International Journal of Research in Advent Technology, Vol.3, No.6, June 2015
E-ISSN: 2321-9637

CFD Application for Coal/Air Flow Analysis in Power Plants: A Review

Sunith Bawankule[1], Dr.RavindraBabuYarasu [2]
[1]*Thermal Engg.,M.Tech. Student*
[2]*Department of Mechanical Engg, Associate Professor*
Email:sunithbawankule@gmail.com[1]

Abstract-Commercially available CFD package is used to simulate the complex flow in transport system. Non-uniform feed rate of coal/air flow in the pipe system feeding the boilers results in non-uniform combustion in furnace, hence an overall lower efficiency of the boiler. The common method to maintain the uniform feed rate is to put orifice flow restrictor in the piping system. 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. This paper represent the research work done by various authors who have done research for deciding the optimum geometry of orifice flow restrictor to maintain uniform feed rate.

Index Terms-CFD, Coal/Air balancing, Mass flux, Multiphase flow, Orifice flow restrictor, Pneumatic coal transport, Power plant

1. INTRODUCTION

R & D of all country efforts in energy industry are devoted to clean coal technologies with higher energy utilization efficiency and lower greenhouse effects. The boilers in the Thermal Power Plants usually consist of large furnaces where pulverized coal is burnt to generate the heat. This heat of combustion is transferred to the circulating water pipelines located in the walls of the boiler thereby producing steam which is subsequently used in the turbines. For the efficient operation of the boilers, it is important that the flame is located centrally in the chamber so that heat distribution is uniform. In order to ensure efficient burning of coal, the pulverized coal is fed pneumatically through a number of coal burners placed at different locations around the periphery of the boiler. It is very essential to ensure that the coal feed rate through all the burners is same so that the flame is located centrally in the chamber. One of the main source of non-uniform combustion is uneven distribution of fuel supply to the various burners of furnace. Number of coal pipes connect the exit of the pulverizer to the individual burners, which are located at the same elevation but horizontal, vertical, and inclined lengths, and bends are different for each system in the furnace for that particular pulverizer. Thus, the resistance to flow is different for each system, causing uneven distribution. 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) emissionsandhigher fly ash unburnt carbon. For this reason, balancing coal flow is a primary objective of combustion optimization. Hence, in order to ensure equal flow rate in different pipelines, it is essential to make the flow resistance equal in all the pipelines. In order to achieve this objective, orifice flow restrictor of various sizes are introduced in the

pipelines so that pressure drops become equal in all the pipes. Presently, the size and geometry of these orifice flow restrictor are decided either empirically or by experience.

The authors have done numerical analyses by using CFD method 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 piping geometry and changing coal/gas loading condition, the results give design criterion and guidelines for reliable and efficient coal-transport system.

2. MODELLING OF COAL/AIR FLOW USING CFD

In power plants, many processes involve multi-phase flow, combustion, phase transformation and complex reactions. Due to the complex nature of these phenomena, it becomes difficult to understand these processes. Therefore, CFD has been used in power industry to gain a qualitative as well as quantitative understanding of these processes. It was shown that the model predictions were in good agreement with the experiment results. The results showed that CFD modelling was well established as a design tool and has been widely used in power generation industry to help engineers reduce emissions, select fuel, increase thermal efficiency and extend pant lifetime, etc. [1]. In this paper, the problem of balancing the coal/air flow i.e. uniform feed rate at burner point is studied using CFD.

Fig. 1 shows typical coal transport system where coal is conveyed pneumatically to burners from pulverizer through coal pipes composed of horizontal,

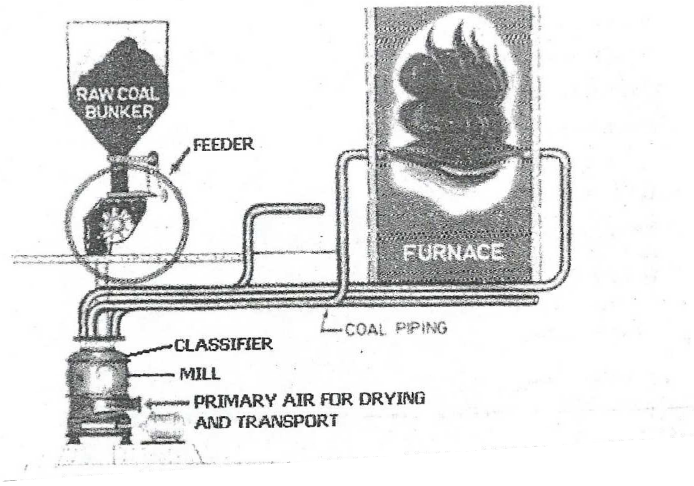

Fig. 1. Coal transport system

vertical and bend pipes. The gas flow and particle track behaviors inside the three key flow elements mainlyaffect the pressure loss, the coal feeding capability and the reliability of coal transport system [3]. Multiphase model is employed to develop a computational model of coal/air flow.

The present study demonstrate the successful use of Computational Fluid Dynamics (CFD) methodology 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.

Almost all the researchers used commercially available CFD package FLUENT or CFX for analyzing the flow of coal/air mixture through complex pipeline network involving orifice flow restrictors. The flow has been assumed to be incompressible, steady, and turbulent. For calculation of the steady state flow in the piping systems, continuity and momentum equations were solved along with the standard k-ϵ turbulence model. The continuity equation, momentum equation, and the turbulence model equation were solved for each phase. Two phase flow calculations were adopted to simulate the air flow and coal particles. Chan Lee *et al.* [3] has considered the effect of turbulence on coal particle behavior by using random walk model. The fluid flow is modelled using Reynolds Averaged Navier-Stokes equations (RANS) solver. R. Vijaykumar*et al.* [2] and Chan Lee *et al.* [3] has done the analysis which are based upon RANS equation solvers coupled with standard k-ϵ turbulence and discrete phase models whereas SowjanyaVijiapurapu*et al.* [1] done with finite volume method on an unstructured mesh with the standard k-ϵ model for turbulence. The boundary conditions were assumed to be uniform velocitydistribution at the inlet and zero gauge pressure at the outlet. The boundary condition for the coal particles was no slip i.e. the particles stick to the surface if they hit the surface. The volume fraction for coal is uniformly distributed at inlet. The coal particles are of uniform size and spherical.

3. CFD ANALYSIS OF COAL/AIR FLOW FOR BALANCING

The current practice of balancing the flow is to put orifice flow restrictors in the coal transport line. These orifice flow restrictors increase the flow resistances of the system. Because of unequal length and different numbers of bend in system, it is necessary to make the flow resistance equal to ensure equal flow rate [2]. The pressure loss equations for horizontal and vertical pipes is given by

$$\Delta p = f \frac{L}{D} \frac{1}{2} \rho V^2 \quad (1)$$

and for elbow pipe is given by

$$\Delta p = k \frac{L}{D} \rho V^2 \quad (2)$$

whereV, L,D and ρ represent gas velocity, pipe length, diameter and gas density respectively, f and k are friction factor and pressure loss coefficient [3].

Many authors co-relate this orifice flow restrictor diameter with the pressure loss in the system and used empirical and CFD studies to determine pressure drop. SowjanyaVijiapurapu*et al.* [1] calculate the orifice diameter from available empirical relation, the commonly used equation is

$$K_{OR} = K_2 - K_1 + \left[\left(\frac{F1}{F0} - 1\right) + \left(0.707 * \frac{FO}{F1}\left(1 - FOF10.3752 , \right.\right.\right. \quad (3)$$

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. This shows that the pressure drop in the system strongly depends on the system geometry.

The exact geometries of the individual system are not always available. This problem is overcome by breaking up the geometry of the system into various component like the horizontal sections, vertical sections and various bends. The pressure across each component is calculated and then put together to give pressure drop along he whole geometry of the system [1].

The alternate method used by R. Vijaykumar*et al.* developed a complete model system in GAMBIT software and repeatedly done analysis on FLUENT with different orifice flow restrictor diameter to equalize the flow [2].

Chan Lee, Jin Wook Lee, Gyoo Tae Kim and Tae Wan Kwon [3] studied 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. With changing coal/gas loading condition, the present prediction results give design criterion and guidelines for reliable and efficient coal-transport system. With changing gas superficial velocity (or changing coal-gas loading ratio) related pressure loss are examined from the CFD results and it is concluded that coal-gas loading ratio should be maintained less than 5 in all the flow elements for continuous and steady pneumatic coal-transport system.

V. Singh and Simon Lo [4] used the commercial CFD software to model pneumatic conveying in a horizontal pipe. The pressure drop increases with increasing solid loading as well as increasing fluid velocity. The pressure drop in the system is dependent on a host of parameters such as particle and pipe diameters, pipe roughness and orientation, particle and fluid properties, pipe orientation and roughness, etc.

The factors which are affecting the flow of pulverized coal carried by air are studied by Basil R. Paynter and B. P. Huynh (2008) [5]. For proper distribution better mixing is required which is related to Reynolds number. As the pulverized flow rate increases, thereby increase in density which effectively increase the Reynolds number with small change in flow velocity. Distribution becomes more even as coal flow increases toward operational (full) capacity, while air flow increases only slightly.

4. CORRELATIONS DEVELOPED FOR COAL/AIR FLOW SYSTEM

Chan Lee *et al.* modified the equation (1) and (2) for coal/air flow system. Based on the computation results solid friction factor(fs) and pressure loss coefficient (Ks) can be deduced under the assumption that overall pressure loss is composed of the gas and the solid ones as follows [3]:

$\Delta p = \Delta p_g + \Delta p_s = \Sigma\,(f_g + f_s)\frac{L}{D}\frac{1}{2}\,\rho_g V^2$(4)

$\Delta p = \Delta p_g + \Delta p_s = \Sigma\,(K_g + K_s)\frac{1}{2}\,\rho_g V^2$(5)

The correlations for solid friction factor and pressure loss coefficients are summarized in Table 1.

5. CONCLUSION

Currently, there are no easy method available for measuring coal/air flow in power plants. In order to balance the coal/air flow i.e. to maintain uniform

Table 1: Correlations for solid friction factor and pressure loss coefficient

Type	Correlation
Horizontal pipe	$f_s = 2.00 \times 10^{5}\ \mu^{1.0} Fr^{-1.34}\left(\frac{\rho s}{\rho g}\right)^{-1.8}\left(\frac{G}{61.6}\right)^{1.5}$
Vertical pipe (upflow)	$f_s = 2.86 \times 10^{-4}\ \mu^{1.0} Fr^{-0.026}\left(\frac{\rho s}{\rho g}\right)^{0.73}\left(\frac{G}{61.6}\right)^{0.16}$
Vertical pipe (downflow)	$f_s = 2.13 \times 10^{-4}\ \mu^{1.0} Fr^{-0.026}\left(\frac{\rho s}{\rho g}\right)^{0.73}\left(\frac{G}{61.6}\right)^{0.16}$
Elbow pipe (upflow)	$K_s = 6.58 \times 10^{11}\ \mu^{1.0} Fr^{-1.31}\left(\frac{\rho s}{\rho g}\right)^{-3.7}\left(\frac{\pi R}{2D}\right)\left(\frac{G}{61.6}\right)^{1.2}$
Elbow pipe (downflow)	$K_s = 4.0 \times 10^{11}\ \mu^{1.0} Fr^{-1.31}\left(\frac{\rho s}{\rho g}\right)^{-3.7}\left(\frac{\pi R}{2D}\right)\left(\frac{G}{61.6}\right)^{1.2}$

whereFr is Froude number, G is mass flux and ρ_s, ρ_g are coal particle and gas densities.

feed rate at the individual burner orifice flow restrictor is used. Orifice flow restrictors sized are based upon calculated pressure drop. The above research studies show that the pressure drop is strongly depends on system geometry, coal gas loading ratio, mass flux. The optimum geometry can be calculated by considering these factors so uniform feed rate is maintained.

Further CFD can also be used to design an adjustable orifice flow restrictor which will maintain a uniform feed rate in spite of gradual abrasion. By fitting flow control element at the exit point so that by measuring flow rate, orifice diameter can be adjusted and balancing is done, but this flow control element should take the effect of both air and coal elements.

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 the system. By using flow 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.

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] 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.

[7] 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.