Numerical simulation of the airflow distribution of WESP

Li Haibo, Yin Cheng, Ren Kai, Guo Chun, Yin Yonghui

Xi'an Xikuang Environmental Protection Co., Ltd, Xi'an 710075,PR China
Corresponding author: *1399281701@139.com*

Abstract The article describes the airflow distribution numerical simulation process of wet electric precipitator (WEPS). The numerical simulation used the SIMPLE algorithm and turbulence model as the model. The monitoring section of the airflow distribution is set at the 100mm distance from the top plane of the anode tube bundle. It can analyzed the internal air flowing condition. By means of the reasonable arrangement of guide plate and the circular hole rate of distribution plate, the airflow distribution on the monitoring section meet the requirements. The results of numerical simulation can provide suggestions for the design of WESP.

Keywords: CFD, Wet electric precipitator (WESP), Airflow Distribution, Numerical Simulation

1. Foreword

Along with the continuous progress of science and technology and the rapid development of social economy, a series of environmental problems have appeared. Cement, metallurgy, electric power, papermaking, and chemical industry are all over china, environmental pollution problems are becoming more serious [1]. The national introduced more stringent standards of the e mission of atmospheric pollutants more stringent standards [2], and the wet electrostatic precipitator is an important means to solve the problem of low emission. It will be used more and more widely.

The uniformity of air distribution is an important factor affecting the efficiency of dust removal for WESP [3]. Usually, the electrostatic wind speed is homogeneous in the shell. But if the air into the shell is not well distributed, the electrostatic wind speed will be deviated greatly from the design average wind speed, even no air flow in some places, thus reducing the specific surface area of the anode tube bundle actually. And the flow rate is too fast in the anode tube bundle at the higher velocity zone, so the charge containing fog droplets can't adsorbed onto the anode tube timely and effectively. These two factors will greatly affect the efficiency of WESP.

Currently, there are two kinds of methods to research the distribution of air flow are mainly model test and numerical simulation at domestic and abroad. Model tests can display the airflow distribution of electric field profile clearly and the result is credible. But numerical simulation can obtain the numerical solution of the flow field and the result is intuitive. Therefore, the numerical simulation of airflow in the scheme design process can avoid the blindness of the design process and ensure the performance of the product. In recent years, the mainly method which researched on the dust collector at domestic is computational fluid dynamics. And great progress has been made in the numerical calculation of the airflow distribution of the WESP [4-6].

2. Numerical Simulation

It has illustrated the application of the numerical calculation of airflow distribution by the computational fluid dynamics (CFD) software FLUENT. The calculation process is shown in figure 1:

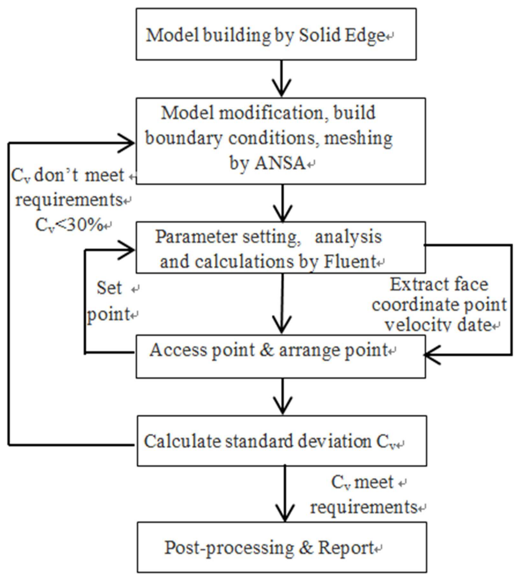

Figure 1. The calculation process

2.1. Computational model

Taking the WESP which after the desulfurization of 410t boiler for example. The anode plate of the WESP adopted the hexagonal tube type, inlet port is upper intake, the horn import is from round change to square, the dimension of round mouth is ∅3800 mm, the size of square shell is 11000×11000 mm, the height of the horn import is 3200 mm. The monitoring section of the airflow distribution is arranged at the 100mm distance from the plane of the anode tube bundle in the shell. After several sets of numerical simulation comparison, taking into account the requirements of the overall resistance of the device, the size of "#" -shaped guide plate and the circular hole type distribution plate combination structure are determined. Figure 2 shows the numerical calculation model of WESP.

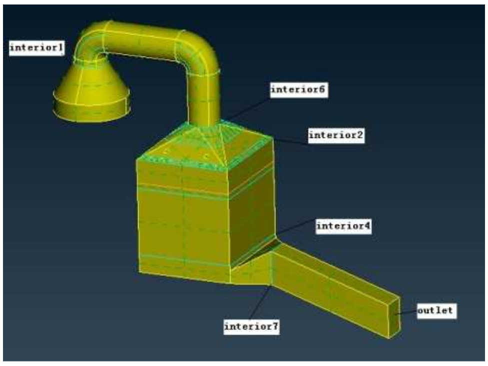

Figure 2. Numerical model

2.2. Controlling equations

During the numerical simulation, it is assumed that the fluid is incompressible and steady flow, the whole process is constant temperature. The controlling equations of the CFD with mass conservation equation (continuity equation), momentum conservation equation, energy conservation equation (k turbulent kinetic energy equation and ε turbulent kinetic energy dispersion rate equation).

2.3. Model processing and boundary conditions

It's preprocessed the model by ANSA, the geometric model and the actual WESP geometry size ratio of 1:1. In order to reduce the influence of the pipeline on the inlet flow, a complete inlet pipeline model was built and the partial FGD absorbing tower model was added, increase the flow guide plate to rectify the airflow. Build a complete export pipeline model on the outlet. In order to reduce the number of meshes and shorten the computing time, all the mesh size should not be less than 200 mm except on both sides of the round hole distribution plate must be refining. Inlet is speed inlet and outlet is pressure outlet; the shell and flow guide plate are treated with no slip wall. The internal pressure plane of the shell is considered to "interior-face". The numerical calculation used the SIMPLE algorithm and the turbulence model is k-ε two equation models. The grid section is shown in figure 3.

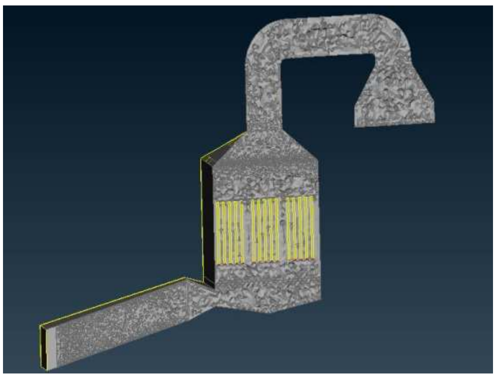

Figure 3. Grid section

2.4. Solve

The convergence of the calculation can be observed by the residual curve. When the iterative residual error is less than the 10^{-3}, it is regarded as the convergence, and the FLUENT stop calculation.

3. Post-processing

Currently, the evaluation parameter of airflow distribution is relative root-mean-square σ, the following is the evaluation standard of the uniformity of parameter:

$$S_r = \sqrt{\frac{1}{n-1}\sum_{i=1}^{n}\left[\frac{V_i - \bar{V}}{\bar{V}}\right]^2} \qquad (1)$$

In formula:

V_i – flow rate on the measuring point (m/s);

$\bar{V}$ – average velocity of cross section (m/s);

n – the number of points on the cross section.

Calculated the relative standard deviation of each test section S_r, and then according to the table 1 look up the values of ΔW (Reaching speed deviation), when $\Delta W \le 12\%$ as qualified.

Table 1.

S_r	5	10	15	20	25	30	32	35	40
ΔW	0.5	1.3	2.5	4.6	7.1	10	12	15	18

After adjust the import pipeline guide plate, the horn mouth inlet airflow uniform distribution; repeated adjustment the size of "＃" - shaped guide plate and the circular hole rate of round distribution plate, make the air velocity distribution optimization. By calculation, the average velocity in the WESP shell is 1.852 m/s, the root mean square value of the velocity distribution is 21.7%, It indicates the airflow distribution is good. The structure of "#" - shaped flow guide plate and circular holes type distribution plate meet the requirement of air distribution. Table 2 shows the value of air flow velocity distribution in the plane of above the anode tube bundle 100 mm. Figure 4 and Figure 5 show the longitudinal and transverse section velocity cloud picture respectively.

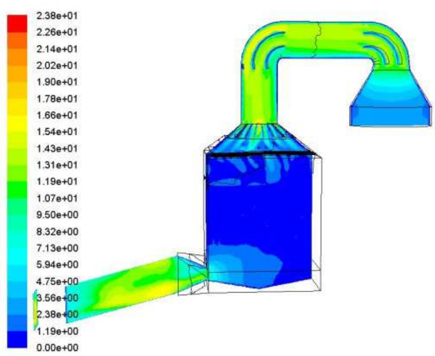

Figure 4. Longitudinal section velocity cloud picture

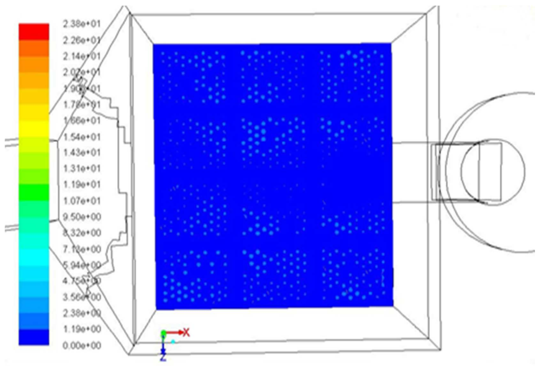

Figure 5. Transverse section velocity cloud picture

Figure 6 is the outlet horn streamline diagram.

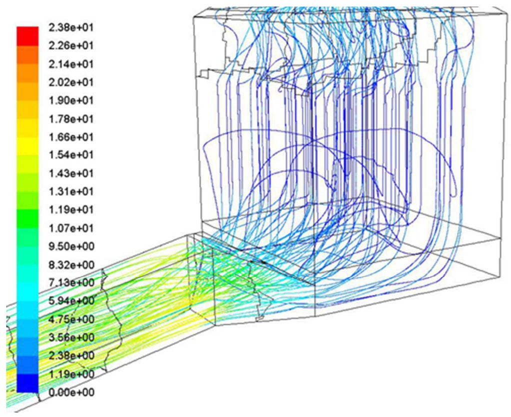

Figure 6.Outlet horn streamline diagram

Table 2. Air flow velocity distribution in the plane of above the anode tube 100mm

Left Chamber1	Section Velocity (m/s)			Lateest 4a						Relative Standard DeviationSr			21.70%
	Row1	Row2	Row3	Row4	Row5	Row6	Row7	Row8	Row9	Row10	Row11	Row12	Mean Value
Line1	2.346	2.836	1.893	2.353	1.762	1.780	1.597	1.777	1.594	2.576	2.307	1.802	2.052
Line2	1.615	2.470	2.047	2.145	1.981	1.941	1.810	1.896	2.208	1.783	1.514	1.615	1.919
Line3	1.653	1.944	2.444	2.094	2.368	2.782	2.138	2.126	1.839	2.150	1.673	1.761	2.081
Line4	1.959	2.307	2.448	2.050	1.971	3.003	1.621	1.991	2.182	2.101	1.768	2.138	2.128
Line5	1.465	1.432	1.634	2.158	1.811	1.659	1.266	1.609	1.414	1.038	1.404	1.479	1.531
Line6	1.348	1.946	1.994	1.967	1.385	1.426	1.460	1.598	1.501	0.952	1.002	0.847	1.452
Line7	1.620	1.676	1.887	1.740	1.242	1.373	1.402	1.514	1.631	0.991	1.038	1.243	1.446
Line8	1.667	1.313	1.673	2.504	2.347	2.483	2.154	1.601	1.474	1.519	1.579	1.847	1.847
Line9	1.804	1.956	2.521	2.332	2.116	2.324	1.815	1.472	2.171	1.941	1.330	1.523	1.942
Line10	1.340	1.832	1.869	2.166	1.938	2.555	1.820	1.943	1.948	2.262	1.624	1.837	1.928
Line11	1.468	1.935	1.872	2.074	1.587	1.776	1.649	1.754	1.651	1.613	1.730	1.965	1.756
Line12	1.805	2.435	2.140	2.428	2.113	2.140	1.640	1.975	1.770	2.661	2.605	1.966	2.140
Mean value	1.674	2.007	2.035	2.168	1.885	2.104	1.698	1.771	1.782	1.799	1.631	1.669	1.852

It can be seen from the outlet horn streamline diagram, the streamlines are smooth, transition velocity is uniform, the whole resistance in the shell and pipeline is 316 Pa, conforms to the resistance of the equipment requirements, so there is no need to add guide plate in the outlet horn.

4. Conclusion

- By way of CFD numerical simulation, we can get basic physical quantity distribution on various locations in very complicated flow field (such as the distribution of velocity, pressure, temperature, concentration, etc.), and these physical quantities changed with time, determine the vortex distribution characteristics, cavitation characteristics and flow separation zone etc. CFD can visually display the flow field of the precipitator. For the high requirements of airflow distribution of the WESP, CFD calculation can avoid the blindness of design and provided the basis for design of the airflow distribution device.
- CFD numerical simulation technology had become increasingly concerned as a new method to study the air flow. With the continuous improvement of technology, CFD will become an important ways to research on the gas flow in the WESP.

References

[1] Qi Juntian. etc., Modern flue gas dust removal technology [M]. Chemical Industry Press, 2008.

[2] GB13223-2012 Emission standard of air pollutants for thermal power plants.

[3] Gao Jianbin, Main influencing factors of wet electrostatic precipitator .Vinylon communication 2009.

[4] Wang Fuju, Computational fluid dynamic analysis [M]. By W.H.Inmon, 2004.

[5] Han Zhanzong, Example and application of FLUENT fluid engineering simulation. Beijing Institute of Technology press, 2010.

[6] Dang Xiaoqing, Li Qianjing, etc., Research on numerical calculation method of air distribution of electric dust collector . Proceedings of the first meeting of the Fourth Committee of the environmental protection branch of China Mechanical Engineering Society[C], 2008.