# CFD Analysis of Coal/Air Flow in Power Plants for Optimum Size of Orifice Flow Restrictor

**Sunith Bawankule**

M.Tech. student, Govt. College of Engineering, Amravati [M.S.], India

**Abstract:** ***Unbalanced coal/air flow in the pipe systems of coal-fired power plants will lead to non-uniform combustion in the furnace, and hence lower the overall efficiency of the boiler. This unbalanced is caused due to the non-uniform flow rate at the burners due to different travelling length. The common method to maintain the uniform feed rate is to put orifice flow restrictor in the piping system and size is decided by using semi-empirical methods. Commercially available CFD package is used to simulate the complex flow in transport system. The two-phase modelling technique was utilized to estimate the pressure drop coefficients with coal/air flows in order to size the orifices. This study represents that the orifice flow restrictor size depends on pressure drop of the system which directly related with coal-gas loading ratio, mass flux and system geometry. CFD analysis is done for deciding the optimum geometry of the orifices to balance the flow in the existing power plant.***

**Keywords:** Balancing flow, CFD, Coal/Air balancing, Fluent, Pneumatic coal transport, Power plant

## 1. Introduction

As global warming continues to impact the environment within which we live, the burning of fossil fuels as a source of energy is being placed under increased political and social scrutiny. As a result, industrial users of coal are being forced to either reduce consumption, or find ways to reduce total $CO_2$ emissions, while maintaining current production rates. In unison, with the globally increasing price of coal and focus on reuse recycling there is a move towards firing waste products, particularly biomass and waste oil, as a substitute for fuels such as coal. One of the clean coal technologies is the fluidized-bed coal gasification which can be very good candidate for utilizing low grade coal. For stable and steady operation of the fluidized-bed coal gasification facility, coal should be transported pneumatically to fluidized-bed reactor without any plugging and discontinuous coal feeding problems.

Poor coal flow distribution to the burners is a common problem in pulverized coal (pc) boilers and has been considered as a potential area that needs to be addressed for improving unit performance, emissions, operations, and maintenance [1]. Coal pipe imbalances among the burners results in deviation from the design values for air- to-fuel ratios in the burners. High coal flow to burners can create carbon-rich zones of reducing atmosphere which leads to increased slagging, increased carbon monoxide (CO) emissions, and increased LOI (loss of ignition). Burners with too little coal flow can create oxygen-rich zones that may increase nitrogen oxides (NOx) emissions. Further, the parameters that affect coal quality (hardness, moisture, energy content, *etc.*) may create fluctuating conditions during boiler operation.This causes an associated deterioration in combustion efficiency due to increased carbon in fly ash level and leads to increased fuel and ash handling cost and possible deterioration in ESP collection efficiency.

In pulverized coal fired utility boilers, coal fineness and uniform distribution of coal flow from pulverizer to burners, lead directly to better control and performance of the firing system [3]. Finer coal particles burn quickly and more efficiently, reducing unburnt carbon in the fly ashwhile maintaining low NOx emissions and increasing boiler efficiency. However, non-uniform pulverized coal flow distribution to the burners is a prominent problem. This results in lower combustion efficiency due to different air/fuel ratios in the burners which can cause unstable combustion with higher carbon monoxide (CO) emissions and higher fly ash unburnt carbon. For this reason, balancing coal flow is a primary objective of combustion optimization. Hence, in order to ensure equal flow in different pipelines, it is essential to make the flow resistance equal in all the pipelines for any given flow rate. In order to achieve this objective, orifices of various sizes are introduced in the pipelines so that pressure drops become equal in all the pipes. The present study demonstrates the successful use of Computational Fluid Dynamics (CFD) methodology for optimizing the design of orifices in the pulverized coal (PC) pipelines feeding the boiler.

Several authors have used empirical and CFD studies for evaluating the pressure drop across the pipe flow. SowjanyaVijiapurapu, Jie Cui, SastryMunukutla (2006)[1] – They proposed the size of orifice for balancing coal/air flow. The resistance of a system is different which is expressed as a dimensionless pressure drop coefficients. For calculating pressure drop coefficients, CFD is used. The diameter of the orifice can be calculated from available empirical equations,

$$K_{OR} = K_2 - K_1 = \left[\left(\frac{F1}{F0} - 1\right) + \left(0.707 * \frac{F0}{F1}\left(1 - \frac{F0}{F1}\right)^{0.375}\right)\right]^2$$

where $\frac{FO}{F1} = \frac{AO}{A1} = \left(\frac{DO}{D1}\right)^2$, Ao, A1 correspond to the area of orifice and pipe and Do and D1 correspond to the diameter of orifice and pipe.

$K_2$, $K_1$ are pressure drop coefficient of system 1 and 2. Orifices size are based on calculated coal/air pressure drop. The results show that the pressure drop in the system strongly depends on the system geometry.

R. Vijayakumar, S.N. Singh, V. Seshadri[2] primary focus on to equalize the flow in all the four pipe lines conveying pulverized coal from mill to the firing elevation in the boiler. For achieving this objective pressure drop in the individual set of pipes has been evaluated for equal mass flow rate, and resizing the orifices has been done to get equal pressure drop among all pipes for same flow rate, using CFD software FLUENT. Chan Lee, Jin Wook Lee, Gyoo Tae Kim and Tae Wan Kwon[3] limit coal/gas loading conditions to secure stable coal feeding and the correlations for pressure losses in horizontal, vertical and elbow pipes, which can be suitable for the design guidelines of actual fluidized-bed coal gasification is provide. Numerical analyses by using CFD method are made to investigate how coal is transported with conveying gas and its particle behavior is related with gas pressure loss inside the key flow elements of coal transport system. In addition, with changing coal/gas loading condition, the present prediction results give design criterion and guidelines for reliable and efficient coal-transport system. V Singh and Simon Lo[4] studied that the pressure drop in the system is dependent on a host of parameters such as particle and pipe diameters, particle and fluid properties, pipe roughness and orientation, etc. It is found that the percentage of wall particle collisions decrease with respect to particle-particle collisions, as the solid loading is increased. The number of particle collisions is also sensitive to particle properties.

Two parameters, "pressure drop along the line" and "minimum conveying velocity" play major roles during the design of reliable pneumatic transport systems. So, one should accurately determine the above two parameters, prior to the design of pneumatic transport system. The literatures related to the above two parameters were studied and discussed, in order to calculate the pressure developed for the pneumatic conveying system, one has to calculate the pressure drop along the line of the pneumatic conveying system. The gas-solidfriction factor could be used to calculate the pressure drop of a solids-laden gas stream flowing in the pipe, considering the effect of Reynolds number, mass flux and the solid-gas loading ratio.

## 2. Description of Pipe Network

The analysis is carried out for 660 MW unit. The exact geometries of the individual systems in a power plant are not always available. This problem was overcome by breaking up the geometry of the system into various components like thevertical section, horizontal section and various bends. The pressure drop across each sections is calculated and then put together to give the pressure drop along the whole geometry of the system. The pressure drops for the horizontal and vertical lengths were calculated initially for a 60D pipe, where D is the diameter of the pipe. The pressure drops across unit length were then calculated and applied to the existing lengths of the pipe. A length of 60D was chosen to ensure that the flow became fully developed. For the current configuration, the flow became fully developed within 30D from the inlet. Therefore, for the bends, the upstream length was assumed to be 40D and downstream to be 20D and the pressure drop across the bend was calculated [1].

The four systems modelled using the numerical method are given in Table 1. As shown in the Table, there are 4 systems that carry coal/air mixture from the pulverizer to the furnace. From Table 1, it is clear that system 3 has longest length and more numbers of bends as compare to other. It means it has higher friction loss coefficients, which is to be maintained in other system for balancing.

**Table 1**: Detail of pipes layout that exit from one pulverizer (660 MW)

| *System* | *Total vertical length (m)* | *Total horizontal length (m)* | *Bends in system* |
|---|---|---|---|
| 1 | 17.83 | 17.05 | $90^0$, $120^0$, $130^0$ |
| 2 | 17.83 | 38.67 | $90^0$, $130^0$, $135^0$ |
| 3 | 17.83 | 77.17 | $90^0$, $140^0$, $90^0$, $155^0$, $135^0$ |
| 4 | 17.83 | 45.17 | $90^0$, $140^0$, $155^0$, $120^0$ |

Diameter = 656 mm

By carrying out general clean air flow test, orifice flow restrictor is fitted in the system. Air flow test is carried out by passing primary air from these system and velocity is measured at the outlet in furnace. The velocity at outlet is different for system due to the difference in length and numbers of bends. To maintain constant air velocity orifice is fitted in the system to make them constant. For these, longest pipe is taken as reference as it is having higher friction loss coefficients. So system 3 is not having orifice, by putting orifice in other system velocity is maintained. This orifice size can be decided empirically or experience. The size of orifice fitted as mentioned in Table 2. These orifices are fitted in system at 1.5 m from the pulverizer. This is for easy replacement of orifices.

**Table 2:** Size of Orifice

| System | Diameter of Orifice (mm) |
|---|---|
| 1 | 552 |
| 2 | 543 |
| 3 | - |
| 4 | 558 |

## 3. Computational Approach

Turbulence consists of fluctuations in the flow field in time and space. It is a complex process, mainly because it is three dimensional, unsteady and consists of many scales. It can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number. The k-ε model of turbulence is widely chosen for fluid flow analysis. 'k' is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. 'ε' is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate).

To simulate the turbulence parameters, a standard k-ε model has been chosen with isothermal heat transfer condition at 300 K. The Solver uses k-ε model with two new variables and the continuity equation is then.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0 \quad \text{……..(1)}$$

And the momentum equation becomes.

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) - \nabla \cdot (\mu_{eff} \nabla U) = \nabla p' + \nabla \cdot (\mu_{eff} \nabla U)^{T} + B \qquad \ldots\ldots(2)$$

The flow–solver CFX-14.5 used for the analysis uses the differential transport equation for the turbulence kinetic energy and turbulence dissipation for analysis. The equation for kinetic energy K is given by

$$\frac{\partial(\rho\kappa)}{\partial t} + \nabla \cdot (\rho U k) = \nabla \cdot \left[ \left( \mu + \frac{\mu_t}{\sigma k} \right) \nabla_k \right] + P_k - \rho\varepsilon \qquad \ldots..(3)$$

The equation for ε without compressibility is given by

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \nabla \bullet (\rho U \varepsilon) = \nabla \bullet \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{K} \left( C_{\varepsilon 1} P_K - C_{\varepsilon 2} \rho \varepsilon \right) \qquad \ldots.(4)$$

Where $\mu$ is molecular viscosity, $\mu_T$ is turbulent viscosity and $C_{\varepsilon 1}$ & $C_{\varepsilon 2}$ are constants with values 1.45 &1.9 respectively. $\sigma_k$ is turbulent model constant for kinetic energy which is 1 and $\sigma_\varepsilon$ is constant for k-ε model which is 1.3.These simulation is done on Intel i5 ($4^{th}$ generation) 2.5 GHz (turbo boost to 3.2) processor with 4 GB ram. Convergence was claimed when stable pressure drop was achieved, usually it takes 7000 iterations. To simulate the flow analysis in the pipe with mesh density of around 4 lac elements and around 80,000 computational nodes is done. The grid is made of unstructured mesh with a tetrahedral shape. From simulation results we get the pressured drop in each system and from these we can calculate pressure drop coefficients.

At the inlet of the pipe, INLET boundary condition is Mass flux of air – 111.012 kg/m$^2$, Mass flux of coal – 59.2 kg/m$^2$ , Temperature – 348 K, Coal particles size – 50 $\mu$m (diameter). At the exit of the pipeline the outlet boundary conditions was applied. The flow is assumed to be fully turbulent and the suspension is uniform having a density of 1.284 kg/m$^3$.

## 4. Results and Discussion

In this study, orifice sizes were calculated for coal/air balancing. Based on the given data, the calculations have been performed with the existing system configuration. Table 3 gives the pressure drop per unit length for the horizontal and vertical components and also the pressure drop across the bends for coal/air flows.

**Table 3**: Pressure drop across various components

| | Horiz | Vert | $90^0$ | $120^0$ | $130^0$ | $135^0$ | $140^0$ | $155^0$ |
|---|---|---|---|---|---|---|---|---|
| ΔP (Pa) | 4059 | 6948 | 9916 | 7447 | 6648 | 6303 | 6154 | 4818 |

Since all the four pipes convey coal from the same pulverizer to the furnace, the pressure losses in each of them have to be equal. Hence the flow velocity in the shortest pipe will be highest where as it will be lowest in the longest pipe. Thus, this leads to non-uniform coal feeding to various burners. In order to overcome this problem, restriction orifices are included in pipe layouts so as to equalize the flow resistances. The size and number of orifices in each pipe are decided on the basis of additional pressure loss required in that pipe. From these results, we are going to calculate the pressured drop coefficients of each system, as calculated for existing model mentioned above. And from these orifice diameter is calculated. Table 4 gives the details of calculation and orifice size.

**Table 4**: Orifice diameter Calculations and Comparison

| System | K | $K_{orifice}$ | Orifice diameter (mm) (by air test) | Orifice diameter (mm) |
|---|---|---|---|---|
| 1 | 2.527 | 1.166 | 552 | 453 |
| 2 | 2.912 | 0.781 | 543 | 504 |
| 3 | 3.693 | - | - | - |
| 4 | 3.295 | 0.398 | 558 | 575 |

The CFD simulations also provide detailed information of the two-phase flow field. Streamlines of the air velocity magnitude for bends with various angles. The air flows from the straight part of the pipe towards the curved part, it is followed by build-up in pressure (elbow heel) along the outer elbow wall (throat) air at the elbow heel gradually turns, while the air stream close to the throat tends to travel in a straight line. Besides, it can be observed that the velocity streamlines are practically parallel along the first straight pipe and are disrupted when the air stream encounters the elbow. The elbow curvature influence isn't restricted to its outlet but an airflow downstream along the second straight pipe resulting in completely distorted streamlines (caused by the secondary flow). Therefore, the 60D pipe length is sufficient to allow a fully developed flow at the elbow inlet but does not allow the airflow redistribution along the second straight pipe. The secondary flow kinetic energy could be totally dissipated by the viscous effects if a more extended straight pipe would be connected to the elbow outlet.

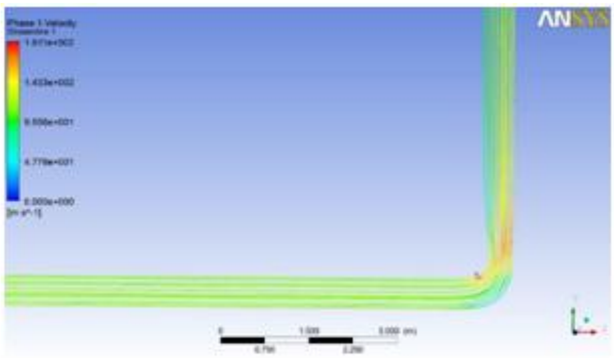

**Figure 1**: Streamlines of $90^0$ Bend

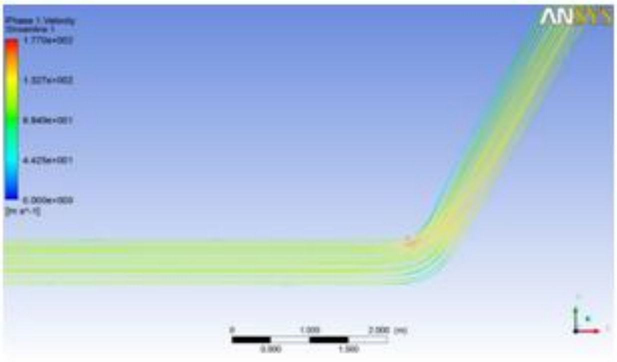

**Figure 2**: Streamlines of $120^0$ Bend

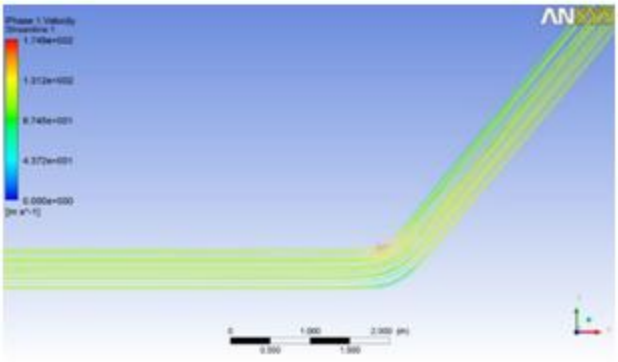

**Figure 3**: Streamlines of $130^0$ Bend

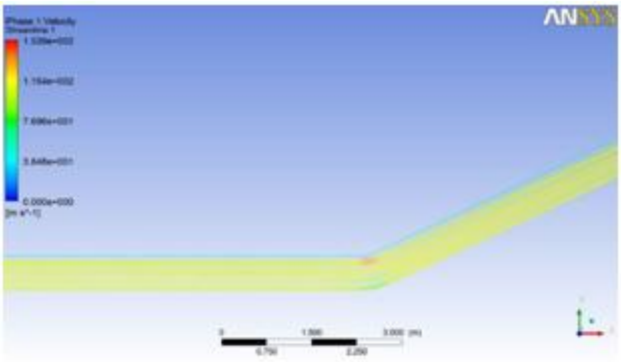

**Figure 4**: Streamlines of $155^0$ Bend

## 5. Conclusions

The mass flux is having considerably effect on the system, which improves the balancing results in increase in combustion efficiency. The current orifice size is calculated for full load. At partial load there will be different coal/air loading ratio and hence mass flux, therefore current configuration unable to maintain balancing. From this it is clear that there will be different orifice size for different load. This can be achieved by using variable or adjustable orifice flow restrictor which adjust the size as per load and balancing is maintained. Variable orifice having one more advantage, gradual erosion of orifice is done due to coal/air flow which create unbalances in the system, using variable orifice it can be adjusted.

Three dimensional CFD computations on coal pipes made have been able to capture the detailed functional features of two phase flow in the current configurations considered to be Fine. Turbulent model based on k-ϵ theory with a RANS code has been used for the CFD predictions of the mass fraction and the coal/air loading ratio has been evaluated leading to bringing out of an optimal design of the orifice for balancing.

Overall the results of present CFD study proves CFD can be used for balancing fuel in the pipelines and the geometry of orifice can be optimized at various stage of operation. At present if adjustable orifice is implanted in the system, we can balance the system as per load. But for this we have to calculate or to know the orifice dimension at every load to balance coal/air flow. By using flow measuring instrument, to measure the coal/air flow at the outlet of burner at four corners we can balance the system as per reading show by flow measuring instrument.

The present system can be modified by changing orifice flow restrictor diameter as per above study which can improve boiler efficiency up to some extent. This will help in to reduce pollution up to certain level and uniform heating can be made in furnace as there will be uniform flow at the coal outlet.

## References

[1] SowjanyaVijiapurapu, Jie Cui, SastryMunukutla, CFD application for coal/air balancing in power plants, Applied Mathematical Modelling 30 (2006), pp. 854-866.

[2] R. Vijayakumar, S.N. Singh, V. Seshadri, CFD Analysis for the Design of Optimum Geometry of the Orifices in the PC pipes for ensuring uniform feed rate of Coal through various Coal Burners in a Boiler.

[3] Chan Lee, Jin Wook Lee, Gyoo Tae Kim and Tae Wan Kwon, Numerical Simulations and Correlations on the coal-conveying gas flow in pipe for fluidized-bed coal gasification facility, Ecology and Safety, Volume 8, ISSN 1314-7234.

[4] V. Singh and Simon Lo, Predicting Pressure Drop In Pneumatic Conveying Using The Discrete Element Modelling Approach, Seventh International Conference on CFD in the Minerals and Process Industries, 2009.

[5] Basil R. Paynter and B.P. Huynh, Distribution of Pulverized Coal Flow in a Power-Station Pipe-Network, 18th Australasian Fluid Mechanics Conference, 2012.

[6] Dr. V. T. Sathyanathan, 'Combustion Tuning in High Ash Pulverized Coal Fired Boilers having Direct Tangential Firing System', http://www.brighthubengineering.com/power-plants/37755-combustion-tuning-in-high-ash-pulverized-coal-fired-boilers, Aug 24, 2014.

[7] Ana Paula P. dos Santos, Claudia R. Andrade, Edson L. Zaparoli, 'CFD Prediction of the Round Elbow Fitting Loss Coefficient', International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering, Vol:8 No:4, 2014.

[8] PandabaPatro and BrundabanPatro, 'Kinetic Theory Based CFD Modelling of Particulate Flows in Horizontal Pipes', International Journal of Mathematical, Computational, Physical and Quantum Engineering, Vol: 7 No: 3, 2013.

[9] Mallick, S.S., 'Modelling of Fluidised Dense-Phase Pneumatic Conveying of Powders university of Wollongong', PhD Dissertation, 2010.

## Author Profile

**Sunith Bawankule**, pursuing M.Tech. degree in Thermal Engineering from Govt. College of Engineering Amravati (2013-2015). He did graduation in B.E. in Power Engineering from National Power Training Institute, Nagpur during the period of 2008-12.