Numerical Investigation of Flow Dynamics and Improvement of Hot Primary-air Pipe Systems in Power Generation Plants

Qingyun Yan\textsuperscript{1}, You Li\textsuperscript{2}, Yuanhong Zhu\textsuperscript{3}, Guoquan Luo\textsuperscript{3}, Haitao Zheng\textsuperscript{3}, Damei Liang\textsuperscript{3} and Xuemin Ye\textsuperscript{2,*}

\textsuperscript{1}Jiangxi Vocational and Technical College of Electricity, Nanchang 330032, China
\textsuperscript{2}North China Electric Power University, Baoding 071003, China
\textsuperscript{3}PowerChina Jiangxi Electric Power Engineering Co., Ltd., Nanchang 330096, China

*Corresponding author email: yexuemin@ncepu.edu.cn

Abstract. The pressure losses of the hot primary-air pipe system have a great influence on the economic operation of a pulverizing system. To balance the flow resistance for different branches, Fluent was applied to simulate flow dynamics and pressure drop for a hot primary-air pipe system, and two improvement schemes were proposed. Numerical simulation reveals that for the original configuration, the flow losses of each branch are quite different, and the large resistance of some branches is mainly caused by pipe tees. After improvement, the flow regime has been significantly polished, and the total pressure drop of different branches is greatly reduced. Compared with the original configuration, the total pressure loss of branch B and branch D in case 2 is reduced by 65% and 53%, respectively, then an approximately equal pressure drop of each branch is achieved after structure improvement.

Keywords: Power generation plants; Hot primary-air pipe; Improvement; Numerical modelling.

1. Introduction

Hot primary-air pipe systems are a crucial component in pulverizing systems, and the flow losses of each branch directly affects the flowrate distribution, which is strongly related to the economic operation of power generation units. Due to the different pressure drop of each branch, a distinctly unbalanced flow is found under throttle valves fully open, thereby influencing the economy of the pulverizing systems to a certain extent. Therefore, the examination of the flow dynamics and pressure drop of hot primary-air pipe systems it is greatly important to improve the original configuration for the economic operation of power generation units.

Presently, some important studies were conducted for exploring the flow dynamics of parallel pipelines. Zhang et al. [1] numerically inspected the impacts of pipe spacing, area ratio and outlet number and Reynolds number on the flow balance and pressure drop of parallel pipelines. Zhou et al. [2] conducted a simulation to examine the flow distribution of all branches in a heat exchanger and found that the variation in pipe diameter and spacing has an apparent influence on the flow uniformity. Additionally, a great impact of pipe tees on the flow balance of the complex pipelines was also found in practical applications [3,4]. Costa et al. [5] experimentally found that for rounded and sharp-angled 90° tees, the minor losses of each branch were higher than main pipe, and the recirculation zone of rounded tee branch pipes was weakened, leading to the reduction of flow resistance. Li et al. [6] numerically and experimentally indicated that after arranging a wedge-shaped element with an suitable height in a tee, the local resistance was reduced by elbow return zone. Taking the biomimicry of the
branched structure of plants into account, Gao et al. [7] utilized a protrusion structure to reduce the flow losses in duct tees. There are, unfortunately, few literatures on investigating the flow distribution of combined tees in complex pipe systems. Hence, the flow dynamics of hot primary-air pipe systems in power generation plants is explored in this study, and then the impact of the upstream pipe tees on the downstream are inspected, finally the improvement measures for the optimization of complex pipelines are proposed.

2. Simulation Strategy

2.1. Physical Model
The geometric diagram of the hot air pipe is presented in Fig. 1. The hot air enters the header from inlet 1 and inlet 2, and then is distributed towards branch pipes. Table 1 presents the practical parameters under typical operating loads. In the present study, the operating condition of boiler maximum continuous rating (BMCR) with check coal was selected for investigation. The same velocity is dictated for two inlets and the temperature of air is 370°C, as well as throttle valves are utilized to balance the flowrate for each branch in practical operation, as shown in Table 1. To simplify the description, the air from inlet 1 to outlet A is named as branch A, and in a similar way, the branch B, branch C, branch D, and branch E are defined. Some assumptions are employed as follows: the inlet flow field of the hot air pipe is uniform, moreover air is considered to be incompressible.

![Figure 1. Diagram of the hot primary-air pipe in a power generation plant.](image)

| Coal          | Load | Operation state | Inlet velocity (m/s) |
|---------------|------|-----------------|----------------------|
| Check coal    |      |                 |                      |
| BMCR          | 2 inlets, 5 outlets | 28.50            |
| BRL           | 2 inlets, 5 outlets | 27.80            |
| BRL           | 2 inlets, 4 outlets | 22.70            |
| Design coal   |      |                 |                      |
| BMCR          | 2 inlets, 4 outlets | 22.10            |
| BRL           | 2 inlets, 4 outlets | 16.80            |
| 75%BMCR       | 1 inlet, 2 outlets | 23.00            |
| 50%BMCR       | 1 inlet, 2 outlets | 24.45            |

For reducing the flow loss of the system, the following schemes are proposed. As is shown in Fig.2, in case 1, the angle between header pipe and branch B is reduced from 90° to 70°, the arrangement of branch C remains unchanged. The outlet position of branch D keeps unchanged, while the elbow at the entrance is changed to a straight pipe, as presented in Fig. 2(b). For case 2, the entrance of branch B is close to main pipe, and the angle between header pipe and branch B is reduced to 48°. And the inlet of branch C is replaced with a divergent pipe, and branch D is the same as case 1.

2.2. Computational Method
ICEM is applied to mesh the 3D physical model of the hot primary-air pipe. In order to promote the calculation accuracy of the simulation, the hexahedral meshes are utilized with the O-type division.
method [8]. Based on the assumption of quasi-steady turbulence, Fluent is used to realize the solution of the continuity equation and Reynolds averaged Navier-Stokes equation. As air flows through a tee, the internal flow field is extremely complex and an obvious recirculation is generated, hence the Realizable \( k-\varepsilon \) turbulence model is employed in this simulation [4]. The coupling of pressure-velocity is utilized with SIMPLEC. When residuals of all parameters are below \( 10^{-4} \), and the mass flowrate difference between inlet and outlet is lower than \( 10^{-3} \), we believe that the simulation is converged [9].

![Image](a) Original

![Image](b) Case 1

![Image](c) Case 2

**Figure 2.** Schematic of the system before and after improvement.

The velocity inlet and pressure outlet are set in the boundary conditions. The intensity of turbulent kinetic energy and hydraulic diameter are selected to define turbulence, and the outlet flowrate is set as the value under the corresponding working conditions. The operating pressure is under the local atmospheric pressure. The solid wall adopts the non-slip condition, that is, the velocity of the fluid on the wall is considered equal to the wall velocity [10]. The absolute roughness of pipe wall is 0.06 mm, and the effect of gravity in this study is ignored.

A tee pipe with 200 mm diameter is modelled with above models and boundary conditions, we found that the results are well consistent with the experiment conducted by Rahmeyer [11] indicating with the maximum of relative deviation of 3.1%. Furthermore, in order to achieve the mesh independence, the meshes of 0.76, 1.41, 2.16 and 3.07 million are carried out for the original configuration. The total pressure drop of the original configuration in different meshes is listed in Table 3. It can be clearly noted that under the meshes of 2.16 million, the pressure drop is approximately same to 3.07 million meshes. Taking the modelling efficiency into account, a mesh number of 2.89 million is used for simulating the original, and other cases also use the same method to achieve the mesh.

| Meshes (million) | A  | B  | C  | D  | E  |
|------------------|----|----|----|----|----|
| 0.76             | 152.1 | 410.2 | 369.8 | 420.6 | 154.9 |
| 1.41             | 154.5 | 413.9 | 367.8 | 424.2 | 152.7 |
| 2.16             | 156.7 | 415.4 | 367.6 | 425.8 | 151.8 |
| 3.07             | 156.8 | 415.6 | 367.7 | 426.0 | 151.8 |

### 3. Results and Discussion

#### 3.1. Internal Flow Dynamics

Different types of shunt and confluence tees are applied in the hot primary-air pipe system in power generation plants. Figure 4(a) shows that the upstream tee has a vital influence on the internal flow of the downstream tees. After air leaving from the main pipe 1 collides the header pipe, then it flows along the direction of the header pipe, and the velocity on the lower side is fast. When air flows into branch B, a sharp turn leads to apparent energy losses. After entering branch B, a distinct low-velocity recirculation zone is formed on the back side, resulting in notable energy losses. The flow field in the middle of the header pipe is relatively uniform and the velocity is low. The low-velocity recirculation zone in branch D is more remarkable, resulting in the decrease in the effective flow cross-area and the increase in the velocity of air, and thereby increasing energy losses.
Figure 3. Streamline distribution of the Z-section for different cases.

After improvement, the low-velocity recirculation zone emerged on the backside of branch B in case 1 is slightly lowered, the effective flow cross-area is increased, and the inlet velocity is diminished, hence the flow resistance of branch B is appreciably weakened. The recirculation zone on the backside of branch D is, however, greatly shrunk, and the effective flow cross-area is increased, which greatly improves the local flow regime in branch D. The left recirculation zone of branch C is decreased, whereas the right recirculation zone is expanded. In case 2, the decrease in the angle between branch B and header pipe leads the branch B inlet to approach the main pipe, hence part of the fluid leaving from the main pipe directly enters branch B, and then the recirculation zone on the backside is apparently reduced, resulting in the reduction of energy losses. After the inlet of branch C is changed to a divergent pipe, the recirculation zone in the vicinity of the inlet is roughly disappeared. Figure 4(a) shows that the high total pressure are found in branch A and branch E have, while the low total pressure are found in branches B, C and D, then low-pressure zones are emerged on the back flow side of these branches. Therefore, it can be concluded that the low-pressure zone generated in branches B, C and D is regarded as the main reason of large energy losses. After improvement, in case 1, the low-pressure zones in the inlet of branch B and branch D are diminished, and the pressure gradient is subsequently reduced; different low-pressure zones are generated on the left side and right side of branch C. In case 2, the low-pressure region at the inlet of branch B is significantly reduced, and the minimum pressure value is increased; the pressure gradient at the branch C inlet is decreased.
### 3.2. Flow Resistance

| Type      | Total pressure drop variation (Pa) |
|-----------|-----------------------------------|
|           | A          | B          | C          | D          | E          |
| Original  | 156.7      | 415.4      | 367.6      | 425.8      | 151.8      |
| Case 1    | 156.6      | 336.1      | 367.6      | 198.1      | 151.7      |
| Case 2    | 156.7      | 145.5      | 305.0      | 198.2      | 151.8      |

Table 3 summarizes the total pressure drop before and after improvement. The results clearly indicate that for the original configuration, the total pressure drop of branches A and E is small and roughly the same, whereas that of branches B and D is large, and that of branch D is slightly larger than branch B. After improvement, in case 1, the total pressure drop of branch D is significantly reduced, and the total pressure of branch B is reduced by 79 Pa. The total pressure drop of branch B in case 2 is greatly decreased, and the total pressure of branch C is reduced by 63 Pa. The pressure drop of branch B and branch C is respectively lowered by roughly 65% and 53%. This is mainly because after improvement, branch B and branch D are close to the inlet of pipe, and the angle variation of branch B and branch D significantly improves the flow regime, thereby reducing the total pressure loss. Although the local flow of branch C inlet is improved, the reduction of the total pressure loss is small due to the high resistance coefficient of the confluence tee. The above results indicate that case 2 is a practical choice for upgrading the configuration of hot primary-air pipe systems.

### 4. Conclusion

In the original configuration of the primary-air pipe system, the inlet of branches B, C and D pipes is the tee structure, so the local flow regime is extremely complex, resulting in apparent energy losses. After improvement, the flow field at the inlet of branch B and branch D is notably improved, then the total pressure drop is significantly reduced. The resistance of branch B in case 2 is reduced by approximately 65% compared with the original layout, and that of branch D pipeline is reduced by about 53%. As the inlet of branch C is a confluence tee and is affected by the position of the steel structure, its flow resistance is only reduced by approximately 17%. Case 2 is a practical choice for promoting the configuration of hot primary-air pipe systems in power generation plants.
References

[1] Zhang W Q, Li A G, Gao R et al. 2018 Effects of geometric structures on flow uniformity and pressure drop in dividing manifold systems with parallel pipe arrays. *International Journal of Heat and Mass Transfer*, 127, 870-81.

[2] Zhou J, Sun Z N, Ding M et al. 2017 CFD simulation for flow distribution in manifolds of central-type compact parallel flow heat exchangers. *Applied Thermal Engineering*, 126, 670-7.

[3] Jiang Y G, Xu Y X, Qin J et al. 2018 The flow rate distribution of hydrocarbon fuel in parallel channels with different cross section shapes. *Applied Thermal Engineering*, 137, 173-83.

[4] Zhang W Q and Li A G 2018 Resistance reduction via guide vane in dividing manifold systems with parallel pipe arrays (DMS-PPA) based on analysis of energy dissipation. *Building and Environment*, 139, 189-98.

[5] Costa N P, Maia R, Proença M F et al. 2006 Edge Effects on the Flow Characteristics in a 90deg Tee Junction. *Journal of Fluids Engineering*, 128(6), 1204-17.

[6] Li A G, Chen X, Chen L et al. 2013. Study on local drag reduction effects of wedge-shaped components in elbow and T-junction close-coupled pipes. *Building Simulation*, 7(2), 175-84.

[7] Gao R, Liu K K, Li A G et al. 2018 Biomimetic duct tee for reducing the local resistance of a ventilation and air-conditioning system. *Building and Environment*, 129(1), 130-141.

[8] Gao R, Wen S H, Li A G et al. 2019 A novel low-resistance damper for use within a ventilation and air conditioning system based on the control of energy dissipation. *Building and Environment*, 157, 205-14.

[9] Gao R, Liu K K, Li A G et al. 2018. Study of the shape optimization of a tee guide vane in a ventilation and air-conditioning duct. *Building and Environment*, 132, 345-56.

[10] Beneš L, Louda P, Kozel K et al. 2013. Numerical simulations of flow through channels with T-junction. *Applied Mathematics and Computation*, 219(13), 7225-35.

[11] Rahmeyer W J and Dent P 2003. Pressure loss data for PVC pipe tees. *ASHRAE Transactions*, 109(2), 252-271.