Article

Effects of a Guide Cone on the Flow Field and Performance of a New Dynamic Air Classifier

Qiang Li, Xinliang Mou and Ying Fang *

Abstract: A new dynamic air classifier was designed to address the problems of uneven material dispersion and high dust concentration in industrial applications of turbo air classifiers. This paper presents a study on the use of guide cones in the new dynamic air classifier. The ANSYS-Fluent 19.2 software was implemented to simulate the airflow in the dynamic air classifier, and the impact of the guide cone size on the flow field and classification performance of the dynamic air classifier was investigated. The simulation results indicated that with the increase in the guide cone height, the flow field distribution becomes reasonable and the velocity distributions become uniform. When the guide cone height is greater than twice the distance between the guide cone and the bottom of the rotor cage, there is no discernible change in the flow field distribution and classification efficiency. When the guide cone diameter is approximately 0.9 times the diameter of the rotor cage, the airflow pathline is more reasonable, and the flow field and velocity distributions are more uniform. An improper guide cone diameter and height will worsen the classification environment, resulting in a significant decline in classification performance. The material experimental and discrete phase simulation (DPM) showed that DPM can anticipate the changing trends of the cut size and classification accuracy. This study provides theoretical assistance for the structural design and optimization of an air classifier.

Keywords: dynamic air classifier; guide cone; numerical simulation; flow field; classification performance

1. Introduction

As essential equipment for powder material preparation, turbo air classifiers are extensively used in building materials, mineral processing, chemical industry, medicine, and other fields. Nevertheless, with the jump in demand for production capacity, some problems have emerged in the industrial application of the turbo air classifier [1–3]. Above all, the dust concentration in the annular area is higher. Particle classification is mainly centered in the annular area. If the material cannot be separated in time, the dust concentration in the annular area will be high, which will increase the chance of particle collision, resulting in the reduction in separation efficiency and accuracy. Secondly, with the increasing demand for production capacity, if only the feed rate on the spreading plate is increased, this will lead to an uneven distribution of materials on the spreading plate, and a thick material curtain will be formed in the annular area, which is not conducive to material classification. Some researchers have carried out structural optimization to address this problem, such as increasing the number of feeding inlets and replacing single-spreading plates with double-spreading plates [4]. However, these methods cannot fundamentally solve it due to structural limitations for the classifier. In addition, because of the asymmetry of the structure of the classifier and the air supply, the air velocity at the intersection of the air inlet and the volute is relatively too large, which leads to serious wear of the guide vanes here by the impact of particles. To address these problems, a three-separation combined air classifier composed of a dynamic airflow classifier and a static classifier was designed, as shown in Figure 1a. The static classifier plays the role of preclassification and dispersing...
materials, which can significantly reduce the content of coarse particles, thereby reducing the dust concentration in the classifying chamber. The feed is conveyed by airflow so that the material is fully dispersed in the classifier. Furthermore, compared with the traditional turbo air classifier, the new dynamic classifier has no guide vanes, which reduces the wind resistance. The spindle of the dynamic air classifier only needs to drive the rotor cage to rotate and does not disperse the material. Therefore, the spindle power is reduced and the fineness control range of end products is increased.

The structure of the classifier determines its flow field distribution, and the rationality and stability of the flow field distribution guarantee the classification performance. With the rapid development of CFD (computational fluid dynamics) techniques, numerical simulation has become the primary choice for researchers to optimize the structure of air classifiers because of its economy, efficiency, and time savings. At present, structural optimization mainly focuses on key components and auxiliary components. The key components have always been the focus of research, which mainly include the rotor cage and guide vanes. Ren et al. [5] designed and simulated a rotor cage with nonradial arc rotor blades. The simulation showed that the flow field distribution in the rotor cage is significantly improved and the inertial vortex between the blades is eliminated. The experimental results of the material classification revealed that the classification accuracy is improved by 10.6~40.8%. Mou found that although the greater the number of blades, the better the flow field, the channel will be too narrow, which affects the discharge of the fine powder [6]. Li et al. [7–10] performed numerical simulations on several specially shaped rotor cage blades. According to the research, the blade shape had a greater impact on the flow field distribution and the classification performance between the rotor cages,

Figure 1. The three-separation combined air classifier (a) and the 3D view of the dynamic air classifier (b): 1—air and material inlet, 2—coarse powder outlet, 3—guide cone, 4—cone area, 5—classifying chamber, 6—rotor cage, 7—air and fine powder outlet, 8—transmission shaft.
and they optimized and improved the blade shape to obtain the optimal blade structure. Bauder et al. studied the effect of the rotor cage geometry on classification efficiency. The results indicated that the rotor cage with radial blades can separate more finely and more sharply than the inclined blades [11]. Liu et al. [12] used CFD techniques to discuss the axial inclined angle of the guide vane. The simulation results revealed that the guide vane with the appropriate axial inclination can decrease the axial velocity in the annular region and increase the tangential velocity, which is conducive to the stability of the flow field. Huang improved straight guide vanes to positively bowed guide vanes and found that the positively bowed guide vane can stabilize the flow field and increase the classification accuracy through numerical simulation [13].

Although auxiliary components are not as significant as key components, they play a crucial function in improving classification performance. Wu et al. [4] designed a new double-layer spreading plate and the discrete phase simulation of the single-layer and double-layer spreading plates was carried out by CFD techniques which can improve the dispersion of materials, thereby improving the classification effect. Wang [14] used the numerical simulation method to compare the effect of the presence or absence of the disturbing cone on the flow field, and showed that the disturbing cone reduces the velocity fluctuation in the annular region and the rotor cage, and lessens the number of high-frequency fluctuating eddies in the elutriation region. Sun et al. [15] studied the effect of the deflector position on the classification performance. It was found that the addition of an upstream deflector can reduce energy loss and enhance the dispersion of materials. Zhou [16] added a triangular turbulence device inside the horizontal classifier to improve the secondary dispersion effect of particles. Betz et al. [17] installed flow baffles inside the rotor cage and simulated the effect of the flow baffle on the flow field. According to the research, the rotating intensity of airflow in the rotor cage is reduced and the pressure drop is decreased by the existence of flow baffles.

As one of the auxiliary components of the new dynamic air classifier, the guide cone contributes significantly to the flow field distribution. However, the action law and effect of the guide cone in the classifier have not been reported. Therefore, the gas flow in the new dynamic air classifier was simulated by using ANSYS-Fluent 19.2 software. The influence of the guide cone size on the internal flow field was investigated. The results demonstrated that a reasonable guide cone size can significantly improve the flow field distribution and classification performance. In addition, the simulation results were tested by material classification experiments. This study not only proposed a method to improve the flow field but also provided theoretical guidance for the design and optimization of classifiers.

2. Description of the Equipment and Calculation Methodology

2.1. Working Principle of the Classifier

The working principle of the three-separation combined classifier follows: after the raw materials are preclassified and predispersed by the static classifier, some coarse particles are collected and discharged from the coarse powder port of the static classifier. The remaining materials enter the dynamic air classifier with the airflow. When the airflow passes through the guide cone, some coarse particles collide with the guide cone due to inertial action, lose kinetic energy, and fall along the cylinder wall, and are discharged from the medium-coarse powder outlet. The remaining materials enter the classification area of the rotor cage with the airflow. The airflow and particles dispersed in the airflow rotate at a high speed driven by the rotor cage. At this point, these particles are mainly affected by gravity, centrifugal force, and air drag force. Because the centrifugal force on the coarse particles is greater than the air drag force, the coarse particles move toward the cylinder wall and hit the cylinder wall, losing kinetic energy and falling under the action of gravity to be collected as the medium-coarse powder. The centrifugal force on the fine particles is less than the air drag force, they are consequently carried by the airflow through the rotor cage blades, leave the dynamic air classifier, and are collected as the fine powder through the cyclone.
The guide cone that existed in the dynamic air classifier does not affect the flow field of the static classifier. Therefore, in the following work, the dynamic air classifier is simulated and analyzed. The 3D view of the dynamic air classifier is presented in Figure 1b.

2.2. Model Description and Simulation Conditions

The main structure of the dynamic air classifier was geometrically modeled using SolidWorks 2021 software. The geometric model and the main parameters for the classifier are presented in Figure 2a. The number of blades is 36, the length is 20 mm, the thickness is 2 mm, and the height is 150 mm, and the blades are evenly distributed radially along the circumference of the rotor cage with a diameter of 256 mm.

![Figure 2. Dimensions (a) and CFD grid (b) of the dynamic air classifier.](image)

The computational model was divided into six regions: air inlet, guide cone, coarse powder outlet, classifying chamber, rotor cage, and fine powder outlet. A tetrahedral grid was used to mesh the irregular coarse powder outlet region, and a hexahedral grid was used for the other regular regions, as shown in Figure 2b. To balance the computational consumption and simulation accuracy, it is necessary to verify the mesh independence after meshing. Five grid sizes were examined under the same conditions with the average static pressure drop between the inlet and outlet as a criterion. The results are shown in Table 1. The results demonstrate that when the number of grids exceeds 1.80 million, there is no distinct difference in the pressure drop. Considering the calculation time and accuracy, 1.80 million meshes were finally selected for the model. Table 2 lists the number of grids in each region.

![Figure 2. Dimensions (a) and CFD grid (b) of the dynamic air classifier.](image)

| Table 1. Effect of the number of meshes on the average static pressure drop. |
|-----------------------------|-----------------|-----------------|-----------------|-----------------|-----------------|
| Mesh Number (million)       | 0.71            | 1.27            | 1.80            | 2.34            | 2.84            |
| ΔP (Pa)                     | 186.56          | 209.81          | 228.72          | 230.14          | 229.81          |
Table 2. Number of meshes in each region.

| Region                  | Mesh Number |
|-------------------------|-------------|
| Air inlet               | 204204      |
| Guide cone              | 307642      |
| Coarse powder outlet    | 359491      |
| Classifying chamber     | 150284      |
| Rotor cage              | 432288      |
| Fine powder outlet      | 342720      |
| Sum                     | 1796618     |

The Reynolds stress model (RSM) was selected for the turbulence model according to the structural characteristics of the dynamic air classifier and the calculation accuracy requirements. Nonslip boundary conditions were adopted for the wall boundary, and the standard wall functions were used for the near-wall surface. The SIMPLEC algorithm was used for the pressure–velocity coupling, and the QUICK difference scheme was adopted for convection and diffusion. The air inlet velocity was 14 m/s and the rotation speed was 500 rpm. The air inlet was defined as a “velocity-inlet” boundary condition and the outlet was defined as an “outflow”. The multiple reference frame model (MRF) was used to simulate the rotational motion of the rotor cage and the rotation direction was clockwise.

2.3. Mathematical Model
2.3.1. Turbulence Model

In the present study, ANSYS-Fluent 19.2 was performed for three-dimensional steady-state simulation. For the incompressible flow, the mass and momentum equations are as follows:

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

\[
\rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ u \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \right] - \rho \frac{\partial u_i'u_j'}{\partial x_j}
\]

where \( \rho \) is the gas density, \( u_i \) is the fluid velocity, \( x_i \) is the position, \( \mu \) is the gas viscosity, \( p \) is the static pressure, and \(-\rho \frac{\partial u_i'u_j'}{\partial x_j}\) is the Reynolds stress term.

The Reynolds stress model (RSM) is a model suitable for describing the anisotropic flow in air classifiers. It can predict the turbulent structure and pressure drop inside the air classifier more accurately [18,19]. Hence, considering the flow characteristics of the classifier, the calculation time, and accuracy, the Reynolds stress model (RSM) was selected as the turbulence model in this paper. The transport equation for the Reynolds stress can be written as:

\[
\frac{\partial}{\partial t} \left( \rho \overline{u_i u_j} \right) + \frac{\partial}{\partial x_k} \left( \rho \overline{u_i u_j} \overline{u_k u_j} \right) = D_{ij} + \phi_{ij} + C_{ij} + \varepsilon_{ij}
\]

\[
D_{ij} = -\frac{\partial}{\partial x_k} \left( \rho \overline{u_i u_j} u_k + \rho \overline{u_i' u_j} \delta_{ik} + \rho \overline{u_i u_j} \delta_{jk} - \mu \frac{\partial u_i}{\partial x_k} \overline{u_j u_j} \right)
\]

\[
\phi_{ij} = \rho \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right)
\]

\[
C_{ij} = \rho \left( \frac{\partial U_i}{\partial x_k} \frac{\partial U_j}{\partial x_k} + \frac{\partial U_i}{\partial x_k} \frac{\partial U_k}{\partial x_j} \right)
\]

\[
\varepsilon_{ij} = 2\mu \frac{\partial u_i}{\partial x_k} \frac{\partial u_j}{\partial x_k}
\]

The two terms on the left side of Equation (3) are the time rate of change in the Reynolds stress and the convection term, respectively. The four terms on the right are
the diffusion term $D_{ij}$, pressure strain term $\varphi_{ij}$, stress generation term $G_{ij}$, and turbulent dissipation term $\varepsilon_{ij}$. $D_{ij}$ appears as divergence and is conservative. It includes three parts: the turbulent diffusion term, pressure diffusion term, and viscosity diffusion term. $\varphi_{ij}$ is composed of turbulent pressure and turbulent strain, also known as the Reynolds stress redistribution term. $G_{ij}$ represents the interaction between Reynolds stress and the mean flow gradient, which improves the source of Reynolds stress. For incompressible flow, $G_{ij} = 0$. $\varepsilon_{ij}$ is composed of the fluid viscosity coefficient and turbulent velocity gradient, and its main function is to consume turbulent energy.

### 2.3.2. Discrete Phase Model

The premise of using the discrete phase model (DPM) is that the volume proportion of the discrete phase is less than 10% of the total volume [20,21]. In the Lagrangian coordinate system, DPM predicts the trajectory of particles by tracking the particles dispersed in the gas flow field. The particle state in a series of integration time steps under the condition of a continuous phase flow field at a certain time can be obtained in a steady-state way, and a series of particle positions can be connected into a motion trajectory. In this paper, the final state of particles (particles reach the coarse powder or fine powder outlet) was obtained. This is a one-way coupling method. Since the volume loading rate of particles is very small, the influence of particles on the flow field is ignored. The particle motion equation follows:

$$\frac{d\vec{u}_P}{dt} = F_D (\vec{u} - \vec{u}_P) + \vec{g} \left( \rho_P - \rho \right) \frac{\rho_P}{\rho}$$  \hspace{1cm} (8)

where $\vec{u}_P$ is the particle velocity, $t$ is the time, $\vec{u}$ is the gas velocity, $\vec{g}$ is the gravitational acceleration, $\rho_P$ is the particle density, $\rho$ is the fluid density, and $F_D$ is the drag force, which is given by

$$F_D = \frac{18 \mu C_D \rho_P}{\rho_D D_P^2}$$  \hspace{1cm} (9)

where $\mu$ is the molecular viscosity of the gas, $D_P$ is the particle diameter, $C_D$ is the drag coefficient, and $\rho_P$ is the relative Reynold number, which is derived by

$$Re_P = \frac{\rho D_P |\vec{u}_P - \vec{u}|}{\mu}$$  \hspace{1cm} (10)

### 2.4. Experimental Setup

The experimental classification system consists of the new dynamic air classifier, static classifier, cyclone collector, pulse bag filter, induced draft fan, and control system. Under the action of the induced draft fan, the whole classification system becomes a negative pressure state. Air is sucked into the classification system from the air inlet of the static classifier due to the negative pressure. The particles to be classified pass through each classification equipment in turn by airflow transport. The coarse powder is collected in the static classifier and dynamic air classifier, and the fine powder is collected in the cyclone and the bag filter. The classification system is shown in Figure 3a. As the standard, the pressure drop ($\Delta P$) is determined by the static pressure difference at both ends of the classifier and was measured by a U-type manometer, as presented in Figure 3b. The particle size of the sample was measured by a Mastersizer 2000 laser particle sizer.
The classification performance was evaluated by the cut size \( d_c \) and classification accuracy \( K \); \( d_c \) is the particle size at a partial classification efficiency of 50\%, and a smaller \( d_c \) represents better classification performance. \( K \) is the ratio of particle size when the partial classification efficiency is 25\% and 75\%. The steeper the partial classification efficiency curve is, the larger the \( K \) value, and the better the classification performance.

3. Simulation Results and Analysis

The particle classification mainly takes place between the classification chamber and the inner surface of the cage. The cut particle size \( d_c \) in the classification area can be obtained by the following formula:

\[
d_c = \frac{3C_D \rho g R v_r^2}{4(\rho_p - \rho_g)v_t^2}
\]

(11)

where \( C_D \) refers to the drag coefficient, \( \rho_g \) refers to the gas density, \( \rho_p \) refers to the particle density, \( R \) refers to the semidiameter of the rotor cage, \( v_r \) refers to the radial velocity, and \( v_t \) refers to the tangential velocity. From Equation (11), when the radial velocity and tangential velocity are uniformly distributed and within a reasonable range, an ideal cut size can be obtained in the classification area. Therefore, it is necessary to analyze the radial and tangential velocities in the classifying chamber and the rotor cage. Secondly, after the particles are classified, it is also necessary to ensure that the coarse and fine powders can be discharged smoothly; thus, the airflow pathlines and axial velocity should be consequently analyzed.

3.1. Effect of the Guide Cone Height on the Flow Field

The distance between the guide cone and the bottom of the rotor cage \( h \) was 40 mm. In order to investigate the influence of the guide cone height on the flow field distribution, three kinds of guide cone heights with \( H = 64 \) mm, \( H = 84 \) mm, and \( H = 104 \) mm were simulated. The ratios of the guide cone height to the \( h \) were 1.6:1.0, 2.1:1.0, and 2.6:1.0. The simulated working conditions were set as 14 m/s-500 rpm.

3.1.1. Airflow Pathline Distribution

The dynamic air classifier is symmetrical about the Z-axis, hence the Y = 0 section is discussed in the following study. The airflow pathline distribution for different guide cone heights is presented in Figure 4. When \( H = 64 \) mm, the airflow first moves to the
cylinder wall after passing through the guide cone, then moves upward along the cylinder wall, and turns to enter the rotor cage after being blocked by the upper cylinder wall. Most airflow enters the rotor cage horizontally from the upper part of the rotor cage. Furthermore, a large vortex is formed near the lower part of the rotor cage, which prevents the airflow from entering and reduces the effective classification height of the rotor cage. The airflow interaction between the rotor cage and the classifying chamber is evident. When \( H = 84 \text{ mm} \), the airflow pathline changes significantly compared with \( H = 64 \text{ mm} \), as shown in Figure 4b. The airflow does not move to the cylinder wall after passing through the guide cone but moves upward along the rotor cage and then enters the rotor cage obliquely upward. Compared with \( H = 64 \text{ mm} \), the vortex for \( H = 84 \text{ mm} \) is formed at the bottom of the rotor cage, and its vortex shape is greatly reduced, which greatly reduces the interaction with the airflow in the classifying chamber. In addition, there is a vortex near the cylinder wall, and its velocity direction is downward along the cylinder wall, which is conducive to the elutriation and discharge of coarse powder [14]. When the height of the guide cone increases to 104 mm, the streamline distribution is nearly the same as that for \( H = 84 \text{ mm} \), which indicates that the airflow pathline is basically unchanged when the guide cone reaches a certain height.

![Figure 4](image-url)

Figure 4. Effect of the guide cone height on airflow movement: (a) \( H = 64 \text{ mm} \), (b) \( H = 84 \text{ mm} \), (c) \( H = 104 \text{ mm} \).

Figure 5 shows the axial velocity distribution for the \( Z = -37.5 \) section, and the positive value represents the movement of the airflow toward the fine powder outlet. When \( H = 64 \text{ mm} \), the axial velocity fluctuates greatly, and the velocity decreases first and then increases from the cylinder wall to the rotor cage. The minimum velocity between the cage blades is almost 0, indicating that the airflow at this height enters the rotor cage horizontally. Note that the axial velocity near the cylinder wall reaches 4 m/s, and the velocity direction remains upward. Excessive positive axial velocities makes it difficult for the coarse powder to fall and be collected. When \( H = 84 \text{ mm} \) and \( H = 104 \text{ mm} \), the distributions of the axial velocities are generally identical. The velocity from the wall to the center of the cage keeps increasing. Both have a large and stable axial velocity near the rotor cage, which ensures the rapid discharge of the fine powder. Furthermore, the axial velocity near the cylinder wall is approximately \(-1 \text{ m/s}\), which is beneficial to the discharge of the coarse powder in time.
When $H = 64$ mm, the radial velocity gradient is large, and the velocity decreases from the upper part of the classifying chamber, indicating that most of the airflow enters the rotor cage in the classifying chamber and the outer surface of the rotor cage is the only way for the airflow to flow to the outlet. We took points at the entrance of the rotor cage and reduced the classification efficiency. Compared with the velocity near the wall, the radial velocities at both heights become positive when the powder powder discharge.

Figure 5. Effect of the guide cone height on the axial velocity ($Z = -37.5$).

3.1.2. Effects on the Radial Velocity and Tangential Velocity

The uniformity of the radial and tangential velocity distributions is a guarantee for obtaining high classification accuracy products [22,23].

Figure 6 shows the radial velocity distribution cloud for different guide cones. The negative value represents that the airflow moves radially to the center of the rotor cage. When $H = 64$ mm, there is a large radial velocity in the upper part of the classifying chamber, indicating that most of the airflow enters the rotor cage in the classifying chamber and the upper part of the cage. The radial velocity distributions for $H = 84$ mm and $H = 104$ mm are almost the same and relatively uniform. The outer surface of the rotor cage is the only way for the airflow to flow to the outlet. We took points at the entrance of the rotor cage along the axial direction to obtain the radial velocity distribution, as shown in Figure 7.

When $H = 64$ mm, the radial velocity gradient is large, and the velocity decreases from the top to the bottom of the rotor cage. Note that the maximum radial velocity is at the top of the rotor cage, reaching more than $-10$ m/s, which leads the coarse powder to be carried into the rotor cage and reduces the classification efficiency. Compared with the velocity change for $H = 64$ mm, the radial velocity distributions for $H = 84$ mm and $H = 104$ mm are more uniform, and the velocity variations are about 6.5 m/s. In particular, when the rotor cage height is 10–90 mm, the variation in the radial velocity is smallest, which remains at approximately $-6$ m/s. The radial velocities at both heights become positive when the rotor cage height is about 119 mm, hence the effective classification height of the rotor cage is 119 mm, which is approximately 15% higher than that for $H = 64$ mm.
Figure 6. Effect of the guide cone height on the radial velocity: (a) $H = 64$ mm, (b) $H = 84$ mm, (c) $H = 104$ mm.

Figure 7. Effect of the guide cone height on the radial velocity of the rotor cage entrance.

Figure 8 shows the tangential velocity distribution for the $Y = 0$ section. As shown in Figure 8a, the tangential velocity changes greatly at the classifying chamber and the upper end of the guide cone, which is caused by the existence of vortices at these two places. However, there is no obvious difference between the tangential velocity distribution clouds for $H = 84$ mm and $H = 104$ mm, and the tangential velocity distributions of the classifying chamber and the upper end of the guide cone are relatively uniform. Figure 9a shows the tangential velocity distribution for the rotor cage entrance. It can be seen from the figure that when $H = 64$ mm, the tangential velocity increases from the top to the bottom of the rotor cage, with its value varying by 4.5 m/s. The tangential velocity at the bottom of the rotor cage reaches $-6.1$ m/s, which is caused by the high-speed rotating vortex in the classifying chamber. When $H = 84$ mm and $H = 104$ mm, the tangential velocity changes are basically the same. Although the velocity variations are also large (this is caused by
the vortex at the bottom of the rotor cage, which is unavoidable), the tangential velocity changes smoothly at the middle and upper parts of the rotor cage (within the effective classification height of the rotor cage), and the velocity is maintained at approximately \(-1.4\) m/s.

\[
\nu_t = \frac{2\pi R n}{60}
\]  

(12)

where \(R\) is the diameter of the rotor cage and \(n\) is the speed of the rotor cage. Substituting the value into the calculation formula gives \(\nu_t = 6.7\) m/s. The tangential velocity for the outer surface of the rotor cage is significantly less than the linear velocity for the outer edge of the rotor cage. The reason is that the tangential velocity of the airflow is generated by the rotation of the rotor cage, while the air has a lag motion due to its small viscosity. However, as the airflow enters the rotor cage channel, the tangential velocity gradually increases and remains stable in the axial direction, as shown in Figure 9b.

According to this analysis, it can be concluded that the increase in the guide cone height is conducive to increasing the effective classification height of the rotor cage and
significantly improving the distribution of the radial velocity and the tangential velocity on the outer surface of the rotor cage and the classifying chamber. When the guide cone height increases to a certain value, the velocity distribution is almost unchanged.

3.1.3. Discrete Phase Simulation Results and Analysis

From the one-phase flow simulation results described above, the flow field with \( H = 84 \text{ mm} \) and \( H = 104 \text{ mm} \) is reasonable and uniform. To further verify the analysis, the discrete phase model (DPM) was used to calculate the partial classification efficiency for different guide cone heights. Calcium carbonate particles with a density of 2750 kg/m\(^3\) enter from the air inlet. Particle trajectories were simulated by tracking 443 particles of each size. The partial classification efficiency curves (Tromp curves) under different operating conditions are presented in Figure 10. As shown in the figure, the Tromp curves for \( H = 84 \text{ mm} \) and \( H = 104 \text{ mm} \) are almost identical, indicating that there is no significant difference in the classification performance. However, the cut size of these two heights is significantly smaller than that for \( H = 64 \text{ mm} \). In addition, the Tromp curves of both are steeper than that for \( H = 64 \text{ mm} \), which means the classification accuracy is higher than that for \( H = 64 \text{ mm} \). Therefore, the classification performance for \( H = 84 \text{ mm} \) and \( H = 104 \text{ mm} \) is better than that for \( H = 64 \text{ mm} \). This can be explained by the fact that when \( H = 64 \text{ mm} \), the vortex in the classifying chamber causes the radial velocity in the classifying zone to be larger and the velocity distribution to be nonuniform, which leads to particle back-mixing. Furthermore, the excessive axial velocity near the cylinder wall causes the coarse powder that has been separated to be brought into the classification zone again, resulting in a decrease in the classification efficiency. After simulation studies, when the guide cone height exceeds 79 mm, the flow field and classification efficiency are virtually identical, indicating that the critical value of the guide cone height is 79 mm, which is about twice the distance between the guide cone and the bottom of the rotor cage.

![Figure 10. Tromp curves of classifiers with different guide cone heights: (a) 14 m/s-500 rpm, (b) 14 m/s-600 rpm, (c) 16 m/s-500 rpm.](image-url)
3.2. Effect of the Guide Cone Diameter on the Flow Field

To investigate the effect of the guide cone diameter on the flow field, four guide cones with different diameters, 196, 216, 236, and 256 mm, were simulated, and the guide cone height was kept at 104 mm. Note that the diameter of the rotor cage is also 256 mm. The operating conditions were 14 m/s-500 rpm.

3.2.1. Airflow Pathline Distribution

The airflow pathlines with different diameters are presented in Figure 11. When the diameter of the guide cone (D) is 196 mm, the airflow hits the bottom of the rotor cage after passing through the guide cone due to its small transverse guiding effect. When encountering obstacles, it first moves to the cylinder wall and then enters the rotor cage at the upper part of the classifying chamber. Note that the vortex center is located in the classifying chamber. When D = 216 mm, the airflow moves upward from the classifying chamber and the cylinder wall after passing through the guide cone, and then turns into the rotor cage, and its vortex center is located between the blades of the rotor cage. When R = 216 mm, after entering the grading chamber, the main airflow mainly moves upward from the middle of the classifying chamber and then enters the rotor cage. The vortex center of D = 216 mm is located between the blades of the rotor cage. When D = 236 mm, the airflow moves upward along the rotor cage after passing through the guide cone due to the increase in the transverse guiding effect of the guide cone, and then enters the rotor cage obliquely upward. Moreover, the vortex center is transferred to the bottom of the rotor cage, and the vortex shape is smaller than the other three. When D = 256 mm, due to the excessive transverse guiding effect, the airflow moves directly to the cylinder wall after passing through the guide cone, and its pathline is similar to that for D = 196 mm. The vortex center is shifted to the classifying chamber again, and its shape is the largest. Based on this analysis, the airflow pathline with D = 236 mm is the most reasonable.

![Figure 11. Effect of the guide cone diameter on airflow movement: (a) D = 196 mm, (b) D = 216 mm, (c) D = 236 mm, (d) D = 256 mm.](image)

From Figure 12, the axial velocity distribution for D = 196 mm is consistent with that for D = 256 mm, with dramatic fluctuations. The axial velocity near the rotor cage remains negative, which is not conducive to the discharge of the fine powder. In addition, the axial velocity near the cylinder wall reaches more than 5 m/s, which causes resistance to the collection of the coarse powder. When D = 236 mm, the axial velocity increases from the cylinder wall to the center of the rotor cage. Importantly, the axial velocity near the cylinder wall remains negative, which is favorable to the fall and collection of the coarse powder. The axial velocity distribution for D = 216 mm is intermediate.
Effect of the guide cone diameter on the radial velocity: (a) $D = 196$ mm, (b) $D = 216$ mm, (c) $D = 236$ mm, (d) $D = 256$ mm.

Figure 12. Effect of the guide cone diameter on the axial velocity ($Z = -37.5$).

3.2.2. Effects on the Radial Velocity and Tangential Velocity

As shown in Figure 13, when $D = 196$ mm and $D = 256$ mm, the airflow in the lower part of the classifying chamber has a larger positive value, indicating that the airflow moves toward the cylinder wall. Figure 14 compares the radial velocity distribution on the rotor cage entrance with different diameters. When $D = 236$ mm, the effective classification height of the rotor cage is higher than the other three and its radial distribution remains the most uniform. However, the radial velocities for $D = 196$ mm and $D = 256$ mm increase continuously from the top to the bottom of the rotor cage, and the velocity variation is more than 16 m/s. In addition, the radial velocity at the top of the rotor cage reaches 12 m/s, which significantly increase the probability of collecting coarse particles as the fine powder, thus reducing the classification performance. $D = 216$ mm still maintains the intermediate state.

Figure 13. Effect of the guide cone diameter on the radial velocity: (a) $D = 196$ mm, (b) $D = 216$ mm, (c) $D = 236$ mm, (d) $D = 256$ mm.
Figure 14. Effect of the guide cone diameter on the axial velocity of the rotor cage entrance.

Figure 15 shows the tangential velocity cloud. Owing to the existence of the rotating vortex, when D = 196 mm and D = 256 mm, the tangential velocity changes greatly in the classifying chamber and near the rotor cage. The tangential velocity distribution for D = 236 mm is relatively uniform, since the vortex with a small shape is at the bottom of the rotor cage and has little influence on the flow field. The tangential velocity distribution for the rotor cage entrance is presented in Figure 16. Influenced by the vortex, the four diameters have large velocity gradients. However, in the middle and upper parts of the rotor cage, the tangential velocity distribution for D = 236 mm remains stable, and the velocity is maintained at approximately −1.4 m/s. The tangential velocities for D = 196 mm and D = 256 mm continuously increase from the top to the bottom of the rotor cage and the variation reaches approximately 4.5 m/s.

Figure 15. Effect of the guide cone diameter on the tangential velocity: (a) D = 196 mm, (b) D = 216 mm, (c) D = 236 mm, (d) D = 256 mm.
The optimum range of the guide cone diameter is 220–242 mm, which is approximately 0.9 times the rotor cage diameter. After simulation studies, when the guide cone diameter is between 220 and 242 mm, there are no significant differences in the flow field and partial classification efficiency. Hence, the optimum range of the guide cone diameter is 220–242 mm, which is approximately 0.9 times the rotor cage diameter.

3.2.3. Discrete Phase Simulation Results and Analysis

Tromp curves under different operating conditions are presented in Figure 17. It can be observed from the figure that the cut size for \( D = 236 \) mm is significantly smaller than that of the other three diameters, and the cut sizes for \( D = 196 \) mm and \( D = 256 \) mm are the largest. Moreover, the Tromp curve for \( D = 236 \) mm is the steepest compared to the other three diameters, which means that its classification accuracy is better. The cut size and classification accuracy for \( D = 216 \) mm are in the intermediate state, which accords with the results of the flow field analysis. Therefore, compared to \( D = 196 \) mm, \( D = 216 \) mm, and \( D = 256 \) mm, \( D = 236 \) mm is the best guide cone diameter. The discrete phase simulation results are consistent with the one-phase flow simulation results. After simulation studies, when the guide cone diameter is between 220 and 242 mm, there are no significant differences in the flow field and partial classification efficiency. Hence, the optimum range of the guide cone diameter is 220–242 mm, which is approximately 0.9 times the rotor cage diameter.

![Figure 16. Effect of the guide cone diameter on the tangential velocity of the rotor cage entrance.](image-url)
4. Comparison of the Simulation and Experiment Results

In order to verify the reliability of the simulation results, material classification experiments were carried out, and the simulation results were compared with the experimental data. An optimized guide cone with 104 mm height and 236 mm diameter was chosen for the experiments. The classification efficiency and pressure drop of the experiments and simulations were compared under different working conditions. The particle size distribution of calcium carbonate as the experimental raw material is presented in Table 3. After each test, the coarse and fine powders were weighed and the particle size of those powders was analyzed by a Mastersizer 2000 laser particle sizer. The pressure drop was measured using a U-type manometer.

Table 3. Raw material particle size distribution.

| Particle Size (μm) | Differential Distribution (%) |
|-------------------|------------------------------|
| 0–10              | 6.4                          |
| 10–20             | 15.2                         |
| 20–30             | 19.7                         |
| 30–45             | 28.9                         |
| 45–60             | 17.1                         |
| 60–75             | 10.6                         |
| >75               | 2.1                          |

A comparison between the simulated and measured pressure drops is presented in Table 4. As the inlet air velocity increases, the pressure drop gradually increases.
average relative error between the experiment and simulation is 5.28%, which shows good agreement. The reason for the relative error is that some complex structures in the classifier were simplified when the simulation model was created. In addition, the simulation was in an ideal state without external interference. The good agreement also verifies the reliability of the simulation model.

Table 4. Comparison of the pressure drop between the experiment and simulation.

| v (m/s)–n (rpm) | Simulation Value (Pa) | Experimental Value (Pa) | Relative Error (%) |
|-----------------|-----------------------|-------------------------|--------------------|
| 12–500          | 184.8                 | 190.5                   | 3.1                |
| 14–500          | 228.7                 | 241.3                   | 5.5                |
| 16–500          | 278.0                 | 292.8                   | 5.3                |
| 18–500          | 335.4                 | 359.6                   | 7.2                |

Figure 18 shows the partial classification efficiency curves (Tromp curves) of the simulation and experiment. It can be seen that there are some differences between the experimental and simulated Tromp curves. The experimental Tromp curves are S-shaped, which is called the fish-hook effect [24,25]. The effect indicates that the classification efficiency first decreases and then increases with increasing particle size in the small range of particle sizes. The DPM model does not predict this effect, since the interaction between particles is ignored in the model. As shown in the Tromp curve obtained by simulation in the figure, the classification efficiency of particles increases with increasing particle size, and there is no fish-hook effect. A specific numerical comparison between the simulated results and the experimental data is given in Table 5. The cut size ($d_c$) and classification accuracy (K) obtained by simulation are better than the experimental values. This can be explained by the fact that the DPM does not take into account the collision and agglomeration between particles. It is worth noting that the $d_c$ and K of the simulation with the change in operating conditions have an identical trend to the experiment. Therefore, the DPM model is helpful for the structural optimization of the dynamic air classifier.
Table 5. Comparison of $d_c$ and $K$ between the experiment and simulation.

| $v$ (m/s)–n (rpm) | Experimental Value | Simulation Value |
|-------------------|--------------------|------------------|
|                   | $d_c$ ($\mu$m)    | $K$              | $d_c$ ($\mu$m) | $K$ |
| 14–500            | 31.7               | 0.62             | 27.4           | 0.68 |
| 14–600            | 28.2               | 0.65             | 22.5           | 0.73 |
| 16–500            | 36.5               | 0.56             | 31.1           | 0.65 |

5. Conclusions

In the present study, the airflow moving in the new dynamic air classifier was numerically simulated by ANSYS-Fluent 19.2 software. The influences of the height and diameter of the guide cone on the internal flow field distribution and classification performance were studied. The following conclusions are drawn:

1. With the increase in the guide cone height, the flow field tends to be stable. When the height reaches the critical value, the flow field distribution does not change with increasing height. With the increase in the guide cone diameter, the uniformity of the flow field first increases and then decreases, indicating that there is an optimal range of the guide cone diameter. When the height and diameter of the guide cone are not within a reasonable range, there is a large axial velocity near the cylinder wall, and the radial velocity in the upper part of the rotor cage is too large, which worsens the classifying conditions. In contrast, when the height and diameter of the guide cone are within a reasonable range, there is a small reverse axial velocity near the cylinder wall, which contributes to the fall and collection of the coarse powder. In addition, the radial velocity and tangential velocity are evenly distributed in the classification area, which helps to improve the classification performance.

2. The vortex inside the classifier cannot be completely eliminated due to structural reasons, and the influence of the vortex can only be reduced through structural improvement. In this study, the diameter and height of the guide cone are within a reasonable range, which can greatly reduce the effect of the vortex on the flow field.

3. The material experiment results reveal that although DPM has errors with the experimental results, it can predict the changing trend of the $d_c$ and $K$, which shows that DPM has good applicability to the new dynamic airflow classification machine.

Author Contributions: Y.F. provided the design idea of the new classifier and supervised the research. Q.L. was responsible for the analysis of simulation and experimental results, writing, and editing the manuscript. X.M. was responsible for model establishment and model verification. All authors have read and agreed to the published version of the manuscript.

Funding: This work was supported by the Priority Academic Program Development of Jiangsu Higher Education Institutions (PAPD).

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Not applicable.

Conflicts of Interest: The authors declared no potential conflict of interest with respect to the research, authorship, and/or publication of this article.

References

1. Jiang, D.; Tao, R. Study on the effect of blade structure of O-SEPA classifier on the flow field in the classification area. Mech. Eng. Autom. 2014, 6, 22–23+26.
2. Petrus, H.T.B.M.; Hirajima, T.; Oosako, Y.; Nonaka, M.; Sasaki, K.; Ando, T. Performance of dry-separation processes in the recovery of cenospheres from fly ash and their implementation in a recovery unit. Int. J. Miner. Process. 2011, 196, 15–23. [CrossRef]
3. Li, H.; Lei, W. Introduction to the vertical rotor and horizontal rotor powder classifiers. Cement 2015, 12, 31–36.
4. Wu, S.; Liu, J.; Yu, Y. Design of a new double-layer spreading plate for a turbo air classifier. *Powder Technol.* **2017**, *312*, 277–286. [CrossRef]

5. Ren, W.; Liu, J.; Yu, Y. Design of a rotor cage with non-radial arc blades for turbo air classifiers. *Powder Technol.* **2016**, *292*, 46–53. [CrossRef]

6. Xinliang, M.; Fangchao, J.; Ying, F.; Chuanwen, C. CFD-Based Structural Optimization of Rotor Cage for High-Efficiency Rotor Classifier. *Processes* **2021**, *9*, 1148.

7. Tong, C.; Li, S.; Qi, H.; Ren, C. Analysis of the influence of the blade structure of vertical mill separator on the velocity field in the classification area. *Chem. Ind. Eng. Prog.* **2012**, *31*, 778–784.

8. Li, J.; Li, S.; Ren, C.; Li, H. Numerical simulation and experimental study on special-shaped blades of vortex classifier. *J. China Powder Technol.* **2009**, *15*, 1–4.

9. Ren, W.; Liu, J. From the source Design of rotating cage of circular arc blade of vortex air classifier. *Chin. J. Mech. Eng.* **2016**, *52*, 195–201. [CrossRef]

10. Li, G.; Han, F.; Nie, W. Numerical simulation of flow field in a new blade turbo classifier. *J. Wuhan Univ. Technol.* **2006**, *2*, 32–35.

11. Bauder, A.; Müller, F.; Polke, R. Investigations concerning the separation mechanism in deflector wheel classifiers. *Int. J. Miner. Process.* **2004**, *74*, S147–S154. [CrossRef]

12. Liu, R.; Liu, J.; Yu, Y. Effects of axial inclined guide vanes on a turbo air classifier. *Powder Technol.* **2015**, *280*, 1–9. [CrossRef]

13. Huang, Q.; Liu, J.; Yu, Y. Turbo air classifier guide vane improvement and inner flow field numerical simulation. *Powder Technol.* **2012**, *226*, 10–15. [CrossRef]

14. Wang, L.; Liu, J.; Zhao, K. Influence of a disturbing cone on the flow field and particle classification performance in a vertical turbo air classifier. *J. Beijing Univ. Chem. Technol.* **2021**, *48*, 87–97.

15. Sun, Z.; Sun, G.; Xu, J. Effect of deflector on classification performance of horizontal turbo classifier. *China Powder Sci. Technol.* **2016**, *22*, 6–10.

16. Zhou, Y.; Sun, G.; Jiang, S. Classification Device: CN 206240790U. Available online: [http://njtech.patsev.com/access/toLogin](http://njtech.patsev.com/access/toLogin) (accessed on 13 June 2017).

17. Betz, M.; Gleiss, M.; Nirschl, H. Effects of Flow Baffles on Flow Profile, Pressure Drop and Classification Performance in Classifiers. *Processes* **2021**, *9*, 1213. [CrossRef]

18. Hoekstra, A.J.; Derksen, J.J.; Van Den Akker, H.E.A. An experimental and numerical study of turbulent swirling flow in gas cyclones. *Chem. Eng. Sci.* **1999**, *54*, 2055–2065. [CrossRef]

19. Guizani, R.; Mhiri, H.; Bournot, P. Effects of the geometry of fine powder outlet on pressure drop and separation performances for dynamic separators. *Powder Technol.* **2016**, *314*, 599–607. [CrossRef]

20. Elsayed, K.; Lacor, C. The effect of cyclone inlet dimensions on the flow pattern and performance. *Appl. Math. Model.* **2010**, *35*, 1952–1968. [CrossRef]

21. Sun, Z.; Sun, G.; Liu, J.; Yang, X. CFD simulation and optimization of the flow field in horizontal turbo air classifiers. *Adv. Powder Technol.* **2017**, *28*, 1474–1485. [CrossRef]

22. He, F.; Fang, Y.; Qin, J.; Chai, S. Numerical simulation of flow field characteristics in the classification area of a new eddy current classifier. *Bull. Silic.* **2016**, *35*, 4128–4132.

23. Ren, W.; Liu, J.; Yu, Y. Effect of Guide-vane on flow field in Turbo Air Classifiers. *Chem. Ind. Eng. Prog.* **2019**, *38*, 31968–33994.

24. Nageswararao, K. A critical analysis of the fish hook effect in hydrocyclone classifiers. *Chem. Eng. J.* **2000**, *80*, 251–256. [CrossRef]

25. Zhang, Y.; Liu, J. Experimental study on “fishhook effect” of vortex air classifier. *J. Beijing Univ. Chem. Technol.* **2004**, *3*, 51–54+58.