Study on Combustion Performances of Burners in-Furnace in Coal Fired Power Plants

Martin Jose[1] Dr. James Varghese[2]
[1,2]Department of Mechanical Engineering
[1,2]SOE CUSAT

Abstract—Steam generation are one of the critical components of a thermal powerplant. Studies of the performance of combustion in a steam generator are very complex in nature. In this work numerical simulations are used to investigate the characteristics of gas and coal particle two-phase flow in boiler furnaces. In order to examine the detailed behavior of the flow different load conditions were given with varying the velocity of flow of air and fuel. Flow characteristics with multiple number of burners and firing at different locations resembeling variation of load is studied. Studies of effect of different operating conditions are studied using commercial code FLUENT. To determine velocity, temperature characteristics a 3D combustor model isused in this case study. Change of size in burners is done in order to find the temperature profiles changes in furnace. Analyses of particle trajectories are done so as to study the emission characteristics, fouling on burners etc.

Key words: ANSYS, CFD, Coal Combustion, Particle Trajectories, Burners

I. INTRODUCTION

Thermal power station is a power plant which prime mover is steam driven. Turbine spins due to the steam generated during heating of water which in turn drives electrical generator. Steam condenses after passing through condenser, and recycled where it is heated, this is the basic principle of Rankine cycle.

The interest on performance optimisation of large utility boilers has become very relevant in recent years. All the optimisation strategies are directed at extending their lifetime, increasing the thermal efficiency and reducing the pollutant emissions. Moreover, efficient use of pulverised coal is crucial to the utility industry. Pulverized coal combustion phenomenon can be divided into two steps; devolatilization and char combustion. As the ignition characteristic depends on the type of coal, it should be reflected in boiler design and the operating conditions should also be determined by preliminarily consideration of the compatibility with the design coal. In engineering practice, it is very difficult to investigate the combustion processes of various kinds of combustibles directly in the boiler. Rather than constructing real boilers and trying to check and improve these characteristics, computers are used to experiment with models of the boilers Simulations made with comprehensive combustion models and codes have led to significantly new physical insights into behavior of complex flows offering great potential for use in optimizing the performance of energy conversion systems. A number of CFD codes, being developed by specialized commercial companies, research organizations and individuals, are available worldwide. The codes provide qualitative, but not necessarily also quantitative characteristics of involving processes. For reliable application of the software packages with the possibility to control and get an insight into the calculation mechanism, a support (usually expensive) of the software owners is required most often. Getting insight into individual segments, as well as making single modifications of the software are not possible. For these reasons, we want, like many others, to provide our own mathematical models, numerical codes and software packages in trying to achieve an optimum usefulness of this powerful for solving technological problems of complex processes.

II. MODELING

A. Furnace Model Generation

A 3D model of boiler furnace, is modelled and 30 burners of diameter 0.2m for coal and primary air inlet and secondary air inlet of 0.3m diameter respectivelly. The furnace geometry of the simulated boiler can be seen in Fig. 1 and Fig 2. The opposite wall fired coal furnace is 46 m high, 19.5 m wide and 18 m deep and with an installed capacity of 500 MW. Thirty burners are arranged in an array of six burners deposited in different levels on two opposite walls of the furnace. The burner geometry specifications are given in the Table 1.

The main boiler operating conditions considered in this paper include base case (corresponding to full load with all burners in service, 3 rows of burners situated on the front wall of the furnace and two rows on the opposite wall), 2–2 firing (two rows of the furnace situated on the front wall and two rows on the opposite wall of the furnace), 2–1 firing (two rows on the front wall of the furnace and one row on the opposite wall in operation), respectively.

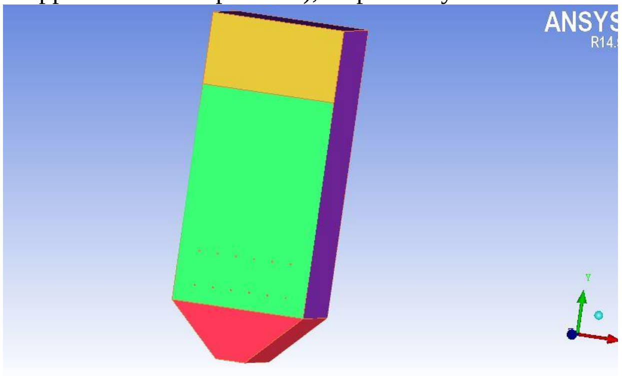

Fig. 1: Front view of furnace

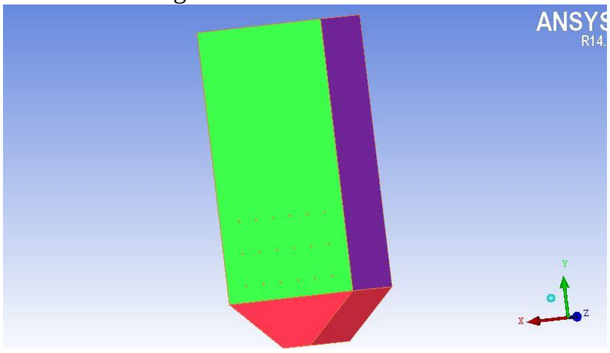

Fig. 2: Back view of furnace

Parameters	Values
Distance between top burner level and furnace outlet	30
Clearance between burners:	
Horizontal	2.29 & 1.98
Vertical	4.57
Burner to wall	2.48
Clearance between bottom burner level and furnace hopper	2.14

Table 1: Specifications of furnace geometry

B. Mesh Generation

The model is developed in ICEM-CFD and unstructured meshing is done. The meshed geometry contained 19,19,750 elements, unstructured meshing with tetrahedral cells. The model is imported to Fluent 14 and boundary conditions in such a way that the primary inlets are represented as velocity inlet with a velocity of 20m/s and secondary outlets are represented with velocity inlets with a velocity of 35m/s respectively. The geometry is as shown below. Fig 1 and 2, shows the front and back view of the furnace respectively. From the figure it can be seen that the ash hooper in the bottom part from both figures.

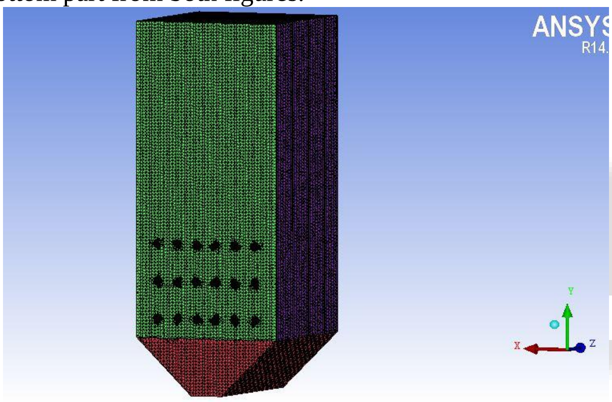

Fig. 3: Meshed front view of furnace

In Fig 1, 18 burners in the 2 front rows are shown and in Fig 4, 12 burners in the back side of the furnace in 2 rows is clearly seen. The outlet can be seen in the Fig 2, outlet is represented as yellow colour. The meshed 3D geometery section of the furnace is as shown in the Fig 3 and 4. Fig 3, shows the meshed front view of the furnace and Fig 4, shows the back views respectively.

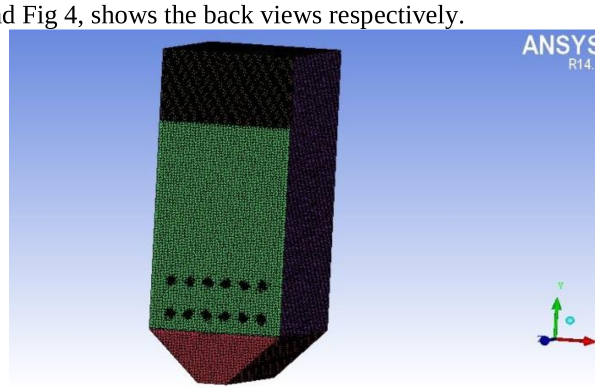

Fig. 4: Meshed back view of furnace

III. MATHEMATIC MODELING

This deals the numerical scheme for modelling combustion process which employs pulverized-coal in a tubular combustor unit. In any particle laden flow, there is exchange of momentum and energy between the discrete phase of particles and the continuous gas phase. The dynamics of gas phase is generally solved in an Eulerian frame using the mass, momentum and energy balance equations along with the equation of state, and additional source terms to represent the exchange of momentum and energy with particles. The particles are tracked from their initial state with respect to time, using a Lagrangian frame. The solutions for the two phases, therefore, are coupled through interactive source/sink terms. The governing equations, discretization of the computational domain and boundary conditions employed for the simulations along with the details of numerical solution procedure are described in this chapter. The validation of the numerical model developed by comparing its predictions with available experimental data from literature, is also presented. Coal combustion involves coupled mass, momentum and energy transfer processes in both the solid and gaseous phases.

The boundary conditions in such a way that the primary inlets are represented as velocity inlet with a velocity of 20m/s and secondary outlets are represnted with velocity inlets with a velocity of 35m/s respectively. The geometry is as shown below. Fig 1 and 2, shows the front and back view of the furnace respectively.

The outlet is given as pressure outlet and given temperature as that of walls which is 600K and emissivity which is equall to 1. The outer walls are given with a temperature of 600K and emissivity as that of 1. The coal properties such as proximate and ultimate analysis values are taken from the paper [3] and sum of proximate and ultimate values are maintaned to 1. The calorific value of fuel is been asuumed.

Proximate analysis	**(% db)**	**Ultimate analysis**	**(% daf)**
Moisture	17	Carbon	74.42
Ash content	4	Hydrogen	4.9
Volatile matter	37	Nitrogen	1.5
Fixed carbon	43	Sulfur	1.0
Total sulfur	0.83	Oxygen	18.13

Table 2: Coal sample

Case description	**Base case 3–2 firing**	**2–2 firing**	**2–1 firing**	**3–2 firing with excess air**
Total primary air flow rate (m3/s) & temperature(K)	19.95 & 343	15.96 & 343	11.97 & 343	28.48 & 343
Total secondary air flow rate (m3/s) & temperature(K)	89.57 & 573 K	71.66 & 573 K	53.74 & 573 K	99.38 & 573 K
VM in coal(%)	37	37	37	37
Total coal feed rate (kg/s)	68.1	54.4	40.8	68.1

Table 3: Boiler operating condition

The mathematical model used here is based on the commercial CFD code, FLUENT, where the gas flow is described by the time averaged equations of global mass, momentum, enthalpy and species mass fraction. The particle phase equations formulated in Lagrangian form and the coupling between phases is introduced through particle sources of Eularian gas-phase equations. The standard k-ε turbulence model, single mixture fraction probability density function (PDF) and the P1 radiation models are used in the present simulations. For the bulk of engineering combustion systems the mixing process proceeds much more slowly than the chemistry and as a result the mixing rate almost always determines therate of combustion. Therefore, for the predominant industry case of diffusion or non-premixed combustion for example, it normally suffices to solve a conservation equation for a mixing variable called the mixture fraction in order todetermine the temperature and concentrations of ma jor species. In this study, a simplified coal combustion furnace is modeled using the non-premixed combustion model for thereaction chemistry.

The coal details like ultimate, proximate analysis (by percentage mass), higher heating value etc. which are used in this stage is given Table 1. The data given in Discrete Phase Modelling and chemical reaction parameters were also set as follows. The number of continues iterations per DPM iterations was set to 40 and number time steps for particle tracking was set at 10000. The injection was chosen group, and rosin rammler distribution was given Parameters of Rossin Rammler are given in Table 4.

Parameter	Value
X position	0 m and 18.5 m
Y position	9.1m and 19.3 m
Temperature	300 K
Total flow rate	2.72 kgs^{-1}
Minimum Diameter	$10*10^{-6}$
Maximum diameter	$150*10^{-6}$
Mean diameter	$54.5*10^{-6}$
Spread parameter	2.51

Table 4: Rossin Rammler distribution date of pulverised coal

IV. RESULTS AND DISCUSSIONS

A. Validation of Furnace Geometry

The computational model has been applied to the furnace of 500 MW boiler fired with high-ash, medium volatile coal. The input data for simulations (including boiler operational conditions) is selected in accordance with data related to the experimental tests of the considered boiler. The properties and the lower heating value of fuel mass as received basis at a locll power station are assumed in calculations. The yield of volatile matter was taken according to the data provided (daf). Although, the full-scale boiler is equipped with 30 conventional burners of the swirling typ e arranged on the front and rear furnace walls in three tiers, only five of the burner sets are in operation at any particular time and one set is used as a standby. Hence, only five sets are considered as full load (hereafter referred to as base case) for mo delling the operation. The calculated primary and secondary air velo cities are 20 and 35 m/s, respectively, which have been determined from the mass flow of combustion air and temperature of preheated air. Another boundary condition is the outflow that is expressed as outlet with the external radiation temperature. This temperature is equal to the wall temperature and approximately corresponds with vaporization of water in evaporator. In this case, the temperature of the walls is equal to 600 K with an emissivity of 1. Fig 6, shows the velocity profile of the coal combustion. It can be seen from the figure that at the velocity ranges from 0-35m/s, having a higher value at the inlet. we can see the vortex formation in between where the two inlet conition meets and a sudden drop in velocity value can be seen. Fig 5, shows the temperature profile respectively. The values ranges from 450-1900 K which is giving a nearer relation as that of the reference paper[3]

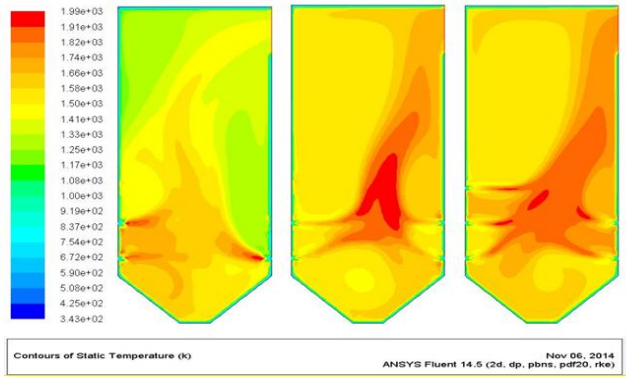

Fig. 5: Temperature profile for different load condition (a)2-1 firing (b)2-2 firing

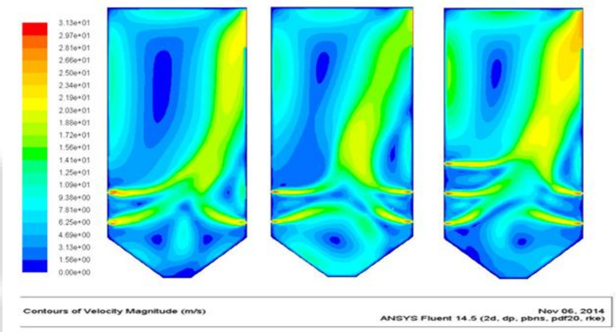

Fig. 6: Velocity profile for different load condition (a)2-1 firing (b)2-2 firing (c)3-2 firing

(c)3-2 firing. The predicted temperature and velocity distribution in a vertical plane crossing the burners is shown in Figs. 5 and 6, for the configurations considered. Initially some general trends observed are reported from the analysis of the temperature distribution, and then the comparison of each case is presented. The temperature distribution for all the cases are shown at a chosen vertical plane, z=5.25m from the left wall, the plane crossing first burner of each row considered. A similar scale range was used for all the cases, except for the excess air case for comparison. I t can be seen that the temperature approaches a maximum in the vicinity of the burner zone or at active combustion zone. The maximum temperature is ab out 1900 K , the upper limit established for the burner zone outlet, i.e., the non-slagging furnace operation is ensured. The figures show that combustion o ccurs mainly in the near burner region. As the furnace chamber is 46 m, the temperatures shown in the vertical plane implies that most of the combustion was occurring about half to three quarters of distance along the chamber.

B. Variation of Burner Geometry

After the validation of the paper, some alterations is been done in the furnace geometry so as to study the combustion performance of burners and its efficency. As a part of it what done is that the geometry of the burner is varied for that the 3 different cases with there different diameter of burners is taken into study. ie the thee cases taken into study are 0.1m, 0.2m, 0.3m dia respectively. The geometry of 3 cases are shown in the Fig 7. The boiler furnace is made in a such a way that the burner dimensions are altered and 3 different geometries corresponding to 3 different cases are created. The geometry is drawed and meshed in ICEM-CFD an Ansys parameter and the mesh is been imported to FLUENT 14. The solution calculation is been done here. The air velocities were kept constant for each 3 cases for both primary as well as secondary. Same coal properties is been used for the analysis such that the temperature distribution variation while using different diameter burners.

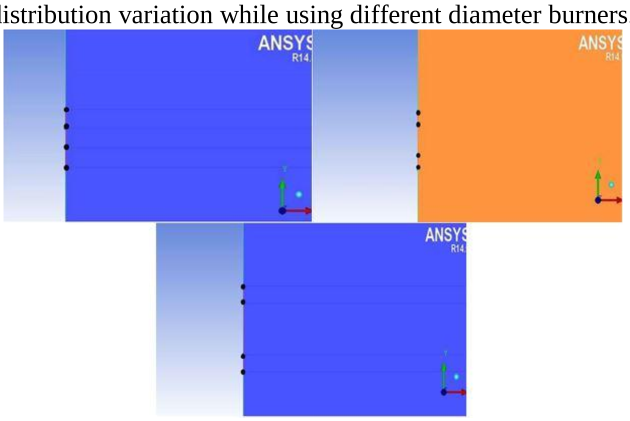

Fig. 7: Burner geometry of 3 cases (i) 0.1 m, (ii) 0.2 m, (iii) 0.3 m

The combustion is been done in such a way that the 2 burners in the front wall is made active and the rest burners were kept idle. This is done because of to reduce the computational time of the program. All boundary conditions were same as that of the previous work. The result obtained is as shown fig 8. From the figure it can be seen that as the diameter of burners increases the temperature will be increasing till the half of third quarter of the furnace. Then it decreases subsequently. By taking the highest temperature points along the interior points a graph plolted Temperature vs Furnace height which will be as shown in the Fig 9.

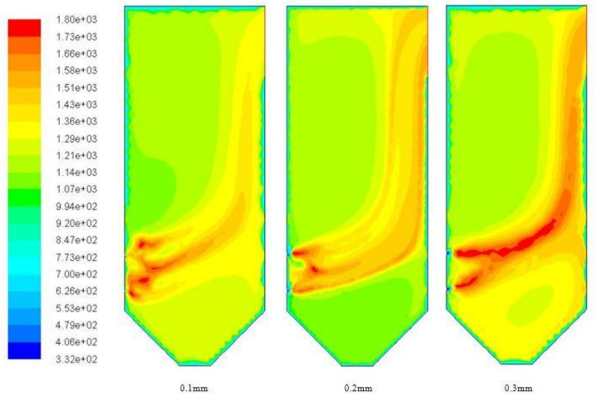

Fig. 8: Temperature profile for three cases (1)0.1m, (2)0.2m, (3)0.3m

From the graph it can be seen that as diameter of the burner increases temperature in the interior position will be higher till be the location of the burner positions. Thereafter the values goes on decreasing till the outlet. From graph it can be seen that for the diameter 0.3m and 0.2m the temperature at the exit will be almost in a similar range. The outlet temperature of the furnace is seem to be of almost same range only 10-20K variation can be seen. From this it can be infer that the size of burner is not a matter for getting higher outer temperature which is provided to the superheaters for heating water inorder to produce steam. Any other parameters such as swirl, preheating of air may give good combustion and higher outlet temperature.

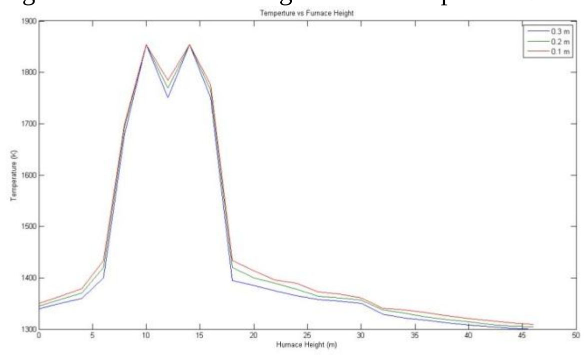

Fig. 9: Temperature vs Furnace height graph

V. CONCLUSIONS

Modelling of a wall-fired furnace is carried out using CFD commercial flow solver FLUENT. Simulation results with full and partial loads with different burners in operation suggested a significant impact on the distribution of temperatures and furnace exit temperatures. This indicates that extreme care must be taken in terms of maintenance and service of burners so as to minimise the effects on combustion and continue smooth operation of boilers. Simulations carried out with excess air indicated that the temperatures within the boiler will increase. This could lead to severe ash deposition problems. In particular, particles emerging out lower burner sections tend to reside within the boiler longer than the particles at other locations. This may lead to poor com-bustion, which is undesirable for efficient operation of power station boilers.

By varying the burner geometry it can be seen that as diameter of the burner increases temperature in the interior position will be higher till be the location of the burner positions. Therafter the values goes on decreasing, so required temperature cannot be obtained at superheater sections for more combustion rate than the usual range.

REFERENCES

[1] Numerical study of flow in a 2D b oiler, Petr Bauer, Vladimir Klement, Pavel Stra chota, Processing of the algoritmy,(2012), pp 172-178.
[2] Study on fluid flow characteristics in different load cases in tangen-tially fired furnaces, Jianren Fan, Xinyu Zhang, Guangda You, Kefa Cen, Chemical Engineering Communications (2010), pp 1348-1370.
[3] Modelling of a wall fired furnace for different operating using FLU-ENT, R.Vuthaluru, H.B.Vuthaluru, Fuel Processing Technology (2006), pp 633–639.
[4] Exp erimental and numerical study on the flow fields in upper furnace for large scale tangentially fired fired boilers, Yuegui Zhou,Tongmo Xu, Shien Hui,Mingchuan Zhang,Applied Thermal Engineering (2009), pp 732–739 .
[5] Saving fuel consumption and reducing p ollution emissions for indus trial furnace, Chien-Li Lee a, Chih-Ju G. Jou, Fuel Pro cessing Technol-ogy 92 (2011), pp. 2335-2340.

[6] Influence of fuel comp osition on chemiluminescence emission in premixed flames of CH4/CO2/H2/CO blends, Tatiana Garca-Armingol, Javier Ballester, Internation journal of hydrogen energy, 39 (2014),pp. 20255-20265
[7] Effect of Preheated Air on the Structure of Coaxial Jet Diffusion Flame, S.Sreenatha Reddy, International Archive of Applied Sciences and Technology, 2 (2013), pp. 70-75