Study on the flow in the pipelines of the support system of circulating fluidized bed

L Meng¹, J Yang¹, L J Zhou*, Z W Wang² and X H Zhuang¹
¹ College of Water Conservancy and Civil Engineering, China Agricultural University, Beijing, 100083, China
² Department of Thermal Engineering, Tsinghua University, Beijing, 100084, China
E-mail: zlj09@263.net

Abstract. In the support system of Circulating Fluidized Bed (Below referred to as CFB) of thermal power plant, the pipelines of primary wind are used for transporting the cold air to the boiler, which is important in controlling and combustion effect. The pipeline design will greatly affect the energy loss of the system, and accordingly affect the thermal power plant economic benefits and production environment. Three-dimensional numerical simulation is carried out for the pipeline internal flow field of a thermal power plant in this paper. Firstly three turbulence models were compared and the results showed that the SST k-ω model converged better and the energy losses predicted were closer to the experimental results. The influence of the pipeline design form on the flow characteristics are analysed, then the optimization designs of the pipeline are proposed according to the energy loss distribution of the flow field, in order to reduce energy loss and improve the efficiency of tunnel. The optimization plan turned out to be efficacious; about 36% of the pressure loss is reduced.

1. Introduction
Circulating fluidized bed (CFB) combustion power generation technology is one of the main clean coal power generation technologies for its advantages such as wide adaptability of fuel, high combustion efficiency and high peaking capacity. The working principle of CFB is: firstly the coal particles below 10mm are fed into the furnace, then with the help of the air send in by the fans, the fuel become fluidized and begin combust. After the mixed gas flow through the cyclone separator the solid particles are then sent back to the furnace for full combustion. In this way, the fuel get fully burned, which decreased the sulphur in the fly ash and improved the combustion efficiency [1, 2].

The pipelines of primary wind which guide the flow from the fan into the circulating fluidized bed is one of the key components that ensure the bed material sufficiently and uniformly fluidized in the thermal power plant [3, 4, 5, 6]. The design of the pipeline structure affects the flow characteristics inside the pipeline. The unreasonable pipeline design even causes vibration and increases energy consumption, and will affect the overall efficiency of the CFB boiler system. So it is necessary to study the flow characteristics in the pipelines and optimize the pipeline design.

In this paper, in order to investigate the flow characteristics and wind loss inside the pipelines with the supporting bars consideration. CFD flow analyses were carried out and new designs were proposed. Compared to the designed pipelines, the new design reduced 36% of the wind loss.
2. Model description and numerical methods
The overall size of the pipelines of primary wind is 46.2×9.73×7.11m as shown in figure 1(a).
The computed flow domain includes two symmetrical inlet sections, one elbow section and one outlet section. The inlet section consists of a diffuser section and a straight section, inside the inlet section there are a lot of supporting bars which make the structure more robust. The elbow section followed the inlet section, combined the two inlet sections and guide the airflow came from X direction into Y direction and then turn another 90 degree into –Z direction. And the structure became complicated because of the supporting bars inside the inlet sections, so an unstructured mesh was used. The elbow section and the outlet section had a relative coarse mesh while the mesh was further refined in the inlet sections, and the total mesh has about 710,000 nodes, 3,880,000 elements, as shown in figure 1(b).

![Figure 1. (a) A view of the Computational domain, (b) Mesh view of the Pipeline.](image)

Initial calculations were performed by using the RNG k-ε and SST k-ω turbulence models. The results showed that the SST k-ω model converged better than the RNG k-ε model, and the energy losses predicted by the SST k-ω model were closer to the experimental results. So the SST k-ω turbulence model was chosen in this paper.

The airflow inside the pipelines of primary wind was treated as incompressible fluid and the dissipated heat created by the viscous force was also ignored. And the thermodynamic properties for real fluids can be closely approximated using the relationships for an ideal gas.

The model used the classic boundary conditions for incompressible flows: the inlet boundary condition was set to constant mass flow rate and the outlet boundary condition is pressure boundary condition by using the default pressure of 0. The other walls were set as fixed walls with a roughness height of 3×10^-5 m, a heat loss rate of 60 w/m^2, a thickness of 0.005 m.

3. Calculation and analysis of the designed model
In order to analyse the flow characteristics of the original model, the mass flow rate of the inlet was set to 147kg/s and the velocity streamline inside the pipeline was obtained as shown in Figure 2.

In figure 2, there are three graphs that represent different perspectives. The inlet part and the elbow part were two key parts that cause the most pressure loss. In the inlet part the airflow firstly become chaotic because of the resistance of the supporting bars, then the airflow from the two inlets converged in the elbow part with opposite speed, colliding with each other and then squashed together into the contraction section. In this process the airflow changed the direction several times and at last flew out. In this way, a large number of collisions, contraction, rotation direction and reflux led to a sharp deterioration of flow regimes and the substantial pressure loss.

![Figure 2. The velocity streamline inside the pipeline.](image)
In order to study the influence of the supporting bars on the flow field, two planes were inserted to the flow field, Plane 1: Y=1.30m and Plane 2: Y=0.95m. Plane 2 come through the centre of the horizontal supporting bar and the Plane 1 just come through the vertical supporting bar.

Velocity and total pressure distribution on Plan 1 is shown in figure 3(a), the airflow firstly flow in the diffuser and the velocity gradient changed a lot, the pressure in the middle was larger than the pressure in the outer side parts which may lead to swirls. When the flow comes out from the diffuser the influence of the supporting bar becomes stronger. In the middle of the plane the air flow across a series of vertical supporting bars, similar to the situation of air flow around a cylinder. Behind the cylinder there is a low pressure part, which led to the reflux, however the pressure gradient was reduced and reduced after flow through some cylinders, so the influence of the bars attenuated as the distance from the bar to the inlet accumulated.

Figure 3(b) shows the velocity and total pressure distribution on Plan 2. Because the plane goes through the centre of the vertical and also the horizontal bars, the flow pattern is more confusing than that in Plane 1. The effect of flow around a cylinder comes up in two directions and superimposed. Large number of reflux and swirls arose and led to significant pressure loss. In this plane the influence of the bars attenuate slower than that in plane 1, and the eddy still appear in a very far distance from the inlet.

In the same way, another two working points were also calculated. In the three simulations, the pressure loss of each part is basically the same, the pressure loss of the inlet part is almost 50% of the total loss, the elbow part contributes 35% of the loss, and the last 15% comes from the outlet part. This means the supporting bars and the elbow part make major contribution to the pressure loss, and optimization work is focused on these two sections.

4. Flow path optimization and analysis
According to the conclusion of the previous section, the selection of the size of the supporting bars and the layouts, the design of the elbow part and the outlet part can be optimized. Considering the time-saving, the optimization of the supporting bars and the other parts were done respectively.

4.1. Optimization of the supporting bars
In order to simplify the model, the inlet section was considered separately from the whole flow path, however the inlet and the outlet of this section was extended in consideration of making the flow sufficiently developed, as shown in figure 4. The optimization focused on the scale of the bars, besides, in order to ensure the same strength, the numbers changed accordingly. The diameter of the initial supporting bars is 125cm, as a comparison, set the supporting bars with diameter 75cm, 100cm and 150cm.

The simulation results are shown in table 1, “D” represents for the diameter of the supporting bars, “N” represents for the numbers of the supporting bars. It can be seen that the pressure loss of the inlet part is proportional to the diameter of the supporting bars, the loss is minimal when the diameter of the bars is 75cm.

The comparison of supporting bars of two diameters is shown in figure 5, it can be seen that when the diameter is 125cm, the flow reflux can be obtained near the exit of the pipeline. When the diameter is 75cm, the flow becomes relatively smooth near the middle of the pipeline, that means the smaller the diameter of the bars is, the influence of the bars to the
flow pattern is slighter. According to this, the diameter of 75cm is the better choice for the pipeline.

![Image](75cm_100cm_150cm.png)

**Figure 4.** Optimization plans of the supporting bars.

| Diameter(cm) | Number | Inlet pressure | Outlet pressure | Pressure Loss |
|-------------|--------|----------------|-----------------|--------------|
| 75          | 38     | 2062.22        | 768.34          | 1293.88      |
| 100         | 29     | 2100.52        | 658.70          | 1441.82      |
| 125         | 23     | 2327.97        | 706.90          | 1621.05      |
| 150         | 19     | 2445.03        | 690.11          | 1754.92      |

**Table 1.** Pressure loss (Pa) of different optimization plans of the supporting bars.

![Image](velocity_distribution.png)

**Figure 5.** Velocity distribution on Plan 2 with bars of different diameters.

4.2. Optimization of the pipeline
The pressure loss was mainly caused by the elbow section and the outlet section when the supporting bars were not included. In order to minimize the pressure loss of each part, some optimization plans are proposed, as shown in figure 6. In Opt. Plan 1 reduced the cone angle of the diffuser of the inlet section, the cone angle is changed from 14.4 degree to 8 degree. In Opt. Plan 2, use arc lines to instead of the fold lines in the corners of the elbow section, which helps to lead the airflow flow smoothly. The Opt. Plan 3 changed the angle of the connection section which leads the airflow flow from the inlet section to the elbow section, in the original model, the angle is 90 degree, the flow collisions was large, in the new design, the angle is set 135 degree. And in Opt. Plan 4, use arc lines to instead of the fold lines in the corners of the outlet section. Besides, the combined optimization plan is name Opt. Plan 5, that is the combination of Opt. Plan 2, 3 and 4.

In Table 2, the pressure loss of each section and the total pressure loss of the pipeline is shown, the bold figures represent for the changed parts, such as the figures in line 3, represent for the pressure loss in each part of the pipeline of Opt. Plan 2, and the second figure 214.75 is written in bold, that means in this plan, the elbow section was optimized. It can be seen that Opt. Plan 1 is not successful in reduce the pressure loss, while other plans achieved the aim. And plan 5 reduced the loss in a large scale.
4.3. Optimization result
Combined the Optimization in the above two subsections, a new pipeline of primary wind is proposed, in which both the design of the pipeline and the supporting bars are optimized, the diameter of the supporting bars is 75cm and the pipeline was optimized as optimization plan 5. The total pressure distribution on the final optimization plan was shown in figure 7.

It can be seen that the total pressure more evenly distributed, the pressure loss caused by the irregular structure was reduced. The total pressure loss of the newly designed pipeline is reduced to 678.29 Pa. while the pressure loss of the primary design is 1059.95 Pa. The final optimization plan reduced 36% of the loss.

5. Conclusion
The study of the flow characteristics was done with the software CFX. Initial calculations were performed by using the RNG k-ε and SST k-ω turbulence models. The results showed that the SST k-ω model converged better than the RNG k-ε model, and the energy losses predicted by the SST k-ω model were closer to the experimental results. According to the calculation, the supporting bars, the diffuser, corners make the flow chaotic, lead to the reflux and the swirls which increased pressure losses and resulting in a waste of energy.
Several optimization plans were proposed, more supporting bars with smaller diameter, arc lines were instead of the fold lines in the corners of the elbow section and in the corners of the outlet section, the angle of the connection section which leads the airflow flow from the inlet section to the elbow section was set 135 degree. And these work turned out to be efficacious, the total pressure loss of the newly designed pipeline is reduced to 678.29 Pa, about 36% of the pressure loss is reduced.

Acknowledgements
The authors thank the National Natural Science Foundation of China (No. 51279205) for supporting present work.

References
[1] Lu C M and Cheng S Q 2003 Circulating fluidized bed boiler equipment and operation (Beijing: China Electric Power Press)
[2] Fan C Z 1995 Boiler Principle (Beijing: Water Power Press)
[3] Hack H, Hotta A and Kettunen A 2008 Ultra- Supercritical CFB Technology to Meet the Challenge of Climate Change (Finland: Foster Wheeler Energia Oy)
[4] Rch L 2003 Development Potentials and Research Needs in Circulating Fluidized Bed Combustion (Beijing: China Particuology) pp 185-200
[5] Cheng L M, Zhou X L and Zheng C H 2008 Journal of power Engineering 28 818-826
[6] Deypere F, pieters J G and Dewettinck K 2004 Powder Technology 145 176-189