International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 06, June-2015

Numerical Analysis of Flow Characteristics and Different Equivalence Ratio for Combustion Chamber of 30MW Power Plant

Patel Vimalchandra S.
Automobile Engg. Department
G.I.D.C. Degree engg. College, Navsari

Patel Mayurkumar R.
Mechanical Engg. Department
G.I.D.C. Degree engg. College, Navsari

Maisuria Snehal P.
Mechanical Engg. Department
G.E.C Valsad

***Abstract*— Combustion chamber is used to burn fuel-air mixture to produce energy. This energy are then supply to enter the nozzle and then into the turbine. Current state-of-the-art in Computational Fluid Dynamics (CFD) provides reasonable reacting flow predictions and is already used in industry to evaluate new concepts of gas turbine engines.**

In parallel, optimization techniques have reached maturity and several industrial activities benefit from enhanced search algorithms. The increase, in recent years, in the size and efficiency of gas turbines burning natural gas has occurred against a background of tightening environmental legislation on the emission of nitrogen oxides. The higher turbine entry temperatures required for efficiency improvement tend to increase NO_x production.

The numerical simulation of can type combustion chamber for Utran power plant is carried out using commercial CFD code CFX. Turbulence is modeled using k-ε turbulence model.

For the liner flow, it is observed that on moving axial from inlet to outlet, velocity, temperature contours become more uniform symmetric in circumference plane. Mass fraction CH_4 and O_2 decreases where concentration of CO_2, H_2O, and NO increases in the axial direction.

Different boundary condition are applied for find out the required exit temperature and reduced nitric oxide (NO) in parts per million.

Keywords*—— *Gas Turbine Combustion Chamber,equivalance ratio, inner liner.

I. INTRODUCTION

Industrial gas turbine combustor technology has been developing gradually and continuously, rather than a dramatic change. As gas turbine technology advances into 21st century; combustion engineers are faced with the challenges of achieving of higher compression ratios, higher turbine inlet temperature in aero gas turbine engines. At the same time, as interest in pollutant emissions from gas turbine increases, combustion engineers are also required to consider new means for pollutant reduction.

There are many types of combustor, but the three major types are tubular, turbo-annular, and annular.

Despite the any design differences, all gas turbine combustion chambers have three features

1. A recirculation zone,
2. A burning zone (with a recirculation zone which extends to the dilution region)
3. A dilution zone.

The function of the recirculation zone is to evaporate, partly burn, and prepare the fuel for rapid combustion within the remainder of the burning zone. Ideally, at the end of the burning zone, all fuel should be burnt so that the function of the dilution zone is solely to mix the hot gas with the dilution air. The mixture leaving the chamber should have a temperature and velocity distribution acceptable to the guide vanes and turbine. Generally, the addition of dilution air is so abrupt that if combustion is not complete at the end of the burning zone, chilling occurs and prevents completion. However, there is evidence with some chambers that if the burning zone is run over rich, some combustion does occur within the dilution region.

Combustor inlet temperature depends on engine pressure ratio, load and engine type, and whether or not the turbine is regenerative or non-regenerative especially at the low-pressure ratios. The new industrial turbine pressure ratios are between 17:1 and 35:1, which means that the combustor inlet temperatures range from 454°C to 649°C. Fuel rates vary with load, and fuel atomizers may be required for flow ranges as great as 100:1.

II. LITERATURE SURVEY

C.Hassa et.al [1] in his isothermal experiments on the mixing of jet rows in cross flow with particular attention to the quench zone mixing of RQL combustors have been carried out. The study with homogeneous cross flow for RQL combustors without cooling air in the primary zone showed optimum mixing with two staggered rows with close axial spacing. The light sheet technique could be demonstrated to give good quantitative results with higher spatial resolution than any other competing technique if applied with the appropriate care. Its application clearly revealed the influence of swirling, re circulating primary zone flow on the jet mixing. This work on RQL concept has been successfully applied in application for nitrogen oxide reduction.

Christopher O. Peterson et. al. [2] in his work to characterize the performance of the RQL combustor at elevated inlet temperature and pressure to determine the significance of various operating parameters on NO_x emission and combustion efficiency. Four objectives were established as follows:

1. Modify an existing testing facility in order to make it capable of simulating acting gas turbine engine combustor operating condition.
2. Develop a model RQL combustor that would simulate the plenum feed characteristic of practical gas turbine combustor hardware.
3. Obtain exit plane emission data for the RQL combustor operated under various conditions.
4. Identify where NO_x production is occurring by operating the model RQL combustor at realistic condition and obtain emission profile at the exit of rich zone and entrance of the lean zone as well as exit plane measurements.

Kulshreshtha D. B. and Channiwala S. A [3] carried out the numerical simulation of the designed can-annular type combustion chamber for small gas turbine engine using commercial CFD code CFX. The exit temperature quality is not uniform. This may be due to very low air flow rates and secondly the intermediate zone is neglected. To overcome this lacuna, a mixing length concept was introduced. The numerical results give near uniform temperatures at the exit.

S N Singh et. al. [4] studied the effect of height of inner & outer annulus for an elliptical dome shape combustor for cold flow simulation using CFD. Prediction has been carried out for air-fuel mixture as working fluid. Primary, secondary and dilution holes are simulated on the inner and outer liner walls with swirler being placed at the centre of the dome. Flow has been analysed in the annulus region. Uniform velocity distribution is obtained in the annulus passage around the liner. For the liner flow it is observed that on moving axially from nozzle to outlet, velocity and temperature contours become more uniform and symmetric in circumferential plane. Mass fraction CH_4 and O_2 decreases whereas concentration of CO_2, NO and H_2O increases in the axial direction.

S. A. Channiwala & Digvijay Kulshreshtha [5] The fuel under consideration is hydrogen and primary zone equivalence ratio varied from 0.5 to 1.6 were simulated. Accordingly, in present study an attempt has been made through CFD approach using CFX 12 to analyse the flow patterns within the combustion liner and through different air admission holes, namely, primary zone, intermediate zone, dilution zone and wall cooling, and from these the temperature distribution in the liner and at walls as well as the temperature quality at the exit of the combustion chamber is obtained for tubular and annular combustion chambers designed for gas turbine engine. The maximum centerline temperature recorded by CFD simulation is in the vicinity of 1876^{o}C while for Experimental Investigations is around 1700^{o}C. The pressure loss along the combustion chamber is 10% of the inlet pressure. The velocity profiles show an increasing trend along the length of combustion chamber, but low velocities are encountered in primary zone which is beneficial for combustion stability.

Paolo Gobbato et. al. [6] studied an experimental and computational analysis of both the isothermal and the reactive flow field inside a gas turbine combustor designed to be fed with natural gas and hydrogen. High combustion efficiency, low pollutant emissions and a reduced flame length are achieved by inducing high turbulent recirculation of hot products in the primary combustion zone. Efforts to understand the influence of combustor design, operating conditions and flow field on pollutant emissions (especially NO_x) have recently intensified due to increasing restrictions imposed to reduce environmental pollution. It is worth nothing that both the measured and the calculated temperatures quoted in these works never exceed 2000 K, while the flame temperature can reach 2500 K in gas turbine conventional combustors.

The paper deals with the CFD analysis of the flow field and NO_x emissions of a single can gas turbine combustor. The computed NO_x concentrations at the combustor discharge are compared with experimental data acquired by an infrared analyser during full-scale full-pressure combustion tests.

A.H. Lefebvre and E. R. Norster [7] presented a method for determining the ideal ratio of flame-tube diameter to air-casing diameter, and the optimum number and size of dilution holes, for the attainment of a well-mixed primary combustion zone and the most uniform distribution of exhaust gas temperature for tubular gas turbine combustion chambers. The Method employs data accumulated from a series of experimental investigations carried out on the flow and mixing characteristics of cold air jets when injected into a hot gas stream. From finding it would appear that a useful yardstick for mixing performance in dilution zones might be the pitch/diameter ratio of the dilution holes. The conclusion have been carried out (1) For any given value of overall pressure-loss factor, the optimum no of dilution hole diminishes as the proportion of air employed in combustion is increased. The practical significance of this the high temperature rise combustor require few dilution holes. (2) For a given chamber temperature rise, the optimum no of dilution holes increase with increase in flame tube pressure loss factor.

Alessandro Marini, et. al. [8] studied a *silo* combustion chamber installed on a 10 MW class heavy-duty gas turbine. The aim of the work was to investigate some modifications for the combustion chamber 100% hydrogen fired in dry operation, in order to reduce the NO_x production. The proposed modifications were analyzed by a 3D CFD RANS reactive procedure based on commercial codes. Full scale tests were performed also on the modified version methane fired. The numerical analysis has shown that the modified version allows a reduction of about 30% on the NO_x emissions. Finally, preliminary considerations related to the fuel injection scheme and to the effect of the main injection condition on the mixing performance, were carried out together with some estimations for NO_x emissions containment.

III. MODELING, MESHING AND BOUNDARY CONDITION

The aerodynamic process plays a vital role in the design and performance of a gas turbine combustion system. The flow in the combustion chamber is generally complex, due to highly turbulent nature of the flow field, coupled with complex geometric configuration. Further, the additional complications of combustion like fuel evaporation, radiative and convective heat transfer, and chemical kinetics are also involved. The interaction between the diffuser and the combustor external flows plays a key role in controlling the total- pressure loss, flow distribution around the combustor liner, durability, and stability.

All dimensions of the can type combustion chamber of Utran gas power plant was taken from the site. As shown in figure 1 is the actual inner liner of the combustion chamber.

Figure 1 shows total length of the inner liner is 702 mm. the inner liner combustion chamber have a first two row is primary hole, third one row intermediate hole and Last one row is dilution hole. As shown in fig 4.2 First row number of 8 hole is 20 mm, second row number of 8 hole is 15 mm, third row number of 8 hole is13 mm, and dilution hole row is 45 mm.

Figure 1 Inner Linear Combustion Chamber

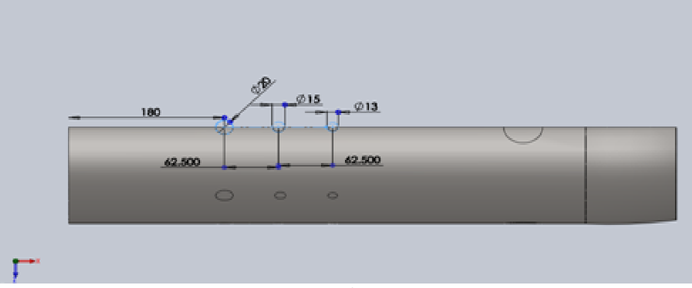


Figure 2 Dimensions of the Geometry used

In 3D computer graphics, 3D modeling is the process of developing a mathematical representation of any three-dimensional surface of the object via specialized software. The product is known as the 3D model.

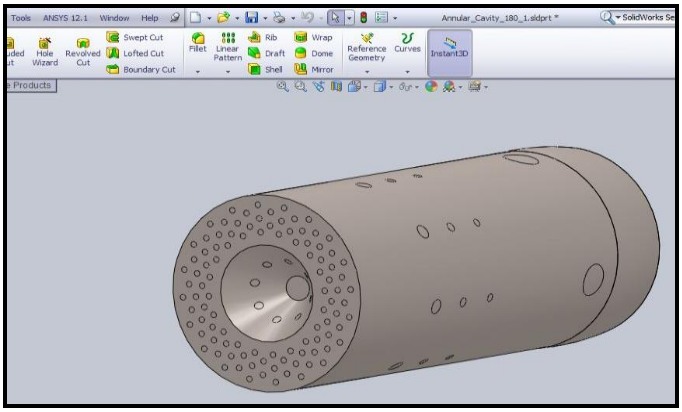


Figure 3 Modeling of Combustion Chamber

Figure 3 shows that by using the all dimension available, the Cavity model of a combustion chamber inner liner, which is to be drawn in Solid Works 2009 X64 editions SPO. This modeling is generated fluid flow volume after transferring to the workbench.

Meshing is the method to define and brake up the model into small elements. In general a finite element model is defined by a mesh network, which is made up of the geometric arrangement of elements and nodes. Nodes represent points at which features such as displacements are calculated. Elements are bounded by set of nodes, and define localized mass and stiffness properties of the model.

Once geometry has been imported, the fluid domain is meshed in workbench's ICEM CFD. Specify the different parameters like element type and size of element. After the select the option create mesh is automatic, tetrahedral, hexahedral, wedge, ect. Figure shows the change the mesh size.

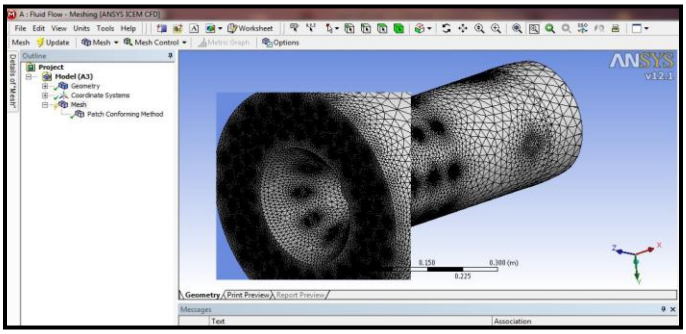


Figure 4 Meshing [ANSYS ICEM CFD]

Figure 4 shown the after completing the create a mesh. The details of the meshed parameters are given in meshing report shown in table 1.

Table 1 Number of Element and Node in mesh

Element Type	Tetrahedral
Number of Nodes	198466
Number of Elements	1072546

After meshing the boundary conditions are specified. Here we are simulating the combustion chamber inner liner of the 30 MW power plant of the Utran. The data obtained from the power plant are used to determine the boundary conditions for the analysis.

Boundary condition calculation from RQL theory

Data from the Utran power station (current working condition)

Gas flow to = 2.403 kg/sec
Fuel gas pressure = 18.64 kg/cm^2
Fuel gas temperature = 100°C
Compressor air exit temperature = 84°C
Compressor air exit pressure = 9 bar
Mass flow rate of air $M_{a\,(total)}$ = 11.34 kg/sec
Mass flow rate of fuel M_f = 0.24 kg/sec
Velocity of fuel = 39.93 m/sec
Mass flow rate fuel air ratio $(M_f/M_a)_{actual}$= 0.0212

I.RESULT AND DISCUSSION

In order to understand the flow of flue gases inside the combustion chamber and the formation of exhaust gases simulation using the ‘ANSYS CFX’ software package is done. As the condition of gases flowing is turbulent, k-ε turbulent model is used to simulate the flow which is readily available in the said software. Best condition which says that complete combustion resulting into less emission is the primary objective of the said work which is achieved by varying different boundary and geometrical conditions. A simulation is considered to have run through its course after a certain number of iteration when the result of the simulation converges, which means that the result of the final iteration is within the specified level of variance compared to the few iterations before it. First of all check verification of model and use the same for further analysis.

Verification of Model

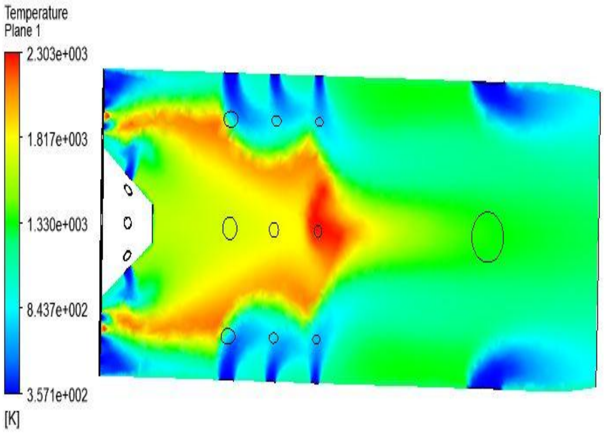


Figure 5 Flame Generation in XY Plane

Boundary condition is applied for actual working condition power plant and equivalence ratio is Ø=1.4. Obtain the result shown in figure 5 flame generation in temperature couture of XY plane generates along the center line. The peak gas temperature is located in the primary reaction zone and after secondary air supplied for the exit of combustor temperature is decrease. The maximum gas temperature for methane combustion is 2046 K and exit temperature is 1230 K. Verification of model is required to obtained result are compared with actual plant condition. Actual plant working condition exit temperature is 1273 K. For the verification of the combustion model, the numerical flame exit temperature for methane combustion was compared to the plant exit flame temperature. CFD and plant results qualitatively match while quantitatively the results differ by around 20%. Our model verified because both result are near about same.

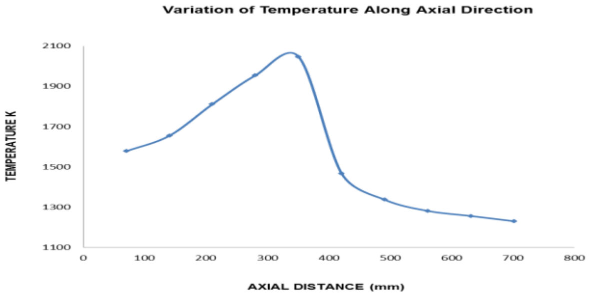


Figure 6 Profile of Temperature Along Center Line

Burning of fuel inside the combustion chamber leads to generation of high flame temperature. As the temperature at the end of the combustion is having limitation by the material used in turbine blades. The peak gas temperature should be in the primary reaction zone and thereafter the gas temperature decreases after the primary reaction zone due to the dilution of the flame with the secondary air. As shown in figure 6 the temperature is varying from inlet to the exit of combustion chamber and it is observed that exit temperature is 1230 K. This temperature is maintained so that turbine blade has no effect.

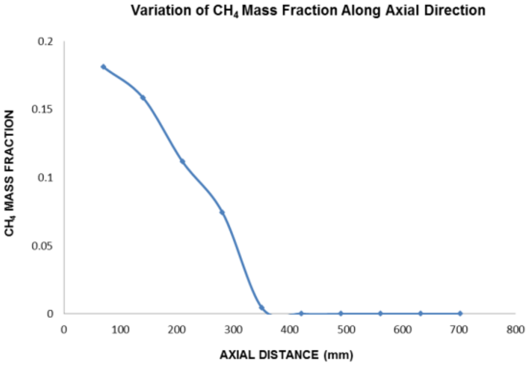


Figure 7 Profile of CH4 mass fraction along center line

Fuel injected in the combustion chamber is methane. Complete combustion of fuel is must in any combustion process as it will lead to less emission. Methane combustion is a chemical reaction that occurs from burning methane gas which is an odorless and colorless fossil fuel causing high amounts of heat and pressure to be produced. Methane by itself cannot be caused to burn effectively without using oxygen as an additive. This combination is what causes methane to begin burning, which is called methane combustion. What is the variation in quantity of fuel as it progress in the combustion process is very difficult to visualize practically but with the help of simulation we can find what mass fraction of fuel is burnt and what is remain as combustion progresses. We can clearly observe that value of mass fraction is higher near to fuel injection zone because, pure fuel is injected there. But we move along center line of combustor, its concentration will significantly reduce due to more and more methane will utilize for combustion Figure 7 shows that at axial distance z=350 mm which representing downstream of secondary air inlet zone, value of mass fraction of methane is approaching to zero. At outlet of combustor CH_4 value is also zero. This indicate that complete combustion has done, there is no fuel present at that region.

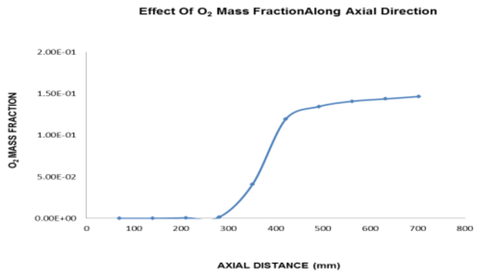


Figure 8 Profile of O2 mass fractions along center line

Shown in figure 8 It is quite observe that concentration of O_2 is higher at exit region of combustor compare to center region because of air as oxidant entering from secondary air inlet region and air composition by weight it contains 23.2 % O_2 and 76.8 % N_2. At center region its value is zero up to axial distance z=300 mm the upstream region of secondary air inlet, further movement after the region of secondary air inlet, increment in mass fraction of O2 significantly due to more and more air is introduced from the secondary air inlet holes. O_2 increased in exit condition gradually.

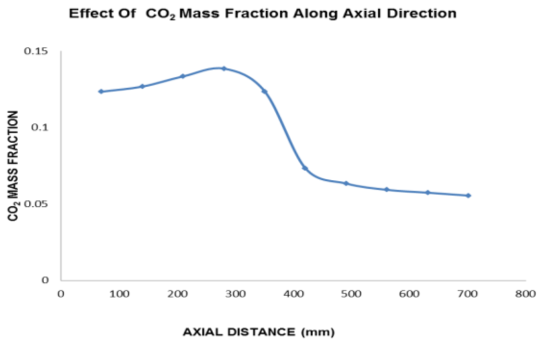


Figure 9 Profile of CO2 Mass Fractions along Center line

Carbon dioxide is a product of the combustion process, and it is primarily mitigated by reducing fuel usage. Carbon dioxide emissions will continue to drop as manufacturers make gas turbine engines more efficient. It is afraid that the atmospheric temperature might be increased on account of the greenhouse effect by CO_2 and the climate might be changed on global scale. Hence it is required to reduce the emission of CO_2 into the atmosphere. Shown in Figure 9 the carbon dioxide (CO_2) produce when the burning the fuel and after the gradually decrease to exit condition.

Shown in Figure 10 the when combustion occurring the H_2O produce. When combustion is occurred CO_2 and H_2O produced the both graph are same for gradually increase mass fraction to intermediate section after reduced to exit condition. Most of the world's nitrogen occurs naturally in the atmosphere as an inert gas contained in air, which consists of approximately 78% N_2 by volume.

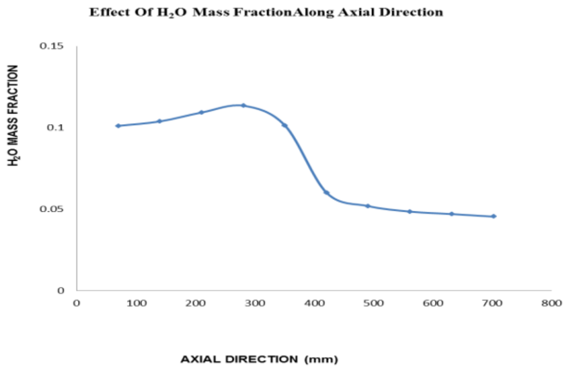


Figure 10 Profile of H2O Mass Fraction along Center Line

NO_x refers to oxides of nitrogen. These generally include nitrogen monoxide, also known as nitric oxide (NO), and nitrogen dioxide (NO_2). The higher turbine entry temperatures required for efficiency improvement tend to increase NO_x production. Shown in figure 11 the nitric oxide is temperature increased value of NO is increased and after secondary air introduces the temperature decrease value of NO decrease. Complex interactions of nitrogen oxides with reactive hydrocarbons and sunlight produce low level ozone, acid rain and smog and legislation reflects the view that any emission of NOx is to be avoided or minimized.

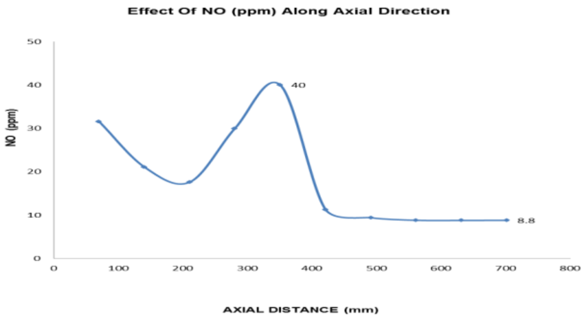


Figure 11 Profile of NO Mass Fraction along Center Line

Emission Nitrogen oxides are affect of the environment so government rules GPCB is limitation of nitrogen oxides 50 ppm. Shown in figure 11 exit condition of NO is 8 ppm so limitation of regulation but also the reduced the NO and produce the minimum Nitric oxide (NO). Plant exit NO_x is 9 ppm.

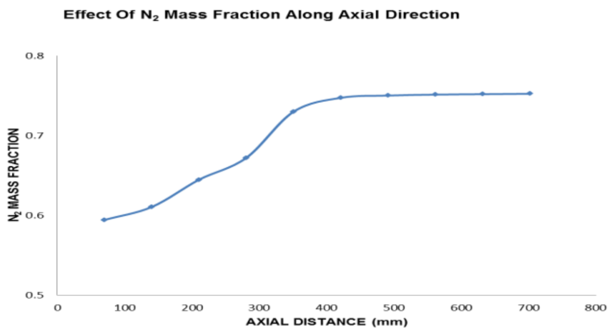


Figure 12 Profile Nitrogen mass Fraction along center line

Figure 12 shows the contour of mass fraction of N2 along variation in profile of mass fraction along center line respectively. It is quite observe from these figure 12 that concentration of N2 is higher at exit region of combustor compare to center region because of air as oxidant entering from secondary air inlet region and air composition by weight it contains 23.2 % O_2 and 76.8 % N2. Further along center line its value is increasing in great amount. It is clear from the profile that the peak value of N2 mass fraction is achieved at the outlet of combustor. The maximum value approaches up to 70 % at outlet of combustor.

The velocity profiles show an increasing trend along the length of combustion chamber, but low velocities are encountered in primary zone which is beneficial for combustion stability. Shown in figure 13 velocity profile in axial direction gradually increase to exit condition. Importance of exhaust gas velocity is turbine speed. Figure 13 shows velocity along center line at axial distance 210 mm it gets lowest velocity value due to primary air inlet swirl effect the recirculation flow is created at this region velocity reduces and also axial distance at 430 mm dilution air inlet swirl effect the recirculation flow is created at this region velocity reduces. After that the velocity gradually further movement in axial direction and gets the peak value of velocity at the exit section of combustor.

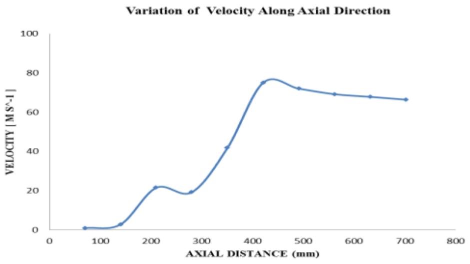


Figure 13 Profile Velocity along the center line

Numerical Simulation using different conditions

As discussed above variation of temperature, velocity, mass fraction CH_4, CO_2, O_2, H_2O, NO, N_2 etc. as per one of the working condition of the plant. Now to obtain the best operating condition of the combustion chamber its boundary condition is changed. This will help us in analyzing the working of combustion chamber at different boundary conditions. After chaninging the boundary conditions its geometry is also varied such as 2 mm decrease the diameter of primary and intermediate hole, 2 mm increase the diameter of primary and intermediate hole**,** and dilution hole shifted to 20 mm. To get the refinement in results the mesh size is also changed.

Variation in operating Condition

The combustion chamber of gas turbine unit is one of the most critical components
to be designed. The reason behind the designing of the gas turbine combustion chamber being critically important is a need for stable operation over wide range of
air/fuel ratios.

The operating condition of combustion chamber is varied by changing the boundary condition which is by changing in equivalence ratio. Table 2 All boundary condition apply and shown the different flow characteristic effect the inner liner inlet to exit condition.

At this changes equivalence ratio the steady state centerline temperature, measured along the length of combustion chamber. Now the equivalence ratios are varied and similar steady state observations pertaining to temperature are recorded.

Table 2 Boundary Condition for Different Equivalence Ratio

Sr. no	Overall Equivalence ratio ($\varnothing$)	$\varnothing_{pz}$	$\varnothing_{iz}$	Ma_{front} Kg/sec	Ma_{pzh} Kg/sec	Ma_{izh} Kg/sec	Ma_{dzh} Kg/sec
1	1.4	0.8	0.6	2.955	2.217	1.7236	1.04
2	1.5	0.8	0.7	2.7586	2.4138	0.7390	2.023
3	1.3	0.7	0.6	3.1830	2.7283	0.98525	1.04
4	1.8	1	0.8	2.298	1.8399	1.0344	2.7675
5	1.6	1	0.6	2.5862	1.5517	2.7586	1.04
6	1.8	-	0.6	2.298		4.598	1.042

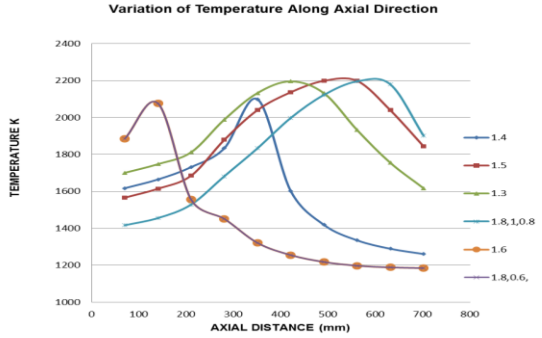
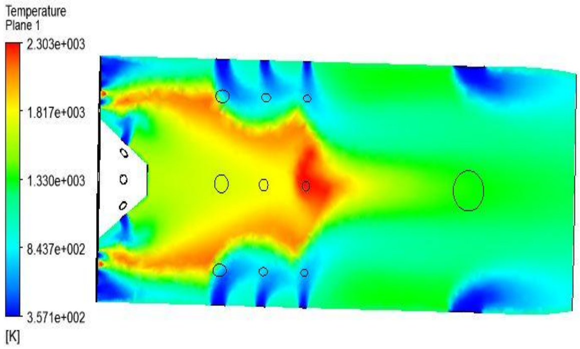


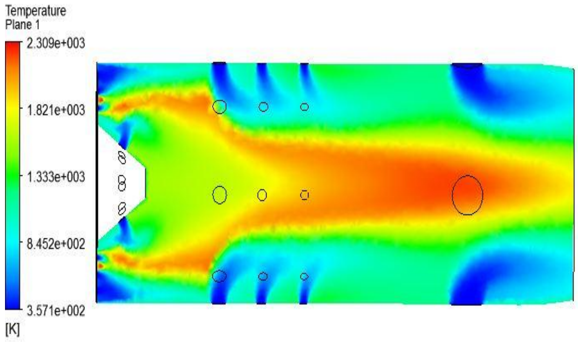
Figure 14 Profile of temperature at Different Boundary Condition along center line

The equivalence ratios have been varied as 1.3, 1.4, 1.5, 1.6, and 1.8. Figure 14 shows the lighted combustion chamber using methane as fuel. What is the variation in quantity of fuel as it progress in the combustion process is very difficult to visualize practically but here see shown in figure 15 with the help of simulation we can find what mass fraction of fuel is burnt and what is remain as combustion progresses. Figure 14 shows effects of profile changes of temperature from inlet to exit combustor at the different equivalence ratio varied from 1.3 to 1.8. Among all the equivalence ratios only one has required exit temperature. For this boundary condition of Ø=1.4, inlet temperature is 1600 K and exit temperature is 1260 K which is desirable at turbine inlet.

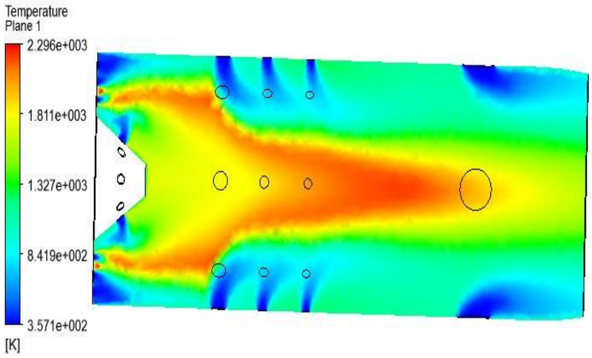
For remaining all other boundary conditions high temperature at inlet and exit condition exists. As shown in figure 14 for Ø=1.3 and Ø=1.5 both produce high exit temperature, which affects the turbine blade material. For the condition of Ø=1.8, $Ø_{pz}$=1, $Ø_{iz}$=0.8 inlet temperature is 1400 K and exit temperature 1900 K this much high exit temperature for combustor i.e. turbine inlet temperature cannot be sustained by turbine blade.



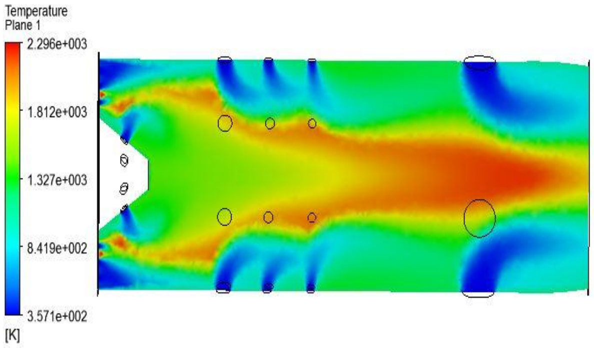
(A) $\varnothing = 1.4$



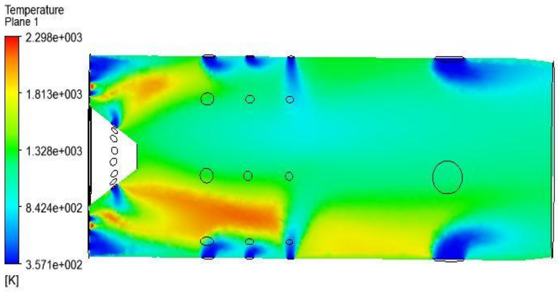
(B) $\varnothing = 1.5$



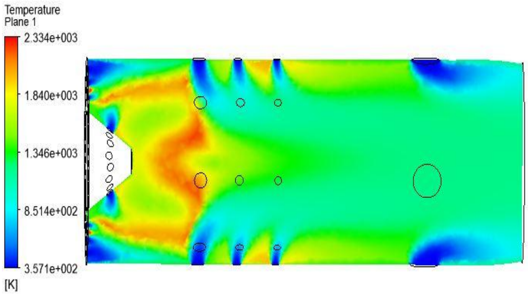
(C) $\varnothing = 1.3$



(D) $\varnothing = 1.8, \varnothing pz = 1, \varnothing iz = 0.8$



(E) $\varnothing = 1.6$



(F) $\varnothing = 1.8, 0.8$

Figure 15 Different Equivalence Ratio Flame Generations

For Another two boundary conditions inlet temperature is too high. Higher exit temperature increases NO_x emission. As shown in figure from 5.11 among all the boundary condition, for Ø=1.4, flame generates required exit temperature. For this boundary condition maximum temperature is 2200 k as shown in figure 15 (A)

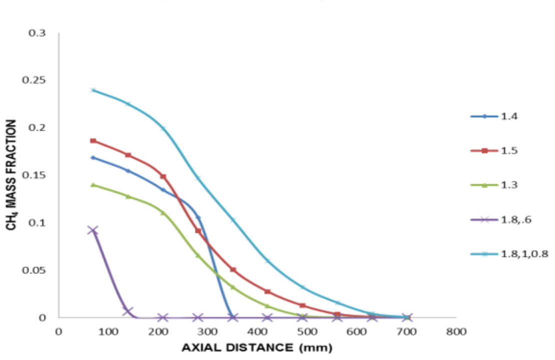


Figure 16 Profile of CH4 at Different Boundary Condition along center line

Changes equivalence ratio centerline methane mass fraction, measured along the length of combustion chamber. Figure 16 shown in change the equivalence ratio methane (CH_4) fuel rate also change. The CH_4 mass fraction totally burning the middle section is only one condition $\varnothing = 1.4$ equivalence ratio. Another all boundary condition is fuel is not burning properly and produced the unburned hydrocarbon also increase the emission value.

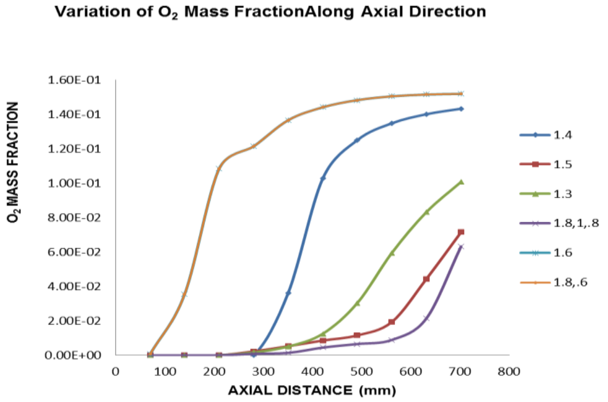


Figure 17 Profile of O2 at Different Boundary Condition along center line

Concentration of O_2 is higher at exit region of combustor compare to center region because of air as oxidant entering from secondary air inlet region and air composition by weight it contains 23.2 % O_2 and 76.8 % N_2. Changes equivalence ratio centerline oxygen (O_2), measured along the length of combustion chamber. At center region its value is zero up to axial distance z=300 mm the upstream region of secondary air inlet, further movement after the region of secondary air inlet, increment in mass fraction of O_2 significantly due to more and more air is introduced from the secondary air inlet holes. Shown in figure 17 oxygen (O_2) at different boundary condition is along axial direction. Primary zone areas till the Oxygen and fuel both are mixed properly and burning so this condition null condition of Oxygen. O_2 increased in exit condition gradually.

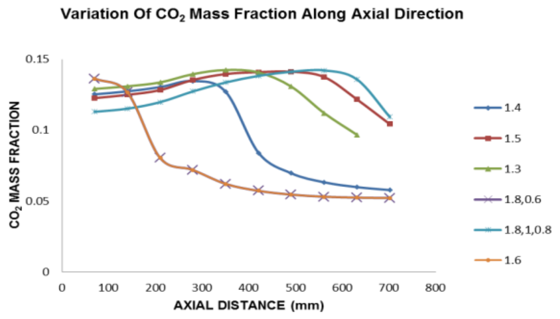


Figure 18 Profile of CO2 at Different Boundary Condition along center line

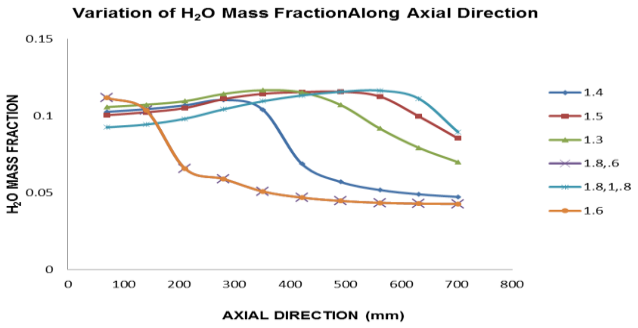


Figure 19 Profile of H2O at Different Boundary Condition along center line

Figure 18 & 19 shown in carbon dioxide (CO_2) and water (H_2O) produces the different equivalence ratio. In exit condition at equivalence ratio $\varnothing = 1.4$ CO_2 and H_2O mass fractions are produced minimum. CO_2 and H_2O mass fraction are also produced minimum at boundary condition $\varnothing = 1.6$ and $\varnothing = 1.8, \varnothing iz = 0.6$ but this boundary condition are not valid because flame are not properly generated.

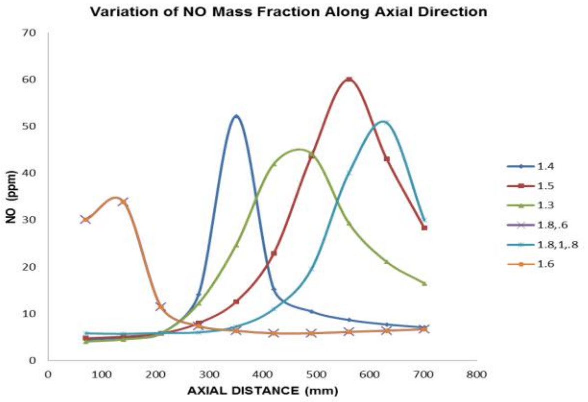


Figure 20 Profile of NO at Different Boundary Condition along center line

Shown in figure 20 Nitric oxide (NO) is produced the different equivalence ratio. In exit condition NO mass fractions are produced minimum at equivalence ratio $\varnothing = 1.4$. NO mass fraction are also produced minimum at boundary condition $\varnothing = 1.6$ and $\varnothing = 1.8, \varnothing iz = 0.6$ but this boundary condition are not valid because flame are not properly generated.

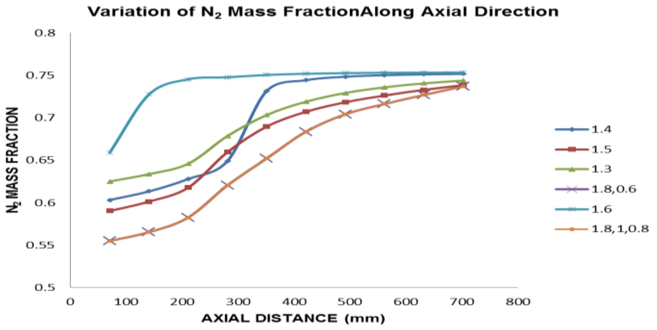


Figure 21 Profile of N2 at Different Boundary Condition along center line

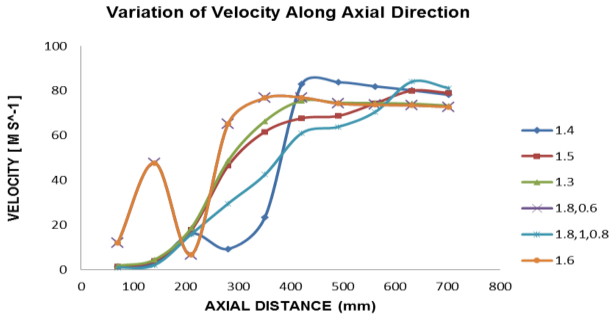


Figure 22 Profile of Velocity at Different Boundary Condition along center line

Figure 21 shows the contour of mass fraction of N2 along variation in profile of mass fraction along center line respectively. It is clear from the profile that the peak value of N2 mass fraction is achieved at the outlet of combustor. The maximum value is achieved at equivalence ratio $\varnothing = 1.4$ boundary condition. N2 mass fraction are also produced maximum at boundary condition $\varnothing = 1.6$ and $\varnothing = 1.8, \varnothing iz = 0.6$ but this boundary condition are not valid because flame are not properly generated.

Figure 22 shows profile of velocity at different boundary condition along center line. We know the here velocity along center line at axial distance 210 mm it gets lowest velocity value due to primary air inlet swirl effect the recirculation flow is created at this region velocity reduces and also axial distance at 430 mm dilution air inlet swirl effect the recirculation flow is created at this region velocity reduces in this situation follow only one boundary condition is $\varnothing = 1.4$ achieved. Another boundary condition are not achieved this velocity.

CONCLUSION

This Paper was carried out to study the characteristics of an inner liner of the Can type of combustion chamber. The design data were available from the Utran power Plant. The boundary conditions applied were the actual boundary conditions of the Utran power plant. RQL (Rich Burn Quick Mix Lean Zone) theory was used in the analysis. Methane is used as the fuel in the combustion chamber. The analysis results are compared with the Actual working conditions of the Power Plant, the exit temperature of the inner liner of can type of the combustion chamber is verified.

It is found that the results obtained from CFD analysis shows close matching with the actual conditions of the inner liner of the combustion chamber.

In this analysis the equivalence ration (Ø) is varied and effect was studied on different parameters. It was found that the amount of NO_x, exit temperature, maximum temperature, inlet temperature, and other exhaust emission parameters varies to great extent than the actual conditions. And it was found that for value of Ø = 1.4 the CFD results are closely matching with the actual conditions.

REFERENCES

1. Hassa C., Miguels C.E. and Voight P.,"Design Principle of Quick Zone of RQL Combustors", *German Aero Space Centre*, DLR, P.P Box 90 60 58 Institute Of Propulsion Technology, D-51170 Köln, Germany May 1998.
2. Christopher O Peterson, William A.sowa, G.S.samuelsen, "performance of a model RQL combustor at elevated temperature and pressure", *University of California*, National Aeronautics and space administration December 2002.
3. Kulshreshtha D. B, Channiwala S. A, Chaudhari K. V & Dikshit S. B, "Design, Numerical Simulation and Experimental Investigations on Can-Annular Type Combustion Chamber for 20 kW Gas Turbine Engine", *International Journal of Dynamics of Fluids*, Vol. 4, No. 2, pp. 175 - 189, 2008.
4. S N Singh, V Seshadri, R K Singh and T Mishra, "Flow characteristic of an annular gas turbine combustor model for reacting flow using CFD", *Journal of scientific & industrial research*, vol. 65, pp. 921-934.
5. Dr. S. A. Channiwala & Dr. Digvijay Kulshreshtha, "Numerical simulation approach as design optimization for micro combustion chambers", *Tenth International Congress of Fluid Dynamics*, December 16-19, 2010.
6. Paolo Gobbato, Massimo Masi, Andrea Toffolo, Andrea Lazarettos, Giordano Tanzini, "Calculation of the flow field and NOx emissions of a gas turbine combustor by a coarse computational fluid dynamics model"10 December 2011.
7. Lefebvre A. H. and Halls G. A., "Some Experiences in Combustion Scaling, AGARD Advanced Aero Engine Testing", AGARD graph 37, 1959, pp.177-204.
8. Alessandro Marini, Alessandro Cappelletti, Giovanni Riccio, Francesco Martelli, "CFD re-design of a gas turbine can-type combustion chamber hydrogen fired", *V European Conference on Computational Fluid Dynamics*, ECCOMAS CFD 2010.