Numerical simulation of dynamic characteristics of bituminite-fired boiler after mixing lignite for combustion

Wang Zemin

1 Huadian Electric Power Research Institute Co., Ltd. HangZhou, Zhejiang, China, Code 310030

Abstract: If the lignite is mixed for combustion in the bituminous-fired boilers in a power plant, less primary air volume is required to avoid burner nozzle coking due to the high volatility of lignite. Stimulate the boiler's dynamic field under the design conditions after reducing primary air rate using the Fluent software, and analyze the diameter of the tangent circle in the furnace and the process of primary air's stiffness attenuation. The result shows that as the primary wind rate and speed decrease, the dynamic field in the furnace changes greatly, while the speed of the adherent wind increases, and the wall brushing phenomenon occurs. In order to ensure a dynamic field in the furnace, it is necessary to place a central cross partition to increase the rigidity of the primary air, and to prevent the primary air from decaying too quickly or deflecting towards the wall. The result can be used as a reference for the design and modification of the same type of boiler burner.

1. Introduction
With the increasingly tight domestic coal supply and the sharp fluctuation of coal prices, more power plants are now unable to depend on the design coal during production and operation. Under the existing coal combustion conditions, a power plant in Northeast China decided to mix lignite, which is an abundant fuel resource, in the walled tangent-circle bituminous coal boiler so as to ensure the combustion stability, safety, as well as economic and environmental requirements. Lignite has a relatively high volatile content and enters the furnace early, resulting in high burner root temperature and easy coking. To avoid this phenomenon, it is necessary to appropriately reduce the primary air volume and put the primary air nozzle in an oxygen-depleted state, thereby delaying the combustion time of coal powder. This paper takes a 600MW unit walled tangent-circle bituminous coal boiler as an object to conduct the power field test and numerical simulation of the power field in the furnace before and after the change of the primary air rate after lignite blending, thereby providing a reference for related research.

2. Device
The 600MW ultra-supercritical boiler (HG-1795 / 26.15-YM1) in the power plant is an ultra-supercritical variable-voltage DC boiler with a single furnace, primary air reheating, balanced ventilation, solid slag discharge, and a full suspension Π structure. The main burner is arranged on four walls. The boiler adopts walled tangent-circle combustion method and takes soft coal for burning. The boiler adopts a positive pressure direct-blown pulverizing system and has six MPS235HP-II medium-speed coal mills. Each coal mill grinds coal powder for the burner on each layer of the furnace. The burner uses a CUF walled tangent-circle large-bellow structure with 6 layers of 24 sets of thick and light primary air vents arranged on the four furnace walls. The OFA burn-out air vent is arranged at the top...
of the main combustion area using a wall arrangement method. There are a total of four layers of burn-out A-A wind nozzles located approximately 5 meters above the uppermost primary pulverized coal spray nozzles, which are arranged in the tangential circle to supplement the air required for the later fuel combustion.

3. Numerical simulation

3.1. Mesh division and mathematical model

![Longitudinal view of boiler](image)

Figure 1 Longitudinal and cross-section grids of boiler

The structural size of the mathematical model adopted in this paper is divided according to the actual size and structure of the boiler. The simulation area starts from the bottom slag hopper to the furnace exit. Due to the complicated structure of the boiler, this paper divides the furnace into 5 parts by using gambit, namely, the cold slag hopper, the main combustion area, the A-A wind area, the area from A-A wind to the furnace arch and area from the furnace arch to the exit.

For the cold slag hopper, the main combustion area and the upper AA wind area adopt the structured hexahedral grid. Since the AA wind adopts a tangential circle arrangement method, which is different from what the burner adopts, this paper uses the paving method for grid division and then uses the cooper method to generate the volume mesh. Except for the main combustion area, the grid in other areas of the boiler is relatively sparse. The burner nozzles in the main combustion area are arranged on four walls, and the jet is sprayed perpendicularly to the wall facing the furnace. Therefore, a hexahedral grid is beneficial to alleviate the pseudo-diffusion phenomenon during simulation calculation. At the same time, the grid of the main combustion area is encrypted, which is conducive to analysis accuracy. The number of grid-point of the whole furnace is 540,000. The grid division of the cross section of the furnace in the main combustion area is shown in Figure 1.

This paper adopts the Fluent software to simulate and study the dynamic characteristics of the boiler. The flow field in the furnace is a complicated three-dimensional turbulent flow. This paper simulates a single-phase turbulent flow while the gas-phase turbulent flow adopts the standard k-epsilon equation model. It is assumed that the furnace is in a steady-state flow, and the finite volume method is used for discrete differentiation. The equation is solved using the simple algorithm, and the difference adopts the Quick format, so that the simulation results can better reflect the flow state in the furnace.

3.2. Boundary conditions and simulation conditions

The burner nozzle arranged on the wall of the furnace is the entrance boundary of the calculation area, which is defined as the velocity inlet. The exit section of the horizontal flue is the exit boundary, which is defined as the free-flow outlet. The hydraulic diameter D and turbulence intensity I of each primary and secondary air nozzle are calculated using the following two formulas:
\[ D = \frac{4ab}{2(a+b)} \quad I = 0.16(Re)^{0.125} \]

In the formula, \(a\) and \(b\) represent the length and width of each nozzle, and \(Re\) is the Reynolds number of the gas flow.

The wind rate and wind speed are shown in Table 1. There are two kinds of simulated working conditions in the furnace, namely, the working condition where the downwind rate of unblended lignite has not changed (condition 1), and the working condition after reducing the primary air rate to avoid coking near the burner nozzle when lignite is mixed for burning (condition 2).

| Project                  | Primary wind rate (%) | Primary wind speed (m/s) | Secondary wind rate (%) | Secondary wind speed (m/s) |
|--------------------------|-----------------------|--------------------------|-------------------------|---------------------------|
| Text (condition 1)       | 31                    | 26                       | 68                      | 46                        |
| Simulation (condition 1) | 31                    | 26                       | 68                      | 46                        |
| Simulation (condition 2) | 28                    | 24                       | 72                      | 47                        |

4. Analysis of simulation results

4.1. Cross tangential section of furnace and comparison of cold test results

Figure 2 shows the comparison between the cross wind speed of the cold test results (solid line) and that of the simulation result (dashed line) of the B-layer burner cross section. In the cold test, the layer B is selected and measured using the ribbon method. It can be seen from the figure that the diameter of the tangent circle measured in the test is slightly smaller than the simulation result. At the same time, the maximum tangential velocities measured in the test and simulation are also basically the same. The adherent wind on the furnace wall is below 3.2m/s, and there is no obvious wall brushing phenomenon. The simulation result is not significantly different from the test result, so the simulation result obtained from this numerical model can well reflect the flow field in the furnace.

4.2. Dynamic characteristics analysis of cross-section of burner nozzle

The reasonable flow of the fluid in the boiler is the key factor for combustion\(^3\). This paper conducts stimulation calculation of the flow field in the furnace area. Figure 3 shows the cloud diagram of the velocity vector distribution on the cross section of the primary wind nozzle of the main burner under conditions 1 and 2 respectively. As can be seen in Figure 3 (a), the fluid has formed a good rotating tangent circle on the cross section of the burner layer. The diameter of the tangent circle is moderate and

![Figure 2: Tangential cross velocity distribution of Layer B in test and simulation](image-url)
the circle center is basically at the center of the cross section. The wind entering the nozzle is intense at first and the intensity continuously decreases along the course, forming a low velocity zone at the center of the tangent circle. The velocity in the flow field is uniformly distributed with good symmetry. In Figure 3 (b), a good rotating tangent circle with a larger diameter is formed, and the wall brushing phenomenon is obvious. This increases the probability of coking of the water-cooled wall of the boiler, which is not conducive to the boiler safety and stability.

Figure 4 shows the diameter of the dimensionless tangent circle in a fluid furnace along the hearth height of the furnace under the above two conditions. The diameter of dimensionless tangent circle is the ratio of the diameter of fluid tangent circle to the equivalent diameter of the cross section of the furnace. It is used to evaluate the strength and weakness of the aerodynamic field in the furnace\([4]\). This paper selects 14 layers of dimensionless tangent circle in the cross section, including the six-layer primary tuyere cross section, the seven-layer secondary tuyere cross section, and the one-layer A-A wind cross section of the boiler. It can be seen from the figure that, under condition 2, the weakening of the jet rigidity has an impact on the flow field in the furnace due to lower primary air rate and lower primary air speed. The diameter of dimensionless tangent circle under two working conditions increases with the height of the furnace, which is due to the influence of greater amount of air under the burner nozzle. That is in line with the characteristics of the dynamic field in the furnace. The diameter of the tangent circle under condition 1 is still within the allowable range \((0.4-0.64)\)[4], while that under condition 2 increases significantly, exceeding the allowable range.
4.3. Characteristics analysis of burner jet

Figure 5 is a comparison chart of the attenuation trend of the jet intensity of the primary air nozzle in the B layer. The primary air jet is relatively rigid with a long range. The attenuation degree of the jet near the primary air nozzle is much smaller than that in the middle and latter part. This is due to the entrainment effect of the jet: as the jet distance increases, the amount of air entrained also increases, which in turn increases the jet width. This is conducive to the mixing of pulverized coal and air. The high-temperature flue gas entrained near the nozzle is beneficial to the stable combustion of pulverized coal[5]. Under condition 2, the primary wind speed, the primary wind momentum, and the jet rigidity are relatively small. With the increase of the range, the stiffness attenuation becomes larger, thereby widening the range of the primary wind. The diameter of the primary air tangent circle in the furnace becomes larger. The adherent wind near the water-cooled wall becomes strong, and wall brushing occurs.
5. Conclusion

Based on the numerical simulation and experimental study of the flow field in a 600MW walled tangent-circle boiler in the power plant when the lignite is mixed for combustion, we obtain the following conclusion.

(1) After blending lignite, which has a high volatile content and enters into the furnace early, the burner foot is high in temperature and is easy to coke. To avoid this, it is necessary to reduce the primary air rate appropriately so that the primary air nozzle can be put in the oxygen-depleted state and the combustion time of pulverized coal can be delayed. Reducing the primary air rate has adversely affected the boiler process.

(2) When the primary air rate, the primary air speed, and the rigidity of the primary air jet are lower, as the distance of the jet increases, the amount of entrained air increases, and the jet attenuation accelerates. The wall brushing phenomenon occurs, which is not conducive to the formation of a good aerodynamic field in the furnace. In order to ensure a good dynamic field, it is necessary to place a cross partition to increase the rigidity of the primary air jet, and to prevent the primary air jet from decaying too quickly or deflecting towards the wall.

(3) The diameter of dimensionless tangent circle is used to evaluate the aerodynamic field in the furnace. After reducing the primary air rate, the diameter of the tangent circle in the flow field increases significantly, exceeding the allowable range.

References

[1] Lockwood F C, Papadopoulos C, Abbas A S. Prediction of a corner-fired power station combustor[J]. Combustion science and technology, 1988, 58(1-3): 5-23.

[2] Ghenai C, Janajreh I. CFD analysis of the effects of co-firing biomass with coal[J]. Energy Conversion and Management, 2010, 51(8): 1694-1701.

[3] Numerical Simulation on Blended Coal Combustion with Anthracite in W-shaped Flame Boilers[J]. Journal of Power Engineering, 2012, 032(005):345-350,367.

[4] Combustion Process Simulation with Duo-mixture Fraction/PDF Approach of Tangentially Coal Blend Fired Boilers[J]. Journal of Power Engineering, 2006, 026(002):185-190.

[5] Structural Features and Performance Analysis of Wall-arrangement Tangential Firing System for a 600 MW Ultra Supercritical Boiler[J]. Journal of Power Engineering, 2011, 031(008):598-604.