A Numerical Report on Burner Combustion
Vishwanath B Walikar1 Dr. Badarinath C2 Syed Sharin3
1M.Tech Student (Thermal Power Engineering) 2HOD 3Assistant Professor
1,2,3Department of Mechanical Engineering
1,2,3The Oxford College of Engineering, Bangalore, Karnataka, India

Abstract— The majority industrial and power developing sector used the application of CFD as advanced resource in Research and Development sector. The design and development of burner for effective combustion of fuel can be carried with CFD approach. The modification in design and understanding the combustion flow variable can be studied without manufacturing the actual model. The present paper present the detailed report regarding the burner combustion approach using numerical methods. The working principle of burner varies according with the design, load, and requirement. The numerical solution and methods involved in solution of general burner combustion is carried with combustible butane gas as fuel source.

Key words: Efficiency, Temperature, Fuel, Burner

I. INTRODUCTION

The burners are the device used to combust the fuel with an oxidizer to convert fuel in to thermal energy. The combustion system may have single or many burner depending on the type of application. The heat from the burner transfer mainly in form of convection and radiation. The heat transfer from burner also depends on design, the type of material used in manufacturing method, size, dimension and application. The alloys of different material like INCONEL, Silicon carbide (SiC) are used to with stand high temperature generated inside burner. The efficient burner combustion reduces the pollutant emission. The use of radiant tube is important because these systems can be coupled with other device, such as recuperative and regenerative heat exchanger which increases the efficiency. These systems have internal recirculation of flue gas to reduce the temperature peak generated during the combustion and better temperature uniformity at the surface of the radiant tube. This reduces the pollutant like NOx emission in to atmosphere.

The recuperative system integrated with a single or multiple radiant tubes are used in many industries. The flue gas moves through the inner tube and flow back along the annulus between the inner tube and radiant tube. This increases the efficiency of the radiant tube.

A. Introduction

The burners are the main source of power production used in industry. Burner always depends on fuel and oxidizer for complete combustion of fuel. The chemical energy from fuel is converted in to useful thermal energy. The thermal energy from the burner is transmitted in form of radiation mainly with convection as an associative form. The transfers of heat in form of radiation depend on type of material used in burner. The thermal and physical property varies with increase in temperature and type of fuel used in combustion. The flexible design of burner effectively reduces the emission of pollutants with use of different fuel. The thermal efficiency can be further increased with incorporating the flue gas recirculation. The recuperative and regenerative heat exchanger system are utilized to increase the thermal efficiency. The use of these system effectively reduces Nox pollutants emission in to atmosphere.

B. Design

The maximum heat transfer and reduction in pollutant emission can be effective with only the design and quality of material used in manufacturing the product. This design also help to reduce emission of pollutants like Carbon monoxide in to the atmosphere. The design of burner reduces the consumption of fuel with increase in the thermal efficiency of burner. The design of burner was carried with modelling tool called as CATIAV5, a drafting and part modeling tool. The effective modelling replicates the exact dimension with virtual 3D display as show in figure 1.

Fig. 1: Burner

The burner consist of equally spaced fuel nozzle for complete distribution of fuel. The butane as fuel is supplied in the inner annulus and air is passed along the outside of the annulus. The dimensional length has been obtained by Tsioumanis et al, (9) (10), for the burner toward the complete combustion of fuel without emission of pollutant such as NOx.

C. Fuel

The complete combustion and thermal efficiency of burner is mainly dependent on the type of fuel used. The quality of fuel also influences the nature of combustion. The Natural gas is the most common and general gaseous fuel used in the industry. The fuel are obtained from the petrochemical industry as bi-product from the petroleum industry. The calorific value in fuel is based on the percentile nature of composition. The effective combustion flame length and quality of flame also dependent on the fuel. The numerical simulation is carried with Butane gas (CH4) as fuel.

D. Oxidiser

The combustion of fuel is aided by Air, it most commonly preferred industrial oxidizer. The availability and least cost has an effective advantage as an oxidizer. The oxidizer also help to reduce the emission of harmful pollutants like NOx in to the atmosphere. The emission NOx is reduced by supplying excess air at the peak temperature zone. The complete combustion of fuel is carried with help of oxygen present in air. The 21% of oxygen present in air undergoes the oxidation and reduction for complete combustion. The intensity of flame is increased by supplying pure oxygen,
which is specially used in glass industries. The mixture of pure oxygen and pure air is used to have intermittent flame temperature.

E. Flue Gas Recirculation

The hot gases after complete combustion of fuel are allowed to pass through the burner system. The efficiency of burner increases by this method, since the flue gas temperature ranges up to 600-800°C. In this study, the flue gas recirculation effect is considered but current study is being worked without this recirculation unit.

F. Literature Review

The combustion of fuel like butane, methane in burner is important to understand the heat transfer in form of conduction, convection and radiation. The single step reaction (1) were included to study the effect of variation in parameters like temperature, mass flux, inlet mass fraction of fuel and wall temperature profile. The shift in flame position among was noted along the tube length. The flame position within the tube were effectively implemented by open-loop and closed loop for thermally stabilized schemes. The one model of plug flow reactor had practical application in heat treatment process. The effect of burning were analyzed with swirl stabilizes combustion with syngas (2). The syngas fuel mixture containing (4% CH₄, 4% CO, 10% CO₂, 2% N₂, 80% H₂) was burnt at ambient pressure in a confined environment by transverse injection. The investigation on turbulent flame with coflow injection of the reactant revealed significant difference in NOx and CO emission compared to other fuels, which have observing changes in the flame structure. The NOx emission scale with adiabatic flame temperature shows the preliminary investigation in burner. The burner operating under very lean condition (below φ =0.1), NOx and CO emission increases by decreasing the global equivalence ratio, lower CO emission as compared to burning NG alone (at φ = 0.6) and lower NOx emission as compare to burning NG+H₂. The optimum syngas composition that yields the lower amount of NOₓ in syngas. The deeper understanding of investigation on lean condition and their effect on CO and NOx emission, flow field and influence of jets velocities on the different fuel injection method.

<table>
<thead>
<tr>
<th>Fuel</th>
<th>Heat of combustion in Mj/kg</th>
</tr>
</thead>
<tbody>
<tr>
<td>LPG</td>
<td>44</td>
</tr>
<tr>
<td>CNG</td>
<td>50</td>
</tr>
<tr>
<td>Kerosene with air</td>
<td>42</td>
</tr>
<tr>
<td>Producer gas</td>
<td>4.2</td>
</tr>
</tbody>
</table>

Table1: Heat of combustion

The table 1 shows the heat of combustion of various different fuel used in understanding the influence on the flame structure, temperature.

A porous solid insertion (3) in to the flame to recycle a portion of the heat by conduction and radiation. The transfer of heat between two phases carries with convection. The design provides an extended lean operational and radiation burner operation with reduced CO and NOx emission. The preheating increases the flame speed inside the porous zone, which can be locally varied in porosity to assign specific flame location. Turbulence contributes to increase in heat and mass transfer rates for flames stabilized outside the porous zone. The global increase in flame speed by increasing the flame surface area. The development of radial flow burner and incorporation of second stage together with the non-premixed flame operation are the advancement in innovation. The effect on the liquid fuel combustion in the porous media is due to favourable influences of radiative heat transfer on the droplet evaporation, premixing and combustion. The swirling flow burner has been essential to premixed and non-premixed combustion system because of their significant beneficial influences on flame stability, combustion intensity and combustor performance. The study shows that the flame characteristic of the low-swirl burner using low calorific gas produced by biomass gasification system. The problems during burning of the mixed gas are the poor distribution of heat released in the chamber. The increase in flame quality or flame strength is necessary to reduce the diameter of the inlet fuel with variation of 6, 8 and 10 blades of the swirlers vanes, with the vanes inclined at 30 degrees from the horizontal axis. Variations of vane design are correlated with quality of the flame, heat release rate and emissions formation in combustion unit. The methodology used includes implementation of three-dimensional (3D) Computational Fluid Dynamics (CFD) simulation using the commercial software FLUENT and gasification experiments which included assembly of a new combustion equipment unit. The experimental results identified the maximum temperature, which occurred at swirl vane of 8 blades at 795°C. The maximum heat release was achieved at 11.15 KJ/s for biomass. The lowest content of CO emission was 0.02% volume of biomass the lowest NOx emission was the 1108 ppm for biomass. Results of this study indicated that the swirl vanes with 8 blades and diameter of 55 mm perform better than other number of blades of swirl vane burners. Swirl burner with number of vane blades of 8 could improve the gas burner performance from existing gas burner. Increasing of flame temperature, heat release rate, combustion efficiency and decreasing of CO concentration.

The comparisons of NO/NO2/NOx and emissions (4) from different lobed and non-lobed fuel-injector configurations illustrate very useful trends with respect to the effect of fuel–air premixing on emissions. The two lobed injector geometries examined and created different local fuel–air mixtures relatively soon after injection, with injector appearing to create a locally leaner (on average) mixture and injector an apparently creating nearer stoichiometric and well mixed with different conditions. These differences appeared to cause increases in NOx emissions for the lobed injectors (compared with nonlobed injector emissions) at higher mass flux ratios M due to lesser sooting and radiative losses and hence higher flame temperatures. The significant reductions in NOx emissions for the lobed injectors at low values and hence at very lean local conditions. The lobed injector geometries with CO emissions become high for locally lean combustion conditions (M<0.45), consistent with lean premixed combustion conditions. Results for both NOx and CO emissions indicate that different degrees of fuel–air mixing (e.g., as seen when comparing lobed injectors A and B) can shift the emissions curves’ dependence on turn-down ratio. The study demonstrates the importance of control of the
local mixture or equivalence ratio in minimizing burner emissions.

The uniformity at the outlet of gas mixing system (GMS) are studied by considering the effect of fan and multi ejectors. The pollutant emission are studied by experiment. The numerical model is validated by the experimental results, the CFD is used to optimize the GMS by improving the uniformity at the outlet of GMS. The optimal structure and geometrical parameters of GMS are obtained. The distribution orifice plate improves the uniformity of velocity and the fuel–gas mixing at the outlet of ejector at different fan speeds. When the fan speed is 5000 rpm, the uniformity of velocity and fuel–gas mixing of single ejector is at least increased by 234.2% and 2.9%, respectively. The proposed diversion plate improves the uniformity of ejectors. The optimal diameter of orifices of the diversion plate is obtained and the optimal diversion plate performs well for all fan speeds studied. For a fan speed of 5000 rpm, the uniformity of flow rate and fuel–gas mixing of ejectors is increased by 1.9% and 2.2% with the optimal diversion plate, respectively. The improvement of the uniformity of GMS results in the reduction of pollutant emission of premixed burner.

G. Burner

The industrial burner are generally forced draft burner. The air and fuel are allowed to flow with pressure using blower. The primary Purpose of burner is to transfer heat from the combustion system to the load. The maximum heat transfer ensures the maximum system thermal efficiency. This reduces the operating cost and indirectly reduces the pollutant emission, since less fuel is required for given combustion rate. The turbulent flow of fluid, chemical reaction leading to high temperature and visible gaseous radiation are complicated heat transfer process from burner.

CFD based application has numerous application in understanding the physics of combustion system. The reaction chemistry and turbulent fluid mechanics has been limited due to nonlinear relationship Combustion-based CFD has become an extremely valuable tool in providing direction to the timely execution of industrial experiments, which further reduces the impact that retrofits and other equipment changes.

The simulations solution of continuity and the Reynolds–Averaged Navier stroke equations were carried out using corresponding codes. In the case of turbulent flow the random nature of flow precludes computations based on a complete description of the motion of all the fluid particles. In general it is most attractive to characterized turbulent flow by the mean values of flow properties and the statistical properties of their fluctuations. The basic general equation used to solve the model using numerical approach. 1) Continuity Equation:
The equation of continuity expresses the conservation of mass and for a steady one dimensional flow it is given by.
\[ \nabla (\rho \vec{V}) \cdot \frac{\partial \rho}{\partial t} = 0 \quad \text{……..(1.1)} \]

2) Momentum Equation (Navier-Stoke) Equation
Momentum equation is derived based on Newton’s second law and relates the sum of the force acting on an element of fluid to its acceleration or rate of change of momentum. The
Newton’s second law of motion \( F = ma \) forms the basis of momentum equation.
\[
\nabla (\rho \vec{V}) + \frac{\partial (\rho \vec{V} \cdot \vec{V})}{\partial t} = \nabla \cdot \tau + \rho \vec{f} \quad \text{(1.2)}
\]

Where \( \mu \) is the turbulent viscosity, obtained from turbulence model contains those parts of the stress term not shown as momentum source.

3) **Turbulence equation:**
\[
\nabla (\rho \vec{V}) + \frac{\partial (\rho \vec{V} \cdot \vec{V})}{\partial t} = S_m \quad \text{(1.3)}
\]

Where, the source \( S_m \) is the mass added to the continuous phase from the dispersed second phase (e.g. due to vaporization of liquid droplets) and any user-defined sources.

J. **Numerical METHODOLOGY**

The Numerical set up contains, the burner along with the different input parameter like air and fuel input in Ansys fluent code. This is shown in figure 1.3.

Fig. 4: Pictorial representation of burner in Ansys tool.

The air and fuel are supplied at nominal rate in burner at 3.235kg/s and 2.12kg/s respectively. The boundary condition to burner is shown in table:

<table>
<thead>
<tr>
<th>Type</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet(Air)</td>
<td>3.235kg/s</td>
</tr>
<tr>
<td>Inlet air temperature</td>
<td>300k</td>
</tr>
<tr>
<td>Inlet(Fuel)</td>
<td>2.12kg/s</td>
</tr>
<tr>
<td>Inlet fuel temperature</td>
<td>300k</td>
</tr>
<tr>
<td>Fuel</td>
<td>Butane</td>
</tr>
</tbody>
</table>

Table 2: Boundary Condition to Burner

The numerical simulation is carried out and obtained using tools like CFX, ANSYS etc. The boundary condition are applied in pre-processor, where the input parameters are fed in form of numerical data. Solver as second step, simulates the input in set of algebraic equation with numerous iteration. The iteration method is carried until the result reaches the exact solution.

K. **Result and Conclusion**

The converged solution is analyzed with graphically i’e in CFD Post, to understand the behavior phenomenon inside the burner. The flame temperature, mass fraction, flame length, velocity, temperature are analyzed.

<table>
<thead>
<tr>
<th>Result</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum temperature</td>
<td>1890K</td>
</tr>
<tr>
<td>Outlet temperature</td>
<td>1850K</td>
</tr>
</tbody>
</table>

Table 3: Results

The outlet temperature is lesser compared to the maximum temperature. The flame temperature can be further increased by increasing the inlet velocity of air and fuel. The maximum temperature is found at 150mm away from center of combustion inside the burner. The temperature of flame is uniform throughout the circumferential burner surface. As the length increases, the flame length of burner decreases. The complete combustion of fuel is notified by absence of CO mass fraction at outlet of burner. The NO\(_x\) can be further reduced by incorporating the flue gas recirculation system.

REFERENCES

[1] S Basu, Control of combustion in a thermally stabilized burner, Department of Mechanical, Materials and Aerospace Engineering, University of Central Florida, Orlando, Florida, USA


[3] M M Kamal, Combustion in porous media, Department of Mechanical Power Engineering, Ain Shams University, Cairo, Egypt


