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

Design and CFD Simulation of Annular Combustion Chamber with Kerosene as Fuel for 20 kW Gas Turbine Engine

K. V. Chaudhari, D. B. Kulshreshtha, S.A. Channiwala
(Department of Mechanical Engineering, GEC Bharuch, Guj. Tech. University, Gujarat, India
(Department of Mechanical Engineering, C.K.P.C.E.T., Guj. Tech. University, Gujarat, India
(Department of Mechanical Engineering, S. V. N. I. T., Gujarat, India)

ABSTRACT

The challenges in designing high performance combustion systems have not changed significantly over the years, but the approach has shifted towards a more sophisticated analysis process. A technical discussion on combustion technology status and needs will show that the classic impediments that have hampered progress towards near stoichiometric combustion still exist. The paper presents the design for annular combustion chamber for annular gas turbine for 20 kW power generation wit kerosene as fuel and its also include the CFD simulation is carried out with the CFD tool ANSYS CFX. It also give results from CFD simulation of temperature distribution at centerline of liner, at liner wall and at exit of combustion chamber.

Keywords – Annular Combustion Chamber, CFD, Gas Turbine, Simulation

I. INTRODUCTION

The technical development of industrial gas turbine gained momentum in the past four decade. In parallel to the technical development, new environmental restrictions have been issued. These restrictions increase considerably the burden on the designer of gas turbine combustion chambers. Greater power density, increased output and cleaner operation have been the central design goals for gas turbine over the past 25 years, in spite of the design complexity; gas turbines offer some clear advantages: high efficiency, low emissions, low installation cost and low power generation cost [1].

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. Also there is a close resemblance among present day combustion chamber to their predecessors. The present day combustion chamber exhibits 100% combustion efficiency over their normal working range demonstrates substantial reduction in pressure loss and pollutant emissions and allows a liner life that is significantly longer than those of many other engine components [1, 2].

The present paper discusses the design of annular type combustion chamber for small gas turbine application. The CFD simulation is carried out for the designed chamber using commercial CFD tool; ANSYS CFX.

II. DESIGN [1]

The design of the combustion chamber was carried out using the initial conditions as obtained from the compressor outlet. The combustor design parameters are listed as from brayton cycle analysis:

Inlet Temperature, T_{03}	:	305 K
Mass Flow Rate of Fuel, m_f	:	8.63×10^{-3} kg/s
Inlet Pressure, P_{03}	:	2.95 bar
Mass Flow Rate of Air, m_a	:	0.48 kg/s
Fuel/Air Ratio	:	8.63×10^{-3} kg/s

The fuel chosen for the engine was kerosene with chemical formula $C_{12}H_{24}$. Other assumed properties of kerosene are [2]:

Density	: 780 kg/m^3
Lower Calorific Value	: 43565 kJ/kg

2.1 Evaluation of Reference Area

The combustion chamber design has to achieve with many constraints. The overall size is dictated by compressor and turbine. The combustor has to depend on the compressor exit conditions and the combustion exit conditions should decided on the required turbine inlet conditions for maximized turbine performance.

The combustion chamber for 20 kW gas turbine engine is designed using two different considerations.

2.1.1 *Aerodynamics*

Aerodynamic process plays a vital role in the design and performance of gas turbine combustion systems. It is probably no great exaggeration to state that when good aerodynamic design is allied to a matching fuel injection system, a trouble-free combustor requiring only nominal development is virtually assured. This means that aerodynamics leads to thermodynamically ideal constant pressure combustion.

2.1.2 *Chemical*

Chemical consideration is used to size the combustion chamber to allow as efficient combustion as possible. Combustion inefficiency represents waste of fuel, which is clearly unacceptable in view of the world's dwindling oil supply and escalation of fuel costs. Another important consideration is that combustion inefficiency is manifested in the form of undesirable and harmful pollutant emissions, notably unburned hydrocarbons and carbon monoxide. These implications relate the combustion efficiency with the casing area of the combustor, which in turn relates it with the liner diameter and performance of the combustion chamber. There are three types of controls as far as combustion is concerned.

1. Reaction Rate Controlled System
2. Mixing Rate Controlled System
3. Evaporation Rate Controlled System

Of the above mentioned controlled system, the reaction rate controlled system is considered. The burning velocity model described by Lefebvre [1] is used to determine the casing area.

The above mentioned considerations of aerodynamic and chemical will give the reference areas for the combustion chamber. The reference area is the casing area for the combustion chamber. The annulus casing area calculated from the above criteria is

$$A_{ref} = 0.0259\ m^2$$

2.2 Liner Area Calculations

It might appear advantageous to make liner cross section area as large as possible since these results in lower velocities and longer residence time with in the liner, both of which are highly beneficial to ignition, stability and combustion efficiency. For a given casing area, the increase in the liner area can be obtained only with reduction in annulus area. This raises the annulus velocity and lowers the annulus static pressure, thereby reducing the static pressure drop across the liner holes. This is disadvantageous as high static pressure drop is required to ensure air jets entering the liner from adequate penetration and sufficient turbulence intensity to promote rapid mixing with the combustion products. A satisfactory criterion for mixing performance is that ratio of the static pressure drop across the liner Δp_L to the dynamic pressure of the flow in the combustion zone q_{pz} should be high. If the ratio of the liner cross sectional area, denoted by $\hat{k}$, then the optimum value of $\hat{k}$ will be such that it gives the highest value of $\Delta p_L / q_{pz}$.

It can be shown that

$$\frac{\Delta p_L}{q_{pz}} = 1 + \frac{T_3}{T_{pz}} \frac{\hat{k}^2}{m_p^2} \left\{ \frac{\Delta P_{3-4}}{q_{ref}} - \frac{(1-m_{sn})^2 + \lambda \left[r^2 (1-\hat{k})^2 - 1 \right]}{(1-\hat{k})^2} \right\} \tag{1}$$

The above equation is used to evaluate $\Delta p_L / q_{pz}$ in terms of $\hat{k}$ by inserting the different parameters. The optimal value of $\hat{k}$ is then obtained by plotting $\Delta p_L / q_{pz}$ against $\hat{k}$. Using this criterion the liner area is calculated as:

$$A_L = 0.01594\ m^2$$

2.3 Liner Air Admission Holes

The need of the liner holes is to provide enough air in the primary zone for complete combustion primary zone equivalence ratio of 0.9 [3,4,5] selected, to provide enough air to the cooling the products of combustion and to provide a uniform temperature profile at the exit in the dilution zone and to cool the liner wall configuration.

The diameter of the air admission holes depends on the maximum penetration required. The effective diameter of the holes will be calculated by the following equations:

$$\frac{Y_{\max}}{d_j} = 1.15 \left(\frac{\rho_j U_j^2}{\rho_g U_g^2} \right)^{0.5} \sin\psi \tag{2}$$

The number of holes can be calculated using one of the forms of continuity equation.

$$n d_j^2 = \frac{15.25\, \dot{m}_j}{\left(P_3 \, \Delta P_L / T_3 \right)^{0.5}} \tag{3}$$

The geometrical diameter can be found using the coefficient of discharge through the air admission holes.

$$d_h = \frac{d_j}{C_D^{0.5}} \tag{4}$$

For the annular combustion chamber the holes of primary zone was selected on the basis of no of fuel injectors used ant they are in double of the no of fuel injectors for getting uniform temperature quality at the exit of the chamber.

Using the above mentioned criteria, the number of holes and their diameter for liner of the annular combustion chamber for different zones are given as:

TABLE 1: Geometrical parameters of air admission holes

Zone	Diameter	Number
Primary	8.42 mm	16
Dilution	12.75 mm	24
Wall cooling	2.527 mm	420

2.4 Length of Liner

The length of liner is a function of the pattern factor and the liner diameter. The length can be calculated using the following equation:

$$L_L = D_L\left(A\frac{\Delta P_L}{q_{ref}}\ln\frac{1}{1-P.F}\right)^{-1} \qquad (5)$$

Using this equation the liner length is 350 mm. the other parameters like the diffuser length and area ratio can be calculated using the continuity equation. The wireframe model of the combustion liner is shown in Fig 1.

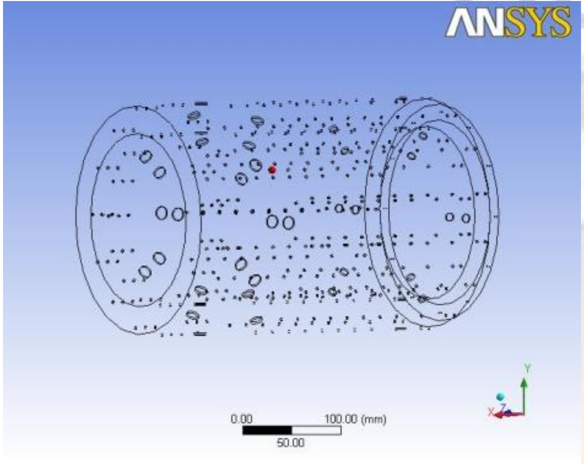

Figure 1: Wireframe model of Liner

III. METHODOLOGY

The designed annular combustor 3-D model shown in Fig. 2 is simulated by CFD tool CFX.

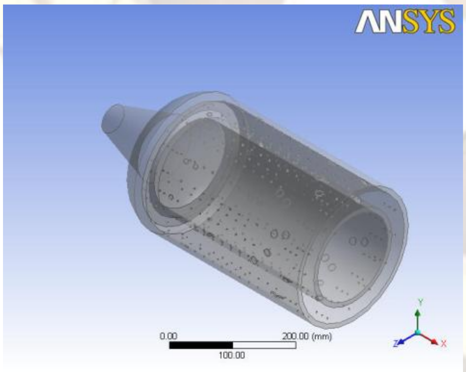

Figure 2: 3-D model of Annular Combustor

3.1 Numerical Simulation

A commercial CFD tool ANSYS CFX is used for the numerical analysis. However, close inspection suggest that many aerodynamic features are common to all systems. In the diffuser and annulus the main objectives are to reduce the flow velocity and distribute the air in prescribed amounts to all combustor zones, while maintaining uniform flow conditions with no parasitic losses or flow recirculation of any kind. Within the combustor liner itself, attention is focused on the attainment of large-scale flow recirculation for flame stabilization, effective dilution of combustion products, and efficient use of cooling air along the liner walls.

Successful aerodynamic design demands knowledge of flow recirculation, jet penetration and mixing, and discharge coefficients for all types of air admission holes, including cooling slots.

The study of flow pattern in the combustion chamber is of vital importance for its successful operation. Fortunately, recent advancement in computational fluid dynamics has made it possible to visualize flow under any condition.

In present study the same approach has been adopted.

3.2 Combustor Model

` For the analysis of the combustion chamber, the commercial CFD code CFX has been used in order to predict the centerline and the wall temperature distribution as well as combustion phenomena.

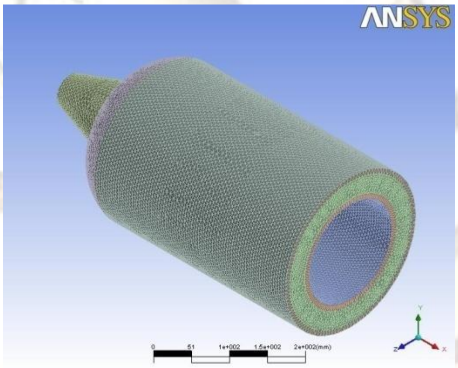

Figure 3: Mesh model of Annular Combustor

Grid generation is very important and time consuming part of the work that has to be done before starting any CFD calculations. For this investigation, the grid generation has been done in CFX- Mesh. The mesh is composed primarily of tetrahedral mesh elements of the numbers of elements were about 5,00,000. The mesh size varies from 0.9 mm at the air admission holes to 6 mm along the length. Fig. 3 shows the generated mesh of the annular chamber using CFX Mesh.

For the 3D calculations with CFX, the adiabatic system model was used because of the wall cooled casing due to large air mass flow.

3.3 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 liner and casing, and from casing into different zones through air admission holes and cooling slots. Such condition is the exact replica of the real case experimentation, 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 the liner.

The boundary conditions are mass flow rate of air at the diffuser inlet as 0.48 kg/s and mass flow rate of the fuel at the fuel injectors as 8.63 x 10^{-3} kg/s. The other boundary condition is pressure of 3 bars at air inlet.

The simulation was specifically targeted to analyze the flow patterns within the combustion liner and through different air admission holes, namely primary zone, dilution zone and wall cooling and from that study the temperature distribution in the liner and at walls as well as the temperature quality at the exit of the combustion chamber using CFX.

IV. RESULTS

Fig. 4 gives the stream lines for wall cooling holes. High velocities are found near the wall regions of the combustion liner.

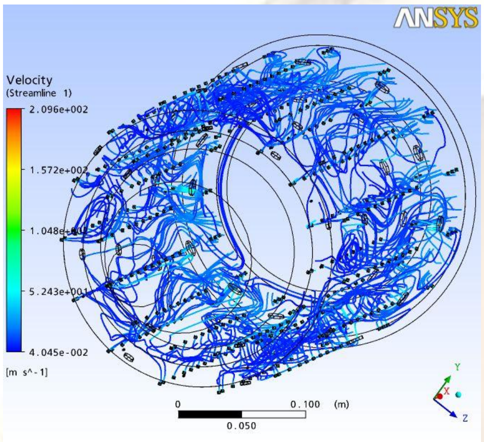

Figure 4: Streamline through the flow domain

This suggests that the static pressure drop through wall cooling holes is more. This advocates for re-design of wall cooling slots. Also, the mass flow rate calculated using CFD is less compared to design value. This may lead to higher liner wall temperatures.

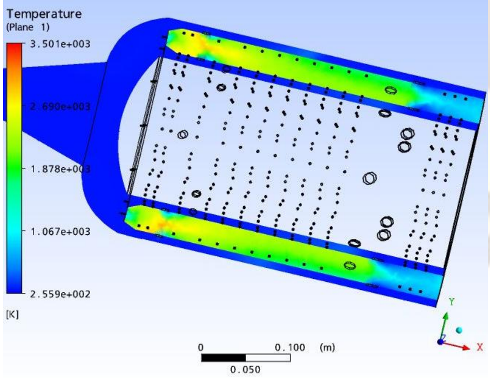

Figure 5: Temperature distribution along the length of liner

Fig. 5 gives temperature distribution in liner, in the entrance region of the combustor, the temperature levels are slightly lower and thereafter increases and reaches maximum and thereafter again decreases. This phenomenon can be explained on the basis of the fact that in entrance region, first fuel mixing and evaporation of fuel takes place thereby indicates reduction in temperature levels, while as evaporation gets completed and more and more air is available from air admission holes, the combustion improves and maxima is reached.

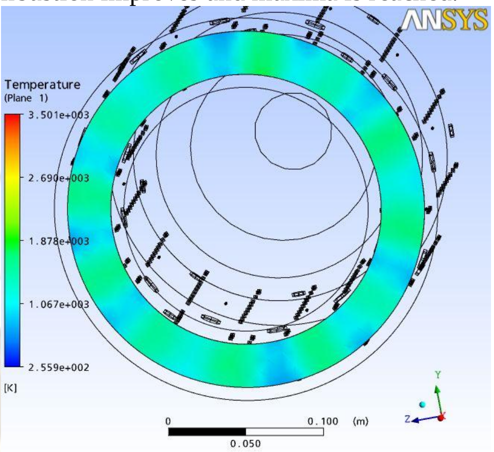

Figure 6: Temperature profile at the exit of the Combustor

Fig. 6 shows, the temperature profile at the exit of the combustor advocates that better mixing in the dilution zone is achieved but due to the air is not properly available for combustion of fuel in primary zone due to less number of primary holes than fuel injectors not uniform distribution of air supply achieved through primary holes in the primary zone found in liner less numbers of holes in the primary zone.

V. CONCLUSIONS

The design of combustion chamber and numerical investigations carried out of annular type combustor. The k-ω model used for analysis [6, 7] and also the mean temperature, reaction rate, and velocity fields are almost insensitive to the grid size [8]. The streamlines from wallcooling holes are not provide proper wallcooling suggest redesign of it. The temperature profile in the annular type combustion chamber is not uniform at exit of the combustor but form Fig. 5 dilution is achieved better so not uniform distribution of air takes place near all fuel injectors in primary zone which suggest primary holes are taken twice the number of injectors so uniform air distribution near each injectors and uniform temperature distribution at exit of chamber was achieved

The numerical simulation state that the flame was touching the liner so chances of burning of combustor is possible if it is made which suggest that durability of chamber liner is decrease.

REFERENCES

[1] Lefebvre A.H., *Gas Turbine Combustion Chamber* (2^{nd} ed., USA: Taylor and Francis, 1999).

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

[3] D. B. Kulshreshtha, Dr. S. A. Channiwala, and Saurabh Dikshit, *Numerical Simulation of Tubular Type Combustion Chamber, Proceedings of ASME International Mechanical Engineering Congress and Exposition*, Florida, 2006.

[4] J. Odgers, *Current theories of combustion within gas turbine chambers, Trans. Am. Soc. Mech. Eng.*, pp 1321 – 1338, 1973.

[5] B. Varatharajan, M. Petrova, F. A. Williams and V. Tangirala, *Two-step chemical-kinetic descriptions for hydrocarbon–oxygen-diluent ignition and detonation applications, Proceedings of the Combustion Institute, 30*, pp.1869 – 1877, 2005.

[6] 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.

[7] L. Eca and M. Hoekstra, *On the Grid Sensitivity of the Wall Boundary Condition of the k-ω Turbulence Model, ASME Journal of Fluid Energy, 126*, 2004.

[8] G.Boudier, L.Y.M.Gicquel and T. J. Poinsot, *Effect of mesh resolution on LES of reacting flows in complex geometry combustor*, Combustion and Flame, *155*, pp. 196 – 214, 2008.

[9] M. Coussirat, J. van Beeck, M. Mesters, E. Egusguiza, Buchlin and X. Escaler, *Computational Fluid Dynamics Modeling of Impinging Gas-Jet Systems: Assessment of Eddy Viscosity Models, ASME Journal of Fluid Energy, 127*, 2005.

[10] 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.