Computational and Cold Flow Analysis on Swirler at Vane Angle 55^0 & 62^0

Amit Kumar
Research Scholar, Dept of Mechanical Engineering, AISECT University,
Bhopal (M.P.) India.

ABSTRACT
This paper brings the computational study on geometric swirler of outlet vane angle of 55^0 and 62^0 for aerodynamic flow properties and cold flow analysis to be done. At mass flow rate of 0.3 to 0.4 kg/s and equivalence ratio are 0.6 to 0.7 ,the cold flow study on CFD (computational fluid dynamics)analysis in swirler is done at 55^0 and 62^0 are respectively of outlet vane angle and inlet vane angle are usually zero. Also will have found aerodynamics and geometric flow properties in swirler at 55^0 and 62^0 vane angles, the swirler numbers are 1.2 and 1.4 respectively is calculated. So purely CFD analysis of aerodynamics properties of flow in swirler at various vane angle(such as 55^0 and 62^0) 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.

Keyword: Swirler, CFD, recirculation zone, Atomization flow properties in swirler .

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 e, 55^0 & 62^0) 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 (55° and 62°), cfd flow condition , boundary condition and swirler number calculation with various vane angle are done.

(a) Flow condition in swirler

T_{inlet}(Tempeture at inlet) = T_{outlet} (Temperature at outlet)= 300 K
P_{inlet} (Pressure at inlet) = 1 bar
P_{outlet}(Pressure at outlet) = 60 K Pa
□(Mass flow rate at inlet or oulet) = 0.3 kg/s
□(Equivalence ratio) = 0.6 to 0.7
Fuel= CH_4 (Methane) or White Kerosine ($C_{12}H_{23}$)

(b) Assumptions and Boundary conditions of swirler

Table-1 for Inlet Boundary Condition of flow parameter as follows:

Condition	Value
Flow regime	Subsonic
Pressure	1 bar
Temperature	300 K
Flow direction	Normal to boundary
Heat transfer	Adiabatic
Wall condition	No slip
Turbulence Intensity	8%

 ISBN: 2278-4187

Table-2 for Outlet Boundary Condition of flow parameter as follow in table:

Condition	Value
Flow regime	Subsonic
Pressure	60 k Pa
Temperature	300 K
Turbulence Intensity	10%

(c) Geometric formation:

$D(D_{sw})$(outer diameter) = 38 mm
$d(D_{Hub})$(inner diameter) = 18 mm
Thickness of vane = 1.5 mm
Height of vane = 8 mm
Length of vane = 20 mm
Number of vane = 8
Vane Angles (θ) = 55°, 62°
L_{SWIR}(length of swirler) = 25 mm
L_{DOME} (length of dome) = 30 mm
D_{DOME} (Diameter of dome) = 50 mm

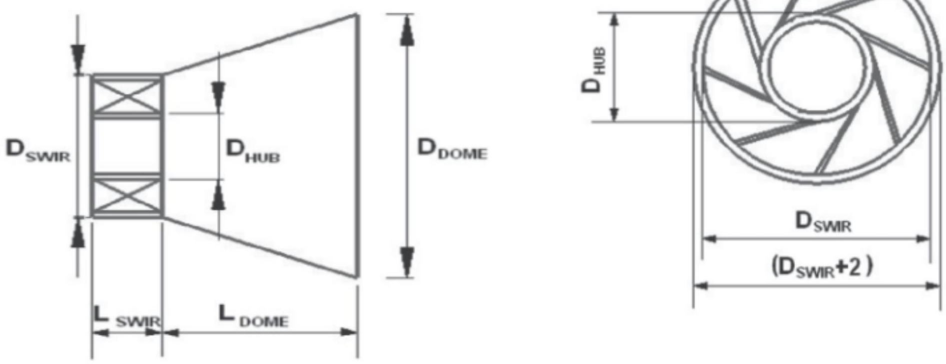

Fig. 1:- Cross-sectional view of SWIRLER

(d) Calculations of Swirler Number

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

Where,

S = swirler number
θ = outlet angle of vane
D_{HUB} = diameter from center line of swirler
D_{SWIR} = diameter of outer of swirler
d/D = 0.47

The different vane angle of swirler numbers are as follows:

S_{40} = 0.6, S_{50} = 0.9, S_{55} = 1.2, S_{60} = 1.3, S_{62} = 1.4, S_{68} = 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 55° and 62° are outlet vane angle of swirler.

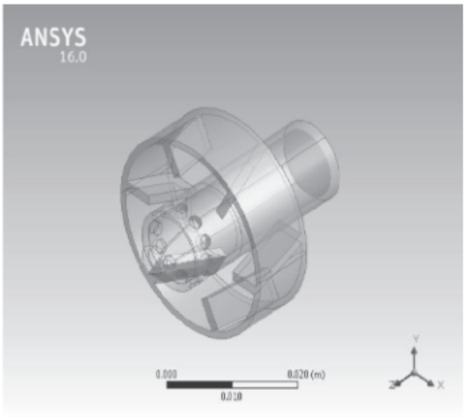

Fig. 2:- Isomeric view of Import geometry in ANSYS at 55°

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 55° and 62° are each meshed as unstructured tetrahedral mesh shaped. So figure of meshing sample is shown in Fig 3.

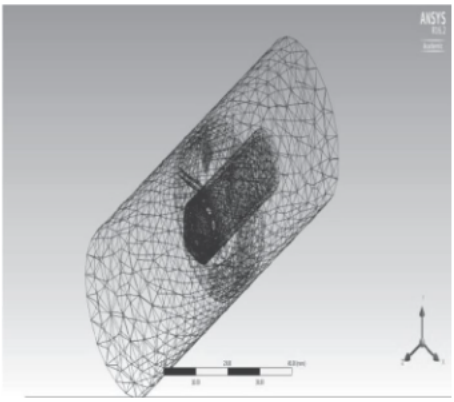

Fig. 3 :- Messing of the SWIRLER geometry

(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.

Table3:
Meshing details as different swirl angle

Swirler angle	Nodes	Elements
55°	5823	25057
62°	6054	26450

V CFD AND COLD FLOW ANALYSIS FOR VARIOUS CONTOUR

CFD analysis of aerodynamics flow properties in swirler with various vane angle with various contour as follows: at 55° and 62° are respective.

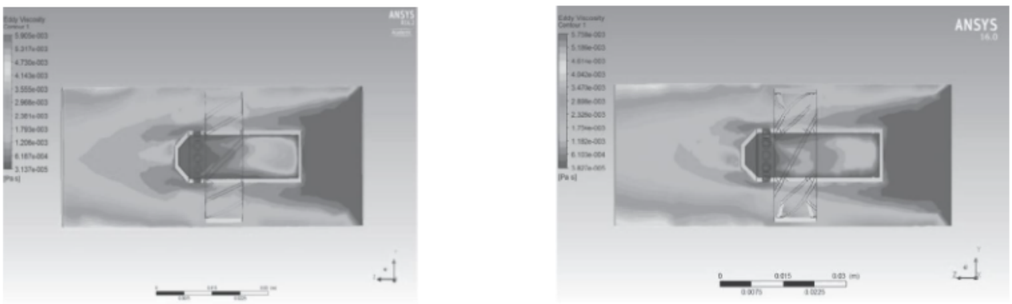

Fig 4: Turbulence contour for van angle at 55^0 & 62^0 respectively.

As Turbulence 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.

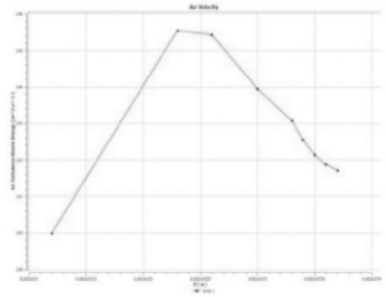

Chart -1

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 55^0, says about velocity of flow in recirculation zone, similar and chart-2 for vane angle 62^0 are plotted to say about performance of swirler and aerodynamics of various vane angle as mentioned.

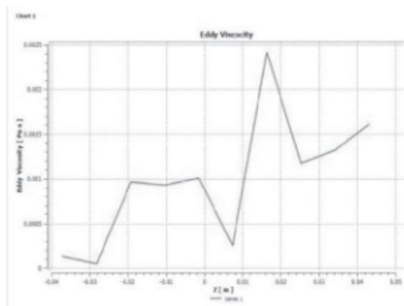

Chart -2

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.*