Flow Field Simulation and Optimization Design of SCR of 660MW Power Unit Based on CFD Study

Dazhou Zhao [*], Da Lin, Zhongping Zhang

Hua dian electric power research institute co., ltd, China.

*Corresponding author e-mail: seudzz@126.com

Abstract. The uniformity of velocity distribution is the key factor that affects the De-NOx efficiency. A three-dimensional SCR model of a 660 MW power unit is established by commercial software Fluent16.0 to obtain the flow field distribution in the reactor. Some optimization measures are put forward for the optimization of flow field. Finally, the distribution of flow field of AIG (Ammonia Injection Grid) section achieved desired effects.

1. Introduction

Selective catalytic reduction (SCR) as a kind of efficiently NOx control technology has been widely used in coal-fired power plants, which use NH_3 to react with NOx and transfer it into harmless H_2O and N_2 between 300℃~400℃ on the surface of catalyst. Under the conditions of catalyst volume and space velocity been confirmed, the uniformity of both velocity distribution and NH_3/NO mole ratio are the key factors to achieve the desired De-NO_x efficiency [1]. The numerical simulation method has the characteristics of high efficiency, economy and speed, which can provide effective guidance for the setting of SCR [2]. Therefore, the flow field distribution in a SCR reactor of 660MW power unit is studied by numerical method.

2. Numerical Model

2.1. Geometric structure of CFD mode

The object is a SCR system of 660MW coal-fired unit, the size of the inlet section of the reactor is 14000mm×9262mm, and the size of the exit section is 11000mm×3000mm. Plate catalysts are used in SCR system with three layers arrangement. The following two layers of catalyst are arranged, the upper layer is a reserved layer for catalyst. Gambit software is used in grid division and model establishment. The total number of grids of the model is about ten million. The following pictures show the details of the model.

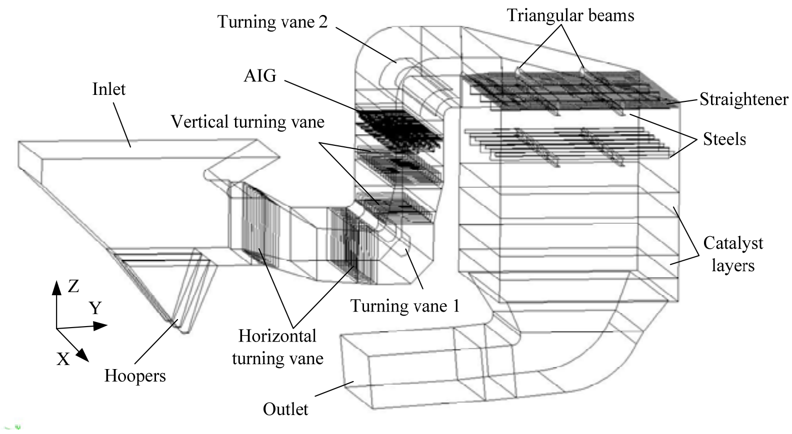

Fig.1. Diagram of the model

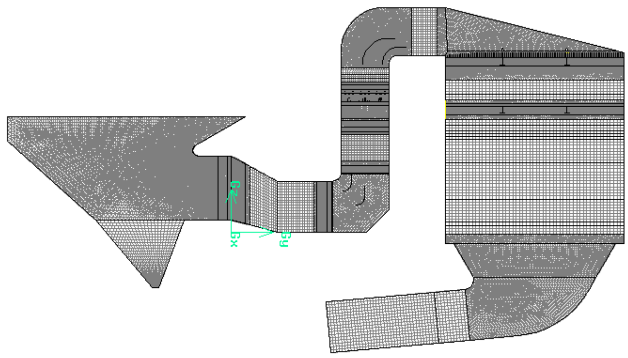

Fig.2. Grid division of the model

2.2. *Mathematics method of CFD model*

The main mathematics models include mass, momentum, energy balances, turbulence and species transport governing equations, which were described as reference [3]. A porous medium model was used to simulate flow in catalyst layers. The commercial software Fluent16.0 was used for simulation.

3. Results and discussion

3.1. *Boundary Conditions*

The boundary conditions are shown in table 1.

Tab1e. Boundary Conditions

Parameter	Values
Inlet velocity (m/s)	4.555
Entrance equivalent diameter(m/s)	11.15
Inlet turbulence intensity(%)	2.83
Inlet flue gas temperature(K)	623

3.2. *Results and Discussion*

The results of the flow field distribution are displayed in Fig.3 and Fig4.

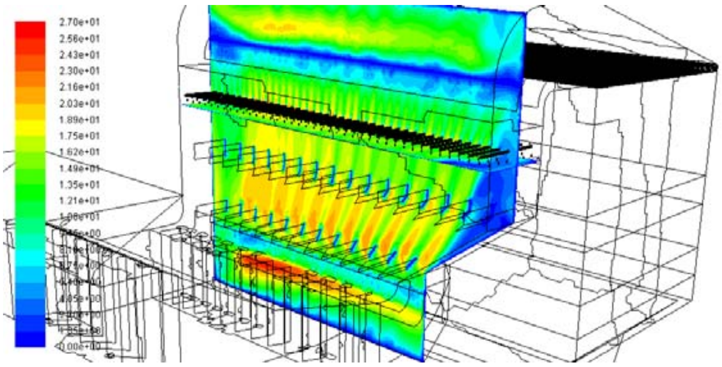

Figure 3. Flow field distribution at Y=8m (m/s)

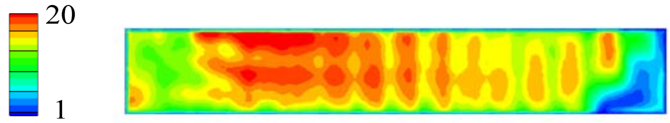

Figure 4. Flow field distribution at AIG section (m/s)

As shown in Fig4, the velocity distribution of AIG section is uneven, the maximum of velocity is about 22m/s, while the minimum of velocity is 0m/s, there are even reflux conditions in some areas. The coefficient of variation of velocity(Cv)is used to express as a percentage of the standard deviation with respect to mean value, Cv is defined as:

$$C_v = \frac{\sigma}{\bar{v}} 100\% \tag{1}$$

Where

$$\sigma = \sqrt{\frac{1}{(n-1)} \sum_{i=1}^{n} (v_i - \bar{v})^2} \tag{2}$$

$$\bar{v} = \frac{1}{n} \sum_{i=1}^{n} v_i \tag{3}$$

The value of *Cv* is 28.5% of the AIG section, however the value of *Cv* is generally set not more than 15% [4]. The velocity distribution has great influence on reactor performance, the more uniform distribution of flow field of AIG section, the more uniform distribution of the ratio ammonia to nitrogen, which can increase the De-NO_x efficiency [5]. Therefore, it is very important to improve the uniformity of velocity distribution of AIG section.

3.3. Optimization measures

The reasonableness of the arrangement of the guide plates is the key factor for improving the uniformity of the velocity distribution. Optimization and reconstruction of the guide plates inside the vertical flue have been taken in this paper. There are 12 guide plates in the original design scheme, and β (the angle between the guide plates and the horizontal direction) vary from 42° to 85°, as shown in Fig5.

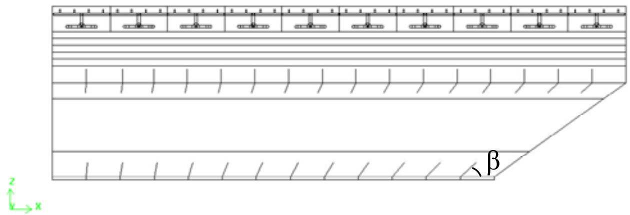

Fig.5. Guide plates inside vertical flue

The value of *Cv* of AIG section is obtained by adjusting both the number of guide plates and the angle between the guide plate and the horizontal direction, the results is shown in Fig.6.

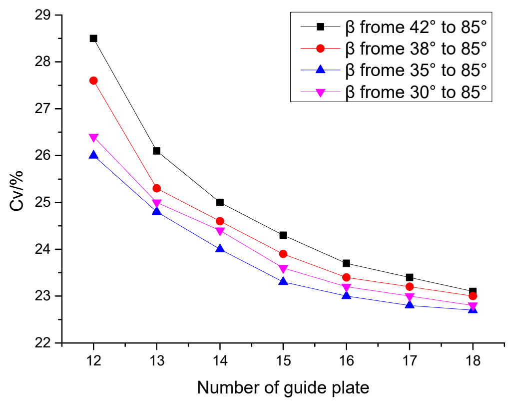

Fig.6. The results of optimization

Fig. 6 show that, as the increasing of the number of guide plates, the value of *Cv* is decreasing step by step, when the number of guide plates is above 16, the value of *Cv* decreases slowly. As the decrease of *β,* the value of *Cv* decreases first and then increases. But, the value of *Cv* is still above 22%. When the number of guide plates is 16 and *β* vary from 35° to 85°, the velocity distribution of AIG section is shown in Fig7.

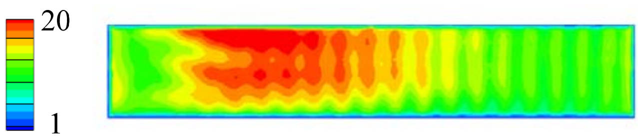

Fig.7. Velocity distribution of AIG section (m/s)

Although the porous plate will increase the pressure drop of the system, but it has good rectifying effect. A porous plate is installed in the vertical flue, which has 50% hole opening rate of the porous plate, and 5mm thickness. Due to the thinner thickness of the porous plate, and small opening size, the porous plate is simplified as a porous-jump model. Its principle can be expressed as formula (4):

$$\Delta p = -(\frac{\mu}{\alpha} v + C_2 \frac{1}{2} \rho v^2) \Delta m \tag{4}$$

Where μ is the laminar fluid viscosity, α is the permeability of the medium, C_2 is the pressure-jump coefficient, v is the velocity normal to the porous face, and Δm is the thickness of the medium.

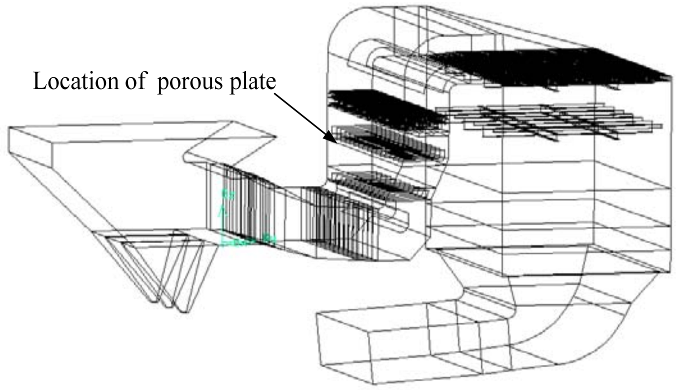

Fig. 8. Location of porous plate

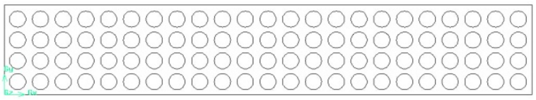

Fig.9. Diagram of porous plate

When using porous plate, the velocity distribution of AIG section was shown in Fig.10, the maximum velocity magnitude is about 18m/s, the minimum velocity magnitude is about 9m/s, the relative deviation coefficient of velocity decreased to 12.5 %. The pressure drop of the system was increased by 180pa.

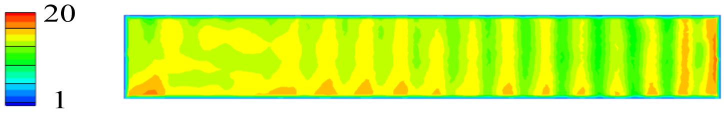

Fig.10. Velocity distribution of AIG section (m/s)

4. Conclusion

A three-dimensional SCR De-NOx reactor model of a 660 MW power unit is established, commercial software Fluent16.0 was used for simulating the flow field distribution in the reactor. The results show that: the velocity distribution of AIG section is uneven, the maximum velocity magnitude is about 22m/s, the minimum velocity magnitude is nearly 0m/s, there are even reflux conditions in some areas, the relative deviation coefficient of velocity is 28.5 %, by optimizing the arrangement of the guide plates in vertical flue, the uniformity of velocity distribution of AIG section is improved, the relative deviation coefficient of velocity is reduced to 23%, when a porous plate having an opening rate of 50 % is arranged, the relative deviation coefficient of velocity is reduced to 12.5 %.

References

[1] Rogers K J,Nolan P S.SCR reactor performance profiling and results analysis[C].The U.S.EPA/DOE/EPRI Combined Power Plant Air Pollutant Control Symposium:"The Mega Symposium",Chicago,2001.
[2] Lei Z,Wen C,Zhang J,Chen B.Selective catalytic reduction for NO removal:Comparison of transfer and reaction performances among monolith catalysts[J]. Ind Eng Chem Res 2011;50(10):5942-5951.
[3] Adams B.Improving design of SCR systems with CFD modeling [C].DOE Environmental Controls Conference,Pittsburgh,2006.
[4] Rogers K,SCR inlet maldistribution-their effects & strategies for their Control[C].NETL 2002 Conference on SCR & SNCR for NOx Control,Pittsburgh,2002.
[5] LEI Da,JIN Baosheng, Influence of the Flue Gas Velocity Field at an Ammonia-injection Grid on Uniform Flows and Reducing-agent Mixing Performance of a High-efficiency SCR(Selective Catalytic Reduction) Device[J]. Journal of Engineering for Thermal Energy & Power, 2009, (1):113-119.