Computational Analysis on Cold Flow in Swirler by Aerodynamics Properties

Amit Kumar[1] Vinay Yadav[2]
[1]Research scholar [2]Associate Professor and HOD
[1,2]Department of Mechanical Engineering
[1,2]AISECT University, Bhopal(M.P.)

Abstract— At mass flow rate of 0.5 kg/s and equivalence ratio are 0.7 to 0.8 ,the cold flow of CFD (computational fluid dynamics)analysis in swirler is done at 400, 500 and 620 are respectively of vane angle. Also find out aerodynamics flow properties in swirler at respective vane angles. At vane angle of 400, 500 and 620, the swirler numbers are 0.6, 0.9 and 1.3 respectively is calculated.So purely CFD analysis of aerodynamics properties of flow in swirler at various mentioned vane angle is investigated and to be designed improvement of swirler with high efficiency of aerodynamics mixing of air and fuel. To support experimental design with CFD analysis will be the key role to predict the improvement of performance and efficiency of swirler.

Key words: Swirler, CFD, recirculation zone

I. INTRODUCTION

Many different types of airflow patterns are employed, so introduce new aerodynamics flow pattern through swirler to improve performance of gas turbine combustor such as tubular (can type) and annular combustor. As per combustor view of gas turbine engine, we have to look after the swirler performance and its design. Looking of swirler design and its performance to do the improvement in its aerodynamic properties of its expected geometry and CFD analysis. So improve design of swirler as improve its performance, required new technique rather than experimental, called CFD analysis of flow in swirler.

To do CFD analysis of swirler, to improve design and its performance, so become strong swirler as aerodynamically. To do CFD analysis of various geometry of different swirler vane (such as 40 degree, 50 degree, 62 degree etc.) at suitable swirl number. As we know swirl number more than 0.6 is good flow mixing in swirler. In CFD analysis of swirler flow, we have to calculate velocity contour and turbulent contour to improve design as experimentally.

II. DESIGN PARAMETER OF SWIRLER

In design parameter include swirler geometry dimension (40o, 50o and 62o), cfd flow condition , boundary condition and swirler number calculation with various vane angle are done.

A. *Flow Condition in Swirler:*

Tinlet(Tempeture at inlet) = Toutlet (Temperature at outlet)= 288 K
Pinlet (Pressure at inlet) = 1 bar
Poutlet(Pressure at outlet) = 50 K Pa
ṁ(Mass flow rate at inlet or oulet) = 0.5 kg/s
ϕ(Equivalence ratio) = 0.7 to 0.8
Fuel= CH_4 (Methane) or White Kerosine ($C_{12}H_{23}$)

B. *Assumptions And Boundary Conditions Of Swirler:*

Condition	Value
Flow regime	Subsonic
Pressure	1 bar
Temperature	288 K
Flow direction	Normal to boundary
Heat transfer	Adiabatic
Wall condition	No slip
Turbulence Intensity	10%

Table 1: For Inlet Boundary Condition Of Flow Parameter As Follows:

Condition	Value
Flow regime	Subsonic
Pressure	50 k Pa
Temperature	288 K
Turbulence Intensity	15%

Table 2: For Outlet Boundary Condition of Flow Parameter as Follow In Table:

C. *Geometric Dimension of Swirler:*

D(Dsw)(outer diameter) = 38 mm
d(DHub)(inner diameter) = 18 mm
Thickness of vane = 1.5 mm
Height of vane = 7 mm
Length of vane = 20 mm
Number of vane = 8
Vane Angles ( ) = 40o, 50o, 62o
LSWIR(length of swirler) = 25 mm
LDOME (length of dome) = 28 mm
DDOME (Diameter of dome) = 52 mm

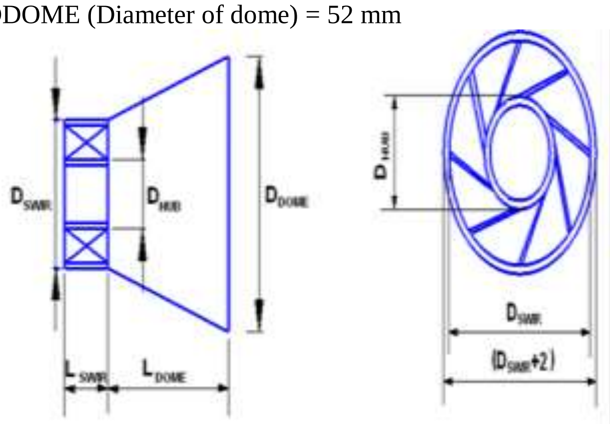

Fig. 1: Cross-sectional view of SWIRLER

D. *Calculations OF Swirler Number:*

$$S = \frac{2}{3}\frac{1-\left(\frac{D_{HUB}}{D_{SWIR}}\right)^3}{1-\left(\frac{D_{HUB}}{D_{SWIR}}\right)^2}\tan\theta$$

Where, S = swirler number
= outlet angle of vane
= diameter from center line of swirler
= diameter of outer of swirler

d/D = 0.47

The different vane angle of swirler numbers are as follows:

S40 = 0.6, S50 = 0.9, S60 = 1.3, S62 = 1.4, S68 = 1.9

*Note: - swirl number of 0.6 is a good mixing for swirler. Angle will increase swirl number will increase, it's a sign of good swirler but at 68 degree angle there will be geometrical obstacle.

III. MODELING OF GEOMETRY

In geometry modeling , we shall see about geometry analysis of swirler design .As we know Radial swirler are more efficient than Axial swirler due to swirl effect of flow and better aerodynamics properties , there will be good recirculation zone to swirl a flow to excellence performance. Per haves our selection is radial swirler. In radial swirler outlet vane will vary the angle with horizontal line of axis or center line of swirler and inlet vane angle starts with is always zero. Here the geometry of 40o, 50o and 62o are outlet vane angle of swirler.

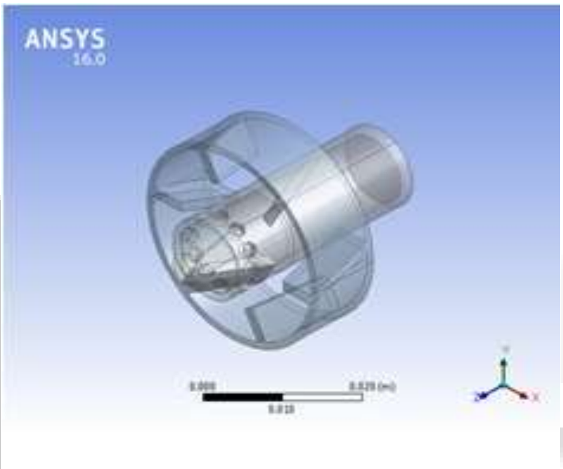

Fig. 2: Isomeric view of Import geometry in ANSYS at 40o

So similarly rest of the geometry draw and processed for analysis.

IV. MESHING THE SWIRLER

In meshing part, as mentioned above and shown geometry such as 40o, 50o and 62o are each meshed as unstructured tetrahedral mesh shaped. So figure of meshing sample is shown in Fig 3.

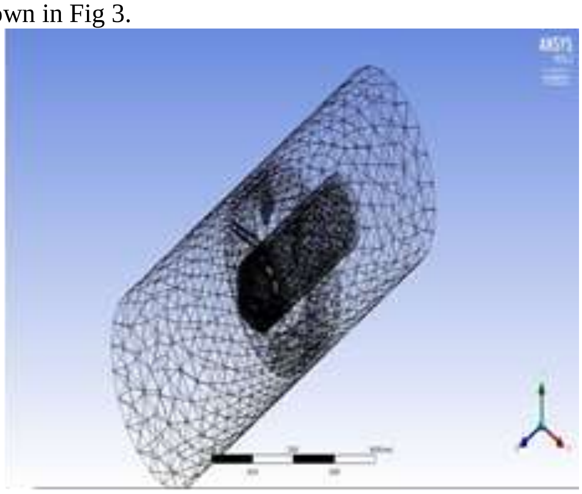

Fig. 3: Messing of the SWIRLER geometry

A. *Grid Details:*

The three dimensional flow regions along with swirler with appropriate swirl angle were modeled using software solidwork. Three dimensional unstructured grids was generated using tetrahedral mesh in ANSYS work bench. The grid cells were refined in the critical regions, like swirler inlet and exit, in anticipation of high velocity and pressure gradient. The solutions were predicted by using ANSYS CFX 16. The grid detail is mentioned in table 3.

Swirler angle	Nodes	Elements
40^o	5963	26437
50^o	5923	26057
62^o	6054	26450

Table 3: Meshing details as different swirl angle.

V. CFD FLOW ANALYSIS IN SWIRLER

CFD analysis of aerodynamics flow properties in swirler with various vane angle with various contour as follows: at 40o, 50o and 62o are respective.

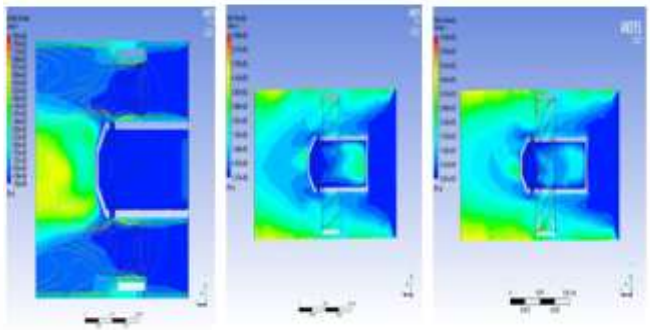

Fig. 4: Viscosity contour for van angle at 400, 500 & 620 respectively.

As Viscosity contour for various vane angle above pasted. The above pasted contour are CFD analyzed flow analysis in swirler with various aerodynamics aspects like more suitable turbulent zone as well as proper recirculation zone should be created to make proper mixing of air-fuel to improve better performance and efficient. Analysis says that as we go increase vane angle more turbulence zone will created as we know swirler number more than 0.6. Viscosity contour is turbulent analysis in ANSYS software.

VI. RESULTS & DISCUSSION

After analysis, will shown detail with performance chart with various vane angle and flow analysis of aerodynamic phenomena in swirler. So in chart-1 as viscosity contour performance for vane angle 400, says about velocity of flow in recirculation zone, similar chart-2 and chart-3 for vane angle 500 and 620 are plotted to say about performance of swirler and aerodynamics of various vane angle as mentioned.

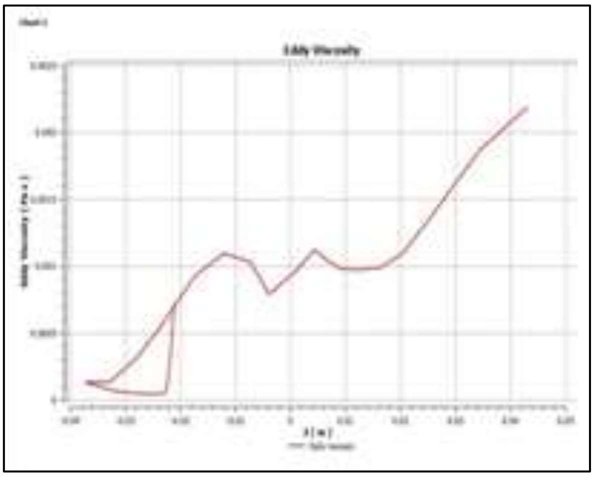

Chart 1

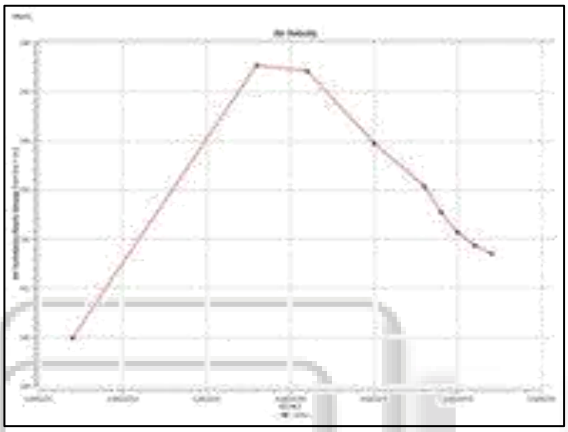

Chart 2:

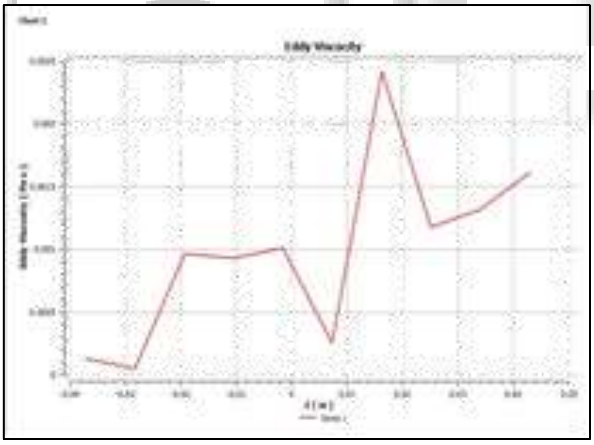

Chart 3:

VII. CONCLUSION

As per various outlet vane angle flow analysis says about it aerodynamic flow characteristics includes various turbulence zone, recirculation zone etc. Once vane angle increase flow properties increase as per our above analysis contour or charts says and experimental prediction through complete CFD analysis. CFD analysis is improved design and performance and to make new efficient design that give strong aerodynamics flow properties of swirler.

REFERENCES

[1] Shah Jagruti, Mansha Kumari, Arvin. S. Mohite ,” Experimental Analysis of Flow through Concentric Vane Swirler in Combustion Chamber Using Atmospheric Air”IJSTA ,vol.-2, issue 1, pp-15-19,2016.
[2] Han Seo, Seong Dae Park, Seok Bin Seo, Hyo Heo, In Cheol Bang ,” Swirling performance of flow-driven rotating mixing vane toward critical heat flux enhancement “IJHMT p-89,2015.
[3] WANG Xiao feng, LIN Yuzhen, ZHANG Chi, TIAN Xing peng ,” Effect of Swirl Cup’s Secondary Swirler on Flow Field and Ignition Performance”JTS, vol 24, 2015.
[4] Alex Borsuk ,” Swirler Effects on Passive Control of Combustion Noise and Instability in a Swirl-Stabilized Combustor” ASME, vol 137, 2015.
[5] Martin Miltner, Christian Jordan, Michael Harasek, “CFD simulation of straight and slightly swirling turbulent free jets using different RANS-turbulence models “Elsevier ATE xxx , p-1-10 ,2015.
[6] M. T. Parra-Santos, V. Mendoza-Garcia, ,” Influence of Flow Swirling on the Aero thermodynamic Behavior of Flames ” CESW, Vol. 51, No. 4, pp. 424–430,2015.
[7] Naitik H. Gor, Milan J. Pandya,” CFD Analysis of Swirl Can Combustion Chamber”IJIRST,vol 1, issue 2, 2014.
[8] Arabi Radwan, Kamal A,” RANS Modeling of Unconfined Swirl Flow “JST ,vol. 143,2014.
[9] Abhilash M , “Study of Swirl and Tumble Motion using CFD”, IJTARME, Volume-2, Issue-1, 2013.
[10] Anusha Rammohan Aditya Bhakta, Vinay Natrajan, “Flow swirl and flow profile measurement in multiphase flow” IJTE vol 2 issue 1, 2012.
[11] S. Mahalingam, M. K. Koyithitta Meethal, A. Banerjee, W. Basu, and H.K. Pillai. “Electrical network representation of a distributed system”. US Patent 8264246. General
Electric Company. 2012.
[12] Lefebvre, A.H., Gas Turbine Combustion.

