CFD Simulation of Circulating Fluidized Bed Combustion
Sudheer¹ M.R.Nagaraj²
¹²Department of Mechanical Engineering
¹²PDA College of Engineering Gulbarga, Karnataka, India

Abstract— Circulating fluidized bed combustion using the fluent software is analyzed. The increase in application of fluidized bed combustion device throughout the world shows that more consideration is given to improve the design and reduce emissions of these. Due to excellent thermal and mixing properties, fluidized beds are generally preferred over the fixed bed combustors. Various industrial boilers play an important role to improve the power generation cycle such as CFBC (circulating fluidized bed combustion), AFBC (atmospheric fluidized bed combustion boiler), co-boiler etc. It is intended to comprehensively give an account of temperature, pressure, in CFBC boiler with due effect of flue gas flow during operation on refractory by using latest technology of cad (computer aided design). The results obtained are helpful to understand temperature, pressure of flue gasses and usage of flue gas to increase thermal efficiency.

Key words: CFD Simulation, Coal, CFBC Loops, CFBC Boiler, Two-Phase Flow and DPM

I. INTRODUCTION

Fluidized bed combustion (FBC) is today a well-established technology for generation of heat, power and a combination of these. Yet, there has been a constant development and refinement of the technology since it reached commercial status in the early 80’s. With respect to the development of the technology, two factors can be mentioned which to a certain extent make the FBC development differ from that of other solid-fuel combustion technologies. First, the fuel flexibility, which is one of the main advantages of the technology, has put focus on different fuels over time since the introduction of the FBC technology. The focus of the development has also been different in different regions of the world, depending on fuel availability. Thus, the various types of fuels yield different demands on the technology (importance of mixing, material issues, heat transfer yield different problems and challenges distribution etc.). Secondly, the two main applications of the FBC technology, smaller heat only or combined heat and power boilers burning renewable and waste fuels and large power boilers mainly burning coal.

A. CFBC Technology

Circulating fluidized bed combustion is the most used & economical technology adopted by the industries. Deterioration of coal quality and pollutant gases (NOx) arising out of burning coal in conventional utility boilers lead to the development of fluidized bed combustion boilers. The main advantages of the fluidized bed combustion boilers are reduced NOx, SOx due to relatively low combustion temperature, better efficiency and reduction in boiler size and design. It has the ability to burn low grade coal and it is less corrosive as the combustion temperature is less when compared to that of an utility boiler. In addition to all of these, the startup and shut down operation of CFBC boilers are much easier. Fluidization is the process by which the solid particles are brought to a suspended state through gas or liquid. When air or gas is passed upward through the solid particles at low velocity, they remain undisturbed. As particles reach the state of "fluidization". A CFBC boiler may be divided into two sections the CFB loop and the convective or back-pass section of the boiler. The CFB loop consists of the following items making up the external solid recirculation system.

1) Furnace or CFB riser
2) Gas–solid separation (cyclone)
3) Solid recycles system (loop-seal)
4) External heat exchanger (optional)

The air system is very important for the CFB boiler, as it consumes the greatest amount of power. A typical utility CFB boiler would use three types of fans/blowers.

1) Primary air fan
2) Secondary air fan
3) Loop-seal air fan or blower

Fig. 1: CFBC boiler general arrangement

Fig. 2: CFBC boiler components

The primary air fan delivers air at high pressure (10 to 20 kpa). This air is preheated in the air preheater of the boiler and then enters the furnace through the air distributor grate at the bottom of the furnace. The secondary air fan delivers air, also preheated in the air preheater, at a
relatively low pressure (5 to 15 kpa). It is then injected into the bed through a series of ports located around the periphery of the furnace and at a height above the lower tapered section of the bed. In some boilers, the secondary air provides air to the start-up burner as well as to the tertiary air at a still higher level, if needed. The secondary air fan may also provide air to the fuel feeder to facilitate the smooth flow of fuel into the furnace. Loop-seal blowers deliver the smallest quantity of air but at the highest pressure. This air directly enters the loop-seals through air distribution grids. Unlike primary and secondary air, the loop-seal air is not heated.

B. Statement of Problem

To study of temperature and pressure of flue gasses which effect on refractory system and waste heat generated by industry is dispersed to environment, even though it could still be reused for some other process and economical purpose. Large quantity of flue gases is produced from boiler, furnaces. By saving this waste heat consider able fuel could be saved.

C. Objective of Study

The objectives of the present work are seen as possible after a thorough literature survey. Inspite of much work has been described on the analysis of cfbc considering different parameters, the analysis considering the parameters such as temperature and pressure. The present work includes:

- To study the temperature and pressure of flue gasses which effect on refractory system.
- The utilization of a three dimensional CFD model to analyze the temperature distribution.
- To minimize energy waste without affecting production and quality and minimize environmental effects.

D. Literature Survey

Konstantin P. Filipov, presented on mathematical model describing two-phase flow in CFB in framework of external model ash was elaborated. It takes into account the all main processes in high-velocity fluidized bed including processes of the coal combustion and the attrition of ash. On base of this model was developed numerical code and carried out some calculations of transient and stationary performances of CFB. [1]

Williams and M. Pourkashanian, worked on the some of the key issues currently being debated regarding the combustion of coal and of some biomass materials. It attempts to summarize the present approaches toward the quantification of the fundamental processes of solid fuel combustion for use in computer models. Some aspects of the various chemical and physical processes are included, such as heating-up of particles, devolatilization, and subsequent char formation. Of particular interest is the prediction of char properties, such as composition, surface areas, and morphology, since these impacts on char combustion. [2]

Z.Guangbo, focused on a steady state model of a coal-fired circulating fluidized-bed boiler, based on hydrodynamics, heat transfer and combustion, is presented. This model predicts the flue gas temperature, the chemical gas species (O2, H2O, CO, CO2 and SO2) and char concentration distributions in both the axial and radial locations along the furnace including the bottom and upper portion. The model was validated against experimental data generated in a 35th commercial boiler with low circulation ratio. [3]

Afsin Gungor, worked on a dynamic two dimensional model is developed considering the hydrodynamic behavior of CFB. In the modeling, the CFB riser is analyzed in two regions the bottom zone in turbulent fluidization regime is modeled in detail as two-phase flow which is subdivided into a solid-free bubble phase and a solid-laden emulsion phase. In the upper zone core–annulus solids flow structure is established. Simulation model takes into account the axial and radial distribution of void age, velocity and pressure drop for gas and solid phase, and solids volume fraction and particle size distribution for solid phase. The model results are compared with and validated against atmospheric cold bed CFB units’ experimental data given in the literature. [4]

II. THE PHYSICAL MODEL

In this method of modeling physical model and co-ordinate system employed for carrying out numerical analysis are described in this chapter along with flue gas recovery is to achieve and maintain optimum energy procurement and utilization of CFBC.

A. Discrete Phase Model (DPM)

In addition to solving transport equations for the continuous phase, FLUENT allows you to simulate a discrete second phase in a Lagrangian frame of reference. This second phase consists of spherical particles (which may be taken to represent droplets or bubbles) dispersed in the continuous phase. FLUENT computes the trajectories of these discrete phase entities, as well as heat and mass transfer to/from them. The coupling between the phases and its impact on both the discrete phase trajectories and the continuous phase flow can be included. FLUENT provides the following discrete phase modeling options:

1) Calculation of the discrete phase trajectory using a Lagrangian formulation that includes the discrete phase inertia, hydrodynamic drag, and the force of gravity, for both steady and unsteady flows.
2) Combusting particles, including volatile evolution and char combustion to simulate coal combustion.
3) These modeling capabilities allow FLUENT to simulate a wide range of discrete phase problems including particle separation and classification, spray drying, aerosol dispersion, and bubble stirring of liquids, liquid fuel combustion and coal combustion.
B. Problem Description

The problem consists of a two phase fluidized bed in which gas and solid (coal) enters at the separate portion of the domain. For a 2D CFB furnace, estimate the primary air is flowing at 503 k and 6m/s which is flowing through the nozzles. The secondary air is injected at the level of 2.3 m from the level of the bottom at 473 k and 2m/s and the fuel feed rate 0.5 kg/s is injected at the level of 1.5m from the level of the bottom, corresponding to very fuel-lean conditions in the flow. The bed cross-section is 3.2 m x15 m below and 6.7 m x15 m above this level. The Reynolds number, based on inlet conditions and the flow is turbulent. The combustion is modeled using the mixture-fraction approach. The bed consists of solid material (coal particles) of uniform diameter and the size of solid particles is 5000 μm. Data for coal analysis is given below. Coal – Lignite (composition on dry) basis C – 66%, H – 6%, O– 26%, N – 1%, S – 1%, HHV – 25MJ/Kg.

C. Analysis Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary air velocity</td>
<td>6-8m/s</td>
</tr>
<tr>
<td>Secondary air velocity</td>
<td>2m/s</td>
</tr>
<tr>
<td>Coal particle size</td>
<td>5000μm</td>
</tr>
<tr>
<td>Primary air temperature</td>
<td>503k</td>
</tr>
<tr>
<td>Secondary air temperature</td>
<td>473k</td>
</tr>
<tr>
<td>Mass flow rate</td>
<td>0.5kg/s</td>
</tr>
<tr>
<td>Oxygen concentration at inlet</td>
<td>21%</td>
</tr>
<tr>
<td>Back flow temperature</td>
<td>1800k</td>
</tr>
<tr>
<td>Initial pressure</td>
<td>1.0x10⁶pa</td>
</tr>
<tr>
<td>Coal heat capacity</td>
<td>1000kJ/kg</td>
</tr>
<tr>
<td>Coal density</td>
<td>1350kg/m³</td>
</tr>
<tr>
<td>Mean mixture fraction</td>
<td>0.09</td>
</tr>
<tr>
<td>Oxide in PDF table</td>
<td>600k</td>
</tr>
</tbody>
</table>

Table 1: Analysis Parameters

III. MATHEMATICAL FORMULATION

A. Continuity Equation

The conservation of mass equation or continuity equation is given by

\[ \frac{\partial \rho}{\partial t} + \nabla(\rho V) = 0 \]  (1)

Where \( \rho \) is the density, \( V \) is the velocity vector.

B. Momentum Equation

Applying the Newton’s second law (force=mass \times acceleration) the conservation of momentum equation is given by,

\[ \frac{\partial (\rho V)}{\partial t} + \nabla(\rho V V) = -\nabla p + \nabla \tau + \rho g + F \]  (2)

Where \( \rho \) is the density, \( V \) is the velocity vector, \( p \) is the static pressure, and \( \tau \) is the stress tensor (described below), \( \rho g \) and \( F \) are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. The stress tensor \( \tau \) is given by

\[ \tau = \mu \left( \Delta V^T + \Delta V \right) - \frac{2}{3} \nabla \cdot V I \]  (3)

Where \( \mu \) is the molecular viscosity, \( I \) is the unit tensor, and the second term on the right hand side is the effect of volume dilation.

C. Energy Equation

Energy is neither created nor destroyed. It is always conserved. The conservation of energy equation is given by

\[ \frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\rho (E + p) V) = \nabla \cdot \left( k \nabla T \right) - \sum \dot{h}_j j + \left( \tau_{eff} \cdot V \right) + S_h \]  (4)

Where sensible enthalpy \( h \) is defined for ideal gases.

\[ h = \sum_{j} h_j Y_j \]  (6)

For incompressible flows as

\[ h = \sum_{j} h_j Y_j + \frac{p}{\rho} \]  (7)

These equations form a set of coupled, nonlinear partial differential equations. It is not possible to solve these equations analytically for most engineering problems. However, it is possible to obtain approximate computer based solutions to the governing equations for a variety of engineering problems. This is the subject matter of Computational Fluid Dynamics (CFD).

D. Energy Equation for Non-Premixed Combustion Model

When the non-adiabatic non-premixed combustion model is enabled, FLUENT solves the total enthalpy form of the energy equation,

\[ \frac{\partial}{\partial t} (\rho H) + \nabla \cdot (\rho H V) = \nabla \cdot (k \nabla H) + S_h \]  (8)

Under the assumption that the Lewis number \( (Le) = 1 \), the conduction and species diffusion terms combine to give the first term on the right-hand side of the above equation while the contribution from viscous dissipation appears in the non-conservative form as the second term. The total enthalpy \( H \) is defined as

\[ H = \sum_{j} H_j Y_j \]  (9)

Where \( Y_j \) is the mass fraction of species \( j \) and

\[ H_j = \int_{T_{ref}}^{T} c_p dT + h_j^0(T_{ref}) \]  (10)

\( h_j^0(T_{ref}) \) is the formation enthalpy of species \( j \) at the reference temperature \( (T_{ref}) \).

E. Standard \( K-C \) Model

The turbulence models are the two-equation models. The simplest one is the standard \( K-C \) model, which is proposed by Launder and Spalding. It is widely used in turbulence simulations because of its general applicability, robustness and economy. The two transport equations for the kinetic energy and dissipation rate are solved to form a characteristic scale for both turbulent velocity and length. These scales represent the turbulent viscosity.

The modeled transport equations for \( K \) and \( C \) in the realizable \( K-C \) model are

\[ C_1 = \max \left[ \frac{0.43}{\eta}, \frac{\eta}{\eta + S} \right] \]

\[ S = \frac{k}{\varepsilon} \frac{d}{dS} = \sqrt{2} S_{ij} S_{ij} \]
\[
\frac{\partial}{\partial x_j}(\rho ku_j) + \frac{\partial}{\partial t}(\rho k) = \frac{\partial}{\partial x_j}\left((\mu + \frac{\mu_t}{\sigma_k})\right) + G_k + G_b - Y_M - \rho \epsilon \quad (11)
\]

In these equations, \( G_k \) represents the generation of turbulence kinetic energy due to the mean velocity gradients. \( G_b \) is the generation of turbulence kinetic energy due to buoyancy. \( Y_M \) represent the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. \( C_2 \) and \( C_1 \) are constants \( \sigma_k \) and \( \sigma_\epsilon \) are the turbulent Prandtl numbers for \( K \) and \( \epsilon \) respectively. \( \sigma_k \) and \( \sigma_\epsilon \) are user-defined source terms.

IV. NUMERICAL METHOD AND SOLUTION PROCEDURE

A. Model Creation and Grid Generation In Gambit

The grids are selected for all the meshes for doing CFD analysis. As the furnace for which analysis is carried out, quadrate type mesh is selected. This specifies that the mesh is composed primarily of quadrate mesh elements. The qualities of the created mesh are checked.

![Fig. 5: View of the model in 2D after modeling in GAMBIT2.3.16](image)

GAMBIT 2.3.16 was used for making 2D furnace geometry with width of 3.2m from the lower part and height 15m. Coarse mesh size of 0.01m was taken in order to have 9365 cells (18952 faces) and 9588 nodes for the whole geometry. It was used in order to have better accuracy. But using mesh results in 9365 cells (18952 faces), which requires smaller time steps, more number of iterations per time step and 4 times more calculation per iteration for the solution to converge.

B. Boundary Conditions

The boundary conditions are as equally important as the selection of the proper mathematical model. Initially, solid particle velocity was set at in minimum fluidization and gas velocity was assumed to have the same value everywhere in the bed. The temperature of the primary and secondary air was also set to 503k and 473k respectively. At the inlet, all velocities of all phases were specified. At the outlet, the pressure was assumed to be Atmospheric pressure. The gas tangential normal velocities on the wall were set to zero (no slip condition). The following boundary equation was applied for the tangential velocity of Particle on the wall.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet1</td>
<td>Inlet1 Primary(Fluidized) Velocity inlet</td>
</tr>
</tbody>
</table>

Table 3: boundary conditions

V. RESULT AND DISCUSSIONS

Analysis of combustion at fluidizing

A. Velocity 6m/s, Total Temperature

The figure shows that the temperature profile in circulating fluidized bed combustor. In this figure, the bed and temperature are increasing as soon as the coal particles are burning and finally obtained the maximum value of total temperature after coal combustion is 1373k.

![Fig. 5.1: Contours of Total Temperature (k)](image)

1) Static Pressure

The value of static pressure is found in the combustor is 10 Pa after combustion process. The static pressure is rapidly increased when air fuel velocity enters in the combustion chamber. In the combustor, maximum static pressure is at the inlet points of the furnace.

![Fig. 5.2: Contours of Static Pressure (Pascal)](image)

2) Turbulence Kinetic Energy

The maximum turbulence intensity is found to be 6.86 e-01 after the combustion. In case of turbulence of kinetic energy, when the velocity of a gas is increased above the minimum bubbling velocity, the bed starts expanding. A continued increase in the velocity may eventually show a change in the pattern of bed expansion. In the turbulent kinetic energy regime, the bubble phase loses its identity due to rapid coalescence and break up the bubbles. This results in a violently active and highly expanded bed with a change in the pattern of bed expansion.
increase in the velocity may eventually show a change in the pattern of bed expansion. In the turbulent kinetic energy regime, the bubble phase loses its identity due to rapid coalescence and break up the bubbles. This results in a violently active and highly expanded bed with a change in the pattern of bed expansion.

B. Velocity 7 m/s, Total Temperature

The figure shows that the temperature profile in circulating fluidized bed combustor. In this figure, the bed and temperature are increasing as soon as the coal particles are burning and finally obtained the maximum value of total temperature after coal combustion is 1376 K.

1) Static Pressure

The value of static pressure is found in the combustor is 13 Pa after combustion process. The static pressure is rapidly increased when air fuel velocity enters in the combustion chamber. In the combustor, maximum static pressure is at the inlet points of the furnace.

2) Turbulence Kinetic Energy

The maximum turbulence intensity is 6.85e⁻¹ found to be after the combustion. In case of turbulence of kinetic energy, when the velocity of a gas is increased above the minimum bubbling velocity, the bed starts expanding. A continued

C. Velocity 8 m/s, Total Temperature

When the fuel and air enter into the combustor, it burns due to high velocity and temperature and then temperature increase rapidly in the combustor. Finally obtained the result, the total temperature of the coal combustion is 1378 K. The figure shows that the temperature profile in circulating fluidized bed combustor. In this figure, the bed and temperature are increasing as soon as the coal particles are burning and finally obtained the maximum value of total temperature after coal combustion is 1378 K.

1) Static Pressure

The value of static pressure is found in the combustor is 13 Pa after combustion process. The static pressure is rapidly increased when air fuel velocity enters in the combustion chamber. In the combustor, maximum static pressure is at the inlet points of the furnace.
The value of static pressure is found in the combustor is 15 Pa after combustion process. The static pressure is rapidly increased when air fuel velocity enters in the combustion chamber. In the combustor, maximum static pressure is at the inlet points of the furnace.

2) Turbulence Kinetic Energy
The maximum turbulence intensity is found to be 6.86e-01 after the combustion. In case of turbulence of kinetic energy, when the velocity of a gas is increased above the minimum bubbling velocity, the bed starts expanding. A continued increase in the velocity may eventually show a change in the pattern of bed expansion. In the turbulent kinetic energy regime, the bubble phase loses its identity due to rapid coalescence and break up the bubbles. This results in a violently active and highly expanded bed with a change in the pattern of bed expansion.

Fig. 5.9: Contours of turbulence kinetic energy (k) (m^2/s^2)

The below fig shows the profile between the total temperature and the position of the combustor on all conditions such as default interior, pressure-outlet, primary air, secondary air and wall but fig. shows the plot the x-y diagram between total temperature and position of the combustor on pressure outlet condition.

Fig. 5.10: Total temp v/s position

D. Behavior of Total Temperature at Different Fluidizing Velocities
Following are the trends of Total temperature v/s inlet air velocity obtained at different inlet fluidizing velocities, which show that temperature increases when air velocity is increased the figure shows that the temperature profile in circulating fluidized bed combustor. In this figure, the bed and temperature are increasing as soon as the coal particles are burning and finally obtained the maximum value of total temperature after coal combustion is 1378 k.

Fig. 5.11: Behavior of temperature at different fluidizing velocities

E. Behavior of Pressure at Different Fluidizing Velocities
Following plots of various pressures vs. inlet air velocity obtained at different fluidizing velocities. This plots show that pressure increases as air velocity is increased. This is because with increase in air velocity and mass flow rate of fuel in the combustor increases thereby increasing the pressure across the combustor. In the above fig when the fluidizing velocity is 6m/s at the inlet of combustor then the static pressure is observed 10 Pascal after coal combustion in the combustor of CFB i.e. the static pressure is increases with increasing the inlet velocity. The same conditions are for total and dynamic pressure.

Fig. 5.12: Behavior of pressure at different fluidizing velocities

VI. CONCLUSION
CFD simulation study has been conducted on full scale model to understand the behavior & flow of flue gases inside the CFBC loop. The model is setup & solved using ICEM CFD (ANSYS) software. Results for temperature and pressure at different velocity are obtained. The results from CFD were compared with the standard & available data from the processing equipment supplier & found within acceptable limits.

In this work, analysis of coal combustion in circulating fluidized bed has been performed at three different fluidizing velocities with fluent software.
Followings conclusions are drawn from the computational analysis in this present work.

Following conclusions are

- In the above results the maximum total temperature for velocities from 6m/s to 7.5m/s is linearly increased and from 7.5m/s to 8m/s is almost constant.
- The maximum static pressure increases from 10Pa to 15Pa, when fluidizing velocity is increased from 6m/s to 8m/s. This increase in the static pressure inside the combustion chamber is because there is no leakage of air from the combustion chamber.
- The velocity at 6m/s to 8m/s, static pressure increases, the resistance to flow of fluid inside the combustion chamber, it means that at these velocities the combustion takes place linearly.
- The velocity from 6m/s to 6.5m/s the dynamic pressure increases rapidly and from 6.5m/s to 8m/s linearly increases. It means that at low velocities the dynamic pressure will be low.
- It has been observed that all the parameters temperature and pressure are better for combustion at fluidizing velocity 8m/s. Therefore, a fluidizing velocity of 8m/s is suitable for fluidized bed combustion as compared to 6m/s and 8m/s.

Acknowledgment

The authors would like to thank to M. R. Nagaraj, Associate professor, Dept. of mechanical Engineering, PDA College of Engineering Gulbarga for an outstanding support.

VII. References


