Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 07, July - 2017

CFD Analyses of the Experimental Setup of a Slinger Combustor

Somanath K Bellad[1],
[1]M Tech Student,
Siddaganga Institute of Technology (SIT),
Tumakuru, Karnataka

T. K. Tharakeshwar[2]
[2]Professor,
Department of Mechanical Engineering,
SIT, Tumakuru, Karnataka.

Abstract: **An annular combustor with rotating fuel injection system is called a slinger combustor, which has been widely employed in small gas turbine engines. In the present study, cold flow CFD analysis of 120^0 sector of a slinger combustor, with inlet and exhaust adaptations required for its experimental characterization, satisfying the periodicity conditions has been carried out for three different test rig conditions. Total pressure loss, variation of velocity components along the measurement planes and mass flow through various features of combustor has been estimated.**

Keywords: Experimental setup, Rotary fuel injection system, Slinger combustor.

NOMENCLATURE

DH	Dome Holes
IAH	Inner Annulus Hole
k	Turbulent Kinetic Energy, m^2/s^2
NGV	Nozzle Guide Vanes
OAH	Outer Annulus Hole
ε	Rate of Dissipation of Turbulent Kinetic Energy, m^2/s^3

INTRODUCTION

In an aero engine, combustor is one of the modules which should withstand high temperature of the hot gases; hence the service life is short. Traditionally, gas turbine combustor development is carried out using experiments, which is time consuming and expensive. Of late, as powerful computational platforms are being realized on a continuous basis, feasibility of using CFD analyses is undoubted for combustor design. Historically, the utilization of gas turbine engine has been in the large scale aircraft applications where the advantages are being high Power to Weight ratio and reliability and its disadvantage being high cost. Advancement in the Technology have made the use of gas turbine engine in the lower power applications like UAV's, cruise missiles, remotely piloted vehicles because of their simplicity and low cost. As a result, there is large demand for small gas turbine engines. Small turbine engines differ from large turbine engines not only in overall requirements, but also in severity of problems associated in achieving these requirements. Basically, difficulty is in geometric scaling. As the power requirement is reduced, the dimensions of the required components of engine also reduce. As a result, compromise must be made which reduce the achievable efficiency and increases the difficulty in achieving the required performance, durability and reliability [[1]].

Small gas turbines are powered by slinger combustors which incorporate rotating fuel orifices or rotary injectors, mounted on high speed rotating engine shaft. Usually the orientation and number of orifices depends on the requirement. These take advantage of centrifugal force of the high speed of rotating engine shaft for atomizing the fuel, thus requires no atomizer. Advantages of slinger combustor is low cost, requires low pressure fuel pump and fuel is more uniformly sprayed into the combustor, leading to more uniform temperature distribution at combustor exit.

Seongman Choi and Donghun Lee [[2]] have proved that the spatial distribution of fuel droplets depends on penetration of fuel spray into the primary zone; also they proved that better ignition could be achieved through increasing the rotational speed and air mass flow rate and the combustion efficiency is smoothly increased from 99% to 99.6%. Bhawan B Patel, Oleg Morenko [[3]] produced the hybrid slinger combustor, which combines two different fuel injection systems, the rotary fuel slinger for spraying fuel in high power requirements and the set of fuel nozzles spraying fuel in the lower power requirements, since the fuel atomization by the rotary fuel slinger at low speed is not satisfactory.

Dahm[[4]] conducted experiments on variety of slinger geometries over a range of operating conditions. Various cross sections of orifice hole like round, square and rectangular slot with round end were subjected to experiments. It was found that, the hole size affects the liquid film thickness and the cross sectional shape has significant effect on the atomization performance. Carmen Sescu, Bogdan R Kucinschi, Abdullah A Afjeh [0] investigated combustor with different dimensions of the atomizing holes (0.9, 2 &3.2mm). The visualization of the liquid flow patterns at the exit indicates that larger hole diameters lead to formation of smaller fuel droplets at lower rotational speeds. Lower flow rates lead to smaller fuel droplets at lower rotational speeds. They concluded that use of larger diameter holes is beneficiary for smaller droplets at lower speeds and lower flow rates. Seong Man

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

Choi, Seong Ho Jang, Dong Hun Lee &Gyong Won You [[6]] shown that the length of liquid column from orifice is reduced with increasing rotational speed. The droplet size, velocity and spray distribution were measured from PDPA (Phase Doppler Particle Analyzer). The spray is visualized using high speed cameras.

Seong Man Choi, Seong Ho Jang [[7]] studied the disintegration of fuel by rotary injector. Diameter of injection orifices was varied from 1 to 5 mm, and the injection orifices were varied from 3 to 12 in number. The droplet size, velocity and spray distribution were measured using PDPA and visualized using Nd-Yag laser flashed photos. The speed of shaft was varied from 10000 to 40000 rpm. It was observed that the length of liquid column decreased with an increase in the shaft speed and with the injection orifice diameter greater than 3mm, the liquid film was spread over wide area and disintegrated into smaller droplets. They also founded that, there is an optimum orifice size to obtain minimum droplet size. With orifice sizes greater than optimum size, it was found that due to irregular spray patterns, small droplets cannot be produced.

CFD ANALYSIS METHODOLOGY

A. GEOMETRIC DETAILS

In the present analysis, 120° sector of slinger combustor with inlet and exhaust system is considered. The geometric model for inlet-exhaust system is created in Gambit 2.4. The inlet and exhaust systems are modeled to be periodic for 60° sector and the combustor for 40° and this makes the entire assembly to be periodic for 120° sector. Figure 1 shows the experimental setup of slinger combustor. The domain at exhaust has been extended to a length which is 20 times the exhaust outer diameter and atmospheric conditions have been specified as boundary conditions at outlets of the extended domain.

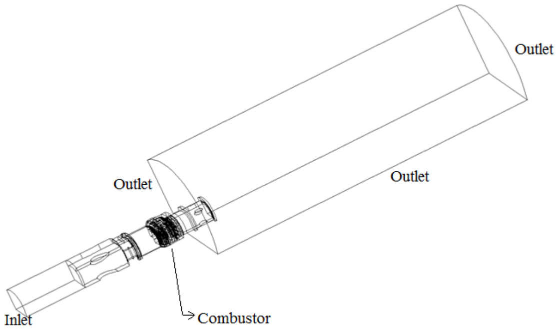

Figure 1: Slinger combustor assembly with inlet and exhaust system

B. GRID GENERATION

The accuracy and correctness of CFD analysis depends on quality of grid. The grid is generated in Gambit 2.4 for the entire model with the combination hexahedral and tetrahedral elements with a grid size of about 14.98 million cells. For combustor, about 98.8% of cells are having equi-size skew less than 0.6 and 99.9% of cells are having aspect ratio less than 20. For inlet-exhaust system, 96.48% of cells are having equi-size skew less than 0.4 and 95.5% cells are having aspect ratio less than 20.

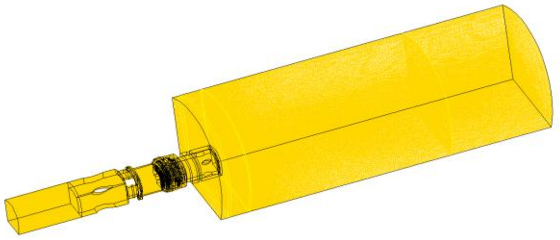

Figure 2: View of 120° slinger combustor grid

Figure 2 shows the grid generated for 120° sector of slinger combustor assembly in Gambit.

C. SOLVER DETAILS

For the present analysis ANSYS Fluent version 17.0 is used. Analysis is carried out considering the flow as steady, compressible and turbulent.

D. TURBULENCE MODEL

In the present analysis, the two-equation Realizable k-ε turbulence model is used which contains a new formulation for the turbulent viscosity. In Realizable k-ε model, new transport equation for the dissipation rate has been derived from an exact equation for the transport of the mean-square vorticity fluctuation [[8]].

E.BOUNDARY CONDITIONS

Boundary conditions specified at the inlet are Total pressure and Total temperature and at the exhaust, atmospheric pressure and temperature are specified. The wall adjacent to orifice and the injection orifices are considered as rotating wall and are rotated with suitable shaft speed. The extreme faces on either sides of circumferential direction are specified as periodic.

F. VALIDATION OF THE CFD CODE

Validation of the CFD code and analysis methodology has been carried out earlier by Srinivasa Rao. M. et al., [[9]].

G. RESULTS AND DISCUSSION

The results of CFD analysis of 120° sector slinger combustor assembly are detailed in this section. The pressure losses across the combustor for three different inlet Mach numbers have been estimated. Radial variation of velocity components have been plotted at inlet and exhaust measurement planes. The mass flow rate through various holes of combustor for three different test conditions has been estimated.

1. COMBUSTOR PRESSURE LOSS

The pressure loss across the combustor for three different inlet Mach numbers has been estimated and its variation is shown in the Figure 3.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

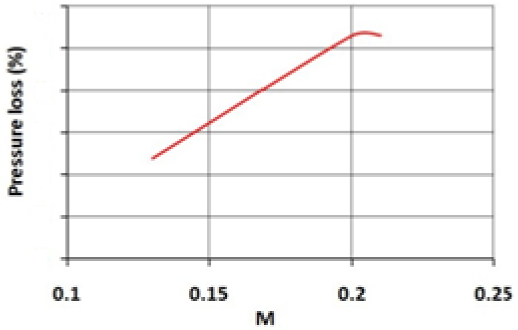

Figure 3: Pressure loss Vs. Mach number

It was observed that, the total pressure loss increases with an increase in the inlet Mach number.

2. VARIATION OF VELOCITY COMPONENTS ALONG THE MEASUREMENT PLANES

Two measurement planes, one at the inlet and the other at exhaust system have been considered for plotting the variation of predicted velocity components at three test conditions.

VARIATION OF AXIAL VELOCITY ALONG MEASUREMENT PLANE AT INLET

Figure 4 shows the radial variation of axial velocity along the centerline at measurement plane in the inlet system. The radial velocity gradually increases from zero at hub and reaches its peak and decreases near the tip. Further it can be observed that, axial velocity increases with an increase in the inlet total pressure.

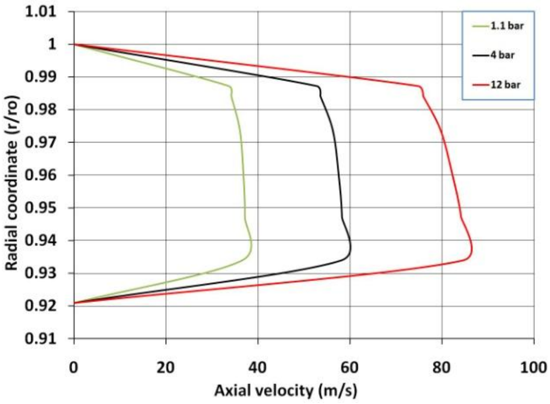

Figure 4: Axial velocity along inlet measurement plane

VARIATION OF TANGENTIAL VELOCITY ALONG MEASUREMENT PLANE AT INLET

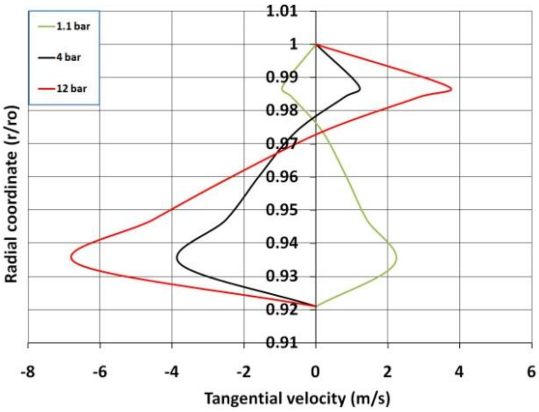

Figure 5: Tangential velocity along inlet measurement plane

Figure 5 shows the radial variation of tangential velocity along the centerline at measurement plane in the inlet system. It can be seen that, higher peak tangential velocity is observed for 12 bar case than that of others. Comparing the absolute values of velocities in figures 4 and 5, it can be stated that, the tangential velocity component is insignificant.

VARIATION OF VELOCITY MAGNITUDE ALONG MEASUREMENT PLANE AT INLET

Figure 6 shows the velocity magnitude along the centerline at measurement plane in the inlet system. It is seen that velocity magnitude increases with an increase in the inlet total pressure and the velocity magnitude is higher for 12 bar case compared to other cases.

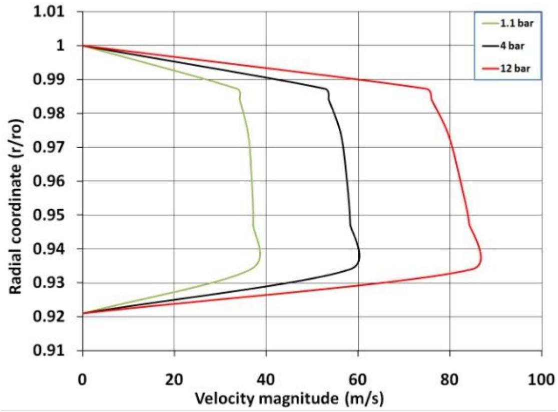

Figure 6 : Velocity magnitude along the inlet measurement plane

VARIATION OF AXIAL VELOCITY ALONG MEASUREMENT PLANE AT EXHAUST

Figure 7 shows the radial variation of axial velocity along the centerline at measurement plane in the exhaust system. The axial velocity gradually increases from zero and reaches peak value in all the three cases and diminishes. It can be seen that, higher peak axial velocity is for higher inlet total pressure.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

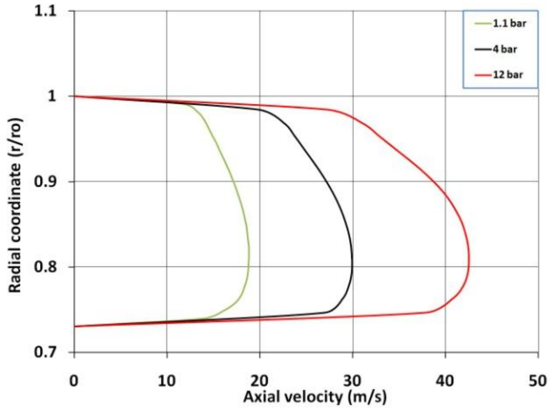

Figure 7 : Axial velocity along exhaust measurement plane

VARIATION OF TANGENTIAL VELOCITY ALONG MEASUREMENT PLANE AT EXHAUST

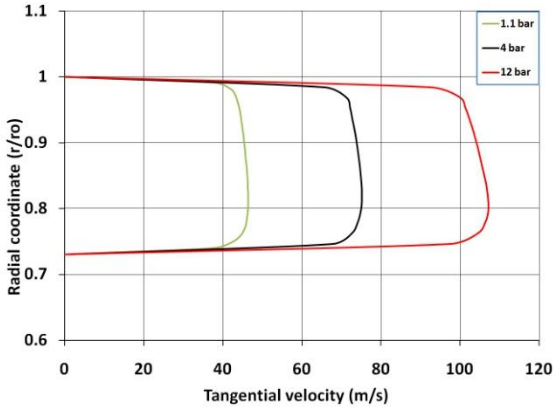

Figure 8: Tangential velocity along exhaust measurement plane

Figure 8 shows the tangential velocity along the centerline at measuring plane in the exhaust system. The tangential velocity increases with an increase in the inlet total pressure. It is evident from figures 7 and 8 that Tangential velocity component is more than the axial velocity component because of the swirl generated by nozzle guide vanes.

VARIATION OF VELOCITY MAGNITUDE ALONG MEASUREMENT PLANE AT EXHAUST

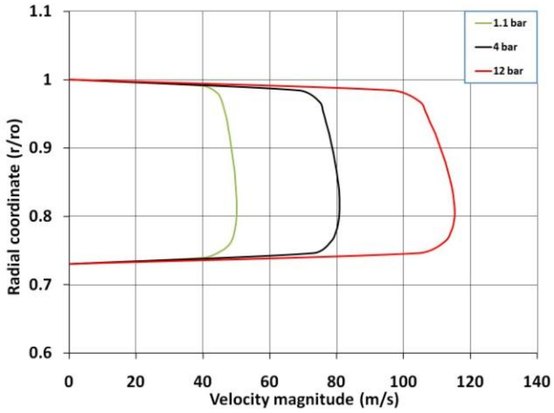

Figure 9: Velocity magnitude along the exhaust measurement plane

Figure 9 shows the velocity magnitude along the centerline of measurement plane in the exhaust system. Velocity starts from zero, gradually increases and again decreases near the walls because of no slip condition. Comparing figures 6 and 9, it is seen that velocity magnitude is higher at exhaust compared to inlet because of presence of NGVs at the exhaust.

3. MASS FLOW THROUGH VARIOUS FEATURES OF COMBUSTOR

Table 1: Mass flow through various features of combustor

SI No.	Features of combustor	% of inlet mass flow rate
		Cases 1 , 2 & 3
1	OAA	5.17 – 5.20
2	OAH2	0.20 – 0.24
3	OAH3	0.55 – 0.68
4	OAH4	0.62 – 0.69
5	OAH5	5.24 – 5.35
6	OAH6	1.67 – 1.68
7	OAH7	0.26 – 0.28
8	OAH8	0.137 – 0.146
9	IAH1	0.33 – 0.34
10	IAH2	0.75 – 0.76
11	IAH3	0.07 – 0.08
12	IAH4	0.24 – 0.25
13	IAH5	0.41 – 0.43
14	IAH6	0.73
15	DH	0.002 – 0.047
16	NGV inner passage	2.62 – 2.70

Mass flow through various holes of combustor has been estimated for three test rig conditions and percentage of inlet mass flow through various holes is shown in the Table 1. It can be seen from the table that mass flow rate through various features of combustor remains invariant irrespective of change in inlet total pressure.

CONCLUSION

Cold flow CFD analysis of 120^0 sector slinger combustor with inlet and exhaust adaptation has been carried out using ANSYS Fluent 17.0. Pressure loss across the combustor, mass flow through various features of combustor and variation of velocity components along the measurement planes for different test rig conditions have been estimated.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

ACKNOWLEDGMENT

The authors would like to thank M. Z. Siddique, Outstanding scientist and Director, GTRE, DRDO for permitting us to carry out the project work at GTRE. Authors would also like to thank Dr. Umashankar, HOD, Mechanical Engineering Department, SIT, Tumakuru for his valuable support and guidance throughout the project. Thanks are due to all members of combustion group for their continuous guidance and support throughout the project.

REFERENCES

[1] E P Demetri, R F Topping, R P Wilson, “Study of Research and Development requirements of Small Gas Turbine Combustors”

[2] Seongman Choi, Donghun Lee, D. & Park , “Ignition and Combustion characteristics of Gas Turbine Slinger Combustors” by, Jouranal of Mechanical Science and Technology , 2008.

[6] Seong Man Choi, SeongHo Jang, Dong Hun Lee, Gyong Won You , “Spray characteristics of rotating fuel injection system of micro-jetengine”, Journal of Mechanical Science and Technology, 2009.

[7] Seong Man Choi, SeongHo Jang, “Spray behavior of the Rotary Atomizer with In-Line Injection Orifice”, Atomization and Sprays, Volume 10, November 10, 2010.

[8] Fluent Theory guide 17.0, Fluent Inc.

[9] Srinivasa Rao. M. et. al., “CFD Analysis of Slinger Combustor”, 14th Annual CFD Symposium, 2012.

[3] Bhawan B. Patel, Oleg Morenko, Publication no. US8677731 B2, 25 March 2014.

[4] Dahm, Patel P &Lerg B, “Visualization and Fundamental analysis of Liquid Atomization by Fuel Slingers in STFE” proceedings of 32nd AIAA Fluid Dynamic Conference.

[5] Carmen Sescu, Bogdan R Kucinschi, AbdollahAfjeh, “Parametric Study of Liquid atomization by Slinger Injectors”, 45th ASEE Joint Propulsion Conference & Exhibit.

.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)