Modeling and Flow Analysis of Can Type Combustion Chamber with N-Butane as a Fuel for Gas Turbine Engine

*V Guru Shanker[1] V. Kiran Kumar [2], A. Anuj Reddy[3] , E. Rakesh[4]
Department of Mechanical Engineering,
Vignan Institute of Technology and
Hyderabad- 508 284, India

Abstract-The combustion chamber of Gas turbine is one of the most critical components to be designed. To convert mass of air from compressor to heat energy combustion chamber plays a major role by burning of fuel, it is a continuous process. The reason behind the designing of combustion chamber being critically important and is a need for stable operation over a wide range of operations, the main drawback of designing combustion chamber is that the temperature, which produce during the process is very high, these high temperatures should withstand the combustion chamber, taking all these considerations, we designed a combustion chamber in catiav5 and also include CFD simulation is carried out with CFD tool GAMBIT-FLUENT. It also gives results from CFD simulation of temperature distribution, pressure distribution and flow visualization.

Key words - Can type combustion chamber; CFD; Gasturbine; 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. This task must be accomplished with the minimum loss in pressure and with the maximum heat release for the limited space available. 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, which is seen by the general public as exhaust smoke [1-3].The challenges in designing high performance combustion systems have changed significantly over the years towards more sophisticated analysis process [4].
The present paper discusses the design and simulation of can type combustion chamber for gas turbine the simulation is carried out with CFD tool.

II.   SELECTION OF COMBUSTION CHAMBER TYPE FOR THE PROJECT

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 modem engines that use can combustors are turbo shafts featuring centrifugal compressors [1].

Keeping their simplicity, effectiveness and their advantages this type of combustion chamber is been selected for making our project.

III.   DESIGN OF A COMBUSTION CHAMBER

o   *Basic combustor design and dimensions*
To document the dimensions of the combustor, we have developed our Combustor formulae from theories available based on our formulae on a metric value which "I" is derived from the inducer diameter of the blower being used to make the formula adaptable to many different sizes. Using centimeters as our measurement value, we took the diameter of the inducer (Value "I" in cm) and based all the combustor measurements to this value. *Fig.1* shows a diagram of the combustor cross section which explains the variables in the formulae. Some values can be adjusted with reasons to the sizes of material available.

o   *Cross section of combustion chamber*
The inside diameter (in cm) of the turbocharger's inducer (inlet) shall be represented by the value 'I'.
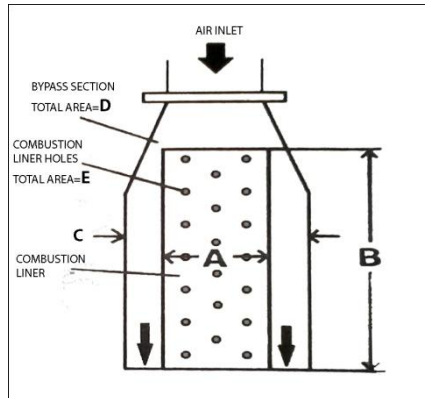
Fig.1. cross section of combustion chamber

- The inside diameter (in cm) of the combustion linen A = 1.3 *I
- The length (in cm) of the combustion linen   B = 3.85 *I
- The inside diameter (in cm) of the combustion chamber   C = 2.1 *I
- The cross-section area (in square cm) of the bypass section   D = 3.6 *I
- The total area (in square cm) of the combustion liner holes E = 4 *I
- The total number of holes (F) in the combustion liner   F = E / 0.33
- The individual size (G) of the liner holes (in square cm)   G = E / F

The overall length of the combustion chamber is varied with the adjustment of the bypass area. There is no set length for the combustion chamber itself, the bypass area in relation to the combustion liner is measured before cutting the chamber to the proper length. The angle of the tapered combustor neck is between 15 and 20 degrees in relation to the sides of the combustor itself.

The Bypass Section of the combustor is the area between the end of the combustion liner and the tapered neck of the combustor. The bypass allows cool incoming air to envelope the hot gas rushing out of the combustor into the turbine. This helps to cool the engine greatly. The bypass area is represented by value "D" and can be adjusted by lengthening or shortening the depth of the combustion chamber itself. This will put the combustion liner closer or farther from the tapered end of the combustion chamber allowing adjustment of the bypass area.
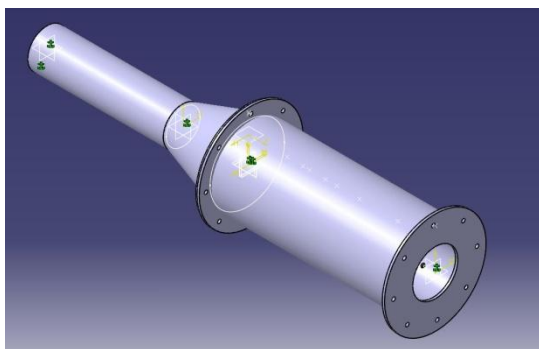

Fig.2 Assembly of combustion chamber

The *fig.2* shows the final assembly of combustion chamber model. The fuel chosen for the engine was n-butane-air with chemical formulae C_4H_{10}

IV. FLOW ANALYSIS OF THE COMBUSTION CHAMBER

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

o *PROBLEM DESCRIPTION*

The problem to be considered is shown schematically in *fig.3*. The combustion fuel enters into the combustion chamber through the fuel nozzle. Through the air inlet air enters from the primary and secondary zones. Both the air and fuel mix in the combustion chamber.
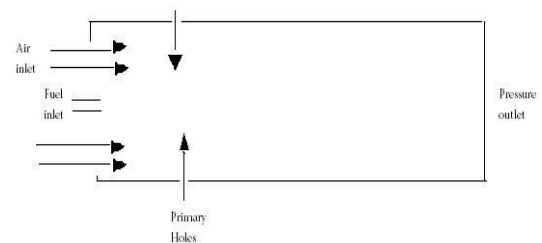

Fig.3 Schematic diagram of combustion chamber

o *CREATING MESH TO THE GEOMETRY*

Grid generation is very important before starting CFD calculations. The mesh created in gambit 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. 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.4* is shows mapped Mesh of combustion chamber.
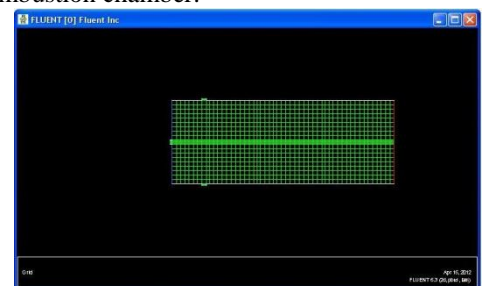

Fig.4.Mapped mesh for the Combustion Chamber

o *BOUNDARY CONDITIONS*

The boundary conditions are given at the inlet of the diffuser 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 5 m/s, turbulence intensity is 10% and Hydraulic diameter 1m. (ii) The boundary conditions for fuel inlet zone are, velocity inlet is 80m/s, turbulence

intensity 10% and hydraulic diameter 0.01m, Mass fraction of C_4H_{10} is 1. (iii) Boundary conditions for outlet zone are, Back flow hydraulic diameter 0.45m ,Turbulence intensity 10%, back flow total temperature 300K, mass fraction of O_2 is 0.23.(iv) Boundary conditions for the primary holes are, velocity is 5m/s Turbulent intensity 10%, hydraulic diameter 1m,primary holes temperature 850K and selected material for combustion chamber as Aluminum. The simulation was specifically targeted to analyze 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 fluent.

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.5* gives the Residual plot and flow distribution along x, and y directions
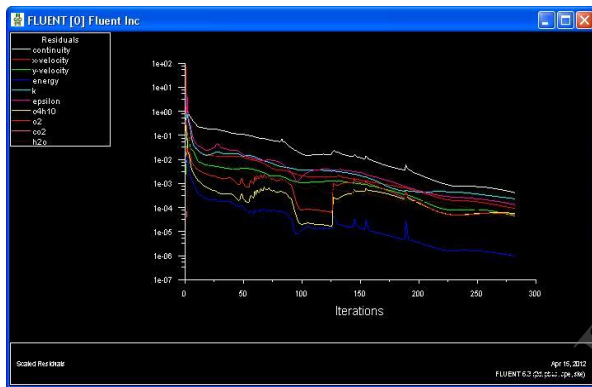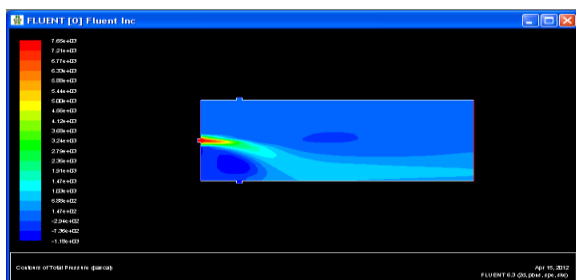

Fig.5 Residuals Plot
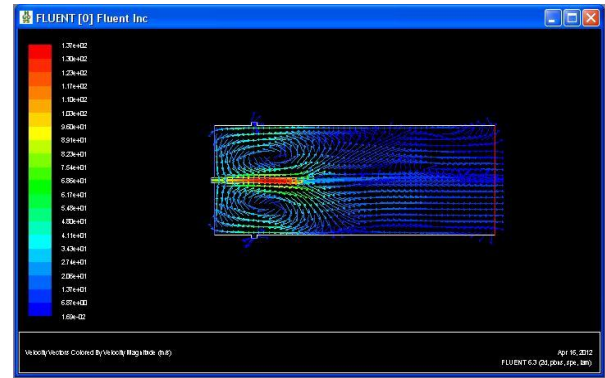

Fig.6 Pressure variation across the chamber

Fig.7 Velocity vectors

The *fig.6* shows that the pressure distribution along the combustion chamber and the fig.7 shows that the velocity vectors along the combustion chamber the velocity vectors solutions for reversed flow.Table-1 shows the the solutions of Residual plot as per the given boundary conditions

RESIDUAL PLOT RESULTS

S.No	Parameters	Values
1	Continuity	4.2062e-04
2	X-velocity	9.2459e-05
3	Y-velocity	4.5626e-05
4	Energy	9.7803e-07
5	K	2.3493e-04
6	Epsilon	1.3493e-04
7	C4 H10	6.7896e-05
8	O2	5.2862e-05
9	CO2	4.181e-05
10	H2O	4.181e-05

VI. CONCLUSION:

The modeling and flow analysis of a combustion chamber have been completed. The k-Epsilon model used for analysis, fig.6 shows 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. Fig.7 shows the velocity vectors graph in that, velocity fields are almost insensitive, and also we can see that the air forms a vortex by the interaction of primary air and the air from primary holes. This process is visualized through fluent analysis.

VII. REFERENCES:

[1] Lefebvre A.H., Gas Turbine combustion   chamber (2nd ed., USA: Taylor and Francis, 1999).

[2] Mellor A.M., Design of Modern Turbine combustors (US: ACADEMIC PRESS INC., 1990).

[3] K.V.Chaudhari, D.B.Kulshreshtha, S.A.Channiwala,Design and CFD simulation of Annular combustion chamber with kerosene as Fuel for 20 kW Gas Turbine Engine IJREC journal ISSN: 2248-9622,Vol. 2,Issue 6Nov-Dec 2012.

[4] W.Shyy, M.E. Braaten and D.L. Burrus, Study of Three-dimensional gas turbine combustor flows, int J. Heat Mass Transfer, vol 32(6), pp.1155-1164,1989.

[5] L .Eca and M. Hoekstra, On the Grid Sensitivity of the Wall Boundary condition of k-Epsilon Turbulance Model,ASME Journal of Fluid Energy, 126,2004.

[6] D.B.Kulshreshtha and S.A.Channiwala, Simplified Design of Combustion chamber for small Gas turbine Application, Proc. Of ASME IMECE2005-79038, Florida USA 2005.