Numerical analysis of the effect of turbulence on particle agglomeration

Weikuo Zhang\(^1\), Guanqing Liu, Chongen Zou

\(^1\)China Huadian Research Institute of Science and Technology. Beijing, 100170, China
\(^*\)Corresponding author’s e-mail: weikuoz@163.com

Abstract. The turbulent agglomeration characteristics of a mechanical elements were investigated by numerical simulation using the SST k-\(\omega\) model. The vorticity field, turbulent dissipation rate and pressure drop in the flow field were obtained. The results show the bluff body flow phenomenon occurs when the incoming flow passes through the spoiler element. On the back of the element, a periodic vortex similar to Karman vortex street is formed. After the airflow passes through the turbulence element, the turbulent dissipation rate in the main flow area is higher, and the particle agglomeration effect is significant. In the vortex street period, due to the viscous dissipation, the dissipation rate decreases exponentially with the increase of the propagation distance, and the agglomeration effect decreases. Due to the resistance of the turbulence element, the pressure drop between the inlet and outlet is up to 143pa, resulting in a large pressure loss.

1. Introduction

Particles in the atmosphere have become one of the main hazards affecting the world's environment and human health\(^1\), and the removal of particles has always been the focus and difficulty of pollution control in coal-fired power plants, especially the fine particles have strong penetration and are difficult to capture. In the past, the main particle pollution control technologies include the following methods\(^2\-\(^3\): mechanical, electric, wet, fiber bag and agglomeration dust removal, etc. Among them, agglomeration dedusting refers to promote the agglomeration of fine particles into larger particles, which are then removed by the dust collector. Turbulent agglomeration technology is an effective method for particle removal due to its simple structure and low cost. Turbulent agglomeration technology has attracted more and more attention, and related researches have been carried out.

Turbulent agglomeration exists widely in natural environments and industrial processes, such as the formation of soot in internal combustion engines\(^4\), the growth of nano-grains, and the condensation and growth of raindrops in high clouds. The turbulent agglomeration device has been studied and partly put into practical application. Zhang et al.\(^5\) carried out experimental and numerical simulation research on a structure of Indigo's commercial agglomerator based on Fluent using the RANS + DPM model, and explored the effect of component arrangement and particle concentration on agglomeration efficiency. Liu et al.\(^6\) studied the influence of particle trajectory and particle charge in a agglomeration device similar to the Indigo structure using k-\(\varepsilon\) combined with DPM model. Kalt\(^7\) pointed out that in shear turbulence with appropriate scale and intensity, significant agglomeration between coarse and fine particles could be achieved, but the agglomeration effect was not obvious when only small particles existed. Liu\(^8\) studied the effect of cylindrical turbulence device on agglomeration effect based on group equilibrium method, and found that although the pressure loss was low, the turbulent dissipation rate was small, resulting in limited agglomeration effect. However, there are still a lot of problems to be
solved in the research on placing of turbulence generated devices in flow field to improve the effect of turbulent agglomeration and reduce the energy loss.

2. Numerical model
The study adopts the numerical simulation method. The velocity of incoming flue gas is 15m/s and the temperature is 403K. The fluid can be treated as incompressible. The theoretical model is as follows:

**Continuous equation**

\[
\frac{\partial \rho}{\partial t} + \nabla (\rho \vec{V}) = 0
\]

**Momentum equation**

\[
\frac{D\vec{V}}{Dt} = \vec{f} - \frac{1}{\rho} \nabla P + \nu \nabla^2 \vec{V} + \frac{1}{3} \nu \nabla (\nabla \cdot \vec{V})
\]

The SST k-ω model combines the advantages of the k-ω model and the k-ε model, including the modified turbulent viscosity formula and effects of turbulent shear stress; SST k-ω model generally can more accurately simulate the separation point and separation zone caused by the adverse pressure gradient. The specific details of this model can be found in the literature [9].

The structure of the spoiler element is shown in Figure 1. The height of the element used in calculation is 0.045m and the headwind angle is 90 degrees. In order to save calculation time, the device is simplified into a two-dimensional structure, and the flow field of flue is approximately treated as two-dimensional. Only one element is selected for numerical simulation. The calculation area is 1.9m*0.19m, in which the width of 0.19m is the height of four elements. In order to prevent the influence of backflow on the flue turbulent flow field, the computational domain length \( L = 1.9 \)m. The periodic boundary condition is used at the upper and lower boundaries, the inlet uses average velocity, and the flow velocity is equal to the wind speed in the actual power plant flue. The equivalent pressure gradient boundary condition is used at outlet. and the wall function is used for the spoiler element wall.

![Figure 1. Schematic of the mechanical element](image)

Because of the special shape of the element, the unstructured grid is used in the simulation. In order to accurately capture the turbulence vortices generated by the element shadowing effect, the mesh is partially encrypted near the element, and the sparse grid is used in the area far from the element. The total number of grids is about 330,000 and the time step is \( 5 \times 10^{-5} \)s. The Fluent software is used to calculate the flow field.

3. Results and discussion
We first study the transient process of turbulent flow formed by the element shadowing effect. Figure 2 shows the transient vorticity field of a single spoiler element with 90° headwind angle. It can be seen that the bluff body flow phenomenon occurs when the incoming flow passes through the spoiler element. The incoming flow is affected by the resistance of the bluff body, and the energy is lost during the flow. A part of the kinetic energy must be taken out to increase the pressure, which causes the speed to gradually decrease to zero, that is the separation point. The airflow accumulates here. The pressure drop continues to rise after this point, forcing the stagnant fluid to reverse the flow and squeeze the mainstream away from the object wall, causing the boundary layer flow to separate, and the vorticity...
also reaches the maximum. A series of vortices are formed at the downstream of the bluff body, and the vortex intensity decreases with the flow propagation. In addition, by comparing the vorticity field in [5] with the same initial conditions, it is found that the overall tendency of the flow field distributions is in good agreement with our results, so the numerical simulation results in this paper are varied.

![Instantaneous vorticity field](image1)

**Figure 2. Instantaneous vorticity field**

In order to analyze the effect of turbulence on particle agglomeration efficiency, Figure 3 shows the transient turbulent dissipation rate in the flow field. The parameter related to the flow field of the classical collision kernel function\(^{10}\) for turbulent agglomeration is the turbulent dissipation rate, which plays a major role in controlling the particle collision rate. The turbulent dissipation rate is usually measured by the turbulence energy lost per unit mass of fluid in one second. The turbulent velocity fluctuates randomly in space, thus forming a significant velocity gradient. Influenced by molecular viscous force, turbulence kinetic energy is continuously transformed into kinetic energy of molecular motion through internal friction.

It can be seen from Figure 3 that after the airflow passes through the spoiler element, the region with the highest turbulent dissipation rate is at the back of the element, and then gradually diffuses to the mainstream area. The turbulent dissipation rate of the mainstream area is also relatively higher, and the distribution range of the high turbulent dissipation area is wide, which can effectively increase the particle agglomeration effect.

![Instantaneous turbulent dissipation rate](image2)

**Figure 3. Instantaneous turbulent dissipation rate**

It can be seen from Figure 1 that the vortex distribution is similar to Karmen vortex street, which is periodic. By monitoring the lift of the flow field, it is found that the cycle of the vortex street is \( t = 0.0145s \). we studied the mass weighted average turbulent dissipation rate of the flow field. At the beginning, due to the influence of the unsteady inlet flow, the monitoring time is greatly different from the vortex period, so the average dissipation rate fluctuates with the flow time; when the monitoring time is equal to the vortex street period, the average turbulent dissipation rate of the flow field approaches constant.

The average turbulent dissipation rate in the vortex period is relatively rough, so it is necessary to reasonably divide the entire area into a series of small regions to analyze the accurate distribution of the average turbulent dissipation rate. It can be seen from Figure 4, when the distance to the spoiler is very short, due to the strong vortex produced by boundary layer separation, the turbulent velocity is fast, lead to large dissipation rates. With the increase of propagation, because of viscous dissipation, the turbulent vortex velocity is greatly reduced, resulting in an exponential decrease of the dissipation rate.
The optimal design guideline for the structure and position of the spoiler element is to enhance the agglomeration effect of the main flow area and reduce the pressure drop. So we need to consider not only the effect of turbulent agglomeration, but also the effect of spoiler elements on pressure drop.

Figure 5 shows the spatial distribution of the pressure. It can be seen that due to the obstruction of the turbulent element, the airflow accumulates in front of the element, causing the highest pressure here, and the pressure in the rear region of the element is negative, thus forming a large pressure differential. As a result, the airflow is reversed at the back of the element, forming a turbulent vortex, and spreading downstream. Therefore, obvious vortex shedding can be observed in the following process. In addition, the pressure difference between the inlet and outlet is 143Pa, so the pressure drop caused by the spoiler is relatively high and energy loss is large.

4. Conclusions
In this paper, the SST k-ω model is used to solve the incompressible and transient turbulent flow field. The influence of the spoiler elements on the turbulent agglomeration is numerically studied. The conclusions are as follows:

1. The airflow passes through the spoiler with a velocity of 15m/s and the phenomenon of bluff body recirculation flow occurs. A series of vortices are formed at the back of the element. It is found that these vortices are similar to the periodic Karman vortex street, and the periodic time $t = 0.0145s$.

2. After the airflow passes through the element, the turbulent dissipation rate in the mainstream region is higher. The distribution range of high turbulent dissipation area is relatively wider, which can effectively increase the particle agglomeration. In the vortex street period, the mass weighted average turbulent dissipation rate of the whole flue flow field approaches a constant. Because of viscous dissipation, the dissipation rate decreases exponentially with the increase of the propagation, and the agglomeration effect decreases.

3. The pressure drop between inlet and outlet is up to 143Pa, and the pressure loss is relatively high.

Acknowledgments
This work has been supported by the Chinese National Key Research and Development Project under Grant No.2017YFB0603203.
References

[1] Wu, G.P., Hu, W., Teng, E.J., et al. Analysis of the effect of air pollution on the adult's respiratory health[J]. Environmental Monitoring in China, 2001, 17(S): 33-38.

[2] Yi, H.H., Hao, J.M., Duan, L., et al. Influence of dust catchers on PM10 emission characteristics of power plants[J]. Chinese Journal of Environmental Science, 2006, 27(10): 1921-1927.

[3] Xiong, G.L., Li, S.Q., Chen, S., et al. Development of advanced electrostatic precipitation technologies for reducing PM2.5 emissions from coal-fired power plants[J]. Proceedings of the CSEE, 2015, 35(9): 2217-2223.

[4] Wen, Z., Yun, S., Thomson, M.J., et al. Modeling soot formation in turbulent kerosene/air jet diffusion flames[J]. Combustion and Flame, 2003, 135(3): 323-340.

[5] Zhang, P.F., Mi, J.C., Pan, Z.M., et al. Influences of elemental arrangement and particle concentration on fine particle amalgamation[J]. Proceedings of the CSEE, 2016, 36(6): 1625-1632.

[6] Liu, Z., Liu, H.X., Feng, X.X., et al. Flow field and particle trajectory numerical simulation of turbulent coalescing[J]. Journal of Chinese Electrical Engineering Science, 2012, 32(14): 71-75.

[7] Kalt, P., Nathan, G., Kelso, R., et al. Assessing the Significance of the Indigo Aerodynamic Agglomeration Technology using Mie/Lif Laser Diagnostics[C]. Proceedings of the Tenth International Conference on Electrostatic Precipitation, Australia, 2006, Paper No. 9B1.

[8] Liu, G.Q., Li, S.Q., Zhang, W.K., et al. Group equilibrium simulation of turbulent agglomeration of fine particles by partition method [J]. in press.

[9] Wu, J., Gu, Z.Q., Zhong, Z.H., The application of turbulence model in vehicle flow simulation [J]. Automotive engineering, 2003, 25(4): 326-327.

[10] Saffman, P.G., Turner, J.S., On the collision of drops in turbulent clouds[J]. Journal of Fluid Mechanics, 1956, 1(1): 16-30.