Combustion Analysis of Gas Fired Burner to Study of NO_x Emission Using Computational Fluid Dynamics

[1]V. A. Amlapure, [2]S. M. Ranade, [3]A. M. Patil

[1]M-Tech, Thermal Student, [2]Professor, [3]Sr. CFD Engineer
[1, 2]Department of Mechanical Engineering, [3]B & H Department
[1, 2]Walchand College of Engineering, Sangli, India
[3]Thermax Ltd., Pune, India

Abstract: Computational fluid dynamics (CFD) is a branch of fluid mechanics that analyzes and solves problems involving fluid flows using numerical analysis and data structures. CFD is a simulation technique used to predict what will occur, quantitatively when fluids flow, often with simultaneous heat flow complication, mass transfer, phase change, chemical reaction, mechanical movement of (pistons, fans, etc.), stress and displacement of immersed or surrounding solids. It helps to simulate vast area of fluid flow problems ranging from simple flow through pipe to multidisciplinary problems involving complex chemical processes. In the existing work by Computational Fluid Dynamics analysis combustion of natural gas is carried out by integrating manual chemical reactions. Given the significance of NO_x emissions in boilers, NO_x predictions may be the most significant improvement in industrial system using CFD analysis. Conditions of combustion within the boiler furnace were simulated by Fluent CFD software at a 30^0, 40^0, 45^0 and 60^0 swirl angles. CFD Fluent software is used as a tool to generate the fired gas burner's three-dimensional model. The CFD simulations provide contours of temperature distribution, contours of CO_2 mass fraction, contours of H_2O mass fraction, contours of NO_x and heat flux profiles to the furnace walls. As the different vane angle geometries are studied, it is found that as swirl vane angle and swirl number increases, NO_x formation decreases.

Index Terms: Swirl number, natural gas, combustion, emissions

I. INTRODUCTION

Gas burners are a kind of equipment mixed with fuel gas and air to jet the flaming gas. They are generally used in heating and associated sectors such as boilers, smelting furnaces and heat treatment. It regulates heat load, distribution of temperature and thermal efficiency as it is the essential component of the entire heating equipment. When the burners work, the combustion efficiency and the amount of fuel and the strength of the flames have significant effects on the quality of the combustion and these parameters above depend on the oxygen supply in the combustion process. The inadequate oxygen supply condition can be due to the absence of air or an inaccurate percentage of primary air mixed with fuel gas and secondary air that contributes to combustion. The primary to secondary air proportion thus has an important impact on flame temperature, flow velocity distribution, and NOx emission in a definite amount of total air. Gas-fired boiler emissions comprise nitrogen oxides (NO_x), carbon monoxide (CO), carbon dioxide (CO_2), methane (CH_4), nitrous oxide (N_2O), volatile organic compounds (VOCs) and particulate matter (PM) [1].

Three distinct processes are essentially involved in forming nitrogen oxides. Thermal NO_x is the main mechanism of NO_x formation in gas combustion. Thermal dissociation and the later response of nitrogen and oxygen molecules in the combustion air cause the process of thermal NO_x. In the high-temperature flame zone, near the burners, the majority of NO_x created via the thermal NO_x mechanism occurs. Three furnace-zone variables adversely affect thermal NO_x: (a) concentration of oxygen, (b) peak temperature, and (c) exposure time at peak temperature. NO_x emission concentrations rise as these three factors rise. For all kinds of natural gas-fired boilers, the emission trends due to modifications in these variables are relatively consistent. The emission concentrations fluctuate significantly with combustor type and size and working conditions (e.g., combustion air temperature, volumetric heat release rate, load, and excess oxygen level). The second NO_x formation mechanism, known as prompt NO_x, appears through early reactions of nitrogen molecules in the combustion air and hydrocarbon radicals from the fuel. Prompt NO_x reactions are generally negligible in flame compared with the quantity of NO_x generated by the thermal mechanism of NO_x. However, with ultra-low-NO_x burners, prompt NO_x concentrations may become important. The third NO_x formation mechanism, called fuel NO_x, arises from the evolution and response of fuel-bound nitrogen compounds with oxygen. NO_x formation through the fuel NO_x mechanism is trivial due to the characteristically low fuel nitrogen content of natural gas [4].

Fluent CFD modelling of the boiler furnace offers a reasonably quick, reliable and cost-effective tool to be used in suitable technical applications and in this case, to study boiler performances. It also provides accurate estimates of how a design will perform under various conditions, including different swirl angle. The computer simulation using the Fluent CFD code provided more information than probable using physical model testing [5].

II. LITERATURE REVIEW

Weibo and Guixiong [1] studied the features of combustion temperature, flow velocity, CO distribution and NO_x emissions of a 10 MW premixed gas burner in the different primary to secondary air ratios are explored numerically using comprehensive models. The simulation method system is determined to simulate the airflow fields inside the gas burner first in order to produce precise predictions and decrease the quantity of calculation. According to the analytical findings, for combustion simulation with a streamlined model, three groups of representative primary to secondary air ratios have been chosen.

Petrica et al. [2] researched CFD Simulation findings indicate that the temperature in the system is decreasing as anticipated as the quantity of steam added rises. The maximum temperature distinction, however, is only about 40 K, and the outlet temperature is scarcely 5 K smaller between the dry case and the case when fuel adds 25% of steam. With regard to the formation of NO_x, it is important to emphasize that, compared to the dry case, the amount of NO_x almost halves. When added to the fuel, the steam addition impact is more intense than added to the combustion air.

Risto and Ilija [3] in this paper, the findings of the temperature distribution, combustion efficiency and heat transfer in the boiler furnace lead to the conclusion that the boiler can be fuelled with natural gas effectively without any further modifications in design.

Timo et al. [4] compared NO_x emission assessment models fired warm water boilers in two comparable 43 MW boilers. The models use online process readings that are always accessible in comparatively small instrumented configurations. The aim of the study was to discover sustainable and transparent but still valid models for NO_x assessment in order to assess the potential of the implementation of the Predictive Emission Monitoring System for NO_x emission surveillance in practical installations in the long term.

Aruna and Nageswara [5] stated Optimizing Bunsen burner Performance using CFD. The adiabatic flame temperature for LPG is the largest temperature reported in the CFD analysis. The losses of radiation are not taken into consideration here. But the major losses happen in tests owing to heat losses from the flame to the burner's cold walls. The high-temperature region in the laboratory flame appears to be slightly smaller owing to the impacts of the burner edge. The concentration measurements of the species will corroborate this. The region with the highest temperatures in both measurements and predictions is between the inner premixed and the central non-premixed reaction area.

Xuedong et al. [6] studied seven boilers of 600MW and 1000MW units installed recently have been studied. The boilers employ different burner types and combustion technologies and burn different coals. The boilers exhibit different performances and NO_x emission. The findings of the experiment and analysis indicate that boiler efficiency and NO_x emissions are highly affected by the configuration of the boiler and the type of burner as well as the type of coal.

Ilker Yılmaz [7] presents the impact of the amount of swirls on combustion features such as temperature, velocity, gas concentrations in a flame of natural gas diffusion. The flame region's combustion reaction system was based on the eddy dissipation model with a global reaction system of one step. The numerical simulations use a standard k-e turbulence model for turbulent closure and a P-I radiation model for flame radiation inside the combustor. In the research, seven distinct swirl numbers including 0, 0.1, 0.2, 0.3, 0.4, 0.5 and 0.6 are used to explore the swirling impact on the combustion features. The results indicated that the combustion features such as the temperature of the flame, gas concentrations including CO_2, H_2O, O_2, and CH_4 are heavily influenced by the swirl number.

Shaiful and Nazri [8] tested eight different air swirlers having 10°, 20°, 30°, 40°, 45°, 50°, 60° and 70° vane angles. For the swirl number of 1.427, the NO_x decrease was achieved by more than 26 percent compared to 0.046. There is a turning point at this stage where the NO_x emission suddenly rises by about 23 percent when the swirl number is 1.911.

Baziar [9] studied the impact of swirl in non-premixed propane-air flames on NO_x formation. The CFD Fluent software performs the numerical simulations. The model k-ε turbulence is used. Both predictions and experiments indicate that as the swirl number increase, the temperature and the heat NO_x first rise slightly and then reduce sharply at the combustion chamber exits.

Harun and Ilker [10] studied the impacts of swirl number and equivalence ratio on combustion and emission conduct of premixed CNG/ H_2/ CO/ CO_2 mixture flames were explored by radial and axial temperature and emission profiles (CO, CO_2, and NO_x) in a laboratory scale combustor.

III. CFD ANALYSIS OF GAS BURNER

3.1 Geometry Modelling of Burner

Flow considered here is single stage natural gas flame in 34 MW swirl stabilized burner. The furnace is horizontally fired. The furnace wall can be refractory-lined or cooled with water. Burner has 8 fuel pipes having 64 fuel outlets, air introduced from one side of wind box and air goes to inside sleeves strike on swirler, a stationary swirl of angle 45^0 having 32 blades is used to impart swirl. Therefore turbulence is created due to swirler.

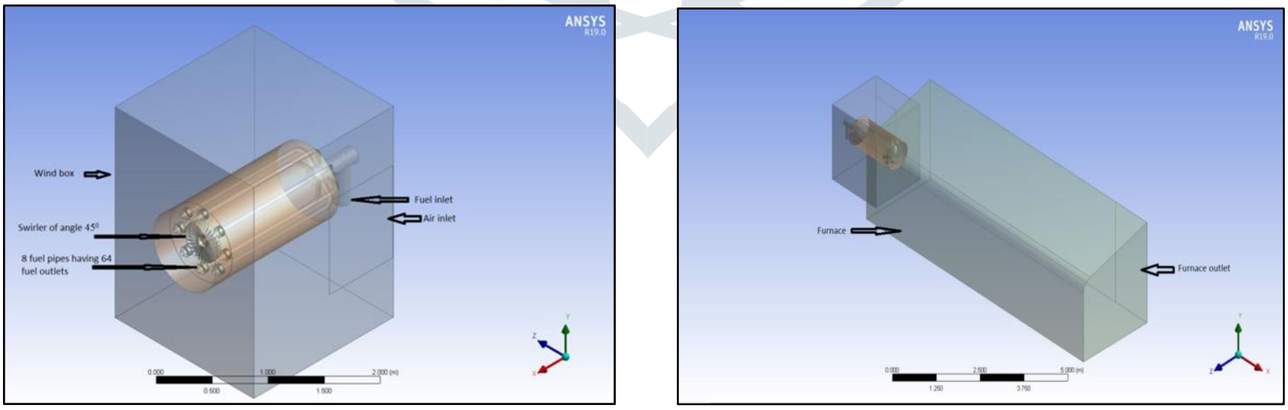

Figure 3.1 Geometry modelling of wind box and furnace

3.2 Meshing

Meshing burner geometry is done in ANSYS Fluent under meshing window. The parameters used for meshing are given as sizing function set as Proximity and Curvature with fine mesh, a global growth rate of 1.2, Patch conforming Method (Tetrahedrons) on air body, All Triangles Methods on surface bodies i.e. swirler blades. Using the sizing parameter, the edges of geometry are divided by giving the number of division between 10 and 80. Body sizing is given of air body and furnace body with an element size of 50 mm and 120 mm respectively. The quality of the mesh is checked by using metrics of skewness and orthogonal quality parameter

The typical details of mesh are as follows.

Number of nodes = 1138431, Number of elements = 6277428, Average Skewness = 0.20957, Average Orthogonal quality = 0.78925

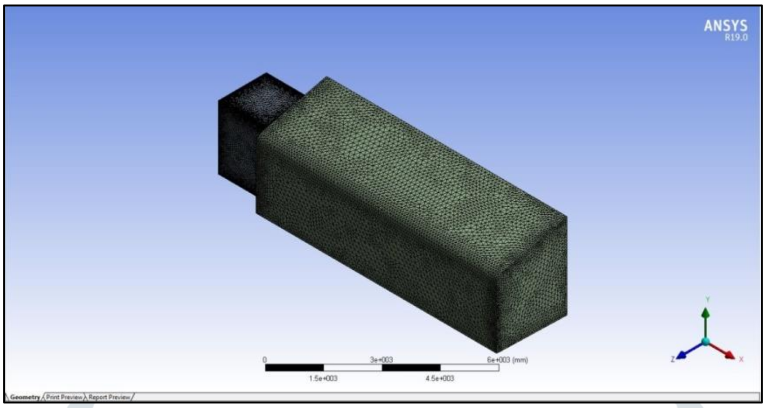

Figure 3.2 Mesh with fluid domain

JETIR

3.3 Fluent setup

The solver settings are set to pressure based solver since the fluid flow involved is incompressible. For this study, the steady-state condition is assumed for all the cases. The energy equation is enabled as the heat transfer is involved. A viscous model with standard wall treatment is set to realizable k-ε turbulence model. Two-equation turbulence models enable both a turbulent length and time scale to be determined by solving two distinct transport equations. In ANSYS Fluent, the normal k-ε model falls within this model category. Robustness, economy and reasonable precision explain its popularity in industrial flow and heat transfer simulations for a wide range of turbulent flows. The species transport under model and eddy dissipation model was selected under turbulence-chemistry interaction in the Fluent solver. Thermal NO_x was enabling for calculating emissions. A discrete ordinate was used in radiation model.

3.4 Boundary Conditions

A mathematical model's boundary conditions are essential components. They direct the flow motion which leads to a unique solution. Standard boundary conditions are inlet, outlet, wall, symmetry etc. Face areas represent boundaries and inner surfaces. Face areas are allocated boundary information. A wide range of types of boundary conditions allows the flow to enter and exit the solution domain. Boundary Conditions are specified in the Fluent solver as shown in Table 3.1

Table 3.1 Boundary conditions

Air inlet (Mass flow inlet) Airflow rate = 12.76 (kg/s) Direction = Normal to the boundary Turbulence intensity = 17% Hydraulic diameter = 0.889 m Air inlet temperature = 312 K Mole fraction of O_2 = 0.21	**Fuel inlet (Mass flow inlet)** Fuel flow rate = 0.6597 (kg/s) Direction = Normal to the boundary Turbulence intensity = 5% Hydraulic diameter = 0.4064 m Air inlet temperature = 308 K Mole fraction of CH_4 = 0.9853 Mole fraction of CO_2 = 0.0145
Furnace Outlet (Pressure outlet) Turbulence intensity = 5 % Hydraulic diameter = 1.792 m Mole fraction of CH_4 = 0 Mole fraction of O_2 = 0.0175 Mole fraction of CO_2 = 0.091 Mole fraction of H_2O = 0.1695	**Wall Boundary Conditions** Air side wall temperature = 312 K Furnace side wall temperature = 1300 K Internal emissivity = 0.6

IV. RESULTS AND DISCUSSION

The blade angles used in simulations are 30^0, 40^0, 45^0 and 60^0.with swirl number 0.39,0.59,0.7,1.22 respectively. The swirler inner diameter is 114.3 mm, and its outer diameter is 402 mm. From Figure 4.1, maximum temperature inside the furnace is 2252.4 K at vane angles of 30^0 and 2213.78 at vane angles of 45^0 which is nearly close to theoretically calculated maximum temperature of 2193 K. From Figure 4.2 it is evident that the maximum values of CO_2 mass fraction are 0.1576, 0.1498, 0.1488 and 0.1643 at vane angles of 30^0, 40^0, 45^0 and 60^0.respectively. So the minimum value obtained at 45^0 vane angle, so that angle is preferred to get less CO_2 formation. From Figure 4.3 it is evident that the maximum values of H_2O mass fraction are0.1199, 0.1141, 0.1132 and 0.1189 at vane angles of 30^0, 40^0, 45^0 and 60^0 respectively. So the minimum value obtained at 45^0 vane angle, so that angle is preferred to get less H_2O formation. From Figure 4.4 it is evident that the maximum values of pollutants mass fraction are 0.00045, 0.002252, 0.000208 and 0.0002621 respectively. So the minimum value obtained at 45^0 vane angle, so that angle is preferred to get less pollutants formation. Figure 4.5 and Figure 4.6 shows that NO_x in ppm at vane angles of 30^0, 40^0, 45^0 and 60^0 and swirl number is 111.673, 89.79, 77.67, 65.64 respectively.

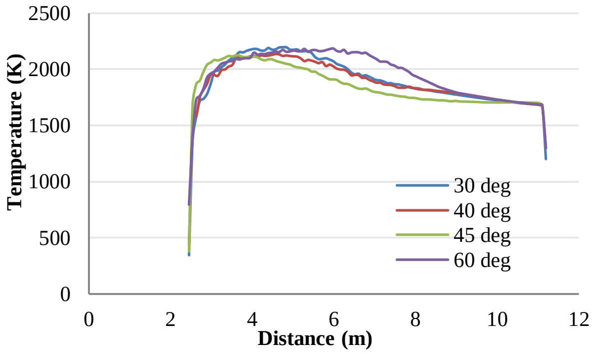

Figure 4.1 Plot between Temperature and distance from furnace inlet

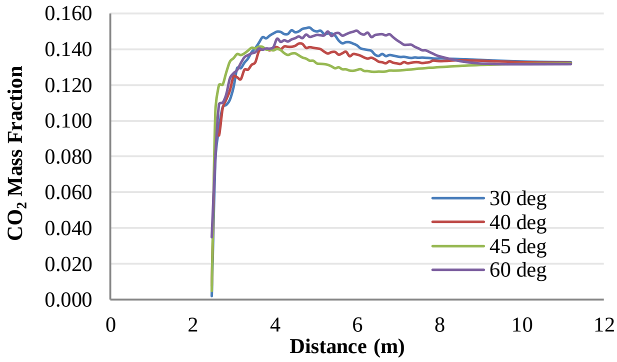

Figure 4.2 Plot between CO_2 mass fraction and distance from furnace inlet

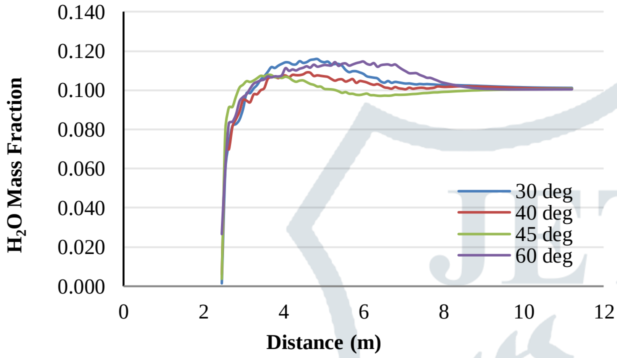

Figure 4.3 Plot between H_2O mass fraction and distance from furnace inlet

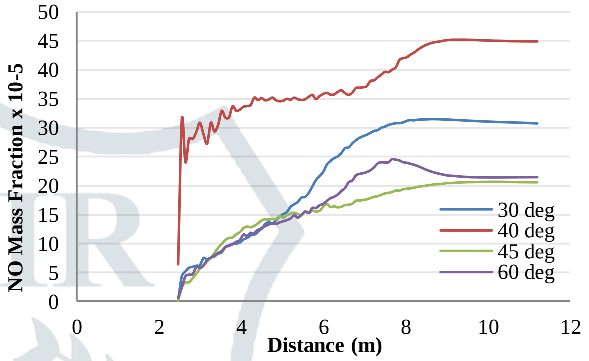

Figure 4.4 Plot between NO mass fraction and distance from furnace inlet

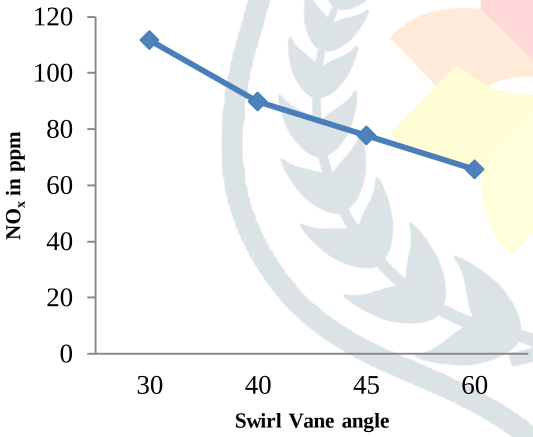

Figure 4.5 Plot between NO_x Vs. Swirl vane angle

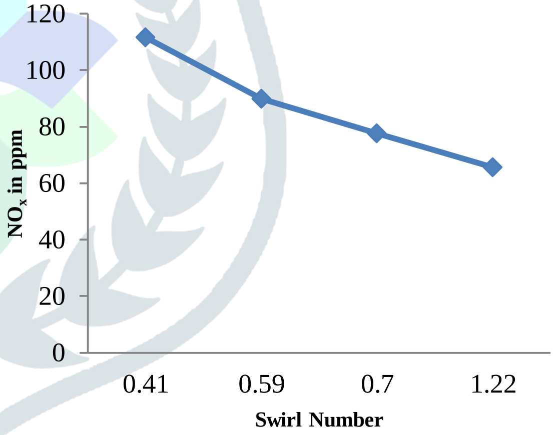

Figure 4.6 Plot between NO_x Vs. swirl number

V. CONCLUSION

As the different vane angle geometries are studied, it is found that as swirl vane angle (swirl number) increases, NO_x formation decreases. At 60^0 vane angle, NO_x ppm is least amongst all geometries simulated. At 60^0 vane angle, due to less availability of the flow area, high turbulence is generated. Turbulence results in excessive vibrations which may cause harm to the structure. Hence 60^0 vane angle is not suitable even though it gives the best results in terms of NO_x emission. 45^0 vane angle has better suitability for the intended purpose of having NO_x formation within acceptable limits along with less formation of CO_2, H_2O, NO_x and reduced vibrations.

VI. FUTURE SCOPE

For all four vane angles of the swirl, a constant air-fuel ratio is analyzed here. The variable ratio for further optimization can be done. Along with the multi-fuel ports, multi air injection can be used. For better combustion, secondary air can be used too after passing primary air.

VII. REFERENCES

[1] Weibo Chen and Guixiong Liu, “Numerical Investigation on the Flow, Combustion and NO_x Emission Characteristics in a 10 MW Premixed Gas Burner”, The Open Fuels & Energy Science Journal, vol.8, pp. 1-13, 2015.

[2] Petrica Iancua, Salvador Vilas-Bonafouxb, Jose M. Iglesias Fernandezb, Valentin Plesua, Jordi Bonetb, Alexandra E. Bonet Ruizb and Joan Llorensb, “Computational Fluid Dynamics (CFD) Simulation of Fuel Gas and Steam Mixtures to Decrease NO_x Emissions of Industrial Burners”, European Symposium on Computer Aided Process Engineering – ESCAPE 27, 2017.

[3] Risto Filkoski and Ilija Petrovski, “Computational Fluid Dynamics in Function of Boilers' Revitalisation”, researchgate publication, 2015.

[4] Timo Korpela, Pekka Kumpulainen, Yrjo Majanne, Anna Hayrinen and Pentti Lautala, “Indirect NO_x emission monitoring in natural gas-fired boilers”, Control Engineering Practice 65, pp. 11-25, 2017.

[5] Aruna Devadiga and Prof. Dr. T. Nageswara Rao, "Optimizing Bunsen burner Performance Using CFD Analysis", International Journal of Modern Engineering Research, vol. 3, pp. 2773-2785, 2017.

[6] Xuedong Wang, Fengxiu Xu, Zhihong Hu and Yuzhen Hao, “Experimental Study on Boiler Performance and NO_x Emission for Coal-Fired Boilers of High Capacity”, the Second China Energy Scientist Forum, 2010.

[7] Ilker Yılmaz, “Effect of Swirl Number on Combustion Characteristics in a Natural Gas Diffusion Flame”, Journal of Energy Resources Technology, December 2013.

[8] Mohamad Shaiful Ashrul Ishak and Mohammad Nazri Mohd. Jaafar “The Effect of Swirl Number on Reducing Emissions from Liquid Fuel Burner System”, Jurnal Mekanikal, No. 19, pp. 48 – 56, June 2005.

[9] P. Baziar, A. Hajipour, A. Fahimirad, M. Mahmoodi Arya, H.M. Heravi, “The Effect of Swirl on NO_x Formation in a Non-Premixed Propane/Air Flame” P. Baziar, vol. 5(15), pp 2175-2185, April 2015.

[10] Harun Yilmaz and Ilker Yilmaz, “Combustion and Emission Characteristics of Premixed CNG/H_2/CO/CO_2 Blending Synthetic Gas Flames in a Combustor with Variable Geometric Swirl Number” Journal of Energy Resources Technology, January 2019.

JETIR