Simulation of Particle Motion Behavior in Fluidized Bed Opposed Jet Mill

Kangwei Yang1, Xuewen He1 *

1College of Mechanical and Electrical Engineering, Jiangxi University of Science and Technology, Ganzhou, Jiangxi, 341000, China

*Corresponding author’s e-mail: 6720190437@mail.jxust.edu.cn

Abstract. In order to study the influence of the internal flow field of the fluidized bed opposed jet mill on the motion behavior of particles, Computational Fluid Dynamics (CFD) and Discrete Element Method (DEM) are used for coupling calculations. By adjusting the nozzle spacing and inlet pressure, Numerical simulation is carried out on the process of particles collisions with each other after accelerating under the high-speed jet produced by the nozzle. The trajectory of the particles in the flow field of the collision area and the change of the collision state of the particles are analyzed. Finally, the best parameters are selected based on the total collision energy. The results show that the particles will gradually shift and spread during the acceleration process. The reduction of the nozzle spacing is beneficial to increase the probability of particle collisions. However, if the spacing is too small, the particles cannot be fully accelerated; the increase in inlet pressure will increase the kinetic energy of the particles, and number of collisions is almost unaffected. By comparing the total collision energy, the best-simulated preparation conditions are selected as 110mm and 1.1MPa.

1. Introduction

The fluidized bed opposed jet mill is the latest equipment in jet mill. It integrates multi-nozzle technology, fluidized bed technology, and turbine classification technology to achieve diversification of the flow field and particle fluidization. The fluidized bed opposed jet mill uses the supersonic airflow generated by the Laval nozzle to accelerate the particles and rub, collide, and collision them in the broken region. Laval nozzle, broken region, and classifier are the three core components of fluidized bed opposed jet mill. The jet mill uses the collision and friction between particles accelerated by the airflow to produce a collision effect. Compared with the spiral, target, and other old jet mills, it has the advantages of low energy consumption, low wear, and no impurities. An important direction for the preparation of ultra-fine powders.

At present, most researches on various types of jet mills are practical applications, which mainly rely on engineering practice to guide the structure design and process parameters of jet mills [1][2]. In the research of numerical simulation of jet mill, currently, Computational Fluid Dynamics (CFD) method is used to simulate the flow field conditions inside the nozzle, the broken region, the transition region, and the classification region [3][4]. CFD method only consider the internal flow field value of the jet mill and rarely consider the movement characteristics of the material particles inside the jet mill. Although the Discrete Phase Model (DPM) can effectively simulate the continuous phase flow field and particle motion in a complex two-phase flow [15][16], it ignores the most important factor in fluidized bed opposed jet mill. The interaction between the particles [5]. The use of DEM can make up for the shortcomings of DPM to obtain mutual collisions between particles [12], in the numerical simulation of jet milling, it can
accurately simulate the movement of materials and collisions with each other\cite{6}\cite{7}. Khakhalev\cite{8} used the CFD-DEM coupling method to study the influence of the mill liner on the particle dynamics parameters. Falah Alobaid\cite{9} applied this method to accurately predict the particle movement and bed pressure gradient, qualitatively analyzed the particle movement trajectory, and quantitatively analyze it. The number of particles in nozzles, volume fraction, and gas velocity distribution proves the feasibility of the coupling method. Therefore, this paper uses the CFD-DEM coupling method to obtain the movement and collision behavior of the material particles in the fluidized bed opposed jet mill, and studies the effects of nozzle spacing and inlet pressure on the movement of particles, and summarizes them.

2. Model building

2.1. Geometry model
The fluidized bed opposed jet mill uses high-pressure gas through a plurality of Laval nozzles placed opposite to produce high-speed jets to carry the material to accelerate the material. The accelerated materials collide with each other at the junction of the jets, causing collision in the particles to break and disperse the agglomerates of particles. The material powder produced by the collision fluidizes through the transition zone and enters the classification area. Under the action of the classification device, it is subjected to gravity, centrifugal force, and fluid drag force. The fine powder is collected through the discharge port of the classification wheel, and the coarse powder is under the action of gravity. Fall back to the broken region and accelerate the broken program again\cite{10}

The fluidized bed opposed jet mills has four nozzles placed at 90° at a height of 300mm. The structure of the fluidized bed opposed jet mill is shown in the figure.

![Fig.1 Fluidized bed opposed jet mill and Laval nozzle structure](image)

2.2. Gas phase physics model
Under the Euler framework, Navier-Stokes control equation is used to solve the continuous phase motion to calculate the fluid motion. Due to the turbulence in the fluidized bed jet mill, the standard k-epsilon turbulence model is used\cite{9}. The mass and momentum equations of the gas phase can be written as:

$$\frac{\partial (\rho \varphi_t)}{\partial t} + \nabla (\rho \varphi_t \mathbf{u}_t) = 0$$  \hspace{1cm} (1)
\[
\frac{\partial \left( \varepsilon \rho_c \mathbf{u}_t \right)}{\partial t} + \nabla \left( \varepsilon \rho_c \mathbf{u}_t \right) = -\varepsilon \nabla p + \nabla \left( \varepsilon \mathbf{\tau} \right) - S + \varepsilon \rho_c \mathbf{g}
\]

Where, \( \varepsilon \) is the grid porosity refers to the volume fraction of the fluid in the grid, \( \rho_c \) is the fluid density, \( \mathbf{u}_t \) is the fluid velocity, \( p \) is the fluid pressure, \( \mathbf{g} \) is the acceleration of gravity, \( \mathbf{\tau} \) is the viscous stress tensor, and \( S \) is the momentum exchange source term. The momentum exchange source direction can be calculated according to the following formula:

\[
S = \frac{\sum_{i=1}^{M} F_{d,ij}}{V_{\text{cell}}}
\]

\( M \) is the number of particles in the grid, \( V_{\text{cell}} \) is the volume of the calculated grid, and \( F_{d,ij} \) is the drag force on the particle in the grid.

### 2.3. Discrete phase physics model

The Lagrange framework uses Newton’s second law to establish force, velocity, acceleration, and other motion information for discontinuous and discrete elements. The movement of particles in the fluid is subject to gravity, buoyancy, aerodynamic resistance, pressure gradient force, additional mass force, Magnus lift, Saffman lift, and Basset force. Due to the preparation of tungsten carbide powder in this simulation, the ratio of tungsten carbide density (15.63 g cm\(^{-3}\)) to air density (0.001225 g cm\(^{-3}\)) is too large (1000~2500), only gravity, drag force, and pressure gradient force are considered, and the rest are can ignore\[11\]. The particle's translational motion equation can be written as:

\[
p m \frac{dv}{dt} = m p \mathbf{g} + F_{gp} + F_{\text{drag}} + F_c
\]

\( m_p \) is the particle mass, \( v_p \) is the translational velocity, \( F_{gp} \) is the pressure gradient force, \( F_{\text{drag}} \) is the drag force, and \( F_c \) is the collision force.

### 2.4. Particle collision model

The collision force in the fluidized bed jet mill is mainly generated by the collision between particles. The particle collision model is divided into the hardball model and softball model. The hardball model does not consider the force of contact particles and calculation directly based on the momentum conservation equation. In the softball model, the particle spheres can be overlapped, and the contact force is calculated according to the physical properties of the contacting particles, the normal overlap amount, and the tangential displacement. The overlap amount is used to characterize the deformation and incorporate it into the contact force calculation. Hertz-Mindlin softball model.

\[
F_c = \sum_{j=1}^{N} \left( F_{ij}^n + F_{ij}^t \right)
\]

The contact force is divided into \( F_{ij}^n \) (normal contact force) and \( F_{ij}^t \) (tangential contact force)

Particle rotation equation:

\[
I_p \frac{d\omega_p}{dt} = \sum_{j=1}^{N} M_{ij}
\]

\( I_p \) is the moment of inertia, \( \omega_p \) is the particle angular velocity, \( N \) is the total number of particles colliding with the particle. \( M_{ij} \) is the torque generated by the collision, \( M_{ij} \) can be written as:

\[
M_{ij} = L_{ij} \times F_{ij}^t
\]

\( L_{ij} \) is the distance from the contact point to the centroid of the particle.
2.5. Gas-solid drag model
The drag force between the gas and solid phases is an important factor in considering the coupling calculation. This paper selects the Gidaspow drag force model which is the most widely applicable in scenarios. The Ergun equation is used when the porosity ($\varepsilon$) is less than 0.8, and the Wen&Yu equation is used when the porosity is greater than 0.8.

$$\beta_i = \begin{cases} 150 \frac{(1-\varepsilon)}{\varepsilon} \frac{\mu_f}{d_p} + 1.75(1-\varepsilon) \frac{\rho_f}{d_p} \left| \mathbf{u}_f - \mathbf{v}_p \right| & \varepsilon \leq 0.8 \\ \frac{3}{4} C_d \frac{\varepsilon(1-\varepsilon)}{d_p} \left| \mathbf{u}_f - \mathbf{v}_p \right| \varepsilon^{-2.65} & \varepsilon > 0.8 \end{cases}$$ (8)

$\beta_i$ is the local momentum exchange coefficient, $\mu_f$ is the dynamic viscosity of the fluid, $d_p$ is the particle size, and $C_d$ is the drag coefficient.

2.6. Software settings
This article uses FLUENT19.0 and EDEM2020 for simulation calculations.

The fluid domain grid size should be set to more than 3 times the particle size, so set to 5mm, 6.3mm, 8mm, and 10mm respectively. The particle velocity distribution in the coupled calculation is basically the same. In order to reduce the amount of calculation, a fluid domain grid of 10mm size is used. Simulation.

The particle time step size is $1 \times 10^{-7}$ s by the Rayleigh wave method. Choose a fluid time step of $1 \times 10^{-5}$s. The total simulation time is 0.1s.

This simulation uses tungsten carbide with a particle size of 2mm to prepare 1μm ultrafine powder as a reference. The density of tungsten carbide is $15.63 \text{g} \cdot \text{cm}^{-3}$, and the mass of each single spherical particle is about 0.0658g. The Poisson's ratio is 0.24, the shear modulus is $6.8 \times 10^{11}$ Pa; the nozzle spacing is 90mm to 150mm; the inlet pressure is 0.8MPa to 1.2MPa; the speed of the classifying wheel is 3000rpm.

Table 1. Simulation conditions

| Label | Nozzle spacing (mm) | Input pressure (MPa) | Label | Nozzle spacing (mm) | Input pressure (MPa) |
|-------|---------------------|----------------------|-------|---------------------|----------------------|
| 1     | 90                  | 0.8                  | 11    | 130                 | 0.8                  |
| 2     | 90                  | 0.9                  | 12    | 130                 | 0.9                  |
| 3     | 90                  | 1.0                  | 13    | 130                 | 1.0                  |
| 4     | 90                  | 1.1                  | 14    | 130                 | 1.1                  |
| 5     | 90                  | 1.2                  | 15    | 130                 | 1.2                  |
| 6     | 110                 | 0.8                  | 16    | 150                 | 0.8                  |
| 7     | 110                 | 0.9                  | 17    | 150                 | 0.9                  |
| 8     | 110                 | 1.0                  | 18    | 150                 | 1.0                  |
| 9     | 110                 | 1.1                  | 19    | 150                 | 1.1                  |
| 10    | 110                 | 1.2                  | 20    | 150                 | 1.2                  |

3. Results and discussion
Based on the Euler-Lagrange theorem, the gas phase motion is calculated under the Euler frame, and the particle phase motion is calculated under the Lagrange frame. The influence of the optimization on particle movement and collision is obtained through multiple sets of simulations to obtain the best simulation parameters.
### 3.1. The effect of nozzle spacing and inlet pressure on particle movement and collision

![Fig 2. Particle trajectory under different nozzle spacing at 0.01s](image1)

![Fig 3. The particle trajectory in the flow field](image2)

After the high-speed jet generated by the oppositely placed nozzles passes through the nozzle outlet, the jet boundary starts to collide quickly, the air flow begins to gradually diverge, and the high-speed jet is asymmetrical under the influence of the staged rotation field and the structure of the broken region. The trajectory of the particles in the acceleration process diverges and spreads under the action of the jet. Fig 3 shows the particle trajectory at a particle spacing of 150mm and an inlet pressure of 1.0MPa. From Fig 2 and Fig 3, it can be seen that the larger the nozzle spacing, the greater the deviation and diffusion of particles, which will affect the aggregation of material particles and collision to produce a broken effect. Shuli Teng [13] found through experiments and numerical simulations that there is a particle concentration layer in the peripheral wall area of the Fluid Energy Mill (FEM) when it is working, and most of the relative collisions of particles that cause particle breakage occur in this collision area. The collision area of the jet type fluidized bed jet mill is an area centered on the intersection of the nozzle airflows. The main collisions to achieve the particle broken effect occur in the collision area where the nozzle airflows meet.

The particles start to be released from 0 seconds, and they are all released at 0.05s. The average particle velocity changes under different nozzle spacings are the same. Fig 4 shows that when the nozzle spacing is 90mm, the average particle velocity is lower than that of other nozzle spacing conditions. The reason is that the nozzle spacing is too small and the particles cannot be fully accelerated in the jet, and the number of particles colliding with each other is significantly greater than other nozzles. The reason is that the particles do not have excessive diffusion and offset during the acceleration process, and the concentrated particle distribution leads to a high probability of particle collision. Figure 5 shows that the average particle velocity gradually increases with the increase of the intake pressure, but the number of particle collisions is hardly affected by the inlet pressure.

![Fig 4. The effect of nozzle spacing on average particle velocity and number of collisions](image3)
3.2. Total particle collision energy

The energy judging method of particle broken is mainly to determine whether the particles have the kinetic energy provided by the airflow generated by the nozzle to reach the broken condition\cite{14}.

\[
E = \frac{1}{2}(m_p + m_g)v^2
\]  

(9)

\(E\) is the collision energy between particles, \(v\) is the relative velocity of the collision between particles.

The total collision energy is affected by the relative collision speed and the number of collisions of the particles in the jet mill. The nozzle spacing obtained from the section 3.1 will affect the relative collision speed and the number of collisions of the particles, and the change in inlet pressure will have almost no effect on the relative collision speed of the particles.

Table 2. The total collision energy under different simulation conditions

| Total collision energy(J) | Inlet pressure(MPa) |
|---------------------------|---------------------|
|                           | 0.8    | 0.9    | 1.0    | 1.1    | 1.2    |
| Nozzle spacing (mm)       |        |        |        |        |        |
| 90                        | 5.96   | 6.67   | 7.80   | 7.77   | 8.15   |
| 110                       | 5.59   | 6.40   | 6.54   | 7.59   | 8.23   |
| 130                       | 4.79   | 5.39   | 5.84   | 7.44   | 7.19   |
| 150                       | 4.87   | 5.22   | 5.31   | 6.68   | 6.18   |

Table 2 lists the total collision energy generated by particles collision with each other in 0.1s under various simulation conditions. It can be found from the table that in the process of increasing the inlet pressure from 0.8MPa to 1.2MPa, the total collision energy to decrease after reaching the maximum value at 1.1MP at 130mm and 150mm, and the overall upward trend at 90mm and 110mm. The total collision energy gradually increases with the decrease of the nozzle spacing, and the input pressure is 1.1 to 1.2MPa. The total collision energy generated by the nozzle spacing of 110mm over 90mm.

4. Conclusion

1. The reduction of the nozzle spacing helps prevent excessive diverges and spreads of particles, which increases the probability of particle collisions, but too small spacing results in insufficient acceleration of particles and low average velocity of particle; The increase in inlet pressure can increase the average velocity of particles, but the number of particle collisions is not affected by the inlet pressure;

2. In the numerical simulation, the total collision energy of particles reaches the maximum value when the nozzle spacing is 110mm and the inlet pressure is 1.2MPa.
3. This article currently only focuses on the numerical simulation of the process of particle motion and collision in the broken region. The subsequent work can simulate the entire process of powder preparation by using CFD-DEM to simulate particles in the transition region and classification region after particle collision.

References

[1] Zhou, X., Cheng, H.Y., Lu, C. Y., Yuan S.L., (2011) Superfine ceria powder produced by fluidized bed jet mill. Powder Metallurgy Technology, 29(01):46-49.

[2] Wang, T., Yang, S.L., Long, J.P., Ma, L., Li, J., Liao, X.J. (2017) Titanium hydride powder grinded by closed-loop jet mill. Materials Science and Engineering of Powder Metallurgy, 22(04):487-495.

[3] Wang, L.W., Pan, J.Z., Wang, Z.G., (2009) Numerical simulation of influence of the inner structure of ultrasonic nozzles on the characteristics of the air mill. Mechanics in Engineering, 31(2):17-21.

[4] Liu, F.H., Zhu, R., Dong, K., (2020) Effect of laval nozzle structure on behaviors of supersonic oxygen jet flow field. Chinese Journal of Engineering, 42(S1):54-59.

[5] Fu, P.B., (2021) DPM simulation of particle revolution and high-speed self-rotation in different pre-self-rotation cyclones. Powder Technology, 394:290-299.

[6] Khakhalev, P. A., (2014) The application of EDEM software for design parameters calculation of a ball mill lining. Research Journal of Applied Sciences, 9(11):869-973.

[7] Li, T.F., Ling, S.Y., Zhang, B., (2009) Study on collisions of steel balls in grinding mill at different rotation speeds. Journal of Central South University (Science and Technology), 50(02):251-256.

[8] Khakhalev, P. A., Bogdanov, V., Kovshechenko, V.M., (2018) Kinetic parameters of grinding media in ball mills with various liner design and mill velocity based on DEM modeling. Lop Conference, 327.

[9] Alobaid, F., Strhle, J., Epple, B., (2013) Extended CFD/DEM model for the simulation of circulating fluidized bed. Advanced Powder Technology, 24(1):403.

[10] Zhu, H.Q., HE, N.D., Ji, G.Z., (2002) Development of QLM series micro fluidized-bed counter impact air-flow mills. Metal Mine, 11:44-46.

[11] Wen, C.Y., Yu, Y.H., (1966) Mechanics of Fluidization. The Chemical Engineering Progress Symposium Series, 62:100-111.

[12] Qiu, K., Wu, F., Yang, S., (2016) Heat transfer and erosion mechanisms of an immersed tube in a bubbling fluidized bed: A LES-DEMP approach. International Journal of Thermal Sciences, 100:357-371.

[13] Teng, S.L., Wang, P., Zhu, K.J., Young, M.W., Young, C., Costas, G. (2009) Experimental and numerical analysis of a lab-scale fluid energy mill. Powder Technology, 195(1):31-39.

[14] Cui, Y., Ge, X.L., (2005) Computer stimulation of air-fluented smashing collide process. China Powder Science and Technology, 11(1):24-26.

[15] Chang, P., Xu, G., Zhou, F., Mullins, B., Abishek, S., (2019) Comparison of underground mine DPM simulation using discrete phase and continuous phase models. Process Safety and Environmental Protection, 127:45-55.

[16] Stone, L., Hastie, D., Zigan, S., (2019) Using a coupled CFD-DPM approach to predict particle settling in a horizontal air stream. Powder Technology, 30:869-878.