Study on the key structure parameters of a gravity settling chamber based on a flow field simulation

Xiaojing Liu, Yicheng Zhang, Qiangyun Wu, Mingfeng Zhang, Fan Liu and Yuanchang Guo
School of Traffic and Transportation Engineering, Central South University, Changsha, People’s Republic of China

ABSTRACT
A gravity settling chamber plays an important role in improving the effect of the dust removal when it is selected as a dust collector in a pneumatic cleaning system for cleaning a subway tunnel. The settling rate and the indexes evaluating the secondary blowing are studied to assess the effect of the dust removal in this paper. Regarding the length of the settling chamber, the height of the inlet duct, and the diameter and height of the outlet duct as the variables of the key structures, a flow field simulation analysis of the gravity settling chamber with different heights of the dust accumulation is carried out, then the key structure parameters are optimized by means of the experimental method of uniform design. Finally, the settling chamber with an excellent dust removal effect is obtained. Major conclusions can be drawn as follows. (1) The maximum velocity and the mean velocity ratio of the airflow near a dust accumulation surface are important evaluation indexes for judging the secondary blowing. (2) The maximum velocity near a dust accumulation surface usually doesn’t appear on the vertical plane through the axis of the inlet duct or the outlet duct. (3) An excellent settling effect can be ensured during the whole cleaning process only when the maximum design height of the dust-accumulation surface is considered as a key working condition. (4) An excellent settling effect can be realized if the settling chamber without any filtering device and affiliated baffle structure are long enough. (5) The secondary blowing can be avoided if the inlet and outlet of the settling chamber are far away from the dust-accumulation surface. This study has a good guiding significance for the design of gravity settling chamber.

1. Introduction
The pneumatic cleaning is an important development trend to replace the manpower or hydraulic cleaning because the many electrical facilities in subway tunnels should be protected from any harmful external force and wet air (Wang & Zhang, 2017; Yang, 2014). The settling chamber designed as a dust collector plays an important role in the pneumatic cleaning system (She, 2015; Yun, 2014a; Zheng, 2016).

Some international scholars have been interested in many kinds of settling chambers. Winfield, Cross, Croft, and Paddison (2012) improved the structure parameters of the cylindrical blast furnace gravity settling chamber, pointing out that the dust removal efficiency can be improved by increasing the diameter and the height of the settling chamber and the angle of the bell mouth. Winfield, Paddison, Cross, Croft, and Craig (2013b) also compared the improved cylindrical blast furnace gravity chamber with a cyclone dust collector in term of the dust removal effect. Winfield, Croft, Cross, and Paddison (2013a) conducted a simulation and experiment research on the effect of the different height of the dust accumulation on the dust removal in the cylindrical blast furnace gravity chamber. The structure parameters of a cylindrical cyclone dust chamber were studied by Surjosatyo, Respati, Dafiqurrohman, and Muammar (2017), Ganegama Bogodage and Leung (2015) and Liu, Zhou, Xie, Lu, and Wang (2015). Kubica, Mokrosz, and Szłek (2017) designed some vertical baffles in a cylindrical dust chamber in order to improve the separation efficiency. Wasilewski and Brar (2017) improved the taper of the top inlet of the cylindrical cyclone dust chamber for a high settling efficiency. The structure parameters of cylindrical cyclone dust chamber such as the overall diameter of the settling chamber, the diameter of the inlet duct, the expanding height of the inlet and the total height are optimized by Misiluia, Elsayed, and Andersson (2017), Sun, Kim, Yang, Kim, and Yoon (2017b) for the highest separation efficiency.

All conclusions of the above studies are beneficial for designing the settling chamber, especially for designing the cylindrical settling chambers with an inlet on the
top. However, the gravity settling chamber studied in this paper is a rectangular chamber with an inlet on the left side and an outlet on the right side, quite different from the settling chambers studied above. So, some studies on the rectangular gravity settling chamber and its affiliated structures were also carried out by some scholars. The influence of the different baffle structures on settling efficiency were studied by Li (2012), Li, Xu, Wang, and Wang (2013), Xie (2016), Li, Deng, Yu, Huang, and Liu (2017), Wei (2016), Hu, Cao, Zhao, and Li (2012), Yun, Yang, Zhou, and Li (2014b), He (2016) and Yang (2016). They also investigated the settling performance by observing the airflow state in the chamber. Wei (2016) and Yun et al. (2014b) also studied the influence of the height position of the inlet duct on the dust settling effect. Li et al. (2013), Li et al. (2017) and Hu et al. (2012) also verified the rationality of the structures according to the exhaust concentration.

Although the conclusions of the above studies on the rectangular gravity settling chamber are helpful in improving the dust removal efficiency, it was found that they were not comprehensively based on the key structure parameters, such as the length of the rectangular gravity settling chamber and the positions of the inlet and outlet duct, and the influence of the dust accumulation in the settling chamber on the settling efficiency was not considered. Also, the above studies evaluated the settling performance only according to the airflow distribution on the vertical plane through the axis of inlet or outlet, and did not analyze the settling effect by observing the airflow near the dust accumulation surface. So, in this paper, the basic structure of the gravity settling chamber, which is installed on a flat car for cleaning subway tunnel, is modeled, and some quantitative indexes to evaluate the settling effect are put forward. Then the gas–solid two-phase flow simulation is performed to show the three-dimensional or two-dimensional states in the airflow field in the chamber. Also, the influences of key structure parameters on the dust settling effect under the conditions of different dust accumulation are studied. Finally, an optimal combination of key structure parameters is obtained.

The remainder of the paper is organized as follows. In Section 2, the structure parameters of the gravity settling chamber are analyzed according to the basic structure of the settling chamber. The theoretical equations for the airflow field calculation are introduced in Section 3. The settling performance of the settling chamber is discussed after the simulation is performed, and the structure parameters of the settling chamber are optimized in Section 4. Finally, some conclusions and future works are summarized in Section 5.

2. Structure introduction

2.1. Basic structure

The basic structure of the gravity settling chamber is a rectangular box with an inlet duct and an outlet duct. Its working principle is that the velocity of dust-carrying airflow off the inlet will drop rapidly in the box whose dimensions are much larger than diameter of the inlet, resulting in the separation of the dust particles from the airflow because of gravity, then falling of dust particles onto the floor of the chamber, finally, the airflow which is relatively clean is exhausted via the outlet duct.

As shown in Figure 1, the basic structure of the gravity settling chamber is not involved in any baffle structure in the box. The internal structure parameters of the box are the width $w$, the length $l$ and the height $h$. The jointing segments of the inlet duct and the outlet duct on the symmetry plane of the box are perpendicular to the box wall. The diameter of the inlet duct and its height are denoted by $d_i$ and $h_i$, respectively. The diameter of the outlet duct and its height are denoted by $d_o$ and $h_o$, respectively. The height of the dust accumulation is denoted by $h_a$. The dust suction port of the inlet duct is a taper mouth with a height of 150mm and a cone angle of 40°. When the box is installed on the subway flat car, the vertical height from the bottom of the settling chamber to the bottom of the dust suction port is 1100 mm according to Beijing Municipal Planning Commission (2014).

2.2. Determination of structure parameters

The area of the vertical-section which is perpendicular to the $X$-axis of the box is a product of the width $w$ and the height $h$, however, the maximums of width $w$ and height $h$ should be limited according to the design specifications of the subway vehicle (Beijing Municipal Planning Commission, 2014). In order to make the speed...
of the airflow entering the box rapidly attenuate and to ensure to load the dust in the box as much as possible, the maximums should be selected, so that the width \( w \) and height \( h \) take their limits, which are 2800 and 2100 mm, respectively (Beijing Municipal Planning Commission, 2014).

During the process of cleaning, if the diameter \( d_i \) of the inlet duct is too small, on the one hand, the cleaning efficiency will be low because of the great gas-flow frictional loss, on the other hand, the inlet duct will be blocked by such a large solid garbage as beverage bottle. If the diameter \( d_i \) of the inlet duct is too large, it will make dust suction port also too large to clean some sharp corner areas as shown in Figure 2. Consequently, 300 mm is selected as the value of diameter according to the common diameter of the inlet duct installed in a majority of the existing road sweepers (Jin, 2008; Yan & Wu, 2007).

As a key structure variable, the outlet duct diameter \( d_o \) will have a value from the range of the diameter of the outlet duct applied to most of the existing road sweepers, i.e. 200–420 mm (Jin, 2008; Yan & Wu, 2007).

Since the gravity settling chamber is installed on the B-type subway flat car, the maximum length of the box is limited by the design specifications of the subway vehicle, i.e. the length \( l \) of the box should be less than 19,000 mm (Beijing Municipal Planning Commission, 2014). Considering a distance to reserve for the driver’s cab, the electrical control room and the negative pressure fan is about 7600 mm, the rest length is 11400 mm. Some scholars used to try to design the baffle structure to improve the settling performance when the length of the box is usually less than 1500 mm. In this paper, an attempt is made to obtain a good settling effect by adopting a suitable length of the box without any baffle structure. Therefore, the length \( l \) of the box is taken as the key structure variable, which ranges from 1500 to 11400 mm.

In order to prevent the dust from blocking the inlet duct or the outlet duct, the surface of the dust accumulation in the box cannot be above the inlet duct or outlet duct. At the same time, the position of the inlet and outlet ducts should be arranged above half of the height of the box in order to ensure that the dust height in the box is close to half the height of the box. Consequently, the heights of the axes of the inlet and outlet ducts should be more than 1050 mm since the box is 2100 mm. And the maximum heights should be 1875 mm because both the inlet duct and the outlet duct should be designed below the top of the box. Consequently, the heights \( (h_i, h_o) \) of the axes of the inlet and outlet ducts as key structure variables range from 1050 to 1875 mm.

3. Airflow field dynamics model

This flow field can be simplified to a case of two-phase flow with an air phase and a discrete particle phase.

(1) The fluid equations

Considering the incompressible airflow through the settling chamber (Wu, Men, & Chen, 2011), the airflow field in the gravity settling chamber follows the continuity equation and the Navier–Stokes equation, they can be written as

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0
\]

(1)

\[
\rho \left( \frac{\partial \bar{u}}{\partial t} + u_k \frac{\partial u_i}{\partial x_k} \right) = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) + \partial R_{ij}
\]

(2)

where \( \rho \) is air density, \( t \) is residence time of air micro-element, \( u \) is air velocity, the subscripts \( i, j = 1, 2, 3 \) represent the direction of \( x, y \) and \( z \), respectively, \( \mu \) is the dynamic viscosity of the air, \( p \) is the pressure acting on the discrete micro-body in the fluid system, and the \( R_{ij} \) is the Reynolds stress tensor.

According to Equations (1) and (2), it can be seen that the Reynolds term may cause the above mentioned system of equations not to be closed. The Reynolds term can be neglected when the airflow is laminar flow. However, the interior airflow in the settling chamber is turbulent flow (Yang et al., 2012), so it is necessary to introduce a turbulence model to close the equations.

Generally speaking, the turbulent flow can be modeled by Reynolds–Averaged Navier–Stokes (RANS), Detached Eddy Simulation (DES) and Large Eddy Simulation (LES). However, the requirements of DES and LES in grids and computing resources are higher than those in RANS (Niu, Zhou, Liang, Liu, & Liu, 2017), so the DES and LES are not suitable for the works presented in this paper. And the simulation models based on RANS equations were widely applied to the flow field in the settling chambers by Yun (2014a), Wei (2016) and Winfield et al. (2012, 2013a, 2013b). The Reynolds Stress model (RSM) is suitable for the accurate calculation of intensively turbulent flow field, such as a cyclone flow field. However,
the simulation is difficult in converging and consumes more time and more computing resources by means of the RSM (Wu et al., 2011). According to the studies completed by Winfield et al. (2012, 2013a, 2013b) and Wu et al. (2011), it is reasonable to use the k-ε model to simulate the turbulent flow in the gravity settling chamber since the swirl number in the chamber is relatively small. The standard k-ε model was proposed by Launder and Spalding (1972) according to an assumption that the viscosity coefficients are same in all directions and was widely used in numerical simulation (Akbarian et al., 2018; Ardabili et al., 2018). However, the turbulent viscosity should be calculated with strain rate instead of a constant according to the study conducted by Shih, Liou, Shabbir, Yang, and Zhu (1995). In order to make the flow conform to the physical laws of turbulence, the realizable k-ε model, therefore, is obtained by modifying the standard k-ε model (Mou, He, Zhao, & Chau, 2017; Niu et al., 2017; Sun, Jia, Xing, & Peng, 2018; Wu et al., 2011; Zhang et al., 2014). In addition, Wu et al. (2011), Qin, Xiao, Zhou, Wu, and Xu (2016) and Wang, Silaen, Tang, Barker, and Zhou (2017) used the realizable k-ε model to research the settling chamber and the simulation results were in accord to their experiment results. Therefore, the realizable k-ε model with swirl correction can be applied to close the time-averaged Navier–Stokes equations in this paper.

The Rij in Equation (2) can be expressed as

$$R_{ij} = -\rho \frac{u_i}{u_j} = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} (\rho k) \delta_{ij}$$

(3)

where $\mu_t$ is the turbulent viscosity, which can be calculated by the turbulent kinetic energy $k$ and the turbulent energy dissipation rate $\varepsilon$, $\delta_{ij}$ is the Kronecker delta, when $i = j, \delta_{ij} = 1$, when $i \neq j, \delta_{ij} = 0$.

$$\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon}$$

(4)

where $C_{\mu} = 0.0845$.

And the transport equations of $k$ and $\varepsilon$ in realizable $k-\varepsilon$ model can be written as

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon$$

(5)

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 E \varepsilon$$

$$- \rho C_2 \varepsilon^2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}}$$

(6)

Where $G_k$ is the generation of turbulent kinetic energy, $\nu$ is the kinematic viscosity, $C_1 = \max(0.43, \eta/(\eta + 5))$, $\eta = E k/\varepsilon$, $\eta = (2Eij \cdot Eij)^{1/2}(k/\varepsilon)$, $Eij = (1/2)((\partial u_i/\partial x_j) + (\partial u_j/\partial x_i))$, $\sigma_k$ and $\sigma_\varepsilon$ are the turbulent Prandtl constants, According to Shih et al. (1995) and Anderson (1995), $\sigma_k = 1.0, \sigma_\varepsilon = 1.2, C_2 = 1.9$, respectively.

(2) The fluid dynamics equation of the particle phase

Wu et al. (2011), Winfield et al. (2012, 2013a, 2013b), Wei (2016), Liu (2017) and Sun et al. (2017b) applied the simulation method of the gas–solid two-phase flow to the flow field mixed with airflow and dust particles in the gravity settling chamber. And the Euler–Euler (E–E) method and Euler–Lagrange (E–L) method may be selected to solve the two-phase flow (Alexander, Bozorgzadeh, Khosousi, & Dworkin, 2018). The different phases are processed into the continuous mediums that penetrate each other in the E–E method. However, for the E–L method, the fluid phase is treated as a continuous phase, which is directly solved by the time-averaged N–S equations, while the discrete phase is obtained by calculating the force balance equation of the particles in the flow field. The discrete phase model (DPM) based on the E–L method is a kind of numerical simulations to observe the trajectories of particles whose volume fraction is lower than 10% (ANSYS Inc, 2014). Meanwhile, the volume fraction of the dust particles in the case of the settling chamber was much lower than 10% (Winfield et al., 2012, 2013a, 2013b). So, the DPM is a good choice to solve this gas–solid two-phase flow problem (Ansari, Goharrizi, Jafari, & Abolpour, 2015; Kosinski & Hoffmann, 2007; Pankajakshan, Mitchell, & Taylor, 2011; Zhou, Mo, & Cen, 2011). Zhou, Flamant, Gauthier, and Lu (2004), Wan, Peng, Xie, Zhang, and Huang (2013), Ma, Liu, Li, Huang, and Su (2015), Liu et al. (2015), Oh, Choi, and Kim (2015), Guizani, Mokni, Mhiri, and Bournot (2014), Baghdar Hosseini, Haghhighi Khoshkhooh, and Javadi Malabad (2017), Xie et al. (2017) and Wang, Zhang, Xie, Zhang, and Gao (2018a, 2018b) used the DPM to simulate the motion of particles in the airflow, and the simulation results were in accord to their experiment results. Therefore, it is reasonable to use the DPM method to research the movement of dust particles in the settling chamber.

The simulation of the discrete phase was performed by using the DPM method. External forces acting on the dust particles include gravity, buoyancy, drag force, pressure gradient force, additional mass force, Basset force, Magnus force and Saffman force. Considering the magnitude of the various forces and their influence, only the gravity, drag and buoyancy were listed below, while other forces were ignored (Wang et al., 2018a, 2018b; Wei, 2016; Winfield et al., 2012, 2013a, 2013b; Wu et al., 2011).

Although the Stokes number of particles has an effect on turbulence, there was no obvious relationship between...
turbulence intensity and Stokes number when the concentration of particles was low (Chen, 2011; Wu et al., 2011; Yun, 2014a). And the interaction among the dust particles and the effect of particle volume fraction on the gas phase were neglected due to the small particle loading (Chen et al., 2016; Sun et al., 2017b). Therefore, the dynamics equation of particles in this paper is presented as follows:

\[
m_p \frac{du_p}{dt} = m_p g - m_p g \frac{\rho}{\rho_p} + \frac{C_d \text{Re}_p}{24} \frac{18 \mu}{\rho_p d_p^2} m_p (u - u_p)
\]  

(7)

where \(C_d\) is the drag coefficient, \(g\) is the gravity acceleration, \(u_p\) is the particle phase velocity, \(m_p\) is the particle mass, \(\rho_p\) is the particle density, \(d_p\) is the particle diameter, \(\text{Re}_p\) is the particle Reynolds number based on relative velocity, and it is defined as follows (Winfield et al., 2012).

\[
\text{Re}_p = \frac{\rho d_p |u_p - u|}{\mu}
\]  

(8)

4. Simulation and optimization of the airflow field

4.1. Evaluation indexes

Several quantitative indexes obtained from the results of the simulation are adopted to analyze the dust removal efficiency.

4.1.1. Settling rate \(B\)

In the manufacturing industry of a dust collector, it is often necessary to monitor the dust concentration in the airflow exhausted off the outlet duct to evaluate the dust removal efficiency. Similarly, the settling rate will be used to examine the effect of dust removal in this paper. The settling rate can be gotten by calculating the ratio of the number of particles fallen onto the bottom of the box to the number of particles in the settling chamber and according to the study conducted by Yun et al. (2014b) and Wei (2016). Therefore, assuming that the number of dust particles in the box released off the inlet duct is \(Q_i\), and the number of dust particles in the outlet duct is \(Q_e\), so the settling rate \(B\) in the settling chamber is defined as

\[
B = \frac{Q_i - Q_e}{Q_i}
\]  

(9)

where \(Q_i\) and \(Q_e\) can be obtained by DPM tracking in the simulation. Equation (9) shows that the larger \(B\) is, the less dust particles escape via the outlet, so the higher the settling efficiency is, indicating a higher dust removal efficiency.

4.1.2. Velocity of the airflow near the dust accumulation surface

The dust particles on the dust accumulation surface are apt to be started by the high velocity airflow near the dust accumulation surface, resulting in a phenomenon of secondary blowing (Gu, 2010), therefore, the velocity of the airflow near the dust accumulation surface is also selected as a quantitative index.

The tiny garbage in the subway tunnels mostly consists of dust particles and iron filings (Zhang et al., 2014). A dust particle is hard to settle down because it is lighter than an iron particle, consequently, the dust particle is prone to a secondary blowing. Therefore, the secondary-blowing velocity of the dust is selected as the critical velocity for judging the secondary-blowing. According to the calculation and research conducted by Wei (2016) and Yun et al. (2014b), the critical velocity of the secondary blowing is set to 5.72 m/s since the density of the dust particle which is difficult to settle is 3020 kg/m\(^3\).

4.1.2.1. Maximum velocity \(V_{\text{max}}\) of the airflow near the dust accumulation surface. According to the theory of secondary blowing (Gu, 2010), the phenomenon of secondary blowing will not occur if the maximum velocity of the airflow near the dust accumulation surface is less than the critical velocity of secondary blowing. Therefore, the maximum velocity \(V_{\text{max}}\) of the airflow near the dust accumulation surface is applied to judge whether or not the phenomenon of secondary blowing can appear.

4.1.2.2. Mean velocity ratio \(\lambda\) of the airflow near the dust accumulation surface. The extent of the secondary blowing, however, is unknown only according to the maximum velocity of the airflow near the dust accumulation surface. There is a linear dependence between the mean velocity of airflow and the degree of secondary blowing according to the study done by Gu (2010). The secondary blowing will become worse when the mean velocity of airflow near the dust accumulation surface is higher. Therefore, the mean airflow velocity can be used to explore the extent of the secondary blowing. However, the mean velocity can only be compared intuitively on value, a dimensionless ratio \(\lambda\) of the mean velocity \(V_m\) of the airflow near the dust accumulation surface to the critical velocity \(V_c\) of secondary blowing is a good choice for measuring the extent of secondary blowing visually. When the maximum airflow velocity near the dust accumulation surface is higher than the critical value of secondary blowing, if the mean velocity ratio \(\lambda\) becomes larger, the dust-blowing will be worse. When it is greater
than or equals to 1, it predicts that any dust particle cannot fall onto the floor in the box. Since the critical velocity $V_c$ of secondary blowing is 5.72 m/s in this study, the mean velocity ratio $\lambda$ is

$$\lambda = \frac{V_m}{5.72} \quad (10)$$

### 4.2. Uniform experimental design scheme

As a type of design of experiments (DOE) techniques, the uniform experimental design method based on Quasi-Monte Carlo algorithm was established by Fang (2001). The main idea of uniform design method is to scatter its design points to be uniformly within the range of variable values. Compared with the orthogonal experimental design method and other design methods, the uniform experimental design can greatly reduce the experiment number under the condition of the same parameter numbers and level numbers (Fang, Liu, & Zhou, 2011). Each variable should have an adequate number of their test levels in the design of the experiment. According to the study completed by Fang et al. (2011) and Zeng (2005), the number of test level of each research variable as many as at least 3 times the number of research variables can ensure the coverage of data sampling analysis within the scope of the study. So, the number of test levels of the variables are 12 when such 4 variables as the outlet duct diameter and the figures in the parentheses are the related parameters.

| Test case number | Box length $l$ (mm) | Inlet duct height $h_i$ (mm) | Outlet duct diameter $d_o$ (mm) | Outlet duct height $h_o$ (mm) |
|------------------|---------------------|-----------------------------|-------------------------------|-------------------------------|
| 1                | 1 (1500)            | 6 (1425)                    | 8 (340)                       | 10 (1725)                     |
| 2                | 2 (2400)            | 12 (1875)                   | 7 (340)                       | 5 (1725)                      |
| 3                | 3 (3300)            | 5 (1530)                    | 10 (400)                      | 6 (1475)                      |
| 4                | 4 (4200)            | 11 (1800)                   | 6 (300)                       | 1 (1050)                      |
| 5                | 5 (5100)            | 4 (1275)                    | 2 (1050)                      | 11 (1900)                     |
| 6                | 6 (6000)            | 10 (1725)                   | 8 (1575)                      | 6 (1475)                      |
| 7                | 7 (6900)            | 3 (1200)                    | 4 (260)                       | 5 (1350)                      |
| 8                | 8 (7800)            | 9 (1650)                    | 12 (420)                      | 2 (1125)                      |
| 9                | 9 (8700)            | 2 (1125)                    | 7 (320)                       | 12 (1875)                     |
| 10               | 10 (9600)           | 8 (1575)                    | 2 (220)                       | 9 (1650)                      |
| 11               | 11 (10500)          | 1 (1050)                    | 10 (380)                      | 6 (1425)                      |
| 12               | 12 (14400)          | 7 (1500)                    | 5 (280)                       | 3 (1200)                      |

### 4.3. Pre-processing

#### 4.3.1. Mesh discretization and independence verification

The 3D airflow field of the gravity settling chamber is meshed with the ANSYS 16.0 MESHING, and the Finite Volume Method (FVM) is adopted. The computational domain is discretized by unstructured tetrahedral elements. As shown in Figure 3(b), some local refinements are applied to the airflow separation region and the zone of high gradient of velocity and pressure, such as the connection regions between the inlet and outlet ducts of the box, in order to capture some subtle flows. In addition, in order to fully solve the airflow near the wall and to capture the separation position of the airflow, eight layers of boundary layer mesh are applied by means of inflation layer, and the growth factor of the mesh height is set as 1.1. For all test cases, the values of $y+$ at the first layer mesh near the wall are within the range from 30 to 100 (Chen, Liu, Jiang, Guo, & Zhang, 2018; Chen, Liu, Zhou, & Niu, 2017; Guo, Liu, Chen, Xie, & Jiang, 2018; Niu, Liang, Xiong, & Liu, 2016; Niu, Zhou, & Liang, 2018). Figure 3 shows the medium mesh of the test case 3 when the dust is accumulated on the floor of the box. The mesh is composed of tetrahedral elements which are inside the settling chamber and hexahedral prism layer cells which are near the wall. And the mesh consists of 1,087,924 cells. The mesh quality meets the simulation requirements since the max and average skewness of the mesh is 0.80734 and 0.20964, and the max and average orthogonal quality is 0.99989 and 0.88541, respectively (ANSYS Inc, 2014).

For example, taking the case of case 3 under the situation of the dust accumulation on the floor for testing, and three kinds of mesh scales of coarse, medium and fine are adopted and the mesh cells of 0.6, 1.08 and 1.3 million are obtained, respectively.

In order to compare the difference among the three kinds of different meshes, the change of velocity with the
height along a vertical line through the centroid of the box is shown in Figure 4, where the coordinate origin (0, 0, 0) is at the centroid of the box. Figure 4 shows that the fine mesh makes a big difference in the velocity from the coarse mesh, no significant difference from the medium mesh. And the height of the high-speed area captured by fine and medium mesh is near the height of the inlet duct axis, and it is reasonable to select the probe point on the axis of the inlet duct in the following grid-independent verification.

In order to verify whether the selected mesh can fully solve the turbulence structure of the airflow field, the grid-independence verification for the 12 sets of test cases are carried out under the situation of the dust accumulation on the floor. When the meshing topology and the mesh height of the boundary layer stay constant, three kinds of mesh scales of coarse, medium and fine are adopted. The turbulent structure of the airflow field is complicated in the region where is applied local refinements shown in Figure 3(b). And it is difficult to solve the velocity in this region (Chen et al., 2017, 2018; Guo et al., 2018; Liu, Chen, Zhou, & Zhang, 2017). Therefore, the probe points are selected on the axis of the inlet duct in this region for grid-independent verification. And the distance between the two probe points and the wall surface of the settling chamber is \( d_i \), and \( 2d_i \), respectively. After the simulation, the velocity of the two points in the X direction are read in the 12 sets of cases under the three grids. And the relative errors of the velocity are obtained and shown in Table 2.

As can be seen from Table 2, the fine mesh makes a big difference in the relative velocity error from the coarse mesh, no significant difference from the medium mesh. Therefore, it is feasible to select the medium mesh to simulate the airflow field in order to improve the calculation efficiency with a better calculation accuracy.

4.3.2. Boundary conditions and algorithms

As can be seen from Table 3, at the inlet of the dust suction port, the boundary condition for the relative pressure is set to 0 Pa, and at the outlet of the outlet duct, the boundary condition for the relative pressure is set to \(-1800\) Pa (Ke, Liu, Wang, & Deng, 2017; Qin et al., 2016; Wei, 2016; Wu, Men, & Chen, 2010; Zheng, 2016). The DPM is used
to simulate the motion of the dust particles, and it is assumed that the discrete phase particles injected into the airflow field be spherical in equal diameter. Since what is studied in this paper is the dust with a difficulty in falling, its particle diameter is defined as 117 μm and its density of the particle is set to 3020 kg/m³ according to Yun (2014a). The dust particles are injected into the domain from the injection surface that is located at the inlet duct of the box. The normal direction of the injection surface is the same as the axial direction of the inlet duct. The interaction among the dust particles and the turbulent dispersion of the dust particles are assumed to be neglected. When the particles fall onto the dust accumulation surface of the box, the dust particles are considered to have been trapped.

The numerical equations are solved by using ANSYS FLUENT 16.0. The realizable k-epsilon turbulence model and the standard wall function are selected. At the same time, the pressure-based implicit solver and SIMPLE algorithm are adopted. The discrete format of the turbulent kinetic energy and the turbulent dissipation rate is set as the second-order upwind style, and the time integration scheme is modeled as the second-order implicit. A transient simulation for the continuous phase is performed and the results are time averaged to obtain the steady-state airflow characteristics with high accuracy and numerical stability (Magnus, Simone, & Lennert, 2018). The total simulation time is 120 s and the time step is 0.001 s, and most of the Courant CFL number in the calculation domain is less than 1.0. And the discrete phase is tracked using DPM with the steady particle treatment to observe the trajectory of particles.

4.4. Results and analysis of the test simulation

For the 12 experiment cases under the three different heights of the dust accumulation shown in Table 1, the simulation results of the airflow field are as follows.

4.4.1. Analysis of settling rate

4.4.1.1. Particle trajectory. In order to visually observe the settling effect of the particles, it is necessary to simulate the steady state of the particle trajectory in the settling chamber, and 2475 particles were injected from the inlet duct. To simplify the calculation, it may be assumed that the dust accumulation surface is horizontal. The three-dimensional simulation results of the particle trajectory are shown under the three different heights of the dust accumulation in Figure 5. The simulation results are arranged from left to right according to the increasing heights of the dust accumulation: the floor of the box, 1/4 height of the box and 1/2 height of the box. The space occupied by the dust is not shown in Figure 5. Although it is hard to display the results in the same scale in Figure 5, the comparative analysis can be correctly made.

By observing the intensity of the particle traces in the outlet duct in Figure 5, the settling effect can be judged intuitively, it means that the more the particles entering
From Figure 5(10) to (12), it can be found that there are no escaping dust particles in the outlet duct at three different heights of dust accumulation, the particles have settled down in the box before reaching the outlet duct. The reason for the phenomena of Figure 5(10) to (12) is that the length of the settling chamber is beyond the maximum distance that the dust particles can probably cover. Consequently, this means that the settling efficiency of the dust particles in the gravity settling chamber can achieve 100% without the baffle structure inside the box if the box is long enough.

In addition, it can be seen from the test in Figure 5 that the space zone at the upper right corner of the box is a hard-to-get-to place of the particles, which indicates that it is beneficial to the settling effect to arrange the outlet duct on the top of the box and far from the inlet duct.

A quantitative research follows in order to further explore the influence of key structure variables on the settling performance indexes.

### 4.4.1.2. Settling rate. (1) Test results

According to the number of dust particles in the inlet and outlet ducts, the settling ratio $R$ can be calculated by using Equation (9), when 2475 particles were injected into the box off the inlet duct, so the $Q_e$
Figure 5. Particle trajectories in the gravity settling chamber.
The settling rates of the test case 1–9 show that each rate monotonously decreases as the height of the dust accumulation increases. These above phenomena happened because the particles had difficulty in settling due to the increasing mean airflow velocity, which was caused by decreasing cross-sectional areas with increasing height of the dust accumulation. It can also be seen from Tables 1 and 4 that the settling rates of the tests case 10–12 are 1 because the boxes are long enough, which is consistent with the case shown in Figure 5(10) to (12).

The above results indicate that both the length of the box and the height of the dust accumulation are important variables that seriously affect the settling effect. To further investigate the sensitivity of key structure parameters on settling rates, a regression analysis is performed as follows.

(2) Sensitivity analysis

The structure parameters, which have little influence on the settling rate or are significantly related to other structure parameters, can be eliminated by means of a stepwise regression analysis. As a result, the expression of the influence of key structure parameters on the settling rate can be obtained. Using the data in Table 4, a high accuracy quadratic polynomial regression model is applied.

\[
B = \beta_0 + \sum_{i=1}^{4} \beta_i x_i + \sum_{i=1}^{4} \beta_i x_i^2 + \sum_{i<j} \beta_{ij} x_i x_j + \varepsilon
\]

(11)

where, \(x_i\) is one of the four key structure parameters \(l, h_i, d_o, h_o\), \(\beta_0, \beta_i, \beta_{ij}\) and \(\beta_{ij}\) are the regression coefficients to be obtained, and \(\varepsilon\) is a comprehensive error.
The settling rates B under three different heights of the dust accumulation are obtained as follows:

\[
B = \begin{cases} 
\text{on the floor:} & 0.6534 + 0.0315l \\
\text{at 1/4 the height of the box:} & 0.5974 + 0.0351l \\
\text{at 1/2 the height of the box:} & 0.6873 + 0.00266l^2 
\end{cases}
\]

where \( l \) is the length of the box and \( h_i \) is the height of the dust accumulation. The fitting degrees of Equation (12) are represented by both the determinable coefficient \( R^2 \) and the adjustment determinable coefficient \( R^2_{\text{adj}} \).

The coefficient \( R^2 \) can be written as:

\[
R^2 = \frac{\sum_{i=1}^{n} (y_i - \bar{y})^2}{\sum_{i=1}^{n} (y_i - \bar{y})^2}
\]

where \( y_i \) is the value that needs to be fitted, \( \bar{y} \) is the mean value of the \( y \), \( \bar{y} \) is the fitted value, and \( n \) is the number of samples to be fitted, in this paper, \( n = 12 \).

The adjustment determinable coefficient \( R^2_{\text{adj}} \) can be written as:

\[
R^2_{\text{adj}} = 1 - (1 - R^2) \frac{(n - 1)}{(n - m - 1)}
\]

where \( m \) is the number of parameters in the function, \( m = 4 \).

The fitting degrees and the significance of Equation (12) are shown in Table 5. It can be seen from Table 5 that the determinable coefficient \( R^2 \) and the adjustment determinable coefficient \( R^2_{\text{adj}} \) are both close to 1, and the \( P \) value is less than 0.01 in the test of significance, indicating that Equation (12) has a good regression effect (Daniel & Chantal, 2017; Sun, Kim, Seung, Chul, & Joon, 2017a).

Equation (12) shows that the influence of the diameter of the outlet duct on the settling rate can be ignored under different dust accumulation heights and when the values of all structure parameters are within the range introduced above. The effects of other key structure parameters are analyzed as follows.

(1) When the dust falls onto the floor, the settling rate is affected by the length \( l \) of the box and the height difference \( (h_1 - h_o) \) between the inlet and outlet ducts. The longer the length of the box is or the more the inlet duct is below the outlet duct, the higher the settling rate is. The coefficients in Equation (12) show a conclusion that the height difference between the outlet duct and the inlet duct has a greater influence on the settling rate than the length of the box. The conclusion can be verified from Figure 5(5) and (9), in which the height differences between the outlet duct and the inlet duct were 525 and 750 mm, respectively, meaning that the axis of the inlet duct was much lower than the axis of the outlet duct, resulting in no particle in the outlet duct. Even if the length of the box was 5100 mm as shown in Figure 5(5), the height difference between the outlet duct and the inlet duct was too large to collect the particles in the outlet duct. The particle motion states from Figure 5(10) to (12) also help to verify the above conclusion with a phenomenon that there was no escaping particles when the dust accumulated on the floor because the length of the box ranged from 9600 to 11400 mm, meaning that the length of the box is long enough.

(2) When the dust accumulates at 1/4 the height of the box, the settling rate is no longer affected by the heights of the inlet duct and the outlet duct, but the rate is directly proportional to the length of the box. This conclusion can be verified with the particle motion states from Figure 5(9) to (12). The box length that ranged from 8700 to 11400 mm resulted in no escaping particle in the outlet duct when the dust accumulated at 1/4 the height of the box because the box length was enough for the settling of all particles.

(3) The settling height in the box becomes minimum when the dust accumulated at 1/2 the height of box, so that the influence of the height of the inlet duct or outlet duct on settling rate can be ignored. The settling rate is directly proportional to the squared length of the box length. It can be verified with the particle motion states from Figure 5. The phenomena from Figure 5(1) to (9) showed that there were the escaping particles in the outlet duct when the dust accumulated at 1/2 the height of box if the box was not long enough. From Figure 5(10) to (12), a phenomenon was observed that there was no escaping particle because of the enough box length ranged from 9600 to 11400 mm.

| Settling rate | The height of dust accumulated | \( R^2 \) | \( R^2_{\text{adj}} \) | \( P \) value |
|--------------|-------------------------------|---------|----------------|-------------|
| B            | on the floor                  | 0.996   | 0.933          | 0.005       |
| B            | 1/4 the height                | 0.985   | 0.939          | 0.007       |
| B            | 1/2 the height                | 0.966   | 0.924          | 0.005       |

4.4.2. Analysis of the secondary blowing

4.4.2.1. Maximum velocity of airflow near the dust accumulation surface. Figure 6 is a histogram of the maximum velocity of airflow near the dust accumulation surface according to the simulation result of the airflow.
field in the gravity settling chamber. For convenience of comparison, a horizontal line in Figure 6 is drawn to represent the critical velocity of the secondary blowing, i.e., 5.72 m/s. Figure 6 shows that there are 3 test cases whose maximum velocities of airflow near the dust accumulation surface are higher than the critical velocity in the case of the dust accumulation on the floor. There is an increasing number of cases, whose maximum velocities of airflow near the dust accumulation surface is higher than the critical velocity, with an increasing height of the dust accumulation. In the same case, the maximum velocity of airflow near the dust accumulation surface increases monotonously as the height of the dust accumulation increases. This phenomenon happens because the cross-sectional area of the box decreases as the height of the dust accumulation increases, resulting in an increase in the mean airflow velocity and secondary blowing.

In addition, as can be seen from Figure 6 that the maximum velocities of airflow near the dust accumulation surface increases significantly at 1/2 the height of the dust accumulation in all test cases except for cases 2, 6 and 10. According to Table 1, it can be known that the inlet duct or the outlet duct is near the surface of dust accumulation at 1/2 the height of the box in cases 3, 4, 5, 7, 8, 9, 11 and 12. On the contrary, the inlet duct or the outlet duct is far away from the surface in the cases 2, 6, and 10 at the 1/2 the height of the box, so their maximum velocities of the airflow near the dust accumulation surface are the lowest among all cases. Thus, it can be seen that the shorter the distance from the dust accumulation surface to the inlet duct or outlet duct is, the higher the velocity of airflow near the dust accumulation surface is, causing a higher the probability of secondary blowing.

In case 1, although the inlet duct and outlet duct are far away from the surface of the dust accumulation at 1/2 the height of the box, the box length is the shortest, resulting in a large value of the maximum velocity near the dust accumulation surface on 1/2 the height of the box. It also shows that a shorter settling chamber needs to be equipped with some baffle structures to avoid the secondary blowing.

4.4.2.2. Ratio of mean velocity of airflow near the dust accumulation surface. As shown in Figure 7, a histogram of the ratio of the mean velocity of the airflow near the dust accumulation surface to the critical velocity of the secondary blowing are obtained according to the simulation results. For convenience of comparison, a horizontal line is drawn to indicate the ratio of 1. It can be seen from Figure 7 that the mean velocity ratio of the airflow near the dust accumulation surface gradually increases as the height of the dust accumulation increases. The values of the mean velocity ratios are all less than 1 except for the case 1. That is, the mean airflow velocity is lower than the critical velocity of the secondary blowing, indicating that the settling is relatively stable, and the secondary blowing doesn’t appear throughout the airflow field in the box. The mean velocity ratios of the airflow near the dust accumulation surface in cases 2, 6, and 10 are affected less by the dust height than those in the other cases. According to Table 1, it can be known that the dust accumulation surfaces in cases 2, 6 and 10 are away enough from the inlet and outlet duct under the three heights of the dust accumulation, as a result, the mean velocities of airflow near the dust accumulation surfaces increase slowly with the increase in the dust accumulation.
Figure 8. Velocity distribution near the surface of the dust accumulation at different heights.
Figure 8. Continued.
4.4.2.3. Velocity distribution of the airflow near the dust accumulation surface. The velocity field in this paper is expressed with dimensionless velocity $U$. The $U$ is defined as

$$U = \frac{u}{U_{\text{ref}}}$$  \hspace{1cm} (15)$$

where $U$ is the dimensionless velocity, $u$ is the airflow velocity of local monitoring points, $U_{\text{ref}}$ is the average velocity of the inlet boundary. The differential pressure is equivalent to the average velocity of 20 m/s since the pressure boundary is adopted at the inlet and outlet duct in this paper.

The velocity distribution of the airflow near the dust accumulation surface is helpful to find the position where the maximum velocity occurs. The velocity distributions near the dust accumulation surface in all test cases at different heights of the dust accumulation are obtained as shown in Figure 8. They are arranged from left to right according to the increasing heights of the dust accumulation: the floor of the box, 1/4 height of the box and 1/2 height of the box. Without affecting the analysis, the velocity distributions are not displayed in the same scale in Figure 8. The axes of the inlet and outlet ducts drawn with dotted lines are added in order to help to determine the locations of the maximum velocities in Figure 8.

According to Equation (15), the dimensionless velocity corresponding to the critical velocity of secondary dust blowing can be calculated to be 0.286. The region where the velocity is more than the critical velocity of the secondary blowing can be visually found according to the colors on a scale. Figure 8 shows that the position where the maximum airflow velocity appears usually keeps changing with the height of the dust accumulation, and the location of the maximum airflow velocity is seldom under the axes of the inlet duct and the outlet duct. The distance between the axis of the inlet duct or the outlet duct and the surface of the dust accumulation, also the space height of the airflow field in the box are changed with the height of the dust accumulation surface, resulting in changing in the velocity distribution. What’s more, when the dust accumulation surface is vertically far away from the inlet duct and outlet duct, the maximum velocity of the airflow near the dust accumulation surface is horizontally away from the axis of the inlet duct or outlet duct due to the turbulence in the box. Therefore, it is unreasonable to study the secondary blowing by examining the maximum velocity of the vertical plane through the axis of the inlet or outlet duct. In addition, it also reflects that the maximum velocity of the airflow near the dust accumulation surface and the mean velocity ratio of the airflow near the dust accumulation surface are the important quantitative indexes for judging the secondary blowing from the side.

4.5. Test optimization and verification

It shows that the settling effect is the worst and needs to be optimized when the height of the dust accumulation is 1/2 the height of the box according to the above researches. As revealed by Table 3, the lower the height of dust accumulation is, the better the settling effect is. Therefore, a better settling effect can be achieved with a suitable optimization method when the dust accumulation surface is at 1/2 the height of the box, which will also lead to the improvement of the settling effect when the dust height is lower than 1/2 correspondingly. According to Equation (12), when the height of dust accumulation is 1/2, there is a settling rate equation: $B = 0.6873 + 0.00266l^2$. If $B$ is 1, then $l$ equals 10843 mm, meaning that the settling effect is the best when the box length is 10843 mm. Considering the secondary blowing, the positions of the inlet duct and the outlet duct should be as far as possible away from the surface of the dust accumulation, so the inlet duct and the outlet duct are arranged at a height of 1700 mm.

A two-phase flow field in the gravity settling chamber is simulated by using the values of the above structure variables. There is a good examination that the 2475 particles are settled down after they are out of the inlet duct. As shown in Figure 9, there isn’t any particle in the outlet duct, i.e. the settling rate is 100%. An investigation from the simulation further indicates that the mean velocity of the airflow near the dust accumulation surface is 1.843 m/s, and the mean velocity ratio $\lambda$ of the airflow near the dust accumulation surface is 0.32, which is less than those in Figure 7. The velocity distribution of the airflow near the dust accumulation surface is shown in Figure 10. Figure 10 shows that the dimensionless velocity $U = 0.175$ (the maximum velocity is 3.50 m/s), which

Figure 9. Particle trace in the optimized settling chamber.

Figure 10. Velocity distribution of the airflow near the dust accumulation surface of the optimized settling chamber.
is less than the critical velocity of secondary blowing and is less than those in Figure 6. So it can be sure that the above optimization is effective.

5. Conclusion and future work

Choosing the box length of the settling chamber, the height of the inlet duct, the diameter of outlet duct and the height of outlet duct as the optimization variables, the settling performance under different conditions of the dust accumulations are studied by means of the CFD simulation and with the uniform design experiment method. The following conclusions can be drawn:

(1) The maximum velocity of the airflow near the dust accumulation surface and the mean velocity ratio of the airflow near the dust accumulation surface are the important quantitative indexes for judging the secondary blowing. The settling rate, the maximum velocity and the mean velocity ratio of the airflow near the dust accumulation surface should be comprehensively studied in order to correctly evaluate the settling performance of a gravity settling chamber.

(2) The maximum velocity of the airflow near the dust accumulation surface, rather than on the vertical plane through the axis of the inlet duct and the outlet duct, should be investigated.

(3) The settling rate of 100% can be expected without any filtering device or affiliated baffle structure if the box is long enough.

(4) A reliable and final settling performance can be guaranteed only considering the maximum height of dust-accumulation while the settling performance is unreliable when considering the height of dust-accumulation lower than maximum height.

(5) The inlet duct and outlet duct should stay away from the highest surface of the dust accumulation in order to avoid the secondary blowing, resulting in high settling performance.

The humidity of the air is not considered in this paper. However, it is possibly high, resulting in the particle agglomerating. Therefore, the influence of the humidity of the air on the settling performance will be carried out in future work. And this paper is not involved to the design of the dust-unload device. Therefore, the dust-unload device will be studied according to accumulation situation of dust particles in the settling chamber.

Disclosure statement

No potential conflict of interest was reported by the authors.

Funding

The research described in this paper was financially supported by the Fundamental Research Funds for the Central Universities of Central South University [grant number 2018zzts511] and 2018 Creative Entrepreneurship Project between Teachers and Students of Central South University [grant number 2018gczd025].

References

Akbarian, E., Najafi, B., Jafari, M., Ardabili, S. F., Shamshirband, S., & Chau, K. W. (2018). Experimental and computational fluid dynamics-based numerical simulation of using natural gas in a dual-fueled diesel engine. Engineering Applications of Computational Fluid Mechanics, 12(1), 517–534.

Alexander, R., Bozorgzadeh, S., Khosousi, A., & Dworkin, S. B. (2018). Development and testing of a soot particle concentration estimator using lagrangian post-processing. Engineering Applications of Computational Fluid Mechanics, 12(1), 236–249.

Anderson, J. D. (1995). Computational fluid dynamics: The basics with applications. New York, NY: McGraw-Hill.

Ansari, V., Goharrizi, A. S., Jafari, S., & Abolpour, B. (2015). Numerical study of solid particles motion and deposition in a filter with regular and irregular arrangement of blocks with using lattice boltzmann method. Computers & Fluids, 108, 170–178.

ANSYS inc. (2014). ANSYS 16.0: Theory guide. Washington, DC: Author.

Ardabili, S. F., Najafi, B., Shamshirband, S., Bidgoli, B. M., Deo, R. C., & Chau, K. W. (2018). Computational intelligence approach for modeling hydrogen production: A review. Engineering Applications of Computational Fluid Mechanics, 12(1), 438–458.

Baghdar Hosseini, S., Haghhighi Khoshkhoo, R., & Javadi Malabadi, S. M. (2017). Numerical study on polydisperse particle deposition in a compact heat exchanger. Applied Thermal Engineering, 127, 330–346.

Beijing Municipal Planning Commission. (2014). Code for design of metro: Vehicle. Beijing: China Building Industry Press.

Chen, Y. (2011). Particle collision theory based on LES/FDF and its application in dense gas-solid multiphase flow (Master’s thesis). Zhejiang University, Hangzhou.

Chen, J., Chu, K., Zou, R., Yu, A., Vince, A., Barnett, G. D., & Barnett, P. J. (2016). Systematic study of the effect of particle density distribution on the flow and performance of a dense medium cyclone. Powder Technology, 91, 16–33.

Chen, Z., Liu, T., Jiang, Z., Guo, Z., & Zhang, J. (2018). Comparative analysis of the effect of different nose lengths on train aerodynamic performance under crosswind. Journal of Fluids and Structures, 78, 69–85.

Chen, Z., Liu, T., Zhou, X., & Niu, J. (2017). Impact of ambient wind on aerodynamic performance when two trains intersect inside a tunnel. Journal of Wind Engineering and Industrial Aerodynamics, 169, 139–155.

Daniel, T. L., & Chantall, D. L. (2017). Data mining and predictive analytics (second Edition): Multivariate regression and model construction. Beijing: Tsinghua University Press.
Fang, K. (2001). *Orthogonal and uniform experimental design: Uniform design*. Beijing: Science Press.

Fang, K., Liu, M., & Zhou, Y. (2011). *Experimental design and modelling: Uniform design*. Beijing: Higher Education Press.

Ganegama Bogodage, S., & Leung, A. Y. T. (2015). Cfd simulation of cyclone separators to reduce air pollution. *Powder Technology*, 286, 488–506.

Gu, Z. (2010). *Wind and dust: Near-surface turbulence and gas-solid two-phase flow: Wind-sand movement mechanism and sand flow*. Beijing: Science Press.

Guizani, R., Mokni, I., Mhiri, H., & Bournot, P. (2014). Cfd modeling and analysis of the fish-hook effect on the rotor separator’s efficiency. *Powder Technology*, 264, 149–157.

Guo, Z. J., Liu, T. H., Chen, Z. W., Xie, T. Z., & Jiang, Z. H. (2018). Comparative numerical analysis of the slipstream caused by single and double unit trains. *Journal of Wind Engineering and Industrial Aerodynamics*, 172, 395–408.

He, L. (2016). *Model study of flow-sharing eddy-breaking device for dust collector* (Master’s thesis). Suzhou University, Suzhou.

Hu, B., Cao, T., Zhao, W., & Li, X. (2012). Design and simulation of dust cover in sweeper waste bin. *Journal of Shandong Jiaotong University*, 20(1), 7–10.

Jin, G. (2008). *Petroleum equipment design selection manual: Dust collector*. Shanghai: Chemical Industry Press.

Ke, J., Liu, H., Wang, G., & Deng, B. (2017). Study on parameter optimization of suction port of trough track sweeper. *Mechanical Design and Manufacture*, 11, 95–99.

Kosinski, P., & Hoffmann, A. C. (2007). An Eulerian–lagrangian model for dense particle clouds. *Computers & Fluids*, 36(4), 714–723.

Kubica, R., Mokrosz, W., & Słzlek, A. (2017). Improving energy and environmental performance of coal fueled boilers – a new type of centrifugal dust separator with external flue recycle. *Energy*, 138, 238–248.

Lauder, B. E., & Spalding, D. B. (1972). *Lectures in mathematical models of turbulence*. London: Academic Press.

Li, C. (2012). *Research on key technologies of full-suction wet and dry sweeper* (Master’s thesis). Jiangsu University of Science and Technology, Zhenjiang.

Li, Z., Deng, G., Yu, C., Huang, H., & Liu, C. (2017). Structure optimization of sanitation vehicle collecting box based on numerical simulation of multiphase flow. *Journal of Wuhan University of Technology (Transmission Science & Engineering)*, 2, 234–238.

Li, L., Xu, C., Wang, Z., & Wang, Q. (2013). Numerical simulation of internal flow field and dust removal efficiency of blast furnace Gas gravity dust collector. *China Metallurgy*, 23(1), 26–29.

Liu, H. X. (2017). *Research and development of track cleaning technology and equipment for modern trams* (Master’s thesis). Southwest Jiaotong University, Chengdu.

Liu, T., Chen, Z., Zhou, X., & Zhang, J. (2017). A cfd analysis of the aerodynamics of a high-speed train passing through a windbreak transition under crosswind. *Engineering Applications of Computational Fluid Mechanics*, 1, 1-15.

Liu, M., Zhou, C., Xie, J., Lu, C., & Wang, Z. (2015). Numerical investigation of performance of a fast gas-solid separator. *Powder Technology*, 275, 30–38.

Ma, W., Liu, W., Li, L., Huang, G., & Su, B. (2015). Numerical simulation of unsteady-state particle dispersion in ferroalloy workshop. *Indoor and Built Environment*, 24(1), 46–55.

Magnus, U., Simone, S., & Lennert, S. (2018). Numerical analysis of a vehicle wake with tapered rear extensions under yaw conditions. *Journal of Wind Engineering and Industrial Aerodynamics*, 179, 308–318.

Misulia, D., Elsayed, K., & Andersson, A. G. (2017). Geometry optimization of a deswirler for cyclone separator in terms of pressure drop using cfd and artificial neural network. *Separation and Purification Technology*, 185, 10–23.

Mou, B., He, B. J., Zhao, D. X., & Chau, K. W. (2017). Numerical simulation of the effects of building dimensional variation on wind pressure distribution. *Engineering Applications of Computational Fluid Mechanics*, 11(1), 293–309.

Niu, J., Liang, X., Xiong, X., & Liu, F. (2016). Influence of external windshield structure on aerodynamic performance of high speed trains. *Journal of Shandong University: Engineering*, 46(2), 108–115.

Niu, J., Zhou, D., & Liang, X. (2018). Numerical investigation of the aerodynamic characteristics of high-speed trains of different lengths under crosswind with or without windbreaks. *Engineering Applications of Computational Fluid Mechanics*, 12(1), 195–215.

Niu, J., Zhou, D., Liang, X., Liu, T., & Liu, S. (2017). Numerical study on the aerodynamic pressure of a metro train running between two adjacent platforms. *Tunnelling and Underground Space Technology*, 65, 187–199.

Oh, J., Choi, S., & Kim, J. (2015). Numerical simulation of an internal flow field in a uniflow cyclone separator. *Powder Technology*, 274, 135–145.

Pankajakshan, R., Mitchell, B. J., & Taylor, L. K. (2011). Simulation of unsteady two-phase flows using a parallel Eulerian–lagrangian approach. *Computers & Fluids*, 41(1), 20–26.

Qin, X., Xiao, Q., Zhou, F., Wu, Q., & Xu, H. (2016). Simulation analysis and design improvement of gas-solid two-phase flow in road sweeper suction nozzle. *Journal of Applied Mechanics*, 33(1), 73–79.

She, G. (2015). *Research and design of mobile dust removal system at the bottom of metro vehicles* (Master’s thesis). Chongqing University, Chongqing.

Shih, T. H., Liou, W. W., Shabbir, A., Yang, Z., & Zhu, J. (1995). A new k-ε eddy viscosity model for high reynolds number turbulent flows. *Computers Fluids*, 24(3), 227–238.

SPSS Inc. (2010). *IBM SPSS 19 core system user’s guide* (2010): *case study*. Chicago: Author.

Sun, S. K., Jia, X. H., Xing, L. F., & Peng, X. Y. (2018). Numerical study and experimental validation of a roots blower with backflow design. *Engineering Applications of Computational Fluid Mechanics*, 12(1), 282–292.

Sun, X., Kim, H. S., Seung, D. Y., Chul, K. K., & Joon, Y. Y. (2017a). Numerical investigation of the effect of surface roughness on the flow coefficient of an eccentric butterfly valve. *Journal of Mechanical Science and Technology*, 31(6), 2839–2848.

Sun, X., Kim, S., Yang, S. D., Kim, H., & Yoon, J. Y. (2017b). Multi-objective optimization of a starmand cyclone separator using response surface methodology and computational fluid dynamics. *Powder Technology*, 320, 51–65.

Surjosatyo, A., Respati, A., Dafigurrohman, H., & Muammar, A. (2017). Analysis of the influence of vortexbinder dimension on cyclone separator performance in biomass gasification system. *Procedia Engineering*, 170, 154-161.
Winfield, D., Cross, M., Croft, N., & Paddison, D. (2012). Numerical simulation of atmosphere migration of uranium tailings grit based on dpm. *Journal of Safety & Environment, 13*(1), 96–101.

Wang, Y., Silaen, A. K., Tang, G., Barker, D., & Zhou, C. Q. (2017). Numerical optimization of a gravity dust-Catcher for improving Operation efficiency. *Asme International Mechanical engineering Congress & Exposition.*

Wang, Y., & Zhang, G. (2017). Simulation of two-phase flow in pneumatic conveying system of cleaning truck. *Energy-saving and Environmental Protection of Traffic, 13*(5), 27–30.

Wang, J., Zhang, J., Xie, F., Zhang, Y., & Gao, G. (2018a). A study of snow accumulating on the bogie and the effects of deflectors on the de-icing performance in the bogie region of a high-speed train. *Cold Regions Science and Technology, 148*, 121–130.

Wang, J., Zhang, J., Zhang, Y., Xie, F., Krajnović, S., & Gao, G. (2018b). Impact of bogie cavity shapes and operational environment on snow accumulating on the bogies of high-speed trains. *Journal of Wind Engineering and Industrial Aerodynamics, 176*, 211–224.

Wasilewski, M., & Brar, L. S. (2017). Optimization of the geometry of cyclone separators used in clinker burning process: A case study. *Powder Technology, 313*, 293–302.

Wei, X. (2016). *Flow field analysis and structure improvement of vacuum sweeping system and dust collection system of suction sweeper* (Master’s thesis). Yanshan University, Qinhuangdao.

Winfield, D., Croft, N., Cross, M., & Paddison, D. (2013a). Incorporating dust lift-off into a cfd model of a blast furnace gravity dust-catcher. *Applied Mathematical Modelling, 37*(16-17), 7891–7904.

Winfield, D., Cross, M., Croft, N., & Paddison, D. (2012). Geometry optimisation of a gravity dust-catcher using computational fluid dynamics simulation. *Chemical Engineering and Processing: Process Intensification, 62*(6), 137–144.

Winfield, D., Paddison, D., Cross, M., Croft, N., & Craig, I. (2013b). Performance comparison of a blast furnace gravity dust-catcher vs. Tangential triple inlet gas separation cyclone using computational fluid dynamics. *Separation and Purification Technology, 115*(2), 205–215.

Wu, B., Men, J., & Chen, J. (2010). Numerical study on particle removal performance of pickup head for a street vacuum sweeper. *Powder Technology, 200*(1), 16–24.

Wu, B., Men, J., & Chen, J. (2011). Study of the particle separation performance of a dust-settling hopper. *Chemical Product and Process Modeling, 6*(1), 16–27.

Xie, S. (2016). *Flow field analysis and structure improvement of vacuum cleaner vacuum system* (Master’s thesis). Yanshan University, Qinhuangdao.

Xie, F., Zhang, J., Gao, G. J., He, K., Zhang, Y., Wang, J. B., & Zhang, Y. N. (2017). Study of snow accumulation on a high-speed train’s bogies based on the discrete phase model. *Journal of Applied Fluid Mechanics, 10*(6), 1729–1745.

Yan, K., & Wu, H. L. (2007). JB/t 7303-2007 People’s Republic of China machinery industry standard road sweeper: Requirement. Beijing: Mechanical industry press.

Yang, C. (2014). Research on particle removal performance of vacuum dust suction mouth. Changsha: School of transportation engineering. *Journal of Central South University, 2014*, 29–38.

Yang, M. (2016). Integral design and vacuum system research of pure electric sweeper (Master’s thesis). Shandong University of Technology, Jinan.

Yang, C., Zhang, Y., Ouyang, Z., Zhang, J., Yang, S., & Zhang, R. (2012). Parameter design of vacuum cleaner vacuum suction port based on flow field simulation. *Journal of Central South University: Natural Science, 43*(9), 385–390.

Yun, X. (2014). Study on the characteristics of dust collection and settlement of road sweepers (Master’s thesis). Central South University, Changsha.

Yun, X., Yang, Z., Zhou, L., & Li, H. (2014). Simulation study on dustfall rate of urban road cleaning vehicle. *Computer Simulation, 31*(10), 191–195.

Zeng, Z. (2005). Uniform design and its application: Uniform design and its application. Beijing: China Medical Science and Technology Press.

Zhang, Y., Yang, C., Baker, C., Chen, M., Zou, X., & Dai, W. (2014). Effects of expanding zone parameters of vacuum dust suction mouth on flow simulation results. *Journal of Central South University, 21*(6), 2547–2552.

Zheng, F. (2016). Design of pneumatic conveying system and Simulation of suction nozzle flow field for sweeper (Master’s thesis). Hunan University, Changsha.

Zhou, H., Flamant, G., Gauthier, D., & Lu, J. (2004). Numerical simulation of the turbulent gas–particle flow in a fluidized bed by an LES-DPM model. *Chemical Engineering Research and Design, 82*(7), 918–926.

Zhou, H., Mo, G., & Cen, K. (2011). Numerical investigation of dispersed gas–solid two-phase flow around a circular cylinder using lattice Boltzmann method. *Computers & Fluids, 52*(1), 130–138.