Design and optimization of distribution structure of fully premixed surface burner

Dong Aihua1 Qiu Penghua2 Liu Hong1 Liu Li2 Sun Zhenglu1 Wang Hui2 Wang Yongtang1* Liu Guichuang2 Shi Yunxin1

1Harbin electric corporation central research institute, Harbin, Heilongjiang province, 150028, China
2Combustion engineering institute, School of energy science and engineering, Harbin institute of technology, Harbin, Heilongjiang province, 150001, China

*Corresponding author’s e-mail: wyongt@harbin-electric.com

Abstract. In response to China’s energy policy and the increasingly stringent NOx emission standards, premixed surface burners have gradually entered the field of vision as an ultra-low NOx emission gas burner. In this paper, the numerical simulation software ANSYS FLUENT was used to analyze the airflow distribution structure of a certain type of premixed surface burner, and the distribution characteristics of the premixed gas flow in the distribution structure during actual operation were also obtained. At the same time, the parameters of the distribution structure were optimized according to the problems appearing in the distribution of airflow by numerical simulation. The results showed that the design and optimization of the distribution structure of the premixed surface burner were reasonable and effective.

1. Introduction

As a high-quality, high-efficiency, clean and low-carbon energy source, natural gas can be controlled in a very low range compared with conventional coal powder if it is equipped with a suitable combustion method. According to the requirements of the 13th Five-Year Plan of the National Development and Reform Commission[1], by 2020, natural gas will account for 10% of primary energy consumption, and the natural gasification rate of urban population will reach 57%. In the Beijing-Tianjin-Hebei, Yangtze River Delta, Pearl River Delta, and other regions, the “coal to gas” project is being implemented on a large scale[2], and has achieved certain results, which will certainly promote the rapid growth of natural gas demand.

Beijing Municipality’s “Boiler Air Pollutant Emission Standard” DB11/139-2015[3], which was revised in 2015, requires a new boiler NOx emission limit of 80 mg/m3 before March 31, 2017. The new boiler NOx emission limit from April 1, 2017 is 30mg/m3.

Low-NOx gas burners currently being developed by European and American companies primarily use semi-premixed combustion technology to achieve low NOx emissions through fuel/air grading and flue gas internal circulation[4]. When implementing more stringent emission standards, such as emission targets of 30 mg/m3, European brands mostly use the technical route of staged combustion plus external flue gas recirculation. This is also the mainstream technology for ultra-low emission burners on the market in China. However, such technologies cannot achieve emission targets below 30mg/m3 for boilers with smaller boiler sizes, while surface fully premixed burners can achieve the goal of low NOx combustion in small and medium boilers.
The surface combustion technology is characterized by fully premixed combustion, which achieves complete uniform premixing of gas and air, while the excess air coefficient in the combustion process is small, and the heat released by combustion can be quickly transferred through radiation and convection. The flame temperature is lower, which can achieve lower NOx levels than other burners[5].

2. Numerical Simulation Study on a Mixture Structure of a Burner

The gas/air mixing technology and the gas distribution technology are two key technical points of the surface fully premixed burner. Bao Tong et al[6] numerically simulated the gas/air mixing structure of a brand surface burner. The results show that the uniformity of gas and air mixing will directly affect the burner combustion requirements and boiler temperature and emissions of nitrogen oxides. The distribution technique is designed to make the mixed gas evenly distributed on the surface of the combustion head, and to avoid uneven distribution of local flame due to uneven flow. Several common distribution structures are shown in Fig.1-Fig.4[7-10]. It can be seen that the characteristics of the distribution structure are mainly concentrated in the following aspects: the number of layers of the distribution structure, the holes distribution and the hole form of the distribution structure, The shape of the burner head.

At present, there are few related studies on the distribution structure of surface burners.

3. New distribution structure

A type of burner is used as a simulation object. The burner distribution structure is located inside the combustion head. It is mainly divided into: the inner layer distribution cylinder with scaly holes, as shown in Fig.5; The outer layer distribution cylinder, and the sintered metal fiber[11] wrapped around the strip hole cylinder, as shown in Fig.6.
In order to accurately simulate the flow of the mixed gas flow inside the burner, the simulation of this paper starts from the inlet of the combustion head, that is, the starting end of the distribution structure, and simulates the flow state of the airflow when both the fish scale hole and the strip hole exist.

The fish scale hole cylinder of one of the distribution structures is an approximate positive 45-sided shape, and about 100 fish scale holes are opened in the longitudinal direction of each side, and about 4,500 fish scale holes are opened in the entire fish scale hole cylinder.

Due to the large number of holes in the distribution structure, in order to reduce the number of meshes and ensure the grid quality to save calculation time, the model adopts a radial thin plate-like unitization model at the center of the fluid domain. Considering the symmetry of the fluid in the entire circumferential direction, only the fish scale holes which are opened in the circumferential direction on one side of the horizontal cross section 45 and the strip holes corresponding to the upper portion thereof are drawn. As shown in Fig.7, the outer cylindrical area is the fluid domain structure inside the actual combustion head, and the red area is based on the fluid domain region as used herein above.

The fish scale holes are opened in the inner cylinder wall, and the number of fish scale holes is 100, and the fluid domain model of the inner cylinder and the outer cylinder fluid domain is built, and the bottom end of the inner cylinder is at a distance of 37 mm from the bottom end of the outer cylinder. The structure is as shown in Fig.8 Shown.

The gas actually flowing in the burner distribution structure should be a mixed gas of natural gas and air. Table.1 shows the relevant physical parameters of natural gas and air under operating conditions.

| Fluid | Temperature(K) | Density(kg/m³) |
|-------|---------------|----------------|
| Air   | 300           | 1.185          |
| Gas   | 300           | 0.7174         |
Since the air volume flow rate during the mixing process of this type of burner is about 14 times that of natural gas volume flow, it can be seen from Table 1 that under the same environmental conditions, the air density is about twice the density of natural gas. Therefore, during the simulation, we set all the gases involved in the distribution to be air. The import and export models used in the simulation process are shown in Table 2.

| Name       | Type          | Velocity (m/s) | Pressure (Pa) |
|------------|---------------|----------------|---------------|
| Mix Inlet  | Velocity-Inlet| 25             | --            |
| Outlet     | Pressure-Outlet| --             | 0             |

In order to facilitate the analysis of the obtained simulation results, as shown in Fig. 9, a plane passing through the center of the fluid domain as indicated by the arrow is established, and the relevant results analyzed are derived from the sections obtained by this position.

The ANSYS FLUENT software was used for the simulation calculation. The five points in the flow field were randomly monitored during the calculation. After the simulation is completed, the velocity nephogram and the local velocity vector diagram on the analysis plane are obtained. As shown in Fig. 10 and Fig. 11, respectively.

It is easy to see from the velocity nephogram that at the end of the combustion head, that is, at the end of the flow of the airflow, there is a region where the airflow velocity is sharply increased due to the sudden decrease in the resistance. However, at the airflow inlet position, the airflow between the inner and outer cylinder is small. The area where the airflow is concentrated in the middle and rear parts of the overall distribution structure. The occurrence of the airflow concentration zone may be caused by the sudden decrease of the resistance of the tail of the combustion head, resulting in a large amount of airflow not flowing out of the fish scale hole and the strip hole, and the reason for the area of small airflow may be due to the special structural form of the fish scale hole. The tendency of the airflow to flow internally causes less flow out of the fish scale hole near the inlet.
4. Numerical Simulation and Optimization of New Distribution Structure

According to the simulation results of a certain type of burner distribution structure, in actual operation, there may be a case where the flow velocity at the inlet end is small and the flow velocity at the tail end is large, which may be a special opening type and scale of the fish scale hole of a certain type of burner. The hole is related to the arrangement of the strip hole. Because the structure of the strip hole is difficult to change under the current conditions, this paper mainly considers the optimization of the air inlet mode and the relevant parameters of the fish scale hole, in order to obtain a more reasonable one.

4.1 Analysis of fluid domain model for increasing airflow inlet

Since a type of burner has a gas flow cell between the distribution interlayers at the inlet, in order to optimize this area, first attempts to improve the low flow area between the fish scale cylinder and the strip cylinder sandwich at the inlet in a manner that increases the inlet area of the gas stream. Fig.12 shows an optimized structure with several circular holes at the inlet of the fish scale to increase air flow at the entrance of the interlayer.

Fig.12 A fluid domain structure model with airflow flowing into the orifice at the inlet mezzanine

The fluid domain meshing and boundary conditions are the same as above. After the simulation is completed, the velocity nephogram distribution on the new reference plane and the local magnification speed nephogram at the entrance position are obtained, as shown in Fig.13 and 14.

Fig.13 Distribution structure velocity map of the mezzanine with airflow flowing into the small hole

Fig.14 Enlarged cloud at the entrance of the distribution structure

It can be seen from Fig. 13 and Fig. 14 that the flow of air into the small hole at the entrance of the two-layer distribution structure is more effective when the airflow just flows in, but as the flow deepens, it improves the overall low flow area of the front section of the interlayer. The effect is not very significant, which may be related to the size of the opening, the arrangement of the holes and other factors.

In order to further verify the effect of increasing the inflow area of the airflow to improve the low flow area, an extreme model is set up, that is, the entrance end, that is, the initial end of the interlayer is also set as the airflow inlet, as shown in Fig. 15, after the simulation is completed, a new one is obtained. The velocity nephogram on the reference plane is shown in Figure 16.
Fig. 15 Fluid domain structure model with inlet section added

Fig. 16 Increased distribution profile of the inlet section

It can be seen from Fig. 16 that the front section of the interlayer is set as the airflow inlet, which also plays a partial improvement role. Due to the effect of the interlayer opening, the low flow area originally located in the front section moves backwards, but the overall has not been reduced. The trend, thus showing that increasing the airflow inlet area is not a better way to improve the low velocity flow zone.

4.2 Fluid Domain Model Analysis of Changing the Opening Area at the Entrance of Fish Scale Holes

One of the reasons for the low flow area between the fish scale cylinder plate and the strip cylinder plate in the inlet section is that there is less airflow from the fish scale hole at the inlet. For this purpose, the structure is increased the opening area of the fish scale hole at a certain length of the entrance of the fish scale hole plate by about twice the original, and aims to increase the outflow of the fish scale hole at the inlet of the orifice plate further serves to improve the flow rate of the inlet. The velocity nephogram distribution and the local velocity of the entrance location are obtained on the reference plane, as shown in Fig. 17 and Fig. 18.

Fig. 17 Overall distribution structure velocity map with increased opening area at the entrance

Fig. 18 Entrance position partial magnification speed nephogram

It can be seen from Fig. 17 and Fig. 18 that after the opening area of the front fish scale hole is increased, the airflow rate in the low flow area is increased to some extent, so the structure has a certain improvement effect on the expected airflow distribution, but it is still necessary to carry out comparative analysis by numerical simulation to increase the opening area and increase the length of the opening.

4.3 Fluid Domain Model Analysis of Reversed Fish Scale Holes

The direction of the opening of the fish scale hole of a certain type of burner will form a flow aid the flow of air flowing into the inside of the combustion head, and the flow direction of the fish scale hole
is set to the opposite direction. The influence of the structure on the flow distribution of the gas flow between the interlayers, and then the relevant conclusions regarding the "the effect of the flow-through effect of the fish scale hole opening form". Fig.19 shows the fluid domain model under this structural condition. After the simulation is completed, the velocity nephogram distribution and local velocity vector on the new reference plane are obtained. As Fig.20 and Fig.21 show.

Fig.19 Fluid domain structure model of fish scale hole opening reverse

Fig.20 Fish scale hole opening reversed distribution structure analysis section velocity nephogram

Fig.21 Fish scale hole opening reversed distribution structure analysis section local velocity vector

As can be seen from Fig. 20, when the fish scale hole openings are all reversed, the front end portion of the combustion head no longer has a very small velocity region, but since most of the gas flows out from the fish scale hole at the front end, and the characteristics of the opening of the fish scale hole can be seen from the velocity vector of Fig.21. The airflow flowing from the fish scale hole at the front end has a tendency to converge toward the middle of the combustion head, so that the flow of air between the distribution interlayers, that is, the middle of the combustion head is the largest. And there is still a certain speed surge phenomenon due to the decrease of the tail resistance.

This kind of structure indicates that the relevant conclusions about the influence of the opening direction of the fish scale hole of a certain type burner on the flow of the inflow airflow are correct. This result also provides a certain idea and direction for the optimization design of the fish scale hole structure.

4.4 Fluid domain model analysis of new fish scale hole opening structure

For the analysis results in Section 4.3, a new type of fish scale hole opening method is proposed, as shown in Fig.22. The structural change of the scale hole is structurally changed. The punched portion is completely perpendicular to the plate surface, and the opening is set as a rectangular opening. The single opening area is 12mm², which is consistent with the opening area of the fish scale hole of a certain type of burner, and the rectangular hole size is 6mm × 2mm. The velocity nephogram of the simulation results is shown in Fig.23, and the local velocity vector diagram is shown in Fig.24.
It can be seen from Figures 22 and 23 that under such opening conditions, the velocity distribution between the distribution interlayers is relatively uniform, the flow rate of the inlet location disappears, and although the velocity surge region still exists in the tail, the airflow velocity can be seen from the nephogram. It is not as obvious as a certain type of burner. This is due to the fact that under the opening structure, the flow of gas flows out more in the length of the whole scale of the fish scale, and the flow of the air flowing out of the tail is less. When the resistance suddenly decreases, although the speed increases rapidly, the whole trend is not big.

This structure can not only prove the principle of the influence of the opening form of the fish scale hole of a certain type of burner on the flow of the airflow, but also provide a reasonable airflow distribution structure.

5. Conclusion
Based on the simulation results of a certain type of burner, this paper conducts an in-depth simulation of the overall distribution structure under the same boundary conditions for the actual situation, and carries out related improvement experiments and analysis on the distribution structure according to the successive optimization experience.

Through the above four sets of simulation results, it can be seen that, without changing the structure of the fish scale hole and the strip hole, attempting to open the holes between the fish scale cylinder and the strip cylinder entrance can improve the distribution interlayer to a certain extent. But the improvement effect is not obvious and may cause tempering under operating conditions. Without changing the strip hole structure, only the structure of the fish scale hole is optimized, and the opening area of the local fish scale hole is appropriately increased, can achieve a certain airflow flow optimization effect but the overall optimization effect is not obvious.

The open hole form of the fish scale hole plays an important role in the flow. According to this conclusion, a new type of fish scale hole opening structure is proposed, and the numerical simulation results are also reasonable. It provides a reasonable idea for the design study of the distribution structure in the future. However, it should be noted that the specific optimization parameters of the new structure still need to carry out a large number of numerical simulations and compare the results to obtain a more realistic value.
References

[1] Gas & Heat. (2017) Notice of the National Development and Reform Commission on Printing and Distributing the National Energy 13th Five-Year Plan. J. Gas & Heat. 37(02):1-16.

[2] Fan, H. (2017) Discussion on Some Important Issues of "Coal to Gas" in China. N. 002.

[3] Beijing Municipal Ecological Environment Bureau. (2015) The standard of boiler air pollutant discharge (DB11/139-2015). Z.

[4] Wang, Z.X., Liao, X. Yang, J.J. (2011) Research on fully premixed metal fiber burner. J. Gas & Heat. (06):27-31.

[5] Hu, B., Lin, L.C., Deng, Y.C., Feng, L., Qi, Y.T. (2015) Study on NOx Emission Characteristics of Metal Fiber Premixed Combustion. J. Shanghai Gas. (06):21-23+26.

[6] Bao, T., Zhang, H.L., Miao, Y.W. (2017) Numerical Simulation of Cold Flow Field of Metal Fiber Gas Burner. J. Shanghai Gas. (06):27-31.

[7] Lonardi, A., Lovato, A. (2005) Premix burner combustion head: EP, EP 1538395 A9. P.

[8] Puzzato, D., Scribano, G., Badoka, L., et al. (2008) Burner, especially premixed burner: CN 101861496 A. P.

[9] Liu, F.G., Zhang, R., Zhang, B.L., et al. (2014) Fully premixed gas burner: CN 203687038 U. P.

[10] Dewaegheneire, G., Marrecau, W., Vansteenkiste, P. (2000) Conical surface burner: US, US 6065963 A. P.

[11] Li, L.J., Tao, Z., Pang, J.L., Long, W.J. (2014) Progress in the application of metal fiber burners in the utilization of low concentration coalbed methane. J. 28(09):35-40