Flow conditions analysis of Gas Turbine Combustor

R Abinaya1, D Rohini2, B Soundarya3, S Balakrishnan4, B Dakshinamurthy5 and P Rajeshwari6

1, 2 Assistant Professor, Bannari Amman Institute of Technology, Department of Aeronautical Engineering, Erode, Tamilnadu, India. 3,4,5,6 Student, Bannari Amman Institute of Technology, Department of Aeronautical Engineering, Erode, Tamilnadu, India. 

abirahul5@gmail.com, rohinidharamaraj90@gmail.com, soundarya.ae19@bitsathy.ac.in, balakrishnan.ae19@bitsathy.ac.in, dakshinamurthy.ae19@bitsathy.ac.in, rajeswari.ae19@bitsathy.ac.in

Abstract. Gas turbines are equipped for working effectively on a wide assortment of energizes. Combustor is directed to a great extent by the requirement for its length and frontal territory to stay inside the cut-off points set by other motor segments by the need for a diffuser to limit pressure problems, another issue of expanding significance is that acoustic reverberation. The decrease of combustor size and weight is another significant necessity for air motors. The aim of the present work is to analyse the combustion chamber geometry using FINE/OPEN of NUMECA software and to predict the hot and cold total pressure loss, (circumferential and radial) pattern factors and combustion efficiency. This paper mainly work towards understanding the effect of cold and hot flow in a practical combustor systems using CFD and to estimate the various parameters like pressure loss, pattern factor (circumferential and radial), velocity distribution through diffuser, dome, swirler, three zones and combustor holes. The flow analysis is further extended to investigate the effects of temperature distribution of the flow that reaches the turbine inlet. The combustor model is created using CATIA and parts are then cleaned individual using CADfix suitable for meshing. The performance of the turbulence model is analysed and compared. Present work deals with the implementation of Computational Fluid Dynamics (CFD) for three-dimensional analysis of flow in an annular combustor of an aero gas turbine engine. FINE/OPEN of NUMECA commercial code will be used to analyse the Combustion chamber geometry. The analysis is carried out both in FLUENT and NUMECA and the results are compared.

Keywords: Combustor, pressure loss, pattern factor, CADfix model, Fluent and NUMECA analysis

1. Introduction

Air-breathing plane impetus began toward the start of the twentieth century. The working liquid enters as natural air which is ducted through a channel diffuser into the motor, the motor fumes gas comprises somewhat of burning gas and half way of air. The fumes gas is extended through a push spout or spouts to surrounding pressure. Thrust can be characterized as the demonstration of pushing or the condition impelled. It states that the study of propulsion includes the study of the propelling force, the motion caused, and the bodies involved. Propulsion involves an object to be propelled and additional bodies, called propellant. Combustor is directed generally by the requirement for its length and frontal region to stay inside the cutoff points set by other motor segments by the need for a diffuser to limit pressure misfortune and by the prerequisite of a liner (fire tube) to give stable activity over a wide scope of air/fuel proportions. During the past 50 years, burning weights have ascended from 5 to 50 climates, delta air temperatures from 450 to 900K and outlet temperatures from 1100 to 1850K. The present combustors
display near 100% ignition productivity over their typical working reach. Moreover, the future of air motors liners has ascended from simply a couple hundred hours to a huge number of thousands of hours. Gas turbines are fit for working productively on a wide assortment of fills: strong, fluid and gas. Another issue of expanding significance is that acoustic reverberation. It happens when ignition insecurities become combined with the acoustics of the combustor. This issue is significant for the future advancement of lean premixed combustors. The decrease of combustor size and weight is another significant prerequisite for air motors. This prompts the exploration on new materials and new techniques for creation. This rearranges essential combustor plan and lessens cost. It has brought about the improvement of cutting edge divider cooling strategies. The metal suitable temperatures can be upgraded by the utilization of warm boundary hard-headed coatings inside the ignition liner. The ignition cycle is significant in a gas turbine cycle as the cycle of compound energy of the fuel is changed over to warm energy. It is changed over into work by the turbine. Burning is maybe portrayed most basically as an exothermic response of a fuel and an oxidant. In gas turbine applications, the fuel might be vaporous or fluid, yet the oxidant is consistently air. Ignition happens in numerous structures, not which are all joined by fire or iridescence.

- Deflagration
- Detonation
- Flame-tube cooling

Requirements of Combustion Chamber,
- Combustion efficiency should be higher (i.e., the fuel should be completely.)
- Ignition should be reliable and smooth
- An outlet temperature dispersion (design factor) that is custom fitted to augment the daily routines of the lives of the turbine cutting edges and spout direct vanes.

Factors affecting combustor design,
- The temperature of the gases after burning must be relatively low to suit the exceptionally focused on turbine materials.
- Toward the finish of the ignition space the temperature dispersion must be of known structure if the turbine cutting edges are not to experience the ill effects of neighborhood overheating.
- Ignition must be kept up in a surge of air moving with high speed in the area of 30-60 m/s and stable activity is required over a wide scope of air/fuel proportion from full burden to sitting conditions.

2. Methodology

The stream in the combustor is profoundly perplexing. The significant plan necessities rely upon the interior optimal design. The combustor design comprises of packaging, liner and fuel injector. The fast stream in the combustor lessens the living time accessible for fuel planning, blending and consuming of fuel with vitiated air.

The blending and burning cycle in a combustor are predominantly affected by stream design, distribution made by fire stabilizer, pressure misfortune and temperature impacts. To accomplish these prerequisites, the significant boundaries to be considered are:
- velocity distribution
- hot and cold pressure distribution
- pattern factor (circumferential and radial)

It is dominantly guided by test techniques and past experience. Anyway trial techniques are naturally moderate and furthermore exorbitant particularly at motor working conditions. These disadvantages and developing need to comprehend the unpredictable stream field marvel included, has prompted the improvement of mathematical strategies for foreseeing streams in combustor. These models are utilized to advance the plan of the combustor and lessen the volume of the trial work and cost included.
So as to reenact the genuine stream circumstance in a combustor, a far reaching model joining every one of these components are communicated through overseeing differential conditions. The arrangement of these conditions is tedious. Notwithstanding, with quick advancement in PC memory and speed, more sensible stream reenactments are progressively endeavored now. In the most recent decade's critical endeavors have been made to create mathematical techniques for anticipating stream in the combustor. Precise expectations of such streams are needed to plan a superior and dependable annular combustor.

3. Design of Model

The computational method dependent on the arrangement of preservation conditions of mass, force, energy and species are utilized. The disturbance impacts are recreated utilizing SpallaTalmalrs condition. The principle steps advanced in the examination of combustor are,

3.1. Software used for working procedure

- **Geometry modeling:** Modelling the Annular Combustor using CAD software (CATIA V5).
- **Geometry cleaning:** Geometry is cleaned using CADfix software that helps meshing easily.
- **Grid generation:** Grid generation for the sector model using HEXPRESS of NUMECA software.
- **Flow analysis:** Flow will be analysed using Fine/Open of NUMECA software.
- **Post Processing:** The results will be obtained using CFView of NUMECA software.

Figure 1: Gas turbine annular combustor

The model consists of a diffuser. The diffuser is attached with a manifold solid and the combustor design consists of an outer casing that is preceded with the diffuser. The combustor design consist of outer casings where the casings are seen as (2 layers at top and bottom) i.e. 2 inner liners which is covered by 2 outer liners. It covers the inner liner of the combustor. The upstream hemisphere end of the liner is called dome. At the center of the dome is swirler is located and within the swirler igniter is present. The swirler is surrounded by the atomizer and are enclosed by the cooling rings and cowl.
Figure 2: Annular combustor with parts

There is a large number of small holes called flare holes to cool the interior surface. The liner and cowl will have heat shields with a large number of holes to prevent them from burning. There are two big holes in the primary zone for allowing air for combustion and cooling processes. At downstream of the liner, there are large number of small holes called muff holes that are located to reduce the temperature of the air that reaches the turbine. At appropriate points, the liner will have slots and louvers to have film cooling air to control the temperature of the inner surface of the liner.

Figure 3: Annular combustor holes

To have an easy meshing, the annular combustor in CATIA can be divided into outer, inner, and cowl parts separately. The inner liner part in CATIA is shown below where the holes are removed, closed, and meshed separately. The combustor is cleaned separately using CADfix.
The CATIA parts are then cleaned singular utilizing CADfix appropriate for cross section. CADfix is driving CAD interoperability instrument which handles the ever present issues of 3D model information trade and re-use between various designing applications. CADfix permits the client to import CAD information, productively fix and adjust it, and fare in the most reasonable structure for reuse in the downstream framework, disposing of the requirement for costly CAD model revise.

3.2. CADfix application in the Model

The combustor model is then imported in CADFIX which helps to visualize and rectifies the surfaces, faces and unwanted double lines of CAD model. When the errors are rectified in CADFIX it is then suitable for importing in HEXPRESS of NUMECA software which requires a CAD clean, water tight model and where grids are generated and meshing is done.

CADfix is a software application that translates, heals and repairs (gaps, silver faces, reversed normal’s, duplicate entities). Minimizes rework associated with CAD data exchange and obtaining usable geometry downstream.

CADfix isn’t a CAD system in which some data is so badly corrupted that it cannot be automatically repaired and its missing data is too complex to re-create in CADfix, e.g., fillets and blends. For meshing the domain (combustor), HEXPRESS of NUMECA software is used. HEXPRESS can only understand closed models and requires cad clean and air tight model.

3.3. Domain discretization

When blunders are eliminated in CADFIX the combustor is then gets appropriate for bringing in HEXPRESS. Utilizing CAD control instrument bar the combustor model is checked. The combustor is imported in HEXPRESS as parasolid space record. For persuade the external packaging, liner, gaps and
cowl part are imported independently in CADFIX and HEXPRESS by shutting the model in CATIA and afterward it is additionally amassed together.

The combustor is divided into 7 domains and grids are generated individually which are then connected using FNMBs. The vertices of the domain are connected to more than 3 edges.

3.4. Grid Generation
For meshing the domain (combustor), HEXPRESS of NUMECA software is used. HEXPRESS can only understand closed models and requires cad clean and air tight model. Hence the model is imported in CADFIX (software that create cad clean model) as separate parts and cleaned separately.

The values for the initial mesh and refinements for different domains in combustor is given in the table 1 and 2.

| Domains          | Initial mesh X direction | Initial mesh Y direction | Initial mesh Z direction | No. of cells |
|------------------|----------------------------|--------------------------|--------------------------|--------------|
| Outer casing     | 16                         | 7                        | 4                        | 448          |
| Inner +heat shield | 19                        | 8                        | 6                        | 912          |
| Cowl             | 6                          | 9                        | 8                        | 432          |
| Swirler          | 4                          | 14                       | 14                       | 784          |
| Swirler path     | 5                          | 9                        | 8                        | 360          |
| Fuel ring        | 10                         | 10                       | 10                       | 1000         |
| Fuel path        | 8                          | 11                       | 11                       | 968          |
Table 2: Mesh requirements

| Domains                  | Max no. of refinements | Curve refinements | Surface refinements | Target cell size |
|--------------------------|------------------------|-------------------|--------------------|-----------------|
| Outer casing             | 5                      | 10,000            | 4                  | 0,0,0           |
| Inner + heat shield      | 7                      | 10,000            | 4                  | 0,0,0           |
| Cowl                     | 4                      | 10,000            | 3                  | 0,0,0           |
| Swrilr                   | 3                      | 10,000            | 3                  | 0,0,0           |
| Swrilr path              | 4                      | 10,000            | 4                  | 0,0,0           |
| Fuel ring                | 3                      | 10,000            | 3                  | 0,0,0           |
| Fuel ring                | 4                      | 10,000            | 3                  | 0,0,0           |

4. **Optimization**

In general, combustion problems are characterized by a strong variation of density. Since combustion in most technical applications takes place at low speeds, the density variations are not caused by the variation in the pressure, but by the heat released from the exothermic reaction between fuel and oxidizer. Most technical combustion problems can therefore be simulated with models developed for low Mach number flow. From a mathematical point of view this means, that the pressure variations in the flow have to be small compared to the thermodynamic background pressure of the system. Based on this definition, even systems with a transient background pressure (e.g., internal combustion engines) can be modeled as low Mach number problems. In the most general approach, combustion is described as a convection-diffusion-reaction problem involving n species and m reactions between these species as described by a reaction mechanism.

The combustion problem in two sub problems,

- Mixture Fraction
- Radiation model

4.1. **Validation case study**

Validation is the process of deciding how much a model is a precise portrayal of this present reality from the viewpoint of the proposed employments of the model. Fuel Methane is provided from the focal aspect of the combustor. The air is provided from the annular surface which is called essential air stream. The essential air is additionally provided from the half of the length of the combustor.
4.2. Boundary conditions

The boundary conditions are used to define the flow and thermal variables on the boundaries of the model. This section gives the details of the boundaries given for the flow simulation. Boundary conditions are employed so as to bring the computational domain in accordance with the real time conditions.

Inlet: Mass flow rate is specified at the core inlet and bypass inlet along with total temperature and turbulent intensity and hydraulic diameter.

Outlet: The boundary condition has been applied at the exit of the domain in the form of pressure outlet. Static pressure and total temperature are defined.

Periodic: The side faces are made periodic in order to map the flow solutions on each other. Periodic are defined on radial sectors to obtain identical solutions on either sides of the sector considered.

Physical Modelling

The time-averaged governing equations for fluid flow in the Cartesian tensor form,

- Continuity Equation
- Momentum Equation
- Energy Equation
- Navier-Stokes Equation

| Parameters          | Inlet            |
|---------------------|------------------|
| Total Pressure      | 17 bar           |
| Total Temperature   | 812K             |
| Outlet Static Pressure | 15,28000 bar  |
| Mass flow           | 2.933 kg/s       |
Table 4: Outlet boundary conditions for the annular combustor

| Regions         | Values   |
|-----------------|----------|
| Core outlet     | 2.2 kg/s |
| Bleed outer     | ½ 0.1965 kg/s |
| Bleed Inner 1   | ½ 0.117 kg/s |
| Bleed Inner 2   | 0.108 kg/s  |

The Boundary conditions applied in the combustor model is listed in the table 1 and the outlet boundary condition for the combustor model is provided in the table 2.

5. Results and Discussion

The comparison of models can also be made by observing the change in pressure from the diffuser to the dilution zone, temperature pattern, Mach no. variations and velocity variations along the annular combustor section.

![Figure 9: Primary Zone](image9)

![Figure 10: Primary Holes](image10)

![Figure 11: Intermediate Zone](image11)

![Figure 12: Exit of Intermediate](image12)
The velocity distribution near the primary zone is obtained as shown above. The velocity at the primary holes at the upper and lower part of the liner is high compared to the flow in the inner part of the liner. The velocity distribution at the cut planes i.e near the primary zones exist and near the holes of the intermediate zone are shown above. The velocity distribution is about 150m/s at the holes and quite low at the other sides. The mass flow is more near the lower part of the swirler exist hence the velocity distribution is more at that point. The velocity vectors and its distribution is low compared to the primary zone.

5.1. Flow Analysis

The combustor works in different modes i.e. the combustion can be premixed non premixed or partially premixed combustion. The analysis done in this report is for non-premixed combustion. The flow inside the combustor is hot and cold flow resulting in hot and cold pressure loss. This is due to the combustion process leading to flow called hot flow and dilution process (flow without combustion can also be included) leading to flow called cold flow.

The inlet total temperature at the primary zone is very high and the total temperature in the region where combustion takes place has a very high total temperature. As the flow advances, the temperature near the wall regions reduces gradually due to the cooling holes and boundary layer interactions and hence, the high temperature is concentrated at the center of flow.

![Figure 13: Absolute Temperature (K) cut section](image1)

![Figure 14: Absolute Temperature (K)](image2)

From the above analysis we can see the difference in temperature before and after combustion. The Absolute total temperature is increases drastically after combustion and reduces by cooling effects of slots and holes of the liner.

![Figure 15: Static temperature (K) Cut section](image3)

![Figure 16: Static temperature (K)](image4)
The pressure at the diffuser increases by decreasing the velocity of the flow. The inlet total pressure gradually increases near the dilution zone. The total pressure loss is obtained by the summation of inlet, annulus and outlet pressure loss.

![Figure 17: Absolute Total Pressure (Pa) Cut Section](image1)

![Figure 18: Absolute Total Pressure (Pa)](image2)

![Figure 19: Static Pressure (Pa) Cut Section](image3)

![Figure 20: Absolute Static Pressure (Pa)](image4)

**Table 5: Pressure Distribution**

| Region           | Static Pressure (Pa) | Absolute Temperature (K) |
|------------------|----------------------|--------------------------|
| Outer Anulus     | 2000000              | 3e^6                     |
| Dome             | 1.6 e^6 - 1900000    | 3e^6                     |
| Swirler          | 1800000              | 2e^6                     |
| Cooling ring     | 1600000              | 2e^6                     |
| Primary Holes    | 1.4e^6               | 3e^6 - 2e^6              |
| Flare Holes      | 1.2e^6 - 1.6e^6      | 3e^6 - 2e^6              |
| Dilution Holes   | 1800000              | 3e^6 - 2e^6              |
| Muff Holes       | 1800000              | 3e^6 - 2e^6              |

The pressure distribution at various parts of the combustor is given in the table3. The Mach number of the flow is obtained. It can be seen that the recirculation zone formed near the end of diffuser section and near the cowl is well defined.
5.2. Comparison between fluent and NUMECA results

The annular combustor is also analysed using fluent software and the pressure loss results obtained from NUMECA is compared with the pressure loss results obtained from Fluent. The pressure loss at inner annulus, outer loss and the overall pressure loss are compared by plotting graph between Mach no. and Pressure loss.

Figure 23: Graph of Pressure Loss at Inner Annulus (Fluent)

Figure 24: Graph of Pressure Loss at Outer Annulus (Fluent)
Figure 25: Graph of Pressure Loss at Overall Combustor (Fluent)

Figure 26: Pressure variation in inner annulus (NUMECA)

Figure 27: Pressure variation in outer annulus (NUMECA)

Figure 28: Pressure variation in overall Combustor (NUMECA)
6. Conclusion

Numerical prediction of flow, combustion and combustion processes are carried out inside the gas turbine combustor under cold flow and hot flow conditions. Emphasis is put on the evaluation of temperature distribution in the dilution zone and pressure loss at the diffuser. The following conclusions are drawn from this study.

✓ The flow decelerates smoothly along the diffuser passage. There is flow separation at the end of diffuser leading to a small recirculation zone created by the swirller increasing turbulence.
✓ The flow field analysis reveals that the air from the compressor enters into the diffuser and then flows through the swirller, heat shield and primary holes, cooling rings and through the dilution slots and holes and finally to the turbine.
✓ The diffuser reduces the velocity. The dome and the swirller creates recirculation zone upstream of the combustor, primary zone.
✓ The flow decelerates in the diffuser from a Mach number of about 0.3 to 0.25.
✓ The pressure loss in cold flow condition is around 7% when measured from inlet to core outlet of the combustor. This pressure loss can also occur due to extra turbulence created by the dome and the swirller.
✓ The pressure loss in outer annulus is 3% and at inner annulus is of 2.8%.
✓ The hot flow analysis reveals that the combustion increases the pressure loss from 3% at the primary zone to 9% at the inlet to the core outlet during mass flow condition (case2).
✓ The maximum temperature on liner surface for both cases is found to be within the service temperature of material selected as the temperature must also be suitable for turbine blades and is of about 812K.
✓ Therefore, an additional volume of air can be used to cool the combustion mixture at the dilution zone to keep the metal temperature within the limit.

References
[1] F.N.Egolfopolous and P.Dagut “Combustion and Flames” is published to have high quality work from experimental, theoretical and computational investigation on the phenomena and closely allied matters.
[2] Marc R.J. Charest, "Design methodology for lean premixed pre-vaporized Can combustor", Ottawa-Carleton Institute, Ottawa, Ontario. pp 103-121 & 162-178, April 2005.
[3] Eric Baudoin, “Large Eddy Simulation of Turbulent Premixed and Partially Premixed Combustion”, Lund Institute of Technology, Sweden. pp 10-25 December 2010.
[4] K. V. Chaudhari, D. B. Kulshreshtha, S.A. Channiwala/ International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 2, Issue 6, November-December 2012, pp.1641-1645 1641.
[5] K. Sreenivasa Rao, " CFD Modeling Of An Aero Gas Turbine Combustor For A Small Gas Turbine Engine", International Journal of EngineeringResearch & Technology Vol. 2 Issue 1, January-2013.
[6] Mohd. Ikhwan Saifulism Bin Roslan, "Computational fluid dynamics (CFD) simulation on gas turbine combustion chamber", University Malaysia Pahang, 28-43 November 2007.
[7] Arthur H. lefebvre, “Gas turbine combustion”, Aerodynamics of combustion chamber, 1999.
[8] Sachin Bhalariao, Dr. A.N.Pawar. "Thermal mapping of a can type gas turbine combustion chamber using CFD" Jawaharlal Darda Institute of Engineering and Technology, Yavatmal India, Government Polytechnic, Amaravati.
[9] F.X. Nicoul, " A study of the influence of a swirled injection on micro scale non- premixed combustion” Fundamental and Applied Energetics Department, ONERA, Palaiseau, France.2009.
[10] S N Singh, V Seshadri, K Saha, K K Vempati, S Bharani "Effect of inlet swirl on the performance of annular diffusers having the same equivalent cone angle” Proceedings of The Institution of Mechanical Engineers Part G-journal of Aerospace Engineering.
[11] P.S. Jeyalaxmi Dr.G.Kalivarathan Research Scholar, CMJ University, Meghalaya, Shillong. IAEME 39 "CFD analysis of flow characteristics in a gas turbine-a viable approach to predict the turbulence". Principal/ PSN Institute of Technology and Science, Tirunelveli, Tamilnadu, Supervisor,CMJ University, Shillong. International Journal of Mechanical Engineering and Technology (IJMET), ISSN 0976 –6340(Print), ISSN 0976 – 6359(Online) Volume 4, Issue 2, March - April (2013).

[12] P.Sravan Kumar1, P.Punna Rao2 " Design and Analysis of Gas Turbine Combustion Chamber " PG Student, Department of Mechanical Engineering, Nimra College of Engineering & Technology. Assistant Professor, Department of Mechanical Engineering, Nimra College of Engineering & Technology, Vijayawada, AP, INDIA.

[13] Syed Alay Hashim "Design and Fabrication of an Annular Combustion Chamber for the Micro Gas Combustion and its control are essential for the existence of an engine" Published in: International Journal of Engineering Research & Technology Website: www.ijert.org Volume/Issue: Vol.2 - Issue 8 (August - 2013) e-ISSN: 2278-0181.

[14] Wang Cheng-jun, "Turbulence combustion modeling in the Gas Turbine combustor" Third International Conference on Measuring Technology and Mechatronics Automation, 2011.

[15] Jack D.Mattingly "Elements of Gas Turbine Propulsion " Published by Tata McGraw-Hill Publishing Company Limited , 7 West Patel Nagar , New Delhi 110 008 , and printed at Sai Printo Pack Pvt. Ltd., Delhi 110 020