Modelling and analysis of coal /air balancing by using CFD in 210 MW thermal power plants

R.Dhamodharan*, E. Natarajan, D.Margabandhu

Mechanical Engineering Department, Institutes for Energy Studies, Anna University Chennai -600025, Tamil Nadu, 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 an overall lower efficiency of the boiler. A common solution to this problem is to put orifices in the pipe systems to balance the flow. It is well known that if the orifices are sized to balance clean air flow to individual burners connected to a pulverizer, the coal/air flow would still be unbalanced and vice versa.

It is now proposed to size the orifices for balancing the coal/air flow and then calculate the unbalanced clean air flow distribution to be known as the ‘‘tailored clean air flow’’. Commercially available ANSYS Work Bench was used for modeling and Computational Fluid Dynamics (CFD) code CFX version 10 was used to simulate the complex flows in the piping systems in a power plant. The two-phase modelling technique was employed to estimate the pressure drop coefficients with both clean air and coal/air flows in order to size the orifices. The results indicate that the pressure drop is strongly dependent on the piping system geometry. With this proposed method, field tests can be conducted to correspond with the tailored clean air flow, and the coal/air flow balancing would be achieved.

Keywords: CFD, Numerical modelling, Coal/Air balancing, Power plant

*****E-mail address*: dhaama10@yahoo.co.in

1. Introduction

Coal –fired boilers use air as the transport medium to convey pulverized coal to the furnace. Modeling the flow of coal/air and calculating its pressure loss is a problematic task. Different mass flow ratio is arise non-uniform combustion in the furnace. Several coal pipes connect the exit of the pulverizer to the individual burner, all of which are located at the same elevation in the furnace for that particular pulverizer (or) mill plant. Pipes have different horizontal, vertical and inclined. So the flow is different for each system.

In the current paper is discussed analytical method of predicting the pressure drop in a system and CFD models to predicting the pressure drop in coal pipes. Both air and coal/air pressure drop calculated separately

Using CFD in power industry is recently make new approach for problem solving and also to gain a qualitive as well as quantitative understanding of the power industry processes.

Multiphase model in the unstructured version of CFX (5) was applied to determine pressure drop pressure drop in the piping system in a power plant

Breaking the components in to individual components calculated pressure drops across various components were then added together, to estimate the pressure drop of the entire system. Orifices in the piping system will be added (or) modified to balance coal/air flow will be achieved.

2. Analytical study

The usual assumption of pressure drop determination in air-coal two-phase flow has been to consider the total pressure drop comprising two components: pressure drop due to the flowing air alone (Δp_f) and the additional pressure drop (Δp_c) due to the presence of coal particles Eq-1 (Bradley, 1990; Bradley & Reed, 1990).

$$\Delta p = \Delta p_f + \Delta p_c \qquad \text{Eq-1}$$

The procedure involved in determination of the air-only pressure drop is quite Straightforward and has usually been represented by the well-known Darcy’s Equation (Massey, 1980):

$$\Delta p = \frac{f \cdot L \cdot \rho_f \cdot V^2}{2 \cdot D} \qquad \text{Eq-2}$$

Similar to the above equation, quite often the pressure drop in pneumatic conveying System has been given by (Weber, 1981)

$$\Delta p = (\lambda_f + \mu\lambda_c)\frac{f \cdot L \cdot \rho_f \cdot V^2}{2 \cdot D} \qquad \text{Eq- 3}$$

3. Principle of the method

Fig .1 shows two systems with the same elevation change, but different horizontal, vertical runs, and bends. Both systems start at the same elevation at the pulverizer exit and discharge at the same elevation in the boiler. System 1 has one vertical run, one horizontal run, and one bend. System 2 has 3 vertical runs, 3 horizontal runs, and 5 bends. The total pressure drop consists of frictional loss over the horizontal and vertical pipe lengths, a loss due to bends, and a loss due to gravity. In general if the total flow rate is Q, with flow rate in system 1 being Q_1 and that in system 2 being Q_2, and the pressure drop coefficients being K_1 and K_2, respectively, we have

$$Q = Q_1 + Q_2$$

$$K_1 Q_1^2 = K_2 Q_2^2$$

$$Q_2 = \frac{Q}{1+\sqrt{\frac{K_2}{K_1}}}$$

$$Q_1 = \frac{Q}{1+\sqrt{\frac{K_1}{K_2}}}$$

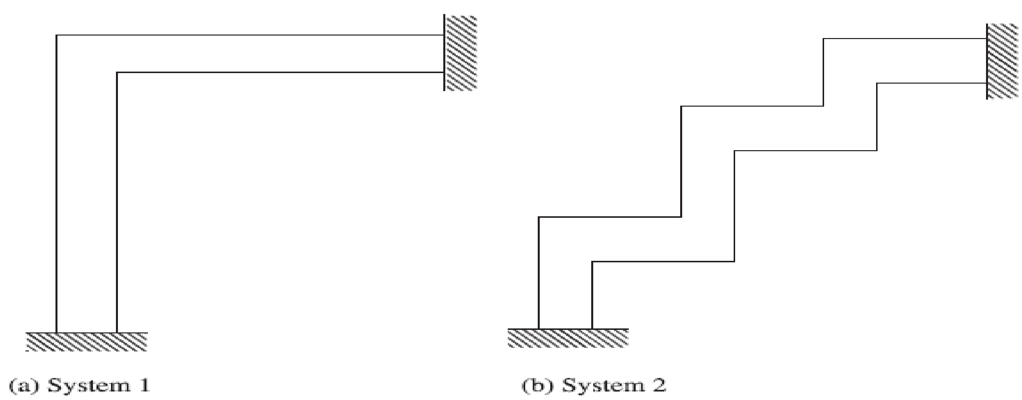


Fig no: 1 shows two systems with the same elevation change, but different horizontal, vertical runs, and bends

Let the pressure drop coefficients for clean air and coal/air for systems 1 and 2 be K_{1A}, K_{2A}, and K_{1C}, K_{2C} respectively. Then the flow distribution with clean air is given as

In most cases, it has been observed that the ratio of K_{2C}/K_{1C} is very nearly equal to K_{2A}/K_{1A}. In other words, clean air and coal/air are unbalanced in a similar manner. The pressure drop coefficient K_{OR} of the orifice plate is calculated as the absolute difference in the pressure drop coefficients of the pipes. In order to achieve clean air balance, an orifice yielding a pressure drop coefficient of $K_{Orf,\,A} = K_{2A} - K_{1A}$ should be introduced. In order to achieve coal/air balance, an orifice yielding a pressure drop coefficient of $K_{Orf,\,C} = K_{2C} - K_{1C}$ should be introduced. As a general rule the pressure drop coefficients for the coal/air (two-phase) flow K_{1C} and K_{2C}, are always greater than the corresponding clean air (single-phase) pressure drop coefficients K_{1A} and K_{2A} Hence, the orifice diameters based on the clean air distribution are larger than those based on the coal/air distributions. As a result of the above, it has been a normal trend in the power industry to under estimate the pressure drop requirement. The diameter of the orifice can be calculated from available empirical Equations .4, one of the commonly used equations is

$$K_{OR} = \left[\left(\frac{F_1}{F_0} - 1 \right) + \left(0.707 * \frac{F_0}{F_1} \left(1 - \frac{F_0}{F_1} \right)^{0.375} \right) \right]^2 \qquad \text{Eq-4}$$

Where $\frac{F_0}{F1} = \frac{A_0}{A_1} = \left(\frac{D_0}{D_1}\right)^2$

A_0, A_1 correspond to the area of the orifice and pipe, respectively, and, D_0, D_1 correspond to the diameters of the orifice and pipe, respectively.

4. System modelled

The four systems modelled using the numerical method is given in Table 1. As shown in the Table 1, there are 4 systems that carry coal/air mixture from the pulverizer to the furnace. The individual lengths of the horizontal and vertical sections between the bends are not known. The only data available is the total horizontal and vertical length of the systems, and the number of different bends in each system. Also, the bend angles for each system are known. If K_1, K_2, K_3, and K_4 are the pressure drop coefficients for systems 1, 2, 3, and 4, respectively, flow distributions V_1, V_2, V_3, and V_4 can be calculated by extending Eq. (3) to a 4-branch system.

$$V_1(\%) = \frac{400}{1 + \sqrt{\frac{K_1}{K_2}} + \sqrt{\frac{K_1}{K_3}} + \sqrt{\frac{K_1}{K_4}}}$$

$$V_2(\%) = V_1 \sqrt{\frac{K_1}{K_2}}$$

$$V_3(\%) = V_1 \sqrt{\frac{K_1}{K_3}}$$

$$V_4(\%) = V_1 \sqrt{\frac{K_1}{K_4}}$$

Note that for a perfectly balanced system $K_1 = K_2 = K_3 = K_4$, and in that case $V_1 = V_2 = V_3 = V_4 = 100\%$. For an unbalanced system V_1, V_2, V_3, and V_4 could be different from 100%, but however, would still add to 400%.

Table no: 1 Details of four systems that exit from one mill plant and feed the boiler furnace at the same elevation

System	Total vertical length (m)	Total horizontal length (m)	Bends in system	Diameters of existing orifices in system (mm)
1	15	30	45°,15°,15°, 90°,90°	406 at 1 m distance 406 at 32 m distance
2	15	68	60°,90°,90°	396 at 1 m distance 396 at 45 m distance
3	15	52	15°,90°,90°,90°	410 at 1 m distance 410 at 64 m distance
4	15	70	90°,90°,15°,15°, 90°,90°	No orifices

Table no: 2 CFD Existing flow distribution for clean air and coal/air flows

SYSTEM	CLEAN AIR FLOW DISTRIBUTION %	COAL/AIR FLOW DISTRIBUTION %
1	93.45	96.61
2	117.74	100.19
3	103.35	111.00
4	85.40	92.16

It can also be observed that the two unbalances are not similar due to the presence of orifices. Hence, it is proposed to retrofit the whole orificing system based on the coal/air balancing. Since the two-phase flow in the existing systems is quite complex and the data from standard handbook is unreliable, it was decided to utilize the two-phase modelling technique to obtain the pressure data in each system. Then the orifice diameters are recalculated based on the CFD results and a well-balanced system is obtained.

Table no: 3 CFD Pressure drop across various components for clean air and coal/air flows

PHASE	HORIZONTAL Pa/m	VERTICAL Pa/m	BEND 90 Pa	BEND 60 Pa	BEND 45 Pa	BEND 15 Pa
AIR	763	173	2905	2504	2971	2742
COAL/ AIR	879	175	3545	9182	5353	2786

5. CFD application

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 the horizontal section, vertical section, and various bends. The pressure drop across each component is calculated and then put together to give the pressure drop along the whole geometry of the system. The geometries used for the modelling are shown in Fig. 2, the pressure drops for the horizontal and vertical lengths were calculated initially for a 70D 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 70D was chosen to ensure that the flow became fully developed. The pressure drop per unit length was calculated for the fully developed region. It was found that for the current configuration and flow condition (Reynolds number 2.3×10^6, surface roughness e = 0.045 mm) the flow became fully developed within 40D from the inlet.

Figure no shows the CFX work bench version 10 is used to create the geometry of the system into various components like horizontal section vertical section and various bends (90º, 60º, 45º, 15º) with pipe diameter of 0.480m.

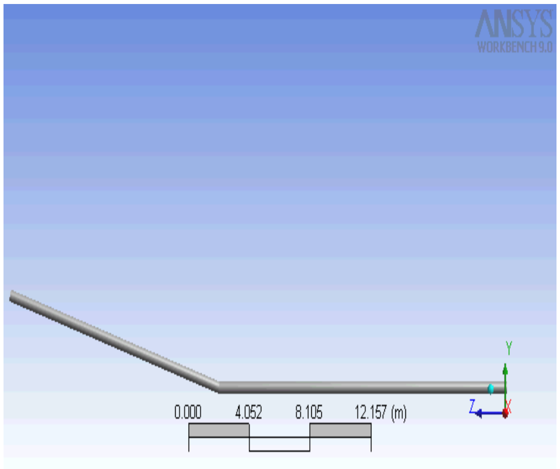
`Therefore, for the bends, the upstream length was assumed to be 40D and downstream to be 30D and the pressure drop across the bend was calculated. For calculation of the steady state flow in the piping systems, continuity and momentum equations were solved along with the standard k–ε turbulence model. Two-phase flow calculations were adopted to simulate the air flow and coal particles. The equation of continuity for a mixed fluid is expressed by Eq. (5) ,where α is the phase, r_α is the volume fraction of that phase, ρ_α is the density of the fluid in the phase α , x^j is the coordinate with the index j ranging from 1 to 3, and $U_\alpha^{\ j}$a is the mean velocity in the phase a along the direction j. The continuity equation expressed for each control volume is shown in Eq. (6)

$$\frac{\partial(r_\alpha \rho_\alpha)}{\partial t} + \frac{\partial(r_\alpha \rho_\alpha U_\alpha^j)}{\partial x^j} = 0 \qquad \text{Eq-5}$$

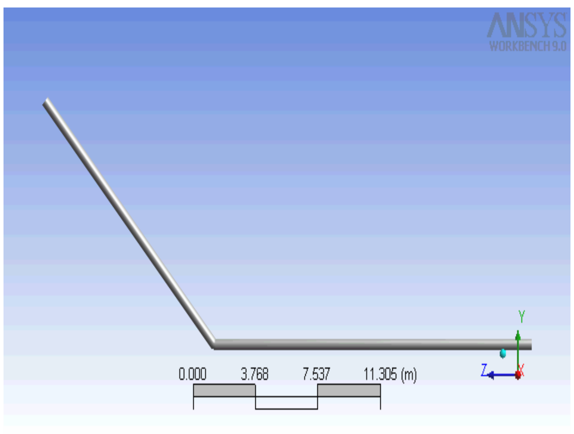
$$\sum_\alpha r_\alpha = 1 \quad \text{Or} \quad \sum_\alpha \frac{1}{\rho_\alpha}\left(\frac{\partial(r_\alpha \rho_\alpha)}{\partial t} + \frac{\partial(r_\alpha \rho_\alpha U_\alpha^j)}{\partial x^j}\right) = 0 \qquad \text{Eq-6}$$

Where repeated indices imply summation from 1 to 3. The equation of motion is expressed by Eq. 7

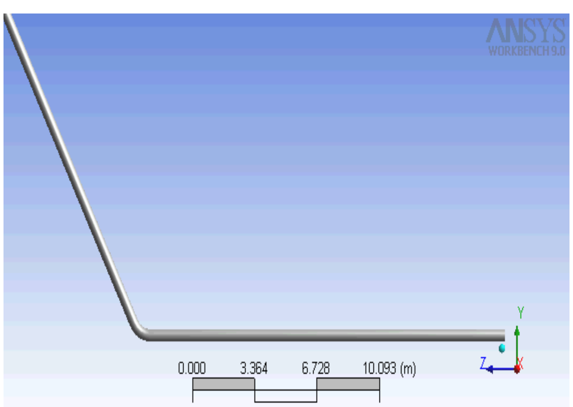
$$\frac{\partial(r_\alpha \rho_\alpha U_\alpha^i)}{\partial t} + \frac{\partial(r_\alpha \rho_\alpha U_\alpha^i U_\alpha^j)}{\partial x^j} = -r_\alpha \frac{\partial p}{\partial x^i} + r_\alpha \rho_\alpha g^i + \frac{\partial(r_\alpha \tau_{ji})}{\partial x^j} \qquad \text{Eq-7}$$



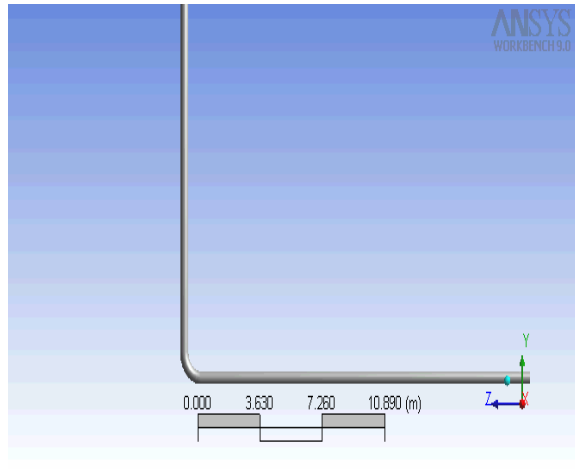
(a) 15°



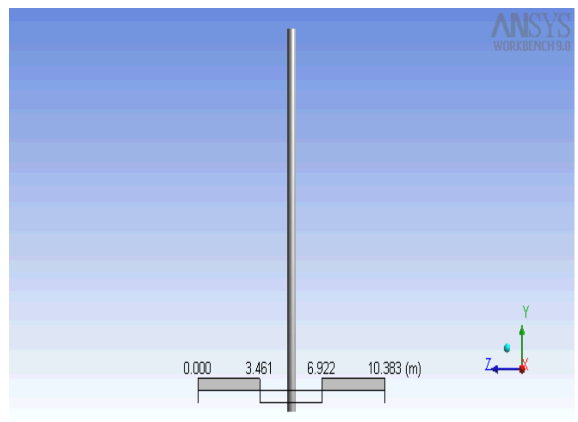
(b) 45°



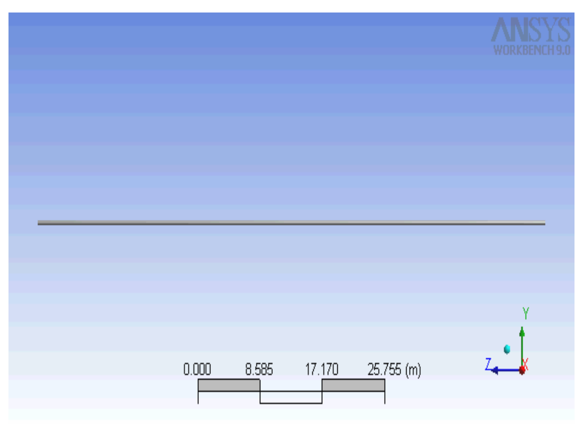
(c) 60°



(d) 90°



(e) Vertical



(f) Horizontal

Fig no: 2 show different bends model and horizontal, vertical

Where p is the pressure and τ_{ij} is the stress tensor given by

$$\tau_{ij} = -\mu_{eff} \frac{\partial U^i}{\partial x^j} + \frac{\partial U^j}{\partial x^i} \qquad \text{Eq-8}$$

Where μ_{eff} is effective viscosity, which is defined as the sum of dynamic viscosity μ and eddy viscosity μ_t , $\mu_{eff} = \mu_t + \mu$ .The eddy viscosity is provided by the $k-\varepsilon$ turbulence model.

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \qquad \text{Eq-9}$$

Where C_μ is a constant and is equal to 0.09, k is the turbulent kinetic energy, ε is the dissipation rate, and both are provided by the $k-\varepsilon$ turbulence model.

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha k_\alpha) + \frac{\partial}{\partial x^j}\left\{ r_\alpha \left[\left(\rho_\alpha U^j_\alpha k_\alpha \right) - \left(\mu + \frac{\mu + \alpha}{\sigma_k} \right) \frac{\partial k_\alpha}{\partial x^j} \right] \right\} = r_\alpha \left(P_\alpha - \rho_\alpha \varepsilon_\alpha \right) + T^k_{\alpha\beta} \qquad \text{Eq-10}$$

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha \varepsilon_\alpha) + \frac{\partial}{\partial x^j}\left\{ r_\alpha \left[\left(\rho_\alpha U^j_\alpha k_\alpha \right) - \left(\mu + \frac{\mu + \alpha}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon_\alpha}{\partial x^j} \right] \right\} = r_\alpha \frac{\varepsilon_\alpha}{k_\alpha} \left(C_{\varepsilon 1} P_\alpha - C_{\varepsilon 2} \rho_\alpha \varepsilon_\alpha \right) + T^\varepsilon_{\alpha\beta}$$

..Eq-11

Where σ_k = 1.0, σ_ε = 1.3, $C_{\varepsilon 1}$= 1.44, and $C_{\varepsilon 2}$= 1.92 are constants from the $k-\varepsilon$ model. $T^k{}_{\alpha\beta}$ and $T^k{}_{\alpha\beta}$ are the terms that represent the inter-phase transfer for k and ε respectively. P represents the shear production due to turbulence for that particular phase, which is

$$P = \mu_t \frac{\partial U^j}{\partial x^k} \left(\frac{\partial U^j}{\partial x^k} + \frac{\partial U^k}{\partial x^j} \right)$$

The continuity equation, momentum equation, and the turbulence model equation were solved for each phase. The pipe diameter was 0.480 m. For clean air flow, the air density was taken as 1.284 kg/m3, which is the value at room temperature. For coal/air flow, the air density was 0.977 kg/m3, which is the value at 84°C, and the coal density, was 1290 kg/m^3. The coal flow rate was 10.27kg/s and the air flow rate was 17 kg/s.

The boundary conditions were assumed to be uniform velocity distribution at the inlet and zero gage pressure (open to atmosphere) at the outlet. The boundary conditions for the coal/air flow were imposed in the same manner as those for clean air flow. The boundary condition for the particles was no slip i.e., the particles stick to the surface if they hit the surface.

The volume fraction for coal at inlet was considered to be uniformly distributed. The coal particles were all considered to be spherical and of the same uniform size (diameter 1 lm). Table 3 gives the pressure drop per unit length for the horizontal and vertical Components and also the pressure drop across the bends for both clean air and coal/air flows.

As can be seen, the pressure drop for the coal/air flow is always greater than that of the clean air flow. It can be said that this is mainly due to the resistance offered by the interaction between air and coal particles and among coal particles themselves, as well as blockage effects.

Reynolds Averaged Navier–Stokes equations (RANS) were solved using finite volume method on an unstructured mesh with the standard k–e model for turbulence.

Table no: 4 CFD Orifice diameter calculation for coal/air balancing

System	K Loss coefficient	Existing orifice K_{OR}	Total K	New K_{OR}	Orifice diameter (mm)
1	66.04	1.014	67.054	6.506	261
2	61.04	1.224	62.27	11.29	234
3	50.31	0.471	50.781	22.78	201
4	73.56	0	73.56	0	0

6. Results and discussion

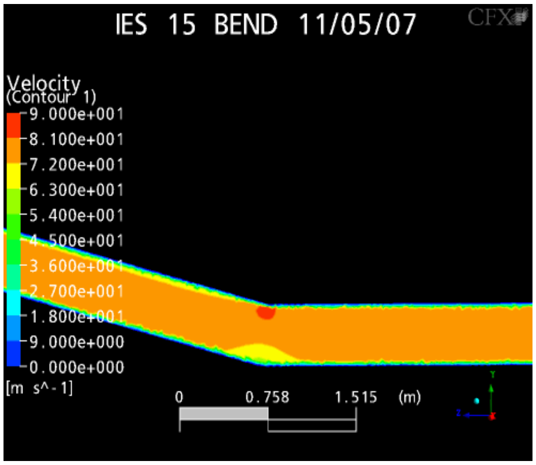
Orifice sizes were calculated for coal/air balancing. Based on the given data, the Calculations have been performed with the existing system configuration. The clean air and coal/ air pressure drop coefficients, and flow distributions were first calculated. Orifice sizes for coal/air balancing were then calculated. Note that in each system if one or more orifices already exist, then there was at least one orifice at 1m distance from the pulverizer exit. It was then tacitly assumed that the new orifice would replace that orifice. Table 1gives the data from the power plant. The existing flow distributions of CFD are given in Table 2 which are based on the pressure drop calculations CFD shown in Table 3 The orifice diameters for balancing coal/air CFD shown in Table.4 The balanced coal/air flow distribution after insertion of new orifices CFD is shown in Table 5 As can be seen from Table 5, The pressure drop for each system is the same and consequently the coal/air flow is balanced. In each instance, an existing orifice at 1 m distance is to be replaced by the new orifice.

Table no: 5 CFD Balanced coal/air flow distribution after insertion of new orifices

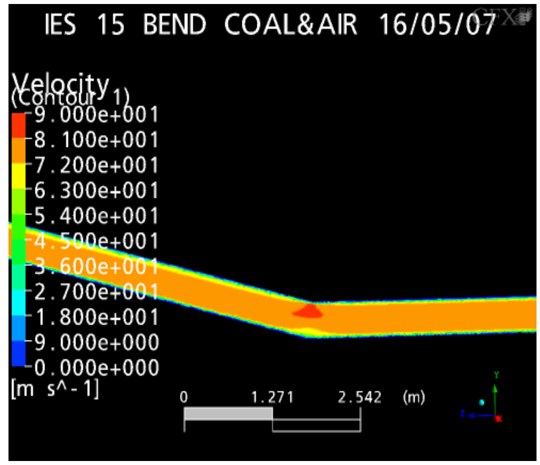
SYSTEM	COAL/AIR FLOW DISTRIBUTION %
1	99.69
2	100.72
3	99.81
4	99.87

The CFD simulations also provide detailed information of the two-phase flow field. Fig. 3&4 shows the close-up view of the air velocity magnitude contours for bends with various angles.
By using CFX-post see the result of air and coal / air Output velocity of air and output velocity coal/air shown in figure for different bends horizontal and vertical section.
The gravity force is vertically downward (in the negative y-direction). All contours shown in Figs. 3 & 4are on the center plane (zx-plane) that passes the axis of the bend. In the horizontal portion of the bend, all flow exhibit similar velocity distributions and there is no variation in the stream wise direction. This is because the flow has gone through 40D upstream length before it reaches the bend, and flow has become fully developed. It is observed that the velocity is lower near the bottom of the horizontal portion
This is due to the higher resistance to the air induced by the coal particles when they deposit to the pipe bottom.

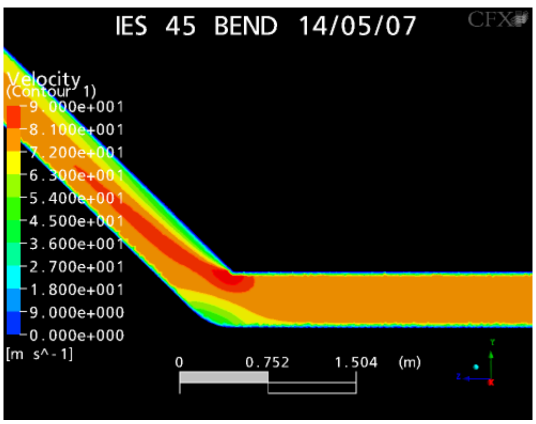
For the flow conditions considered in the current model, the effects of coal particle deposition are significant. It can be clearly seen from Fig. 3 at left hand side column that the characteristics of the air flow downstream of the bend strongly depend on the bend angle
For sharp turns (Fig.No.3&4) from angle 90°,60°,45°), the flow has to change direction quickly and the higher velocity flow is shifted towards the outer radius of the downstream bend. While near the inner radius of the bend, a lower velocity region can be identified.



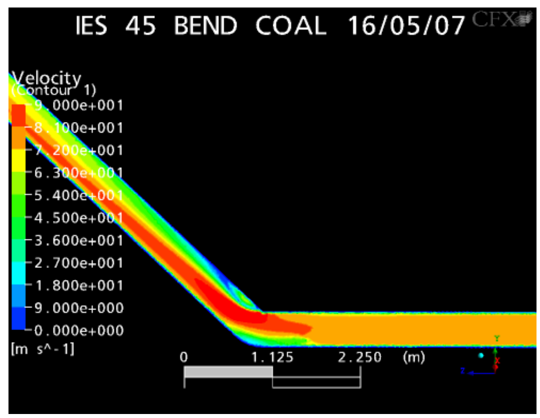
15° air



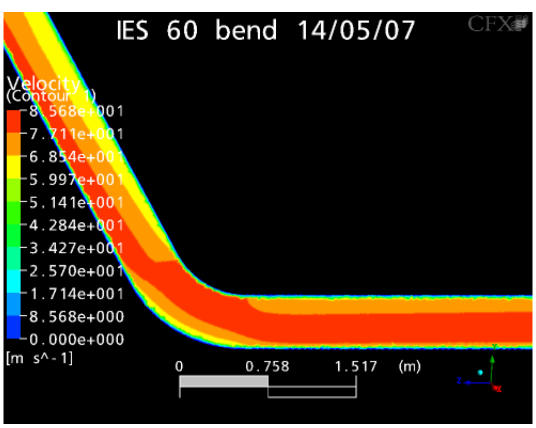
15° coal/air



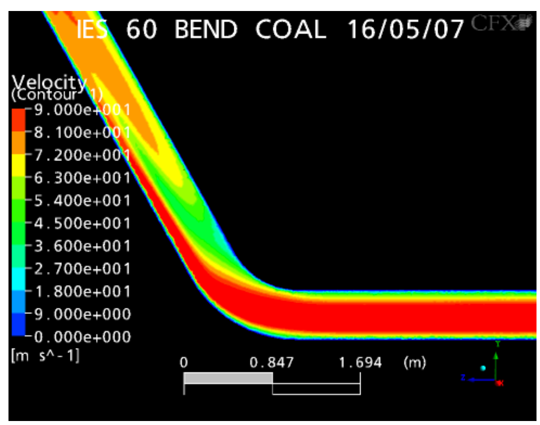
45° air



45° coal/air

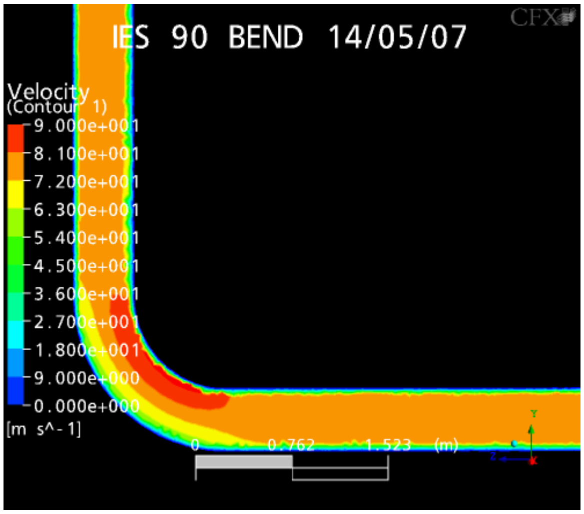


60° air

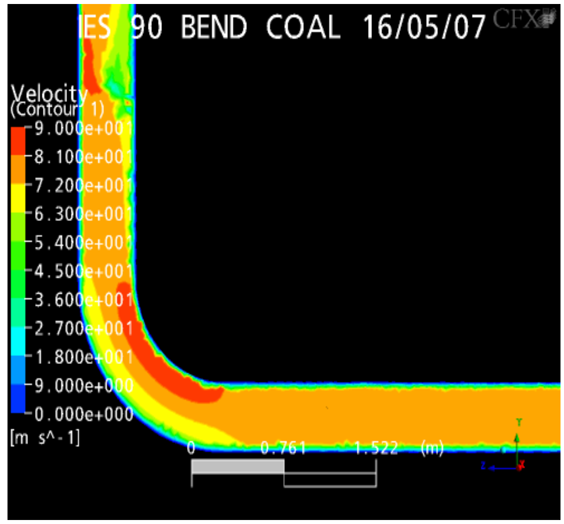


60° coal/air

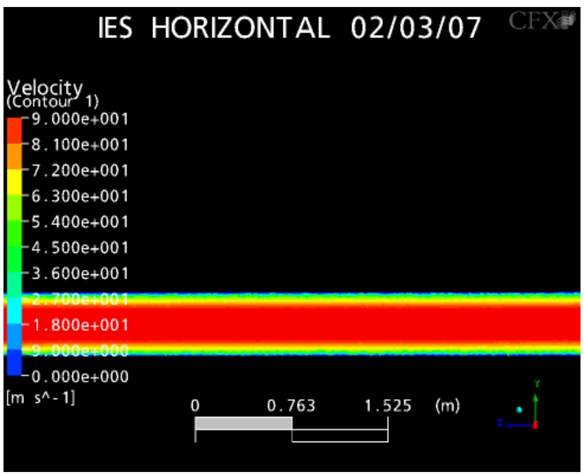
Fig no: 3 Air (left hand side column) and coal/air (right hand side column) velocity magnitude contours for bends with various angles.



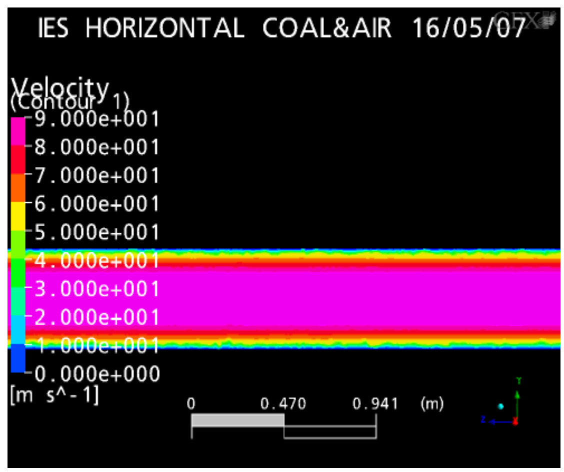
90° air



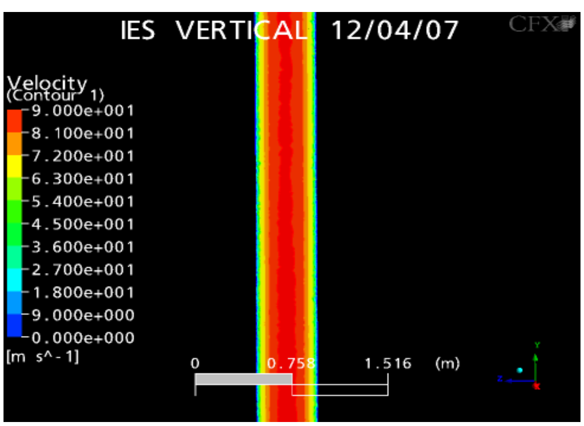
60° coal/air



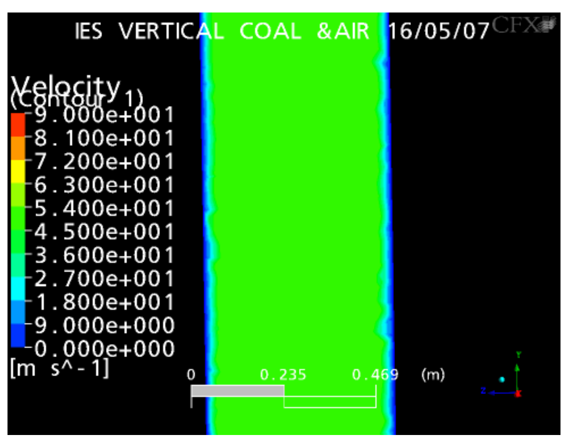
Horizontal air



Horizontal coal/ air



Vertical air



Vertical coal/air

Fig no: 4 Air (left hand side column) and coal/air (right hand side column) velocity magnitude contours for bends, horizontal, vertical section

As shown in Table.3, this sharp flow direction change results in significant pressure loss, with the 90° bend giving the highest. For higher bend angles (Fig.3 15°), the flow goes through the bend quite smoothly, and the pressure loss is insignificant compared to the sharp turning cases.

Higher coal VOF (volume of fluid) is found at the lower part of the upstream pipe. This is again attributed to the deposition of the coal particles. In the bend, maximum VOF of coal can be identified near the outer radius of the bend. This is explained by the flow impingement in the bend, and the attachment and accumulation of coal particles at this location. For sharp turning bends, the overall VOF of coal becomes much less downstream of the bend, indicating significant deposition and accumulation of coal particles in the bend and upstream pipe

While for smooth turning bend, (15°) the effects of coal particle deposition and accumulation are not as prominent (Fig.3). It should be pointed out that for the 90° bend, immediately downstream of the bend and close to the inner radius, this is the indication of flow separation. The local recirculating flow traps a significant amount of coal particles and a region of higher VOF of coal is formed.

7. Conclusions

There is currently no easy way of measuring coal/air flow in a power plant. In order to balance the coal/air flow to the individual burners, it would, therefore, be necessary to rely on clean air tests.

Commercially available software CFX 5 was used to calculate pressure drops in systems. Using this, several geometries involving any number of independent lines starting from a mill plant and discharging to a given furnace can be handled. The results show that the pressure drop in the systems strongly depends on the system geometry. Orifices are sized based on calculated coal/air pressure drops, and finally imposing a tailored imbalance in clean air flow distribution leading to a balanced coal/air distribution. This current paper demonstrated that CFD can be used as an effective tool for design and research for power industry applications.

References

1. Achim .D and Easton.A.K and Schwarz. M.P and Witt. P.J and Zakhari.A, (2002), 'Tube erosion modelling in a fluidised bed', -Appl. Math. Modell. vol- 26 pp 191–201.

2. AU-FRG, Institute for CAD/CAM (2007), 'Computational Fluid Dynamics Using CFX', Student Course and Manual, Anna University, Chennai-25

3. Brown .G.J., (2002), 'Erosion prediction in slurry pipeline tee-junctions', -Appl. Math. Modell. Vol - 26 pp 155–170.

4. Datta .K.biplab and Ratnayaka. C (2003), 'A Simple Technique for Scaling up Pneumatic Conveying Systems', Particulate Science and Technology, vol-21, pp 227–236

5. Eastwick. C.N. and Pichering. S.J. and Aroussi .A, (1999) 'Comparisons of two commercial computational fluid dynamics codes in modeling pulverized coal combustion for a 2.5 MW burner', -Appl. Math. Modell. Vol - 23 pp 437–446.

6. Good fellow .H and tahti E, (2001), Ed, 1 'Industrial Ventilation –Design Guide Book', pneumatic conveying, Academic Press ch-14, pp1317-1356

7. Govier.G.W and Aziz.K, (1977), 'The Flow of Complex Mixtures in Pipes',Robert.E.krieger puplishing company,Huntington, New York

8. Ratnayake .C and Melaaen. morten C.and Datta.k (2004) ,' Pressure Drop Determination Across a Bend Using CFD'A Programme of Research in Powder Science and technology, News Letter No. 22 .

9. Sowjanya Vijiapurapu and Jie Cui and Sastry Munukutla ,(2006), 'CFD application for coal/air balancing in power plants' Applied Mathematical Modelling vol - 30 ,pp 854–866

10. Sudharsan .Natteri M, 'Turbulent flow', Lecture Notes,Mechanical Engg Department Anna University