Scientific Journal of Impact Factor(SJIF): 3.134
e-ISSN(O): 2348-4470
p-ISSN(P): 2348-6406
International Journal of Advance Engineering and Research Development
Volume 2,Issue 3, March -2015

MODELING AND CFD ANALYSIS OF SWIRL CAN TYPE COMBUSTION CHAMBER

Naitik H. Gor[1], Milan J. Pandya[2]

[1] *PG Student (Thermal Engineering), Mechanical Engineering Department, L J Institute of Engineering and Technology, Ahmedabad, Gujarat.*

[2] *Asst. Professor, Mechanical Engineering Department, L J Institute of Engineering and Technology, Ahmedabad, Gujarat.*

Abstract: *New computational methods are continuously developed in order to solve problems in different engineering fields. One of these fields is gas turbines, where the challenge is to make gas turbines more efficient and to reduce emissions which affect the environmental balance. One of the main parts of a gas turbine that can be improved is the combustion chamber. In order to optimize the combustion chamber, both experimental and numerical methods are called for. Another topic of interest is to change the mixture in combustor and analyse its effects for reducing emissions. This paper describes a model of Swirl Can Type Combustion Chamber that is numerically reliable, mesh independent and fast. This paper presents a computational study of the flow field generated in Can combustion chamber and how that flow field convicts through the swirler. Specifically, the effect that the flow field acting at the outlet is studied. This paper presents the results of computational simulations done in parallel with e experimental. In comparisons of computational predictions with experimental data, a reasonable agreement of the mean flow and generated swirling effect in the combustor after changing the swirl angle, pressure loss at outlet may be changed.*

Keywords: *Can type combustion chamber, swirl angle, CFD, Gas turbine, simulation.*

I. INTRODUCTION

The combustion chamber has the difficult task of burning large quantities of fuel, supplied through the fuel spray nozzles, with extensive volumes of air, supplied by the compressor, and releasing the heat in such a manner that the air is expanded and accelerated to give a smooth stream of uniformly heated gas at all conditions required by the turbine. The combustion chamber must also be capable of maintaining stable and efficient combustion over a wide range of engine operating conditions. Efficient combustion has become increasingly important because of the rapid rise in commercial aircraft traffic and the consequent increase in atmospheric pollution

This paper presents a computational study of the gas turbine combustor and how to increase the performance using computational fluid dynamics.

The combustion chamber is the place where two major events take place; at the inlet fuel will mix completely, or to a sufficient degree, with air. In some combustors fuel mixes with air before combustors, however, in order to achieve a smooth burning, air and fuel should be mixed before burning. Second event is burning. In the combustion chamber, due to the high temperature, the gaseous mixture which consists of fuel and air will ignite and raise the temperature. Rise in temperature will increase the volume which will drive the fluid forward. There are number of facts that make this part of gas turbine important. In order to make this clear, we will address problems in a poorly designed combustion chamber

Can type combustors are self-contained cylindrical combustion chambers. Each "can" has its own fuel injector, igniter, liner, and casing. The primary air from the compressor if guided into each individual can, where it is decelerated, mixed with fuel, and then ignited. The secondary air also comes from the compressor, where it is fed outside of the liner (inside of which is where the combustion is taking place). The secondary air is then fed, usually through slits in the liner, into the combustion zone to cool the liner via thin film cooling.

In most applications, multiple cans are arranged around the central axis of the engine, and their shared exhaust is fed to the high pressure turbine. Can type combustors were most widely used in early gas turbine engines; owing to their ease of design and testing (one can test a single can, rather than have to test the whole system). Can type combustors are easy to maintain, as only a single can needs to be removed, rather than the whole combustion section, Most modern gas turbine engines (particularly for aircraft applications) do not use can combustors, as they often weigh more than alternatives. Additionally, the pressure drop across the can is generally higher than other combustors (on the order of 7%). Most modern engines that use can combustors are turbo shafts featuring centrifugal compressors. Fig 1 shows the schematic Diagramme of can combustion Chamber.

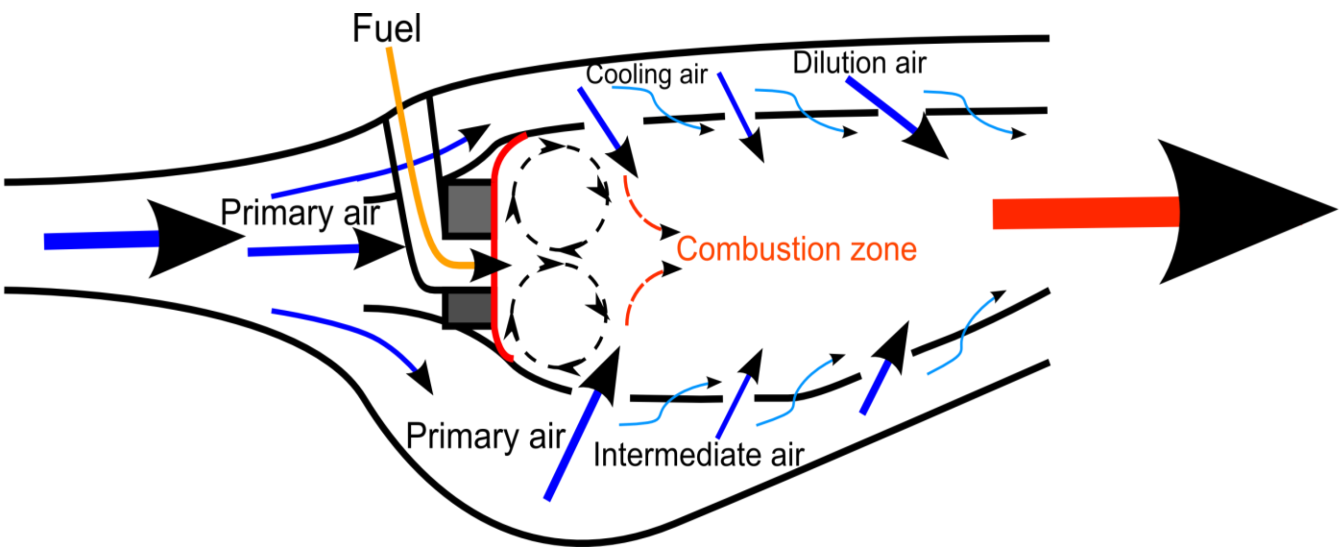

Fig 1 Schematic Diagram of Can Combustion Chamber.

II. LITERATURE REVIEW

[1] V Guru Shanker et al, "Modelling and Flow Analysis of Can Type Combustion Chamber with N-Butane as a Fuel for Gas Turbine Engine."
In this paper author shows the modelling and flow analysis of a combustion. The k-Epsilon model used for analysis. They shows that the pressure and temperature distributions along with results the temperature profile in a can type combustion chamber is not uniform at exit of the combustor to the grid size. Also the velocity vectors graph in that, velocity fields are almost insensitive, and also the air forms a vortex by the interaction of primary air and the air from primary holes.
In the study they observed the pressure and temperature distributions and Flow visualization (i.e. velocity vectors), across the chamber. Fig 2 shows that the pressure distribution along the combustion chamber.

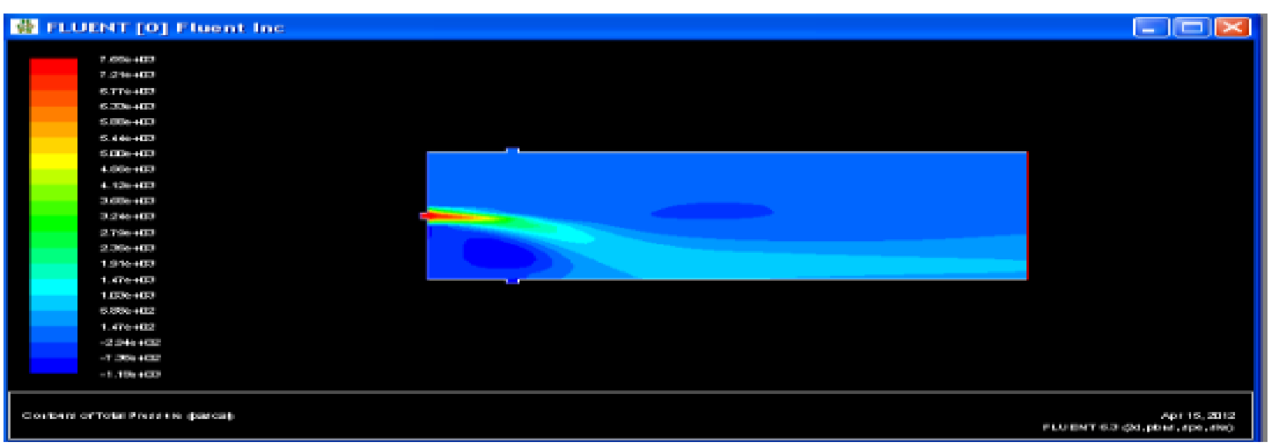

Fig 2 Pressure distribution along the combustion chamber.

[2] Guessab A et al, "Effect of Fuels on Gas Turbine Can-Type Combustor using CFD Code."
This paper describes reacting flow analysis of a gas turbine combustion system. The method is based on the solution of Navier-Strokes equations using finite volume method. The turbulence effects are modeled through the renormalization group k-ε model. The method has been applied to a practical gas turbine can-type combustor.

They begin the study of different flow configuration of various swirl angles (35°, 40° and 45°) are compared in the primary chamber between a swirling angular jet (Primary air) and the non-swirling jet (fuel). Swirling air flow is a key feature in many types of combustors.
Numerical investigation on Can-type combustion chamber shows that biogas 3 (70% CH4 + 30 % CO2) is giving less NO emission as the temperature at the exit of combustion chamber is less as compared to gas natural, biogas 1 and biogas 2. Temperature profiles shows increment at reaction zone due to burning of air-fuel mixture and decrement in temperature at the downstream of secondary inlet holes due to supply of more to dilute the combustion mixture.
The results from the parametric studies indicate that the calculation of NO emission serves to develop low emission combustor.
Figure 3 presents a set of measurements of toxic compounds or mixtures of natural gas (100% CH4) and biogas 1, 2 and 3 burnt in strong swirl flows. The values measured for NO correspond to those taken 30mm from the rectangular.

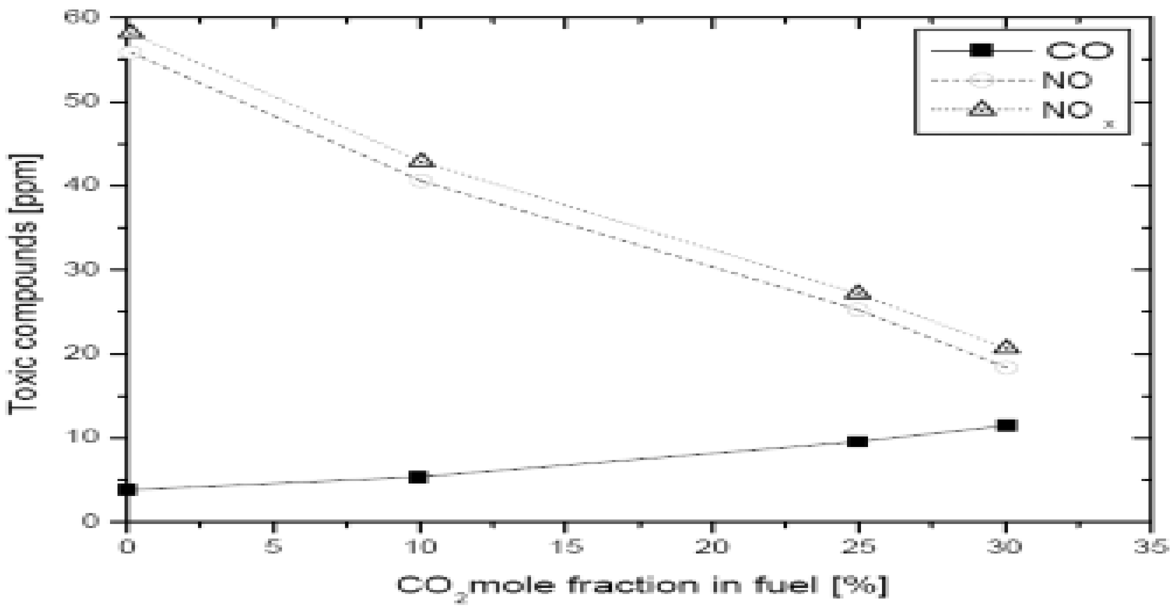

Fig. 3 Emission of NO, NOx and CO vs. CO2 content in Combustion mixture

[3] P. Di Martino et al, "Reactive CFD Analysis in a Complete Combustor Module for Aero Engines Application"
In this paper an improved version of BODY3D CFD in-house code is described to calculate the full model combustor, from compressor diffuser exit to turbine inlet. The coupled model accomplishes the following two main objectives: (1) implicit description of air flow splits and flow conditions for openings into the combustor liner, and (2) prediction of pressure losses distribution from the diffuser to the combustor exit. Remaining difficult issues such as generating the computational grid and modelling effusion/impingement cooling systems are also discussed.
[4] M. Khosravy el Hossaini, "Review of the New Combustion Technologies in Modern Gas Turbines."
In this chapter, a short introduction of combustion process and then a description of some new pioneer combustor have been presented. As gas turbine manufacturers are looking for continuous operation or stable combustion, satisfactory emission level, minimum pressure loss and durability or life. Hence, the advanced combustor might include all of these criteria, so some of them are selected to discuss in details.
A review of technologies for reducing NOx emissions as well as increasing thermal efficiency and improving combustion stability has been reported here. Trade-offs when installing low NOx burners in gas turbines include the potential for chamber. Mass splits of the total flow through the outer annuli and inner annuli respectively have been found to be: primary, 23.06, 22.51; secondary, 23.17, 22.93; and dilution hole, 53.77, 54.56%. Recirculation zone forms just downstream of the primary and secondary holes at both the inner and outer wall. There is no flow reversal at downstream of the dilution holes at the outer wall; however, a large reverse flow is seen at the inner wall. This phenomenon suggests the necessity for modification of the liner shape decreased flame stability, reduced operating range and more strict fuel quality specifications. In the other word, although, the turbine inlet temperature is the major factor determining the overall efficiency of the gas turbine but higher inlet temperatures will result in larger NOx emissions. So the essential requirement of new combustor design is a trade-off between low NOx and improved efficiency.
[5] S N Singh et al, "Flow characteristics of an annular gas turbine combustor model for reacting flows using CFD."
An attempt has been made to simulate the phenomenon of reacting three-dimensional turbulent flow in the combustion

chamber using CFD. Methodology allows parametric investigation for optimizing the design of combustion. The flow spreads uniformly in the axial direction and velocity contours change from circular to elliptical shape in the circumferential plane quantifying the spread rate. The temperature contours are circumferentially more uniform and symmetric. Temperature was found to be maximum at the outlet of the liner. The mass fractions of CH_4 and O_2 decrease whereas concentration of CO_2 and H_2O increases as combustion products move from the inlet to the outlet.

III. MODELLING OF CAN COMBUSTION CHAMBER

As per standard data we have made one cavity model in the solidworks.in this model we changed the swirl angle at inlet from 15 ° to 20°.also change secondary air holes distance from 180 mm to 200 mm. Fig.4 shows a cavity model of can combustor.

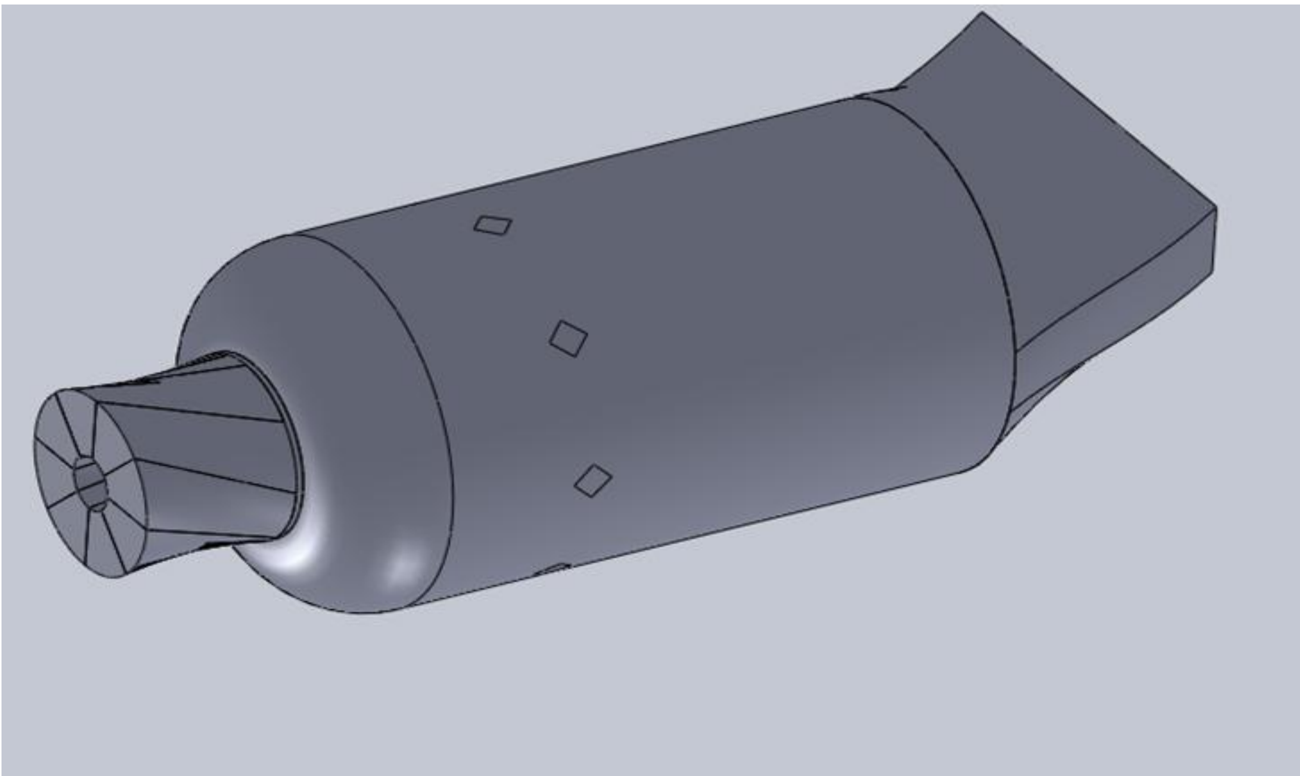

Fig. 4 Cavity Model of Can Combustor

IV. FLOW ANALYSIS OF THECOMBUSTION CHAMBER

We use ANSYS CFX 12.1 to import the geometry for combustion chamber and then generate a mesh. Then this mesh is imported into fluent solver for analysing the flow properties.

CREATING MESH TO THE GEOMETRY

Grid generation is very important before starting CFD calculations. The mesh created in ANSYS is intended for use in fluent, so it must be a single block, structured mesh. However, this mesh can also be used in any of the other fluent solvers. Before meshing we set the fluid domain in combustor. Fig 5 shows the fluid domain in combustor.

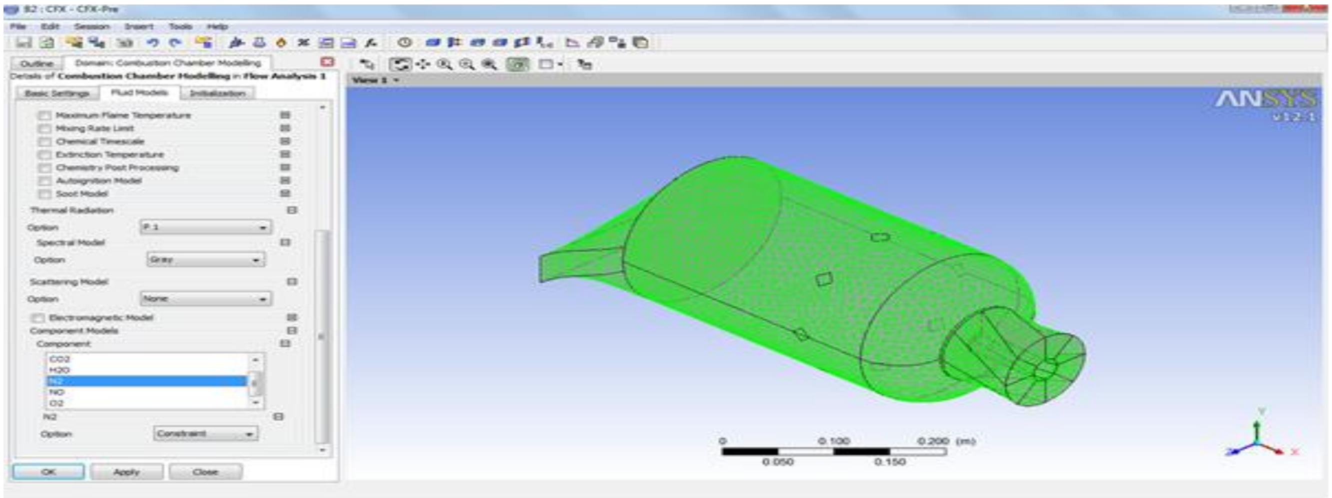

Fig.5 Fluid domain for the Combustion Chamber

This type of mesh is sometimes called a mapped mesh, because each grid point has a unique i, j, k index. In order to meet this criterion, certain additional steps must be performed in gambit. The Fig 6. shows Mesh of combustion chamber. In this mesh we discretise the model in 5000 nodes and 14000 tetrahedral elements.

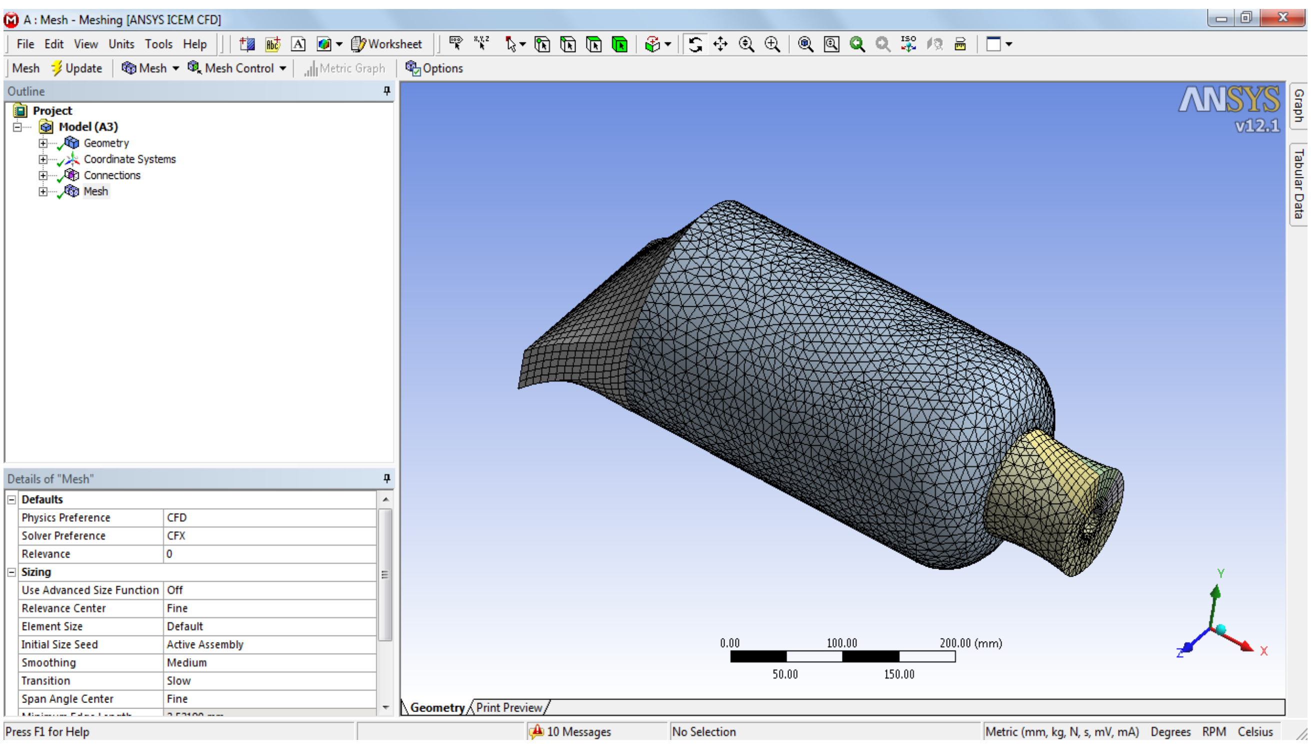

Fig.6 Mesh Generation for the Combustion Chamber

BOUNDARY CONDITIONS

The boundary conditions are given at the inlet of the Combustor and the fuel injectors. The flow is allowed to divide itself into linear casing, and from casing into different Zones (i.e. Primary and Secondary) through air admission holes, in which the air is supplied at the inlet diffuser with known conditions of pressure, Temperature and velocity, and then

allowed it to divide by itself between the casing and liner.
(I)The boundary conditions for air inlet zone are in flow air velocity is 40 m/s, turbulence intensity is 10% and Hydraulic diameter 1 intensity 10% and hydraulic diameter 0.01m, Mass fraction of C4H10 is 1.
(ii) Boundary conditions for outlet zone are, Pressure is 1.013 bar
(iii) Boundary conditions for the primary holes are, velocity is 6m/s, primary holes temperature 850K and selected material for combustion chamber as Aluminium. The simulation was specifically targeted to analyse the flow patterns in the combustion liner and through different air admission holes namely primary zone and dilution zone and from that study the temperature distribution in the liner and it walls as well as the temperature quality at the exit of the combustion chamber using ANSYS CFX. Fig 7 shows Boundary Condition.

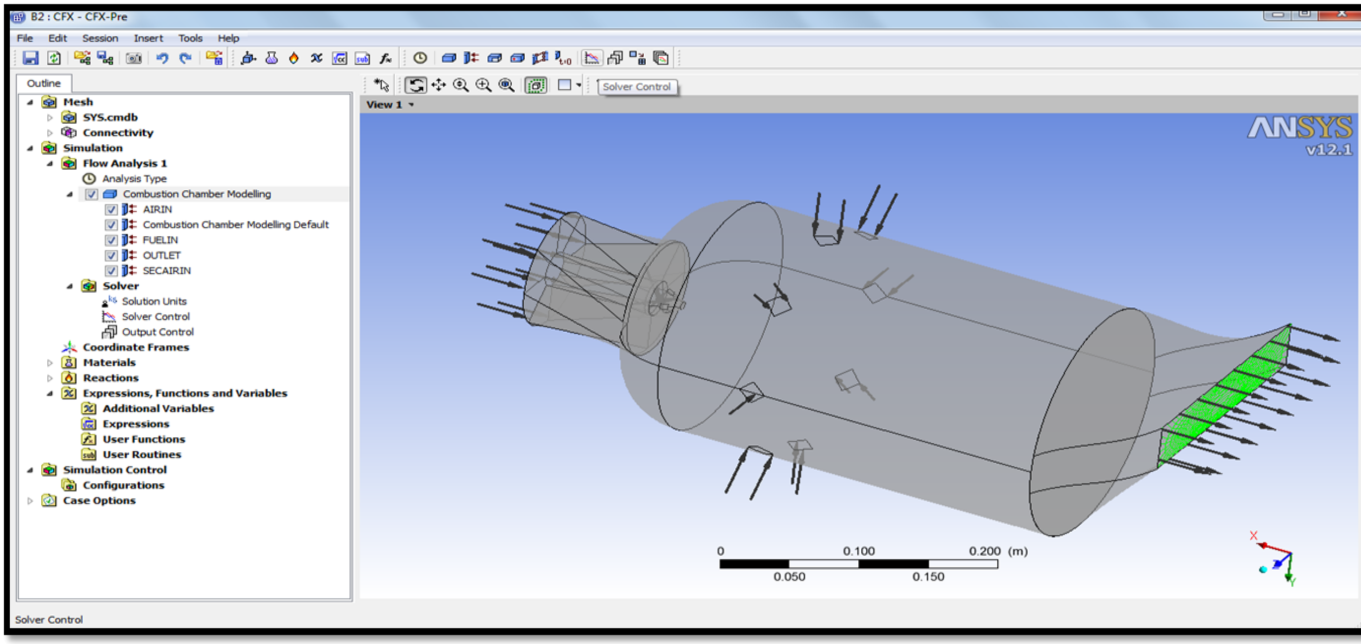

Fig.7 Boundary Condition for the Combustion Chamber

V. RESULTS AND DISCUSSION

In the study we have observed the pressure and temperature distributions and Flow visualization (i.e. velocity vectors), across the chamber. The fig.8 shows the CFX SOLVER ANALYSIS of can combustion chamber. In this study we choose three parameters. (1) Swirl angle (2) secondary air hole distances (3) Inlet velocity. By changing this parameters we get the different values of CO_2 emissions, outlet pressure. For that purpose we choose different cases by combination of three parameters and after analysis we get best result which can improves the combustor performance.

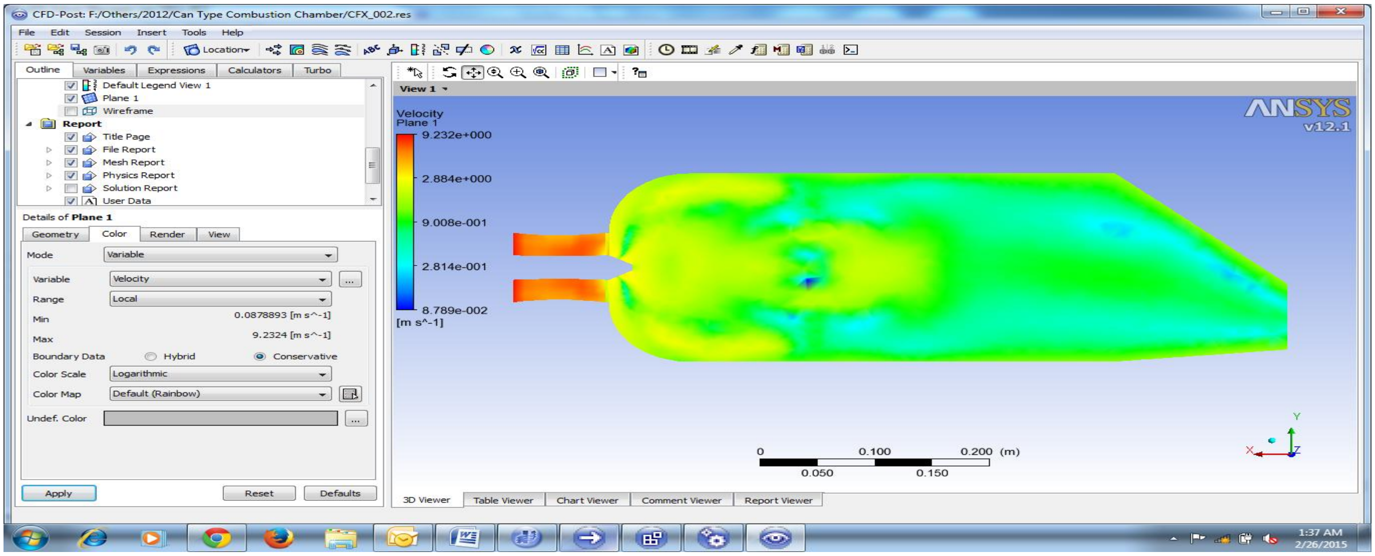

Fig 8. CFX SOLVER result

This results shows that by changing the swirl angle we can reduce the pressure loss at outlet of combustor.in this analysis pressure drop reduced from 2 bar to 0.3 bar and the value of velocity at outlet is 0.9m/s^2

VI CONCLUSION

The modeling and flow analysis of a combustion chamber have been completed. The k-Epsilon model used for analysis, fig.8 shows pressure and velocity distributions along with results the velocity profile in a can type combustion chamber. In this study our main focus was swirl angle. Because by increasing swirl angle we can increase the swirling effect in combustor which creates higher turbulence. Which gives better combustion rate. Also by changing distance between secondary air holes we can improves the mixing process of secondary air in the combustor. So by using this parameters we can increases combustor performance which gives higher rate of power production.

REFERENCE

[1] V Guru Shanker ,V. Kiran Kumar , A. Anuj Reddy, E. Rakesh, “Modeling and Flow Analysis of Can Type Combustion Chamber with N-Butane as a Fuel for Gas Turbine Engine”, International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181, Vol. 3 Issue 12, December-2014.
[2] Guessab A., Aris A. Benabdallah T. and Chami N , “Effect of Fuels on Gas Turbine Can-Type Combustor using CFD Code”, Applied Numerical Mathematics and Scientific Computation , ISBN: 978-1-61804-253-8, November, 2014.
[3] P. Di Martino, G. Cinque , A. Terlizzi , G. Mainiero , S. Colantuoni “Reactive CFD Analysis in a Complete Combustor Module for Aero Engines Application”Universita Deegli Studi Di Napoli Federico II, April 26-28 , 2009.
[4] M. Khosravy el_, “Review of the New Combustion Technologies in Modern Gas Turbines”, Intech open science, ISBN 978-953-51-1166-5, Published: June 19, 2013.
[5] S N Singh, V Seshadri, R K Singh and T Mishra , “Flow characteristics of an annular gas turbine combustor model for reacting flows using CFD”, Journal of Scientific & Industrial Research Vol. 65, November 2006, pp. 921-934
[6] R.K.RAJPUT Internal combustion engines : (including air compressors and gas turbines and jet propulsion) , . Bangalore, India : Laxmi Publications, 2007.
[7] V. Ganesan , Gas Turbines 3E , Tata McGraw-Hill Education, 01-Apr-2010
[8] John D. anderson ,Computational Fluid Dynamics Jr. McGraw – hill Education (India) Private Limited