Design and Wall Temperature Optimization of Combustor for Low Calorific Value Gases Application using Numerical Analysis.

Raghavendra Rao R. [1] Vijayakumar G Tile[2] Nagesha S P [3]
[1] Assistant Professor, Dept. of Mechanical Engg. MCE, Hassan.
[2]Assistant Professor, Dept. of Mechanical Engg. MCE, Hassan.
[3]Scientist F, LRDE, DRDO, Bangalore.

***Abstract**:-* **The world is facing the grave problem of energy crisis because of over usage of conventional fuels in automobile, power generation and various thermal applications. Heat transfer analysis is carried out inside the combustor by using the industrial standard RANS (Reynolds Average Navier Stokes) tool for design optimization of combustor. The combustion difficulties for low heating value (LHV) gases like producer gas derived from biomass have led to more investigations into LHV gas combustors. A geometry optimized combustor of 20 kW to provide good air/fuel mixing with long residence time for melting application with producer gas is to be designed. High Alumina Ceramic liner is used to reduce radiant thermal losses. The casing temperature has to be maintained below 600 K for which CFD analysis is carried out to study the effect of flow parameters for a Mach number lesser than 0.1 considering smooth wall conditions. The combustor to be modeled using the CFD program CFX. As chemistry models, the chemical equilibrium, eddy dissipation and k-ε model were applied. Gas combustors, providing information from which practical combustors have been designed and applied in industrial furnaces and heating equipment. The combustors consist of combinations of an injector, in which low pressure air injects fuel gas, followed by a refractory-lined combustion chamber i.e. high alumina ceramic liner in which the air/gas mixture is burned, and the burned gases, at or near flame temperature, are accelerated through a exit or outlet of combustor. The studies included investigations of the fluid dynamics of combustion chamber, thermal losses of combustor and completeness of combustion.**

Key words: Temperature, Turbulence, Heat, Combustion, Computation Fluid Dynamics, Fluid Dynamics

1. INTRODUCTION

The promotion of low calorific value (LCV) gas derived from biomass is nowadays a vital alternative for fossil fuels, of which oil and natural gas would last for no more than a few decades. It is likely that a mismatch between fossil fuel energy (oil and natural gas) supply and demand will cause serious problems in this century. One of the ways to avoid such a crisis is to increase the share of biomass. Air blown gasification of biomass is now one of the most promising and efficient ways to use the alternative energy sources, which results in so called LCV gas. Being potentially sustainable, and already renewable, biomass has the advantage that it is practically neutral regarding CO_2 emissions; as the same amount of CO_2 consumed by the crops during their growth is released later during the combustion process.

However, the major problems related to the use of LCV gas in combustors are high emissions, especially of NO_x. The design of modern industrial combustor, is almost exclusively based on operating experience with the combustion of natural gas and distillate oil in view of the foregoing, there appears to be an incentive to investigate designed gas combustors using LCV fuel gas. Material and energy losses during these process steps represent inefficiencies that waste energy and increase the costs of melting operations. Modifying the design and/or operation of any step in the melting process may affect the subsequent steps. It is, therefore, important to examine the impact of all proposed modifications over the entire melting process to ensure that energy improvement in one step is not translating to energy burden in another step.

2. EXPERIMENTAL PROCEDURE

2.1Modeling

Gas combustor CFD modeling has become an important combustor design tool in the past few years. Generally the numerical models are limited to the reactive flow field inside the combustor liner with the inlet mass flow rates being derived from 1D flow network analysis and used as boundary conditions for the internal-flow-only combustor CFD prediction. Although strongly coupled in reality, the two regions have rarely been coupled in CFD modeling for industrial applications.

The preliminary considerations that take place during the early design stages for a combustor. The overall combustor design involves three steps: preliminary design, detailed design, and validation. The overall dimensions of the concept are determined during preliminary design, after which its performance is verified and validated by detailed design and testing, respectively. Experimental testing is not included in the scope of this thesis.

Combustor sizing refers to the definition of the reference area (or diameter) to provide sufficient stability without incurring excessive pressure losses. All operating points (i.e., idle, full power) are considered and the smallest size that provides stability over the entire range of operation is chosen a premixed combustor must be scaled from an equivalent conventional combustor.

The liner diameter must be chosen carefully. It would appear desirable to select the largest diameter possible and reduce the velocity of the flow within the liner. This increases the residence time in the combustor and promotes stability. However, for any given casing area, increasing the liner diameter results in a reduction of the annulus flow area The smaller area results in higher annulus velocities that decrease the static pressure in the annulus. Therefore, a larger liner diameter is undesirable since a high static pressure drop across the liner admission holes is necessary to provide adequate penetration of the jets.

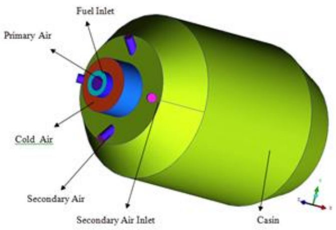

Figure 1 Combustor Casing

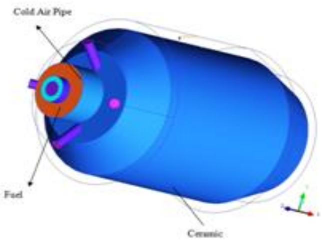

Figure 2 Combustor Liner

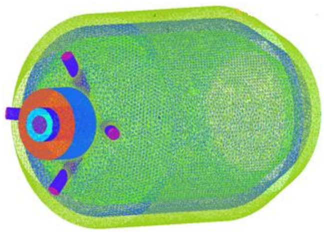

Figure 3 Meshed model

The Figure 1 and Figure 2 gives 3D view of combustor. The length of the combustion chamber is 120 mm and the diameter of 70 mm with 5 mm thick ceramic liner. The primary air and the fuel entry is axial to the combustor. The secondary air entry is at 45 deg and was selected on the trial and error basis for a better mixing of air and fuel. The inner diameter of casing of 90 mm was selected based on the velocity requirements. Total number of tetrahedral elements 1.8million. Total number of nodes: 0.65 million and it can be seen figure 3.

2.2 Problem Statement:

A combustor domain encompasses with two inlets, middle part of the combustor has small tube and it control fuel flow and around the tube ambient air passes (Fig.1 & 2), which are designed for air to fuel ratio for control supply to the computational domain. Inner cylinder has perforated holes for pass the secondary air to control the wall temperature. The fluid is considered as incompressible air and producer gas. The combustion simulation uses the standard RANS equations for mass, momentum and energy and the turbulence is simulated with the k-epsilon model known to have given the most acceptable results in the validation made in the earlier investigation [4]. A no-slip and adiabatic wall conditions are imposed at the wall of the combustor domain. Outlet pipe is located at exit part of the combustor for maintain the uniform temperature across the outlet and it can connect for any application like boiler, domestic purpose, etc., The numerical code uses an industry standard CFD code solving 3D RANS equations with a second order upwind scheme. The solution sought is for steady state and a diminishing residuals falling by 1×0^{-6} is set as the criteria for convergence. For a clarity the main equations considered for the solution is provided below.

1. The Continuity Equation:

$$\frac{\partial \bar{U}_j}{\partial x_j} = 0 \qquad (1)$$

2. The Momentum Equation:

$$\frac{\partial}{\partial t}(\rho \bar{U}_i) + \frac{\partial}{\partial x_j}(\rho \bar{U}_i \bar{U}_j) = -\frac{\partial \bar{P}}{\partial x_i} - \frac{\partial}{\partial x_j}\left(\bar{\tau}_{ij} + \rho \overline{u_i'' u_j''}\right) \qquad (2)$$

The Energy Equation:

$$\frac{\partial}{\partial t}(\rho \bar{h}) + \frac{\partial}{\partial x_j}(\rho \bar{U}_j \bar{h}) = -\frac{\partial}{\partial x_j}\left(Q_j + \rho \overline{u_i'' h'}\right) \qquad (3)$$

3. RESULTS AND DISCUSSIONS

The below figure-4 shows a temperature profile obtained of the entire combustor. It gives us an indication of how the flame burns and how the temperature varies throughout the combustor. We can see a maximum temperature of 1920K is obtained at the location of the flame. The blue color indicates 300K hence showing that the combustor walls are not being exposed to high temperatures that can result in failure.

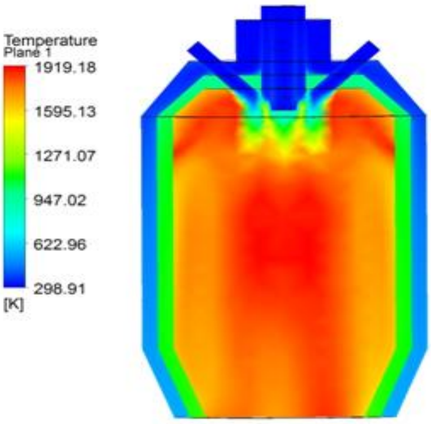

Figure. 4 Temperature distributions inside the combustor

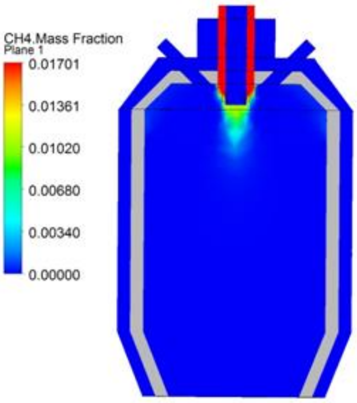

Figure.5 CH_4 distribution

Figures 5, 6 and 7 show the mass fractions of CH_4, H_2 and CO respectively. The figures show how the concentration of these gases reduces over the length of the combustor. The Blue color shows that the concentration of gas is zero indicating complete burning. Under complete combustion CH_4 when burnt produces CO_2 and H_2O, H_2 produces H_2O and CO converts to CO_2. This can be seen from figures 8 and 9 where we see a constant increase of CO_2 and H_2O during and after burning of fuel.

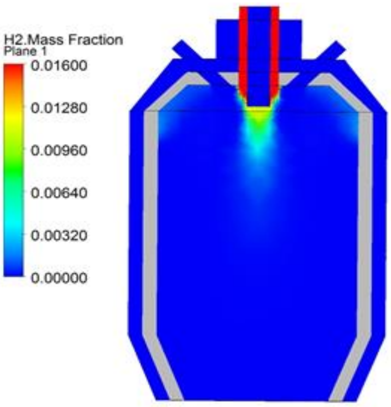

Figure.6 H_2 distribution inside the combustor

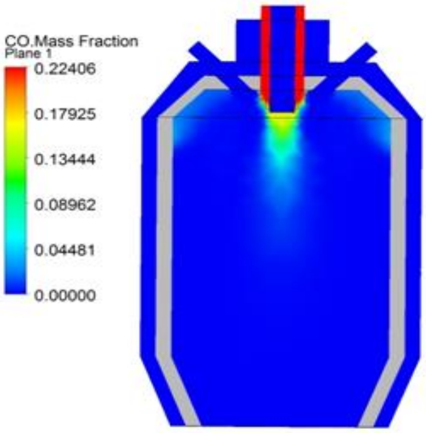

Figure.7 CO distribution

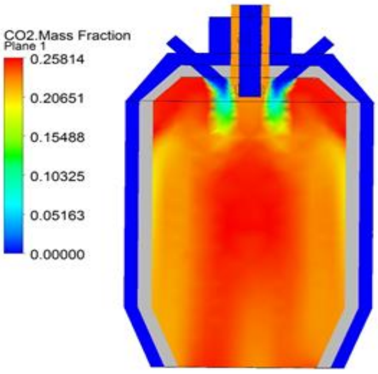

Figure.8 CO_2 distribution inside the combustor

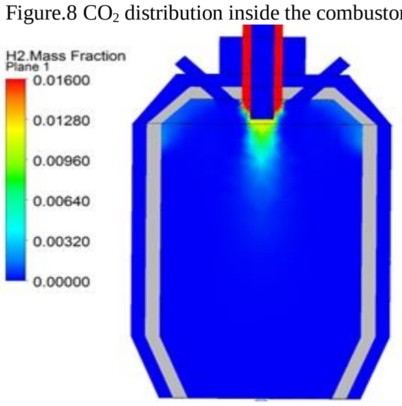

Figure.9 H_20 distribution

Figures 10 provide us the results that are obtained at the wall temperature of the combustor. The temperature profile shows us the variation in temperature along the wall and at the outlet. A variation of 100K is obtained which is within the tolerance limits. If the temperature variation is too high it can result in damage of the wall due to high thermal stresses being developed in them.

Figure 11 provides us an idea of the velocity obtained at the outlet and pressure profile.

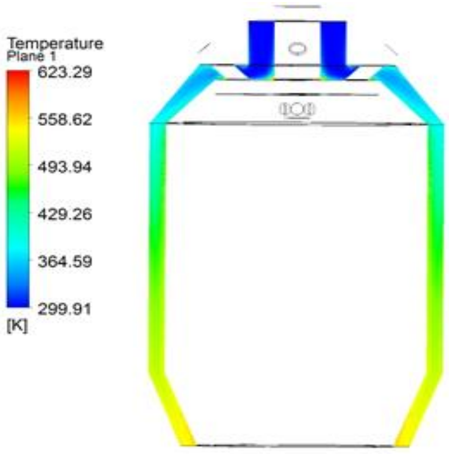

Figure.10 Combustor wall temperature profile

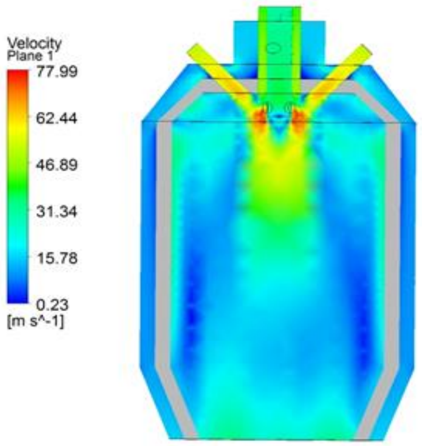

Figure .11 Combustor velocity profile

4. CONCLUSION

Motivated with the literature survey, initial dimensions of the combustor were arrived at. Several combustor concepts were developed after considering all the specifications finally freeze this concept to finalize this concept for various thermal applications. After numerous modifications based on results obtained, the best performing model was in reasonable agreement with the theoretical calculations and technical specifications were selected and taken to the detailed design stage and drafts of parent and child parts were prepared for fabrication. During analysis of the best performing models the following points were observed**:** The Emissions are nil, which indicate complete combustion. The adiabatic flame temperature calculated theoretically and observed during analysis nearly converges. The wall temperature of the chamber is maintained below 600K as shown by the analysis results as a safety precaution. The outlet temperature difference also, as shown in the analysis is maintained well laminar flow, and temperature difference is not observed.

5. REFERENCES

[1] Arthur H. Lefebvre and Dilip R. Ballal *Gas turbine Combustion, Alternative fuels and emissions* (3rd Edition) CRC Press Taylor & Francis Group, 2010

[2] John D. Anderson, Jr. Department of Aerospace Engineering , University of Maryland , *Computational Fluid Dynamics , The Basics with Applications.*

[3] G Sridhar, H V Sridhar, S Dasappa, et al, Combustion Gasification and propulsion Laboratory, Indian Institute of Science, *Development of Producer Gas Engines.*

[4] Oboetswe Seraga Motsamai, University of Pretoria**,** *Optimisation Techniques for Combustor Design*

[5] Rolls Royce, *Gas turbine Technology, Introduction to a jet Engine*

[6] Combustor, Available**:** http://en.wikipedia.org/wiki/Combustor