Research on the Airflow Distribution in the Fabric Area of Electrostatic Fabric Precipitator

Chaofan Sun[1,a], Xinglu Yu[1,2,b], Xinfeng Long[2,c], Bo Lou [2,d]

[1] Electric Power Research Institute of Guangdong Power Grid Corporation, CSG, Guangzhou, 510080, China

[2] South China University of Technology, Guangzhou, 510640, China

[a]sun-cf@163.com, [b]yuxinglu66@163.com, [c]cexflong@scut.edu.cn, [d]loubo@scut.edu.cn

Abstract: Numerical simulation method was used to calculate the airflow distribution in the fabric area. We established the 3-D structure for Electrostatic Fabric Precipitator(EFP) of 600MW unit by the Pro/Engineering software and dealt with the data collecting from the simulation with the Tecplot 360. Data about airflow distribution in the filter area were obtained by comparing different operation conditions. In the filter area, the biggest pressure is 530Pa and the speed in the bags was changing from 1m/s to 4m/s. Bags located at the front of fabric area are tend to be damaged due to the fast flue gas and those bags in the second filter area would be used for more time because of the smaller flux than the first one. The numerical simulation results are reasonable and could be used as the reference to the designing of Electrostatic Fabric Precipitator.

Keywords: numerical simulation; filter bag; airflow distribution; electrostatic fabric precipitator

Introduction

Currently, the rules for the control of soot particle produced by power plant is becoming more and more severe in China. In order to reduce the volume of gas, many manager chose the Electrostatic Fabric Precipitator(EFP) to removal ash in the flue gas because of its good performance. In the past, many researchers had made a lot achievements on the study of Electrostatic Precipitator or Fabric Filter. In the contrary, the number of papers about EFP was less than those. As we know, Electrostatic Fabric Precipitator is one kind of new filer which combined the advantage of Electrostatic Precipitator with the Fabric Precipitator. However, the mechanism of Electrostatic Precipitator is different from the theory of Fabric Filter[1],[2] and the airflow is also different from each other. The airflow direction of Electrostatic Precipitator is horizontal while that of Fabric Filter is vertical. People are lack of knowledge about airflow distribution due to the less research on the EFP, especially the research on the airflow uniformity of Fabric area of EFP. According to the theory of Electrostatic Fabric Precipitator, about eighty percent of particle in the dust would be removal in the Electrostatic area, and twenty percent of particles in the flue gas would be intercepted. In other words, most of particles whose diameter were between 10μm and 1μm could be intercepted because of the function of Fabric[3]. In this way, the pressure of operation and cost of equipment would decrease. In order to gain one kind of uniform airflow, many measures have been taken. Now three flow deflectors were placed in the inbox of EFP in the purpose of uniform airflow distribution. Therefore, influence on the Fabric area has not been researched. So this paper puts much more attention to the airflow distribution in the Fabric area.

If the airflow distribution in the Fabric area was not uniform, a lot of bags would be destroyed, and the efficiency of Fabric area would decrease, which would affect the operation of EFP.

Numerical Model

In the numerical simulation, the boundary of bags was defined as the porous-jump. In the CFD, the porous medium model is used in the simulation of models with many holes in one face or body. In this model, we could set permeability as the percentage of opening. As we know, the momentum equation of porous medium model has a momentum source in the numerical model. The momentum source is consisted of two parts, one is viscosity losses (Darcy), and another is internal losses[4].

$$S_i = \sum_{j=1}^{3} D_{ij}\mu\upsilon_j + \sum_{j=1}^{3} C_{ij}\frac{1}{2}\rho|\upsilon_j|\upsilon_j \quad (1)$$

Where, S_i - i direction momentum source; D and C are matrix.

In each porous medium model, the momentum losses is contribution to the pressure gradient, the pressure drop and fluid speed (or speed gradient) are in proportion [5].

For the simple uniform porous medium

$$S_i = \frac{\mu}{\alpha}\upsilon_i + C_2\frac{1}{2}\rho|\upsilon_j|\upsilon_j \quad (2)$$

Where, α-permeability; C_2 – internal resistance factor; D and C are diagonal matrix 1/a and C_2, others are 0.

When the airflow was laminar flow, the pressure drop and velocity are in proportion. In the equation (2), C_2 could be set as 0, ignoring the convection acceleration and spread. We could simple the porous-jump as the Darcy rule:

$$\triangle P = -\frac{\mu}{\alpha}\upsilon \quad (3)$$

When the speed of flow in the porous medium was high, the C_2 in the equation (2) could adjust the internal losses. This constant could be regard as the loss coefficient in each unit length whose direction is along with the flow direction. So we could regard the pressure drop as the function of pressure[6].

Therefore, we could call the porous medium with one dimensional model as porous jump, which is used in the simulation of film whose pressure drop and speed have been known. This method is much more reliable than porous medium. As the thickness of film is limited, we could define the connection between Darcy rule and internal losses by the change of pressure.

$$\triangle P = \left(\frac{\mu}{\alpha}\upsilon + C_2\frac{1}{2}\rho\upsilon^2\right)\times\triangle m \quad (4)$$

Where, △p – pressure losses, Pa, μ – fluid dynamic viscosity coefficient, Pa • s; α – permeability, m^2, which is used as the coefficient to define the permeability strength of porous medium; C_2 – resistance coefficient; υ – velocity, m/s, which is perpendicular to the filter medium surface; △m – the porous medium thickness, m.

3-D structure model of EFP

This paper is focus on the airflow distribution in the fabric area of large scale EFP, which is designed for the 600MW power plant. Every unit has two EFPs, and we use one of them as the model to calculate in the purpose of gaining the results quickly. In this model, each EFP has 4 electrostatic fileds and 2 fabric areas. The 3-D structure model of EFP has been established with the software called Pro/Engineering. The inlet was 4.2m wide and 4.1m long. In the electrostatic field, the height is 14.6m, the width is 15.4m and the length is 9.8m. While the fabric area is 15.4m wide and 6.3m long, but the height is different from each other, one is 13.3m and the other is 11.1m. Also, 250 bags have been placed in the fabric area, and the number in total is 500. Each bag is 8m long and the diameter of bag is 300mm.

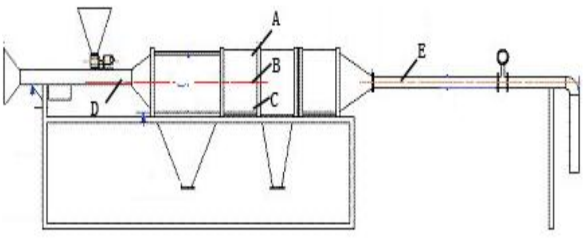
Meshing and Setting of boundary

In order to get a better simulation, the inlet box was meshed by the hexahedron in consideration of the symmetry of inlet velocity, some tetrahedrons were also used to mesh the flow deflectors, in the meanwhile, the electrostatic field was meshed with asymmetric grid in order to control the density of local grid. In the fabric area, every bag has been mashed with 770 tetrahedron grids, and the total number of grid in the fabric area is 5,100,000. The grid test result shows that the grid meshing is satisfy the computational accuracy.

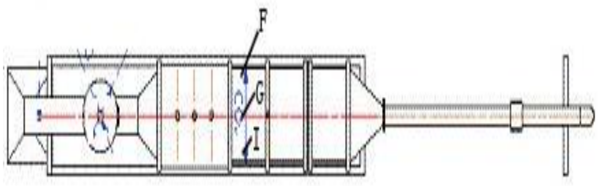
In the setting of boundary condition, we define the inlet as the velocity-inlet, and outlet was defined as the pressure-outlet, in the meanwhile, define the flow deflector as wall, bags as the porous-jump. According to the data collected from the power plant, the speed in the inlet was 14.38m/s, the operation temperature was 20℃, hydraulic diameter is 4.149m, and the turbulence intensity is 3.3%. Also we have chosen the K-ε equation as the computing method. And the permeability of bag is 1.5×10^{-10}, the thickness of bag is 1mm. We supposed that the fluid in the EFP is steady and the time has nothing to do with the motion parameters. And the fluid cannot be compressed in consideration of its low speed. Also the whole simulation process is isothermal.

Verification of Numerical simulation

One test model of EFP is established in this paper in order to verify the numerical simulation. 8 points (in Fig.1) have been chosen to test their velocity and pressure.



(a)



(b)

Fig. 1 Position of measuring points

In this model, the quantity of flow is 230m^3/h. As we see in the tab.1, the big error is 9.79% and the small one is only 0.64%. These are the types of errors caused by human error or fraud.

Tab.1 velocity and error

Items	V_{inlet}	V_A	V_B	V_C	V_F	V_G	V_I	V_{outlet}
Test (m/s)	5.42	0.20	0.70	0.82	0.55	0.60	0.55	12.72
Simulate(m/s)	5.50	0.2586	0.776	0.776	0.5465	0.5834	0.5465	13.00
absolute error	0.08	0.0586	0.076	-0.044	-0.0035	-0.0166	-0.0035	0.28
relative tolerance	0.0145	0.2266	0.0979	-0.0567	-0.0064	-0.0284	-0.0064	0.0215

Also more information could be gained in Fig.2. The velocity of inlet and outlet are uniform, the average velocity is 0.5m/s and 13m/s. It meets conservation law in value. In the airflow area, the velocity changes but its distribution is symmetrical. In electrostatic area, the smallest velocity located at the middle, which is only 0.2586m/s. At the same time, the biggest one is 0.7760m/s, which is located at the bottom. As we know, when flue gas cross the airflow area, it tends to flow along the border.

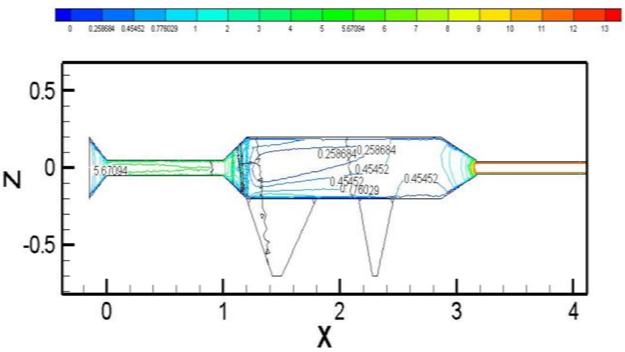


Fig. 2 Y=0 Velocity contour of EFP test model

Pressure changes when flue gas flow in the EFP. As we see in Fig.3, the pressure of EFP is subpressure. It is -1000pa in the outlet and the ash bucket is 0. The results of numerical simulation matches up with the date of test model.

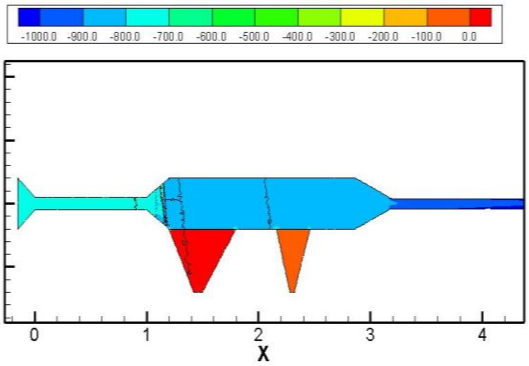


Fig.3 Pressure contour of EFP test model

We could make one conclusion that the numerical simulation could calculate the velocity distribution of EFP in this paper.

Simulation and Discussion

This paper focus on different operation conditions on the purpose of airflow distribution in various places. This data include the impaction to the bags.

1. Pressure Contour

Data about the pressure in different positions were calculated in the computer. We draw several plots about pressure contour when it was x=10, 13, 19. By comparing these plots, we could make a conclusion that when airflow went to the faces of bags, the biggest pressure was produced in the middle of filter area, while a small pressure existed in the left and right side of the filter area, which was mainly caused by the flow deflector located at the middle of EFP. The biggest one is about 530Pa. And a bigger pressure would lead into the damage of bags. In the vertical direction, pressure in the gas-collecting chamber was only 350Pa and the airflow in the bags were uniform. However the pressure in the below of bags are smaller than it in the face of bags. We can explain this phenomenon by the theory of resistance.

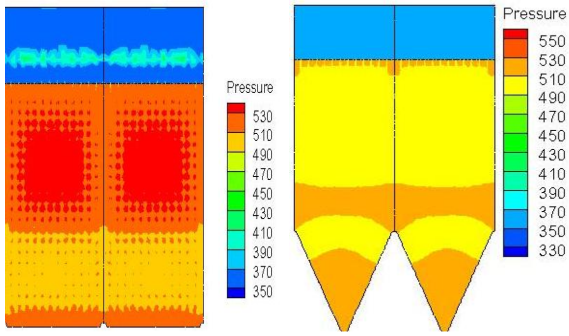


Fig.4 The pressure in the X=10and X=13

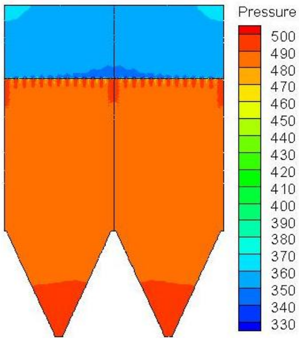


Fig.5 The pressure in the X=19

Comparison of pressure in the horizontal direction was made. We chose 3 figures shown in the follow pictures which were located at Y=0.1, 0.3, 7. Conclusion is made by comparing data, pressure in the filter area becoming small as well as the position of airflow. Especially when it is in the middle of fabric area, the pressure become bigger because of the flow deflector located in the front of the second filter area.

We could gain some results by analysis the pressure distribution that some structures whose pressure were greater than others should be optimized.

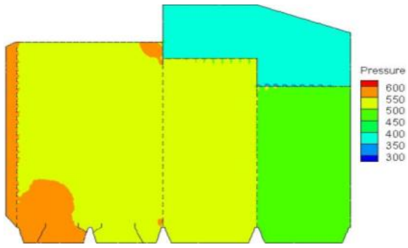


Fig.6 The pressure of Y=0.1

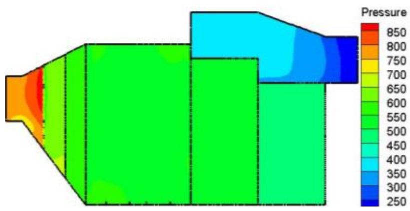


Fig.7 The pressure in Y=0.3

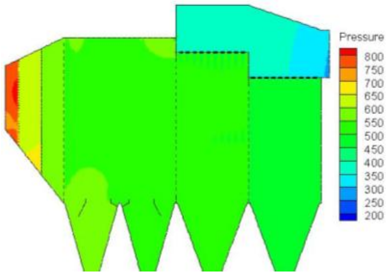


Fig.8 The pressure in the Y=7

2. *Velocity Contour*

Three Figures were drew to gain the detail data about velocity. These velocity contour figures were located at Z=-6, 0, 5. When it was Z=0, the speed of airflow in the bags was 1m/s but the velocity located in the flow deflector added to 3m/s. These figures also show that the flow deflectors could reduce the damage of airflow on the bags effectively. And the speed in the boundary of filter area was smaller than it in the middle.

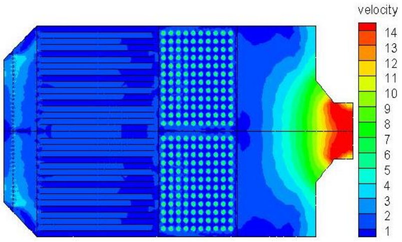


Fig.9 The velocity in the Z=-6

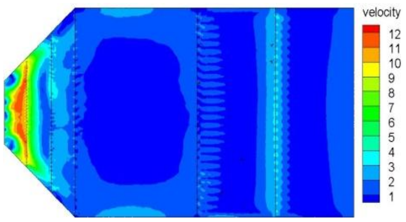


Fig.10 The velocity in the Z=0

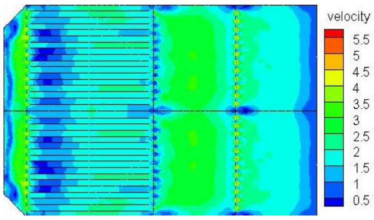


Fig.11 The velocity in the Z=5

In the meanwhile, velocity in the X direction was changing, the higher position, the bigger speed. Velocity in the bags was bigger than it in the gap of bags. For more information, we drew other 6 plots. We could make one conclusion that bags closed to the flow deflector are tend to be damaged because of the bigger speed. As we see in the Figures, the biggest speed was 4m/s located in the entrance of flow deflector. And bags in the second filter area would be used for more time due to the smaller flux than the first fabric field.

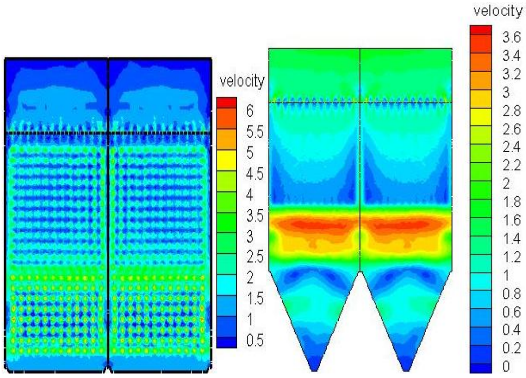


Fig.12 The velocity in the X=10 and X=13

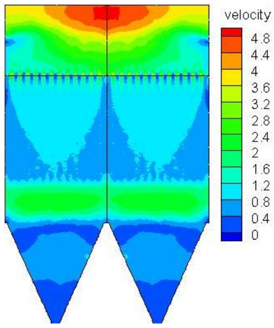


Fig.13 The velocity in the X=19

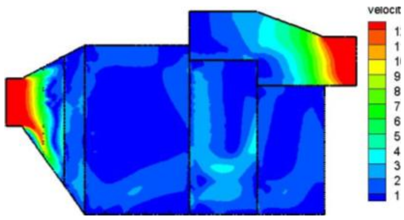


Fig.14 The velocity in the Y=0.1

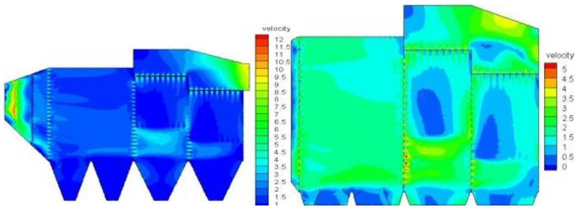


Fig.15 The velocity in the Y=0.3 and Y=7

3. *Velocity Vector*

In this part, three vector plots are drawn. See Fig.16 and Fig.17, the airflow went across the electrostatic fabric precipitator as the X direction, but some reflux would be generated due to the boundary effect. Also most of gas came out from the bag export, and the velocity was bigger than it in the space in the filter area. An eddy may produce at the end of baffle.

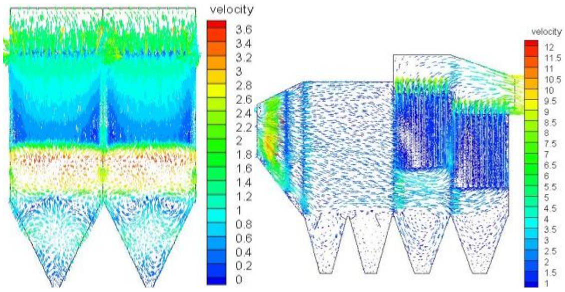


Fig.16 The velocity vector in the X=13 and Y=0.3

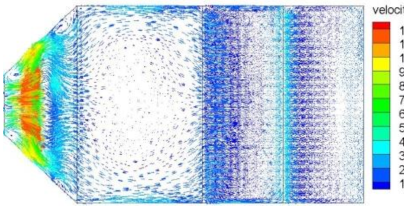


Fig.17 The velocity vector in the Z=0

4. Pressure distribution of single bag

Some specific information is obtained in the numerical simulation. As we know in the Fig.18, the airflow produce different pressure on the bags located in different rows, the biggest one is 510Pa, which is different from it in the filter space. The pressure in the exit is 460 Pa.

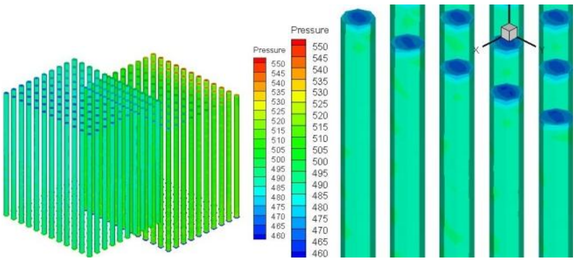


Fig.18 The pressure on the single bags

Conclusions

A) Some results could be obtained quickly and accurately by the method of numerical simulation. And some detail data could be gain by this way. Numerical simulation could reflect the airflow distribution in the fabric area which is beneficial to the design of EFP.

B) As we see in these figures, airflow distributions are nonuniform in the bags, and different pressure would lead to different damage on the face of bags, which would affect the operation of electrostatic fabric precipitator.

C) The biggest velocity in the fabric area is 4m/s and the low speed is only 0.5m/s. Also we could make one conclusion that the flow deflector should be optimized to adjust the airflow distribution.

References

[1] J.J.Yu, Z. Ying, S.G. Gang, W. Jun. Study of the separation efficiency and the flow field of a dynamic cyclone. Separation and Purification Technology. 49(2006): 157-166.

[2] Chaofan.Sun, Xinglu.Yu,Xinfeng.Long, Bo.Lou.Study on the operation characteristics of Electrostatic Fabric Precipitator. Advanced Materials Research Vols. 616-618 (2013) pp 2013-2016.

[3] L.Z. Wei, Y.Qiang. Numerical simulation of the flow and the collection mechanism inside a scale hybrid particulate collector. Power Technology. 215-216(2012): 26-27.

[4]K.S.P. Nikas, A.A. Varonos, G.C.Bergeles. Numerical simulation of the flow and the collection mechanism inside a laboratory scale electrostatic precipitator. Journal of Electrostatics. 63(2005): 423-443.

[5] H.K. Versteeg, W. Nalalasekera. An Introduction to Computational Fluid Dynamics: The Finite Volume Method[M]. London (1995).

[6] J. Subrenat, P.L. Cloirec. Chemical Engineering Sci. 4965-4973(2003), p.58.