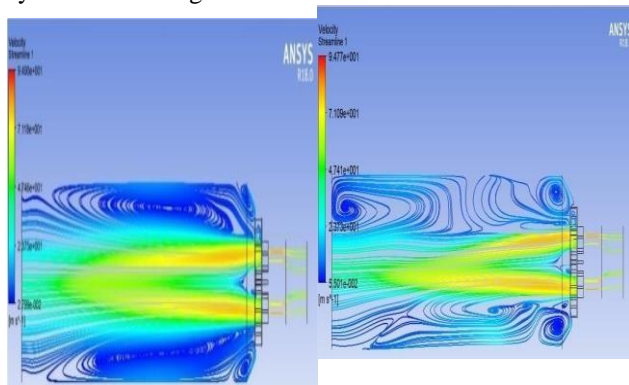

Figure 2. k-ε Turbulence model at 90°.   Figure 3. k–ε RNG Turbulence model at 90°.

Here, the recirculation zones are formed randomly on either side that is observed in figure 3. The first recirculation zone is formed at a distance of 55 mm from

the swirler. The other recirculation zones are formed at 200 mm and 760 mm from the swirler.
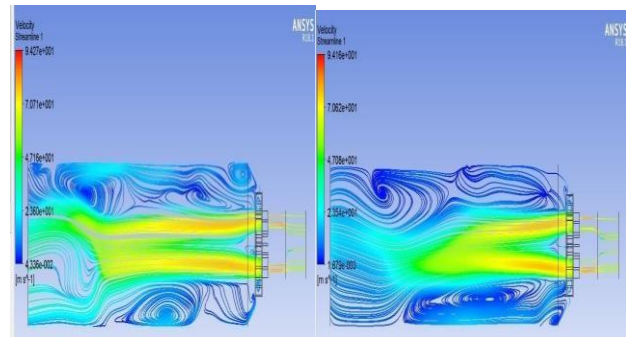

Figure 4. BSL RSM Turbulence model at 90°.   Figure 5. SSG RSM Turbulence model at 90°.

In this model, 8 recirculation zones are formed randomly in figure 4. First set of recirculation zone is formed near the tangential vanes of swirler and at the mid of domain. The other recirculation zones are formed at about 55 mm, 100 mm, 175 mm, 340 mm and 650 mm from the swirler. From figure 5 we witness the recirculation zones are formed along the tangential vanes, middle of domain and at about 250 mm and 675 mm from the swirler randomly.

7.2 Effect of Streamline with respect to various Turbulence models to 54^0 Doublet swirler

In this model from figure 6, almost 3 asymmetric recirculation zones are formed at about 150mm, 275 mm and 675 mm from the swirler. Here, recirculation zones are formed at about 50 mm, 100 mm, 125 mm and 780 mm from the swirler. And all these recirculation zones are asymmetric and random.
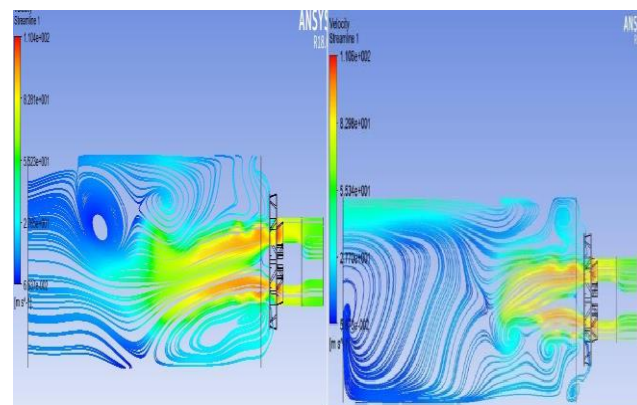

Figure 6. k–ε Turbulence model at 54°.   Figure 7. k–ε RNG Turbulence model at 54°.

Here, 5 random oriented recirculation zones are formed randomly at about 55 mm, 100 mm, 130 mm, 275 mm and 650 mm from the swirler in figure 8.
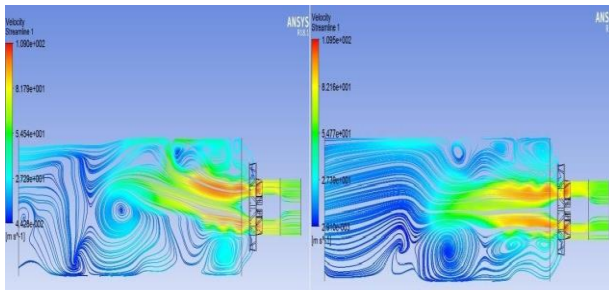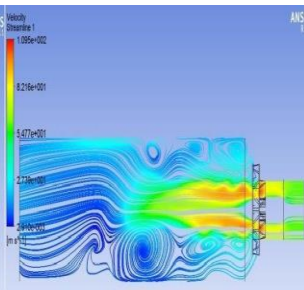
Figure 8. BSL RSM Turbulence
model at 54°.

Figure 9. SSG RSM Turbulence
model at 54°.

Here, 7 asymmetric recirculation zones are formed and that is observe in figure 9 at length about 50 mm, 60 mm, 85 mm, 120 mm, 150 mm, 280 mm and 400 mm from the swirler.

8. Conclusion

The evident result obtained from the CFD Analysis on a different angle of swirl vane with various Turbulence models. That, the result predicted by BSL RSM Turbulence model agrees best with the other turbulence models with the comparison of parameter velocity, pressure, and TKE. Adding to that, the value of pressure drop is greater in angle 540 model compared to angle 900 model. The velocity value of angle 540 modelhas a large velocity drop than angle 900 model. The highest value of TKE for both angle 540 and angle 900 model is almost same. Ultimately the angle 540swirler model is more efficient in mixing characteristic and better than angle 900model.

References

[1] Guessab A, TahaJanan M, Mokhtar M Aris A 2017 Simulation of Turbulent Swirling Flame in 3-D Can Combustion Chamber using 2D and 3D URANS RSM/SST. 2,11-14

[2] Douglas L Straub Geo Richards A 1999 Effect of axial swirl vane location on combustion dynamic. 2, 99-109

[3] Hiregoudar Yerrennagoudaru Dr Shiva Prasad desai 2014 Inlet Air Swirl On Four Stroke Single Cylinder Diesel Engine Performance. 2, 2347-6435

[4] Lavanya D, Raghunathan S, Jagadeesh N 2016 Computational Fluid Dynamic Analysis of Swirl Vane Gripper. 6,1313-1325

[5] HolgerWerschnik, Marius Schneider, Janina Herrmann, Dimitri Ivanov, Heinz-Peter Schiffer ChristophLyko 2012 The Influence of Combustor Swirl on Pressure Losses and the Propagation of Coolant Flows at the Large Scale Turbine Rig (LSTR) Experimental and Numerical Investigation. 2,12

[6] Dombard J, Poinsot T, Moureau V, Savaryk N, Staelbach G, Bodoc V 2012 Experimental and numerical study of the influence of small geometrical modifications on the dynamics of swirling flows. 4,469-482

[7] KhairulFikriTamrina, Nadeem Ahmed Sheikhb, BahbibiRahmatullaha 2012 Numerical analysis of swirl intensity in turbulent swirling pipe flows. 78, 5-10

[8] Manshakumari, Shah Jagruti, Arvind.S.Mohite 2014 Experimental analysis of flow through rotating swirler with effect of guide vane. 3,14958-14964

[9] Mansha kumara 2014 CFD Analysis of Flow Through Rotating Combustion Swirler. 5,820- 823

[10] Mohamad Shaiful AshrulIshaka, Mohammad NazriMohd Jaafarb 2014 Effect of Velocity Variation at High Swirl on Axial Flow Development inside a Can Combustor.71,19-24

[11] Mark Turrell D, Philip Stopford J 2004 CFD simulation within and downstream of the high swirl lean premixed, gas turbine combustor.4,31-38

[12] Muthukumar P, Balakrishnan S R 2013 CFD Analysis of recirculating flows induced by Axial Swirler. 2,86-90

[13] NaitikGor H, Milan Pandya J 2014 CFD Analysis of Swirl Can Combustion Chamber. 1,52-58

[14] Naveen J, Krishnasairam K, Sureshratankumar M, Vinayababu Y, Jayadeep P 2017 Swirl velocity visualization by the less in combustion chamber. 8,451-456

[15] Prasanna gouda, Srinivasan, Sivaramakrishnan, Shashishekar S 2016 CFD analysis of gas turbine combustor primary zone using different axial swirler configurations. 4,35-38

[16] Prashant Singh, SomeshMurarka, KalpitKaurse P, Brijesh Patel 2015 Review of Swirl Generator Used In Gas Turbine Combustor. 2,811-816

[17] Prashant Singh, SomeshMurarka, 2015 Comparative Analysis of Mixing Using CFD for Various Swirl Number. 1,111-120

[18] Raj Kumar A, Dr.JanardhanaRaju G Dr.Hemachandra Reddy K 2016 Experimental investigation of in-cylinder air flow to optimize number of guide vanes to improve performance and emissions of di diesel engine.3,2800-2805

[19] Thundilkaruppa raj R, Ganesan V 2009 Experimental study of recirculating flows induced by vane swirler. 16,14-22

[20] Thundilkaruppa raj R, Ganesan V 2008 Study on the effect of various parameters on the flow development behind vane swirlers. 47,1204-1225

[21] Teresa Parra, Ruben Perez, Miguel Rodriguez A, Francisco Castro, Robert Szasz Z, ArturGutkowski 2015 Numerical Simulation of Swirling Flows - Heat Transfer Enhancement. 2,1-6

[22] WazeemNishad, Shamnadh M 2016 CFD Analysis of a Noncontact Gripper with Rotor. 2,1069-1072

[23] YehiaEldrainy A, Mohd Fairuz bin Ahmad, Mohammad NazriMohdJaafar 2009 Investigation of Radial Swirler Effect on Flow Pattern inside a Gas Turbine Combustor,3,21-30