INVESTIGATION ON GAS TURBINE COMBUSTION CHAMBER FOR MICRO MIXING USING HYDROGEN AS FUEL

Nilesh N. Patel[1], Digvijay B. Kulshreshtha, Gaurang C. Chaudhari[2*]

[1] Research Scholar, Gujarat Technological University, Ahmedabad, Gujarat, India,

Email: nilesh.patel.mech@scet.ac.in, patelnilesh3800@yahoo.in

[2] C.K. Pithawala College of Engineering and Technology, Surat, Gujarat, India

Email: digvijay1401@gmail.com, gaurang.chaudhari@ckpcet.ac.in

* Corresponding Author

Received: 11.04.2024 Revised: 16.06.2024 Accepted: 24.06.2024

ABSTRACT

Considering the inherent advantages of small-scale combustors compared to the electro thermal batteries such as higher energy density, higher heat and mass transfer coefficients and shorter recharge times, a considerable interest in its development has been observed lately. This paper is an attempt to present the creation of a micro combustor design in which CFD was used to simulate the combustion process. The function of the combustor swirler in the change the overall direction of the air flux in order to improve the mixing with the fuel. Swirl is the possible factor for increasing the turbulent burning velocity at the flame base which leads to stable combustion. Swirl is employed as a support for effective clean combustion and in non-premixed combustion can help in reducing reduction of pollutant emissions, mostly that of NOx. Two different streamlines from the gas turbine's combustion chamber's inlet diffuser portion. Flow is split into two streams, one going through the liner and the other through the casing. Most of the mixing takes place close to the walls, some of the fuel remains in the chamber's middle. The results shows that when flame propagates along the length of chamber, temperature of the flame comes down. The exit temperature profile is nearly unvarying, which is a crucial parameter in reducing the thermal stresses on the turbine blades.

Keywords: *Micro Gas Turbine, Micro Mixing, Hydrogen, Swirler, CFD*

1. INTRODUCTION

Contemporary experiments and the gas turbine combustors' theoretical study have been greater comprehension of the actual processes that are taking place inside combustor chamber. The simulation tools are useful for giving information about a particular design that is being simulated, but they offer no indication of the optimum designs.

The process of an exothermic chemical reaction between an oxidant and fuel that results in the production of heat and the conversion of chemical species at the microscopic level is known as micro-combustion.

One of the most important components to be constructed for a gas turbine unit is the combustion chamber. Because of the essential requirement for table operation across a wide range of air/fuel ratios, it has been critically important to design the gas turbine. [1] To ensure that the temperature is consistent at the exit, the combustion device is designed for continuous operation with controlled heat release. This points out that the design of gas turbine combustor is primarily related to stable combustion with high efficiency and better temperature quality at exit [2]. To achieve stable combustion, the design procedure is carried out in such a way that the combustion reaction starts and ends in the primary zone. Afterwards, the next zone helps to attain uniform and sustainable temperatures for turbine blades. One of the important criteria in primary zone design is the mixing procedure of fuel with accurate proportion of air.

Pressure loss, temperature rise and exit temperature are the foremost parameter that define the combustion chamber design. Aerodynamics of flow through combustor defines the pressure loss correlations whereas this pressure loss within the combustor takes place due to skin function losses in the diffuser and casing as well as pressure drop through air admission holes.

The purpose of the current effort was to develop a micro combustor, evaluate its performance, and utilize it for improvements in design. This is simulated using computational fluid dynamics (CFD) models for hydrogen fueled micro combustor.

2. DESIGN OF COMBUSTION CHAMBER

To comprehend the various combustion chamber components, it is necessary to understand the basics of combustion chamber terminology. The cross-section of a typical diffusion flame combustion chamber is shown in Fig 1. The casing and liner regions are the combustion chamber's primary dimensions. The other dimensions depend on these domains and consequently, the following section provides the methodology for design. [3,16]

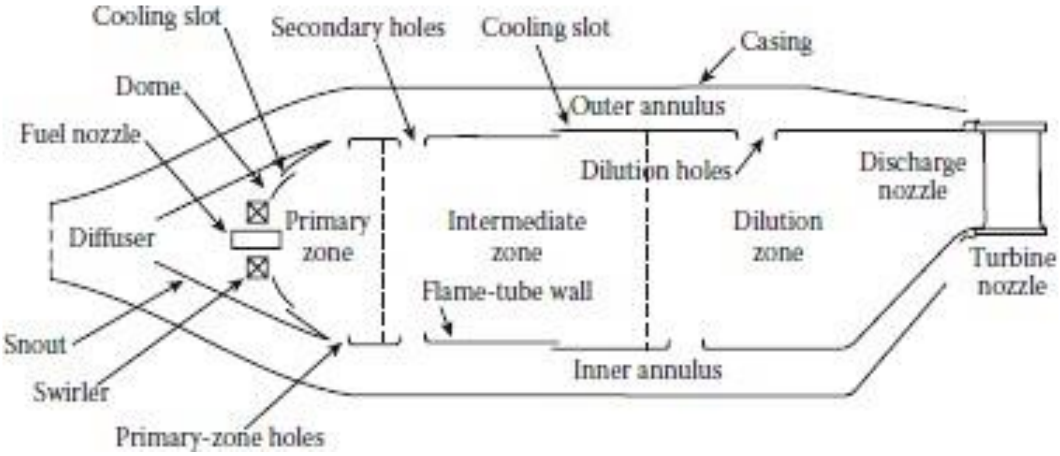


Fig. 1 Typical Combustor Cross Section [3]

Design work has been carried out for the combustor with the known condition ratio, temperature rise and pressure at inlet using hydrogen as fuel. The design of tubular and annular type combustors is initiated by calculating the reference area from both the aerodynamic and chemical perspective.

2.1 Pressure Loss Method

In order to ensure that the aerodynamic consideration for the chamber can be satisfied, the Pressure Loss method is opted. Under such circumstances, the overall pressure loss points out the size of casing. Below is the equation to find the reference area (A_{ref}). [3, 4]

$$A_{ref} = \left[\frac{R}{2}\left(\frac{\dot{m}_a T_i^{0.5}}{P_i}\right)^2 \frac{\Delta p}{q_{ref}}\left(\frac{\Delta p}{p_i}\right)^{-1}\right]^{0.5}$$

2.2 Chemical Method

To get the complete chemical reaction, chemical method helps to obtain the required size. The residence time predicts the size by chemical method. The measure of complete combustion process is combustion efficiency which is correlated to combustion loading parameter with inlet pressure and temperature as: [3, 5 & 6]

$$\theta = \frac{P_i^{1.75} \times A_{ref} \times D_{ref}^{0.75} \times exp^{T_i/b}}{\dot{m}_a}$$

The liner area is decided by two major factors, namely, static pressure drop and low velocity. Low velocity is required in the liner for increasing the residence time and thereby allowing the fuel and air to react. This ensures stable and efficient combustion, the static pressure drop must be high. The jets' penetration from air admission holes and consequent mixing with products of combustion necessitates higher static pressure drop within liner. Consequently, the ratio of liner area to reference area is the function of static pressure drop and mass flow rate entering the liner from swirler.

$$A_L = K_{opt}\, A_{ref}$$

The air into the primary zone was previously selected as 25% of the total flow of which 50% enters from the front of the chamber. The remaining air is admitted through the air admission holes to produce swirling by opposite jets for flame stabilization.

The air distribution for intermediate zone, dilution zone and wall cooling is to be selected from remaining 75% of the overall air supply. The wall cooling air percentage is found to be 30% [7]. The remaining 45% of air is to be divided into dilution 30% and intermediate zone 15%.

The effective diameter for air admission holes is given by the equation [3]

$$n\, d_j^2 = \frac{15.25\, \dot{m}_j}{\left[P_3\, \frac{\Delta p_L}{T_3}\right]^{0.5}}$$

Geometrical diameter (d_h) of holes is then calculated using coefficient of discharge (CD) from equation

$$\therefore\ d_h = \frac{d_j}{C_D^{0.5}}$$

2.3 Length of liner

Length of liner is an important parameter as it dictates the exit temperature quality of the combustor. The exit temperature quality, known as pattern factor, depends upon mixing of air from dilution holes with combustion products. The jets' penetration in the cross flow is determined by the static pressure drop that occurs across the dilution hole. Therefore, the length of liner is dictated by pattern factor and is then a function of:

$$L_L = D_L\left[A \times \frac{\Delta p_L}{q_{ref}} \times ln\left(\frac{1}{1 - P.F.}\right)\right]^{-1}$$

In this case A is a constant, for tubular liners it is 0.07 and for annular liners it is 0.05.

2.4 The sizes of various Zones

The empirical relations for the various zones can be used to evaluate the length of the various zones. Table 1 displays the empirical relationships.

Table 1 sizes of various Zones

Zone	Empirical Relations
Intermediate	$L_{Iz} = 0\ to\ 0.5\ D_L$
Dilution	$L_{Dz} = 0.5\ to\ 1.5\ D_L$
Primary	$L_{Pz} = L_L - (L_{Dz} + L_{Iz})$

2.5 Diffuser Design

Diffuser plays a vital role as it is the entrance of the combustion chamber that allows a compressed air entering from compressor which further mixes with fuel to form a combustible gas; and before mixing with fuel, velocity of has air is required to be reduced as it allows residence time for mixing with fuel which gives better mixing of air with fuel, and therefore diffuser design is an significant task while designing for a combustion chamber.

AR is area ratio, it is a crucial factor in the design of the diffuser and snout for the combustion chamber of a gas turbine since it is directly related to the diffuser's primary function.

For conical units, the length of the diffuser can be calculated from the equation given below.

$$AR = \frac{A_{ref}}{A_1} = 1 + 2\,\frac{L}{R_1}\sin\theta + \left(\frac{L}{R_1}\sin\theta\right)^2$$

Thermodynamic Cycle analysis and performance parameters of individual gas turbine engine components are provided by Saravanamutto et al. [8] the cycle analysis performed by these researchers provides the information needed to design a tubular type combustion chamber and it is displayed in Table 2.

Table 2 – Combustion Chamber Design Parameters

Combustor Inlet Pressure	3.8 bar
Combustor Inlet Temperature	504 K
Air Mass Flow Rate	0.77 kg/s
Fuel Mass Flow Rate	0.004075 kg/s
Combustor Exit Temperature	950 K

The dimensions of the combustion chamber for the hydrogen-fueled micro gas turbine is displayed in Table 3 while Fig 2 shows the dimensional drawing of the combustion chamber.

Table 3 – Combustion Chamber Dimensions

Chamber	Casing area (m^2)	Liner area (m^2)	Air admission holes					
			Primary Zone		Dilution Zone		Wall cooling	
			n	d_h (mm)	n	d_h (mm)	n	d_h (mm)
Tubuler	0.00975	0.00682	18	9.50	11	13.30	320	2.53

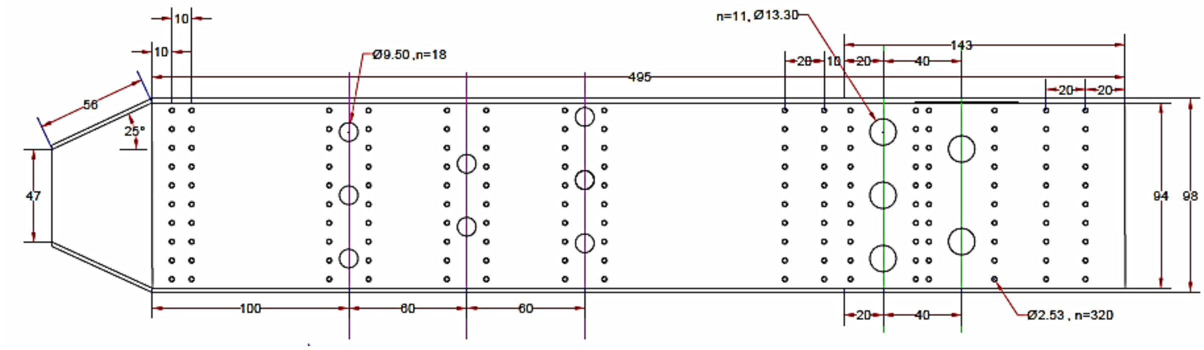


Fig 2 (a) Dimensional drawing of liner

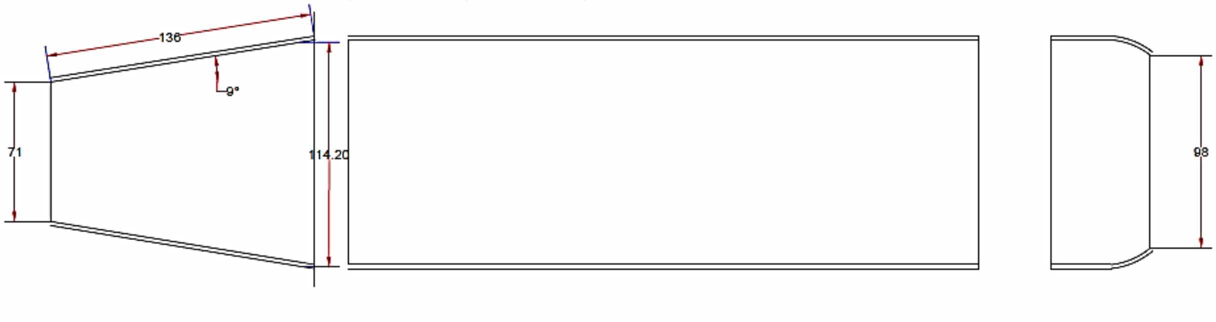


Fig 2 (b) Dimensional drawing of Casing

3. NUMERICAL SIMULATION

Combustion chamber can be analyzed numerically in two different ways. The very first step is to set up input conditions at each air admission hole and at the inlet according to the design conditions. But it is impossible to regulate how the flow is distributed throughout all the different zones. The major disadvantage of using various inputs at various air entrance holes is this. Another method is to provide a single intake at the diffuser's inlet and let the flow naturally separate into a liner and a casing, and then a casing into various zones through air inlet holes and cooling slots. Figure 3 shows the mesh details of combustion chamber and figure 4 and 5 shows the 3D model of combustion chamber and swirler respectively. This scenario is a perfect representation of the real-world experiment, in which the air is delivered at the inlet diffuser at known pressure, temperature, and velocity parameters, and is then permitted to divide between the casing and the liner with fuel injection at liner entrance [17].

3.1 Basic Assumptions and Boundary Condition

- One entrance at the diffuser is used to evaluate the combustion chamber.
- The flow is permitted to separate between the liner and the casing on its own, and from the casing into various zones through air admission holes and cooling slots. [17]
- This situation is a perfect representation of the real-world experiments, in which the air is fed at the inlet diffuser under known pressure, temperature, and velocity conditions before being allowed to separate between the casing and the liner.
- Most scientists who work in the field of computational combustion have chosen k-e model to capture physics off turbulence [9, 10, 11]. Accordingly in the present case also k-e turbulence model is selected.
- In present case, simulation done with fluent, the huge mass flow of air through the annulus, which maintains the wall cooled almost at ambient temperature, justifies the adoption of the adiabatic system model.

3.2 Grid Independency

With the help of the built-in software Design Modular, the three-dimensional flow zones and swirler were modeled. Tetrahedral mesh was used in the ANSYS work bench to create three-dimensional unstructured grids. In areas where strong velocity and pressure gradients were expected, such as the swirler's input and exit, the grid cells were improved.

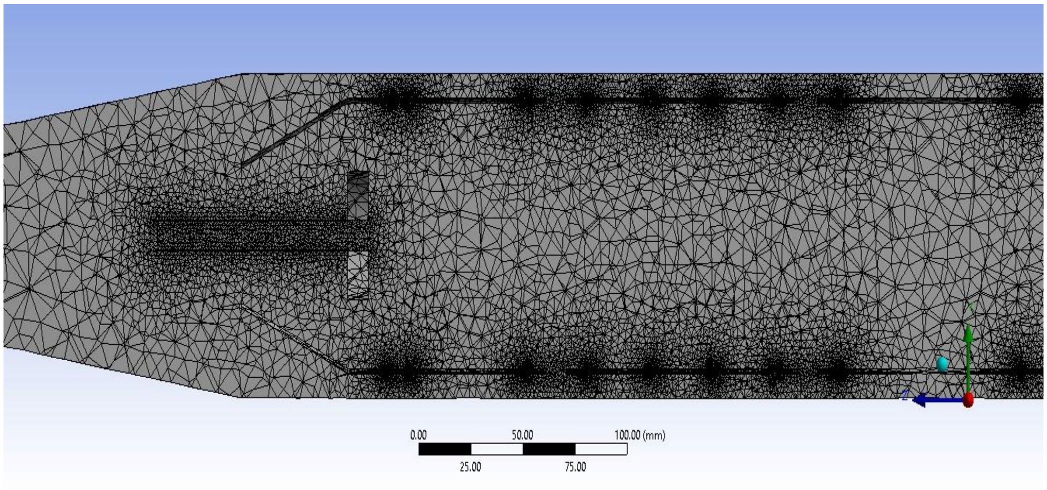
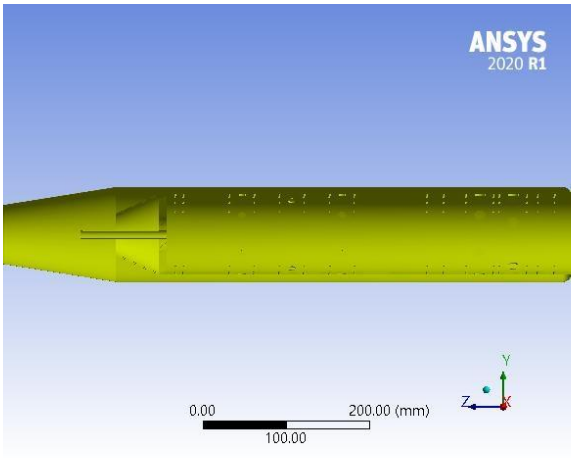


Fig 3 Mesh details

3.3 CFD Modelling

A non-premixed model was adopted because the mixing process takes place in the combustion chamber, which is also a component of the simulated region. The hydrogen and air mixture combustion in the gas phase reaction model.



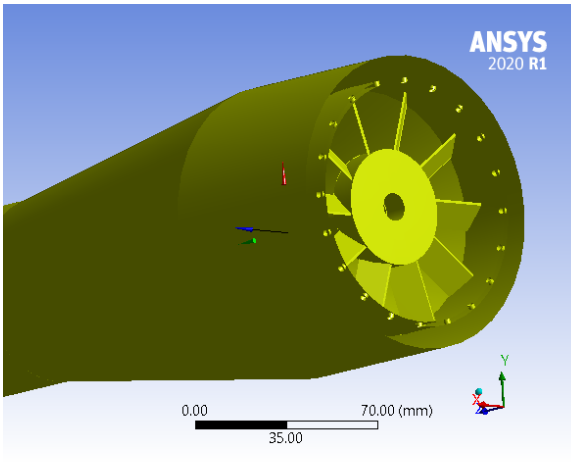


Fig. 4 3D model of Combustion Chamber

Fig. 5 3D model of Swirler

4. RESULTS & DISCUSSION

The major goal is to get the highest possible swirling effect with strong turbulence and no pressure losses.

Figure 6 shows the streamlines from the gas turbine's combustion chamber's inlet diffuser portion. The fact that the flow is split into two streams, one going through the liner and the other through the casing, is interesting to note which is quite logical. The streamlines from the swirler output in Fig. 6 make this fact clear.

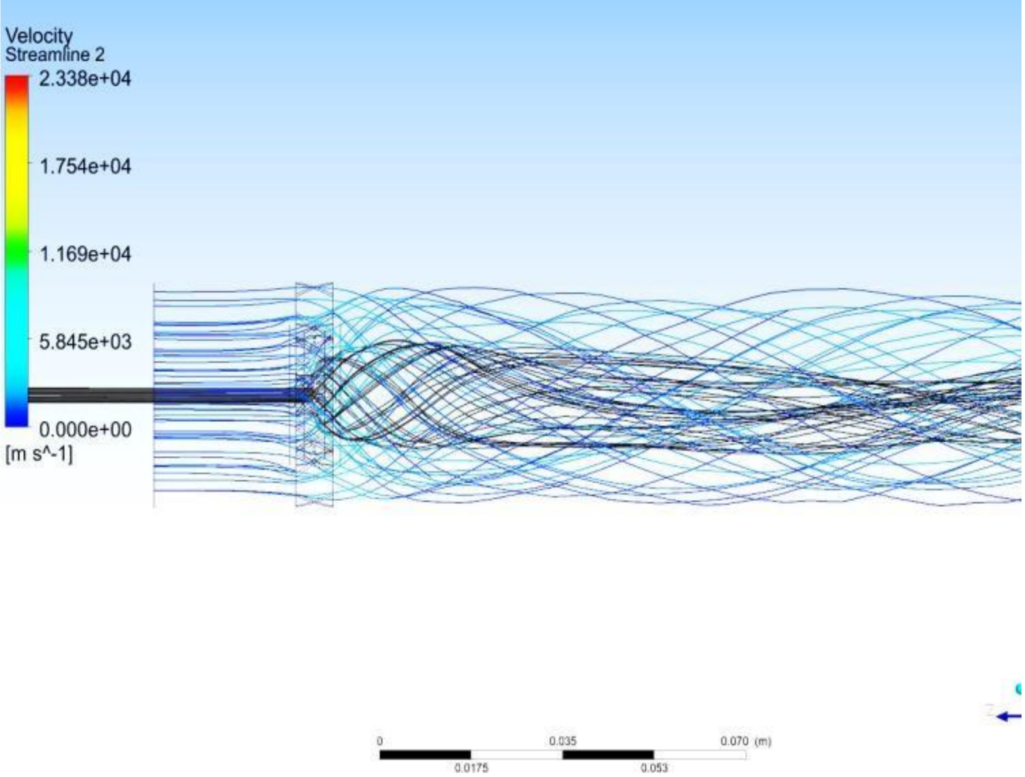
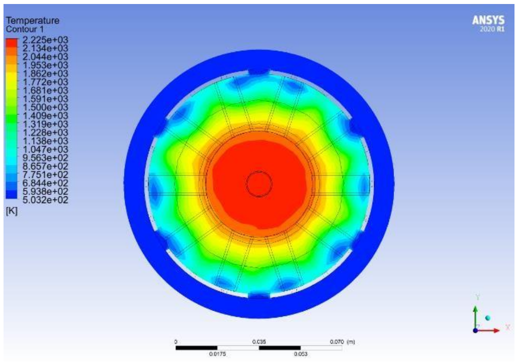


Fig. 6 Velocity streamlines

There is clear complete mixing, which is advantageous for better fuel and air mixing inside the primary zone. At the central core, there is intense mixing and recirculation, which could provide a stable narrow flame. [13, 14].

It is interesting to notice the mixing of the two groups of streamlines. In this most of the mixing takes place close to the walls, some of the fuel remains in the chamber's middle. The mixing of the fuel must occur in the center of the chamber and must be completed in the lowest amount of time possible.



.

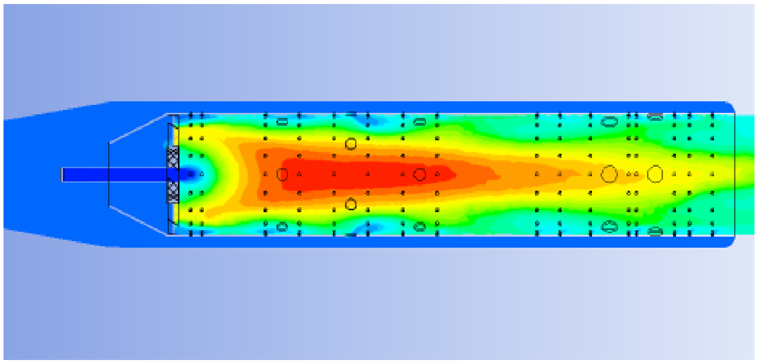


Fig. 7 Temperature Contour swirler

The function of the combustor swirler in the change the overall direction of the air flux in order to improve the mixing with the fuel [12]

Figure 7 shows the flame is developed near the fuel nozzle. Complete mixing is evident which is beneficial for uniform temperature at exit of combustion chamber. Where a consistent temperature gradient is attained at the combustion chamber's exit.

The temperature is lower in the entrance area near the fuel nozzle at first and then rises to its highest point. As more air becomes accessible from the wall cooling zone and dilution zone, the temperature levels drop once more. The analysis of the flame structure reveals that the current design offers a narrow flame in primary zone with the highest temperature being of the order of 2274K. K. H J. Tomczak et al. [13] has obtained these maxima as 2330 K. Jinsong Hua et al. [14, 15] has stated that this peak temperature was between 2200 and 2400K.

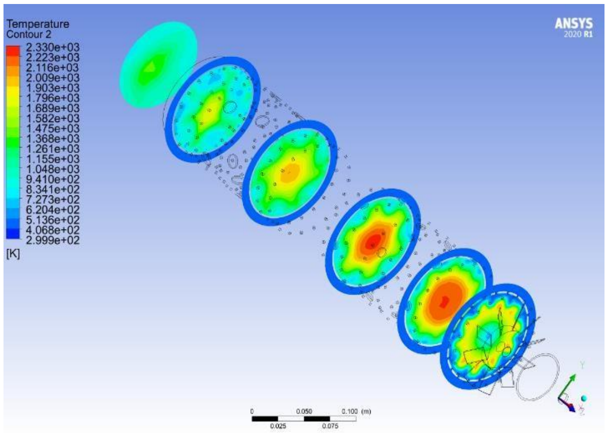


Fig. 8 Temperature Graph

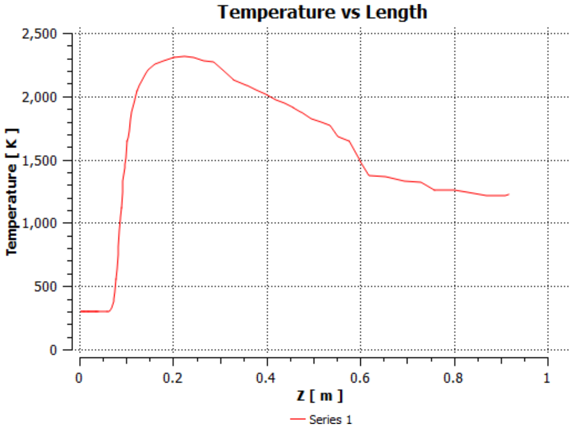


Fig. 9 Temperature Distribution at exit

The pattern factor distribution is shown in figure 8. As shown in figure 9 the exit temperature distribution is found to be uniform but on a higher side of around 1300K, as against the designed value of 950K. The exit temperature levels as observed by Jinsong Hua et al. [14,15] are in the range of 1900K, under no heat loss condition.

The results shows that when flame propagates along the length of chamber, temperature of the flame comes down effect of diffusion holes present in primary and secondary zone.

5. CONCLUSION

Small gas turbine combustion chamber design employ hydrogen as a fuel in their combustion. Understanding the temperature contours, flow phenomena and air flow distribution for combustion chamber can be accomplished by using CFD analysis. The mixing of the two groups of streamlines is happens by this micro mixing concept. In this most of the mixing takes place in the center of the chamber, some of the fuel remains near chamber wall. This results combustion in the lowest amount of time possible. The function of the combustor swirler in the change the overall direction of the air flux in order to improve the mixing with the fuel. As swirl is the possible factor for increasing the turbulent burning velocity at the flame base, it is believed to be stabilizing factor. To increase flame stability as a result of toroidal recirculation zones developing in highly swirling regions. In particular, swirl is employed as a support for effective clean combustion in a range of real-world applications. Using swirl in non-premixed combustion can help in reducing reduction of pollutant emissions, mostly that of NOx. The proposed method derives to the conclusion that a reduced temperature was found in the flame in the primary zone which leads to reduction in NOx productions. The exit temperature distribution is around 1300K. The exit temperature profile is also almost uniform, which is a significant parameter in reduction of the thermal stresses on the turbine blades to prolong their life and functionality.

Funding

No funding is received for the research work.

REFERENCES

1. D. B. Kulshreshtha, S. A. Channiwala, K. Chaudhari and S. Dikshit, 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, Dec, 2008.
2. A.M.Y. Razak, Gas turbine combustion. Industrial Gas Turbines, 2007.
3. A. H. Lefebvre, Gas Turbine Combustion. Taylor and Francis (3rd Ed). ISBN: 978-1-4200-8605-8, 2010.
4. A. M. Mellor, Design of Modern Turbine Combustors. Academic Press INC, 1990.
5. V. W. Greenhough and A. H. Lefebvre, Some Applications of Combustion Theory of Gas Turbine Development, 16th Symposium on Combustion, pp. 858-869, New York, 1957.
6. A. H. Lefebvre and G. A. Halls, Some Experiences in Combustion Scaling, AGARD Advanced Aero Engine Testing, AGARDO graph 37, pp. 177-204, 1959.
7. D. A. Nealy and S. B. Reider, Evalution of Laminated Porous Wall Materials for Combustor Liner Cooling, J. Eng. Power, vol. 102, 268-276, 1980.
8. H. Saravanamutto, H. Cohen, G. Rogers, Gas Turbine Theory (6th ed). ISBN 978-0-13-22437-6, 2008.
9. H. Keshtkar, The Effect of Flow Pattern on Combustion and Pollution Generation in a Jet Engine. J. Astrophys. Aerosp. Technol. 2018, Volume 6, 1–4, 2018.
10. N. A. Chigier, Progress in Energy and Combustion Science, V- 9, Oxford, Pergamon Press, UK. (1983)
11. C. John Marek, Timothy D. Smith, and Krishna Kundu, Low Emission Hydrogen Combustors for Gas Turbines Using Lean Direct Injection. 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, Tucson, Arizona, July 10–13, 2005.

12. A. K. Gupta, D. G. Lilley, and N. Syred, “Swirl Flows Abacus Press,” Tunbridge Wells, England, 1984.
13. H. J. Tomczak, G Benelli, L Carrai and D Cecchini, Investigation of a Gas Turbine Combustion System Fired with Mixtures of Natural Gas and Hydrogen, IFRF Combustion Journal, 2002.
14. Hua J, Wu M, Kumar K, Numerical Simulation of The Combustion of Hydrogen –Air Mixture in Micro-scaled Chambers Part I: Fundamental Study, Chemical Engineering Science, Vol. 60, pp. 3497-3506. (2005)
15. Hua J, Wu M, Kumar K, Numerical Simulation of The Combustion of Hydrogen –Air Mixture in Micro-scaled Chambers Part II: CFD Analysis for Micro Combustor, Chemical Engineering Science, Vol. 60, pp. 3507-3515. (2005)
16. D. B. Kulshreshtha, S. A. Channiwala, Simplified Design of Combustion Chamber for Small Gas Turbine Applications. Proc of International Mechanical Engineering Congress and Exposition 2005.
17. D. B. Kulshreshtha, S. A. Channiwala, Hydrogen Fuelled Micro Gas Turbine Combustion Chamber. The Hong Kong Institution of Engineers Transactions, Vol. 18, No 2, pp19-25. (2010)