Analysis of Flow Characteristics of Different Shape Particles Chasing

Xiaodong Xu, Zhuoxiong Zeng*, Kaifang Guo

College of Power and Mechanical Engineering, Shanghai University of Electric Power, Shanghai, 201306, China

*Corresponding author’s e-mail: zengzhx@163.com

Abstract: in order to study the gas flow characteristics of particles with different shapes in the pursuit process, spherical, ellipsoidal and square particles are selected for numerical simulation by fluent. The results show that when the pursuit speed is the same, the streamline produced by square particles in the pursuit process is denser than that of spherical and ellipsoidal particles. Particle movement creates a certain vacuum on both sides and rear of the particle, causing the surrounding air flow to enter the areas on both sides of the particle, resulting in strong turbulence. After the rear particles catch up with the front particles, the changes of the surrounding flow field and streamline are similar to the inverse process before catching up, indicating that the particle motion has a certain regularity, and the square particles have the largest turbulent kinetic energy among the three particles.

1. Introduction
In the boiler with fluidized bed combustion, the flow of air and fuel particles on the furnace fluidized bed is gas-solid two-phase flow. The motion law of pulverized coal in the fluidized bed is very complex. Pulverized coal particles are dispersed phase, large and small, and have different shapes. The flow condition in the furnace is usually turbulent. Therefore, the research on the characteristics of gas-solid two-phase flow is the structural optimization of fluidized bed has great engineering practical value.

Di,Y.T., et al. [1] found that compared with the traditional cfd-dem method, the cfd-dem-ibm fluid structure coupling numerical simulation method can accurately consider the interaction between particles and between fluid and solid through the simulation of the settlement behavior of single, double and group particles. Liu,K., et al [2] simulated the process of coke particle settling combustion on the sub particle scale by using the finite element virtual region method, and obtained the calculation results of coke particle settling combustion. Wachs [3] simulated the settlement of a single particle in a vertical channel and proved the relationship between the resistance coefficient and Reynolds number and particle shape. Lv,K., et al [4] used the CFD DEM coupling method to numerically simulate and record the particle settlement process to verify the accuracy of the drag coefficient model. Chhabra, R.P., et al [5] simulated the drag coefficient of non spherical particles in incompressible viscous fluid. The results show that the influence of particle shape on the drag coefficient increases with the increase of Reynolds number. Qiu Yi [6] used the meshless method to simulate the drop of single particle in gas-solid two-phase flow, and obtained the results of the change of particle velocity with time in the process of particle drop.

Chang Fu et al. [7] found that the drag force on non spherical particles is quite different by comparing and calculating three kinds of drag coefficient correction relations. Huang Haibo et al. [8]...

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd
studied the settlement of elliptical particles in circular and square tubes by numerical method, and obtained the settlement curve varying with pipe diameter. Wang Kun et al. [9] studied the fluidization characteristics of turbulent fluidized bed through numerical simulation, and obtained the laws of pressure, particle velocity and particle size distribution in the bed. The moving grid method used for particle catching up draws lessons from the numerical method of vehicle catching up [10-13].

The particle motion studied above is the falling of a single particle or the flow around non-spherical particles, and the motion state of particles with different shapes has not been reported.

2. calculation model and grid division

2.1 geometric model

According to the results of reference [7], the selection of calculation area has a great impact on the force of research particles. The calculation domain used for three different shapes of particles in this paper is shown in Figure 1. The calculation region is a cuboid. After calculation, the setting of the calculation domain meets the requirements of blocking ratio [14] and will not affect the particle motion. The particles catch up from bottom to top. The particles are divided into square particles, spherical particles and ellipsoidal particles. In the past, the numerical calculation of irregular particles mostly adopts the law of approximate spherical particles.

There are two ways to realize this approximate process: ① introducing "equivalent diameter", that is, the force on irregular particles is equal to that on spherical particles with the same equivalent diameter. The definitions of the equivalent diameter of irregular particles mainly include equal volume equivalent diameter $d_v$, equal surface area equivalent diameter $d_s$ and equal specific surface area equivalent diameter $d_{sv}$. ② the sphericity is introduced and the force coefficient of particles is corrected. At present, the former calculation method is widely used, and the latter is rarely used because it is difficult to measure the sphericity of complex particles. In this paper, method ① is used for calculation. The force on the particles is regarded as the force on the spherical particles with the same equivalent diameter. Therefore, the volume of the three particles is the same, which is set as $4100 \text{mm}^3$. The spherical diameter is $R = 16 \text{mm}$, the equatorial radius of ellipsoid type is $a = 12 \text{mm}$, $b = 4.5 \text{mm}$, the polar radius is $c = 4.5 \text{mm}$, and the square side length is $l = 16 \text{mm}$. $L$ is the distance between adjacent particles. In order to control the spacing variable of particles [14], the distance between the three particles is set as $L = 16 \text{mm}$. The calculation area grid is tetrahedral unstructured grid [15].

![Fig.1 Calculation area and particle shape](image-url)
2.2 numerical model and boundary conditions

The turbulence model is standard K-\(\varepsilon\)Model, the transmission equation of the model is as follows:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \\
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho u_i \varepsilon) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{k}{\varepsilon} (G_k + C_{2\varepsilon} \varepsilon) - C_{2\varepsilon} \rho \varepsilon^2 - S_\varepsilon
\]

The standard wall function method is used near the wall. The diffusion term adopts the second-order central difference, and the convection term adopts the second-order upwind difference. The pressure velocity coupling adopts the simple method [16].

The pressure inlet is adopted for the inlet interface and the pressure outlet is adopted for the outlet interface. The non-slip wall boundary condition is adopted at the wall. The front particles advance at the speed of 20m/s and the rear particles catch up at the speed of 25m/s. The catch-up simulation is realized by using the dynamic grid [17].

3 calculation results and analysis

3.1 streamline diagram of different particles at different times

Fig.2 flow lines of three particles with \(T = 0.003s\)  
Fig.3 flow lines of three particles with \(T = 0.015s\)

Fig.4 flow lines of three particles with \(T = 0.02s\)  
Fig.5 flow lines of three particles with \(T = 0.025s\)
As can be seen from Figure 2, when t=0.003s, the vortices on the left and right sides of spherical and ellipsoidal particles are symmetrically distributed, and regular streamline appears on the outside of the two particles. Because the particles are in the process of continuously discharging the fluid when moving, the streamline shows a closed curve from top to bottom (starting at the top of the particles and closing at the bottom of the particles).

As shown in Figure 3, after t = 0.015s, the vortex inside the particle decreases due to the movement of the particle on the inner side, the turbulence area on the left side of the front particle changes due to the emergence of the rear particle. As a result, the negative pressure on the left side of the front particle and the right side of the rear particle decreases and the turbulence phenomenon weakens. When the rear particle gradually approaches the front particle, the air flow in front of the front particle will not change, the pressure in front of the front particle will hardly change, and the gas turbulence will not change.

When the particles run to t = 0.02s, as shown in Figure 4, compared with ellipsoid and sphere, the streamline generated by square particles in the process of catching up is denser. This is because the flow around the square corner can produce a large reverse pressure gradient, and the separation layer forms dense vortices. These vortices make a very large suction near the separation point, so the speed is large. At this time, the front pressure field and the rear pressure field of the two particles are roughly the same. It can be seen from the figure that the left and right flow lines of the three particles are basically symmetrical, so the lateral force coefficients of the front and rear particles are basically the same and the direction is opposite.

When the particles run to t = 0.025s, the rear particles have overtaken the front particles. With the increase of the distance between the particles, the middle streamline gradually recovers, indicating that the interaction between particles is gradually weakened.

There are many similarities among the three particles. Along the front of the particle movement, there is no trajectory of the particle streamline, because there is no interference of external air flow in the movement process. The particle movement generates vacuum on both sides and rear of the particle, causing the surrounding air flow to enter the areas on both sides of the particle, causing strong turbulence.
From Fig. 6 to Fig. 9, it can be found that the turbulent kinetic energy contours of the three particles tend to expand in the process of 0.003s to 0.025s. During the movement of the particles, the back particle turbulent field of square particles gradually converges with the front particle turbulent field, and the spherical and ellipsoidal turbulent fields always maintain two turbulent fields. The change of the minimum turbulent kinetic energy of the three particles is small, and the change of the maximum turbulent kinetic energy is shown in Figure 10 below. The change of the turbulent kinetic energy of spherical particles is the smallest at different times, and the value is also the smallest. The turbulent kinetic energy of ellipsoidal particles tends to be stable after gradually increasing, and the turbulent kinetic energy of square particles is the largest and the change range is also the largest. This is because the flow around the square corner can produce a large reverse pressure gradient, there is a very large suction around it, so the speed is large.

3.2 Velocity distribution at the same position at different times

In order to explore the disturbance of different particles to the same position at different times, the velocity distributions around the three particles in the X direction were compared at four times of T = 0.003s, 0.015s, 0.02s and 0.025s. The position in X direction is as follows (taking spherical particles as an example)
It can be seen from the first figure in fig.11 that the vertical distance between the two particles is large, and the velocity fluctuation in the X direction is mainly affected by the wake of the front spherical particles. Therefore, it can be seen in fig.12 that because the spacing between the square particles and the spherical particles is the same, the two particles reach their maximum velocity at the same position, and the maximum velocity of the ellipsoidal particles is right due to the long distance.
between the ball centers; When the particles run to t = 0.015s, as shown in fig.13, due to the close vertical distance between the two particles, the velocity distribution in X direction is affected by the front and rear particles, and the center distance of ellipsoidal particles is large. Therefore, the maximum velocity in the front section of rear particles is left than that of spherical and square particles, and the maximum velocity in the rear section of front particles is right than that of spherical and square particles; When the particles run to t = 0.02s, as shown in fig.14, the velocity distributions in the X direction of square and spherical particles almost coincide. Because the center distance of ellipsoidal particles is large, the maximum velocity in the X direction of the two particles is different from that of other particles; When the particles run to t = 0.025s, as shown in fig.15, the rear particles surpass the front particles, the velocity distribution is similar to fig.13, and the velocity of square particles is always the maximum.

4. Conclusion
In this paper, the turbulent flow of particles with different shapes is numerically simulated, and the following conclusions are obtained:

1. At the beginning of particle movement, the vortices on the left and right sides of spherical and ellipsoidal particles are symmetrically distributed, and the vortices on the inner side of square particles are denser. The velocity distribution diagrams of different particles at different times and at the same position also well illustrate that the velocity around square particles is greater.

2. When the rear particle is about to catch up with the front particle, the turbulence area on the left side of the front particle changes due to the emergence of the rear particle. As a result, the negative pressure on the left side of the front particle and the right side of the rear particle decreases and the turbulence phenomenon weakens. After the latter particles catch up with the former particles, the change of the surrounding streamline is similar to the reverse process before catching up, indicating that the particle movement has a certain regularity.

3. In the process of particle movement, the minimum turbulent kinetic energy remains almost unchanged, among the maximum turbulent kinetic energy of several particles, the spherical particles are almost unchanged, the ellipsoidal particles first increase and then decrease, and the square particles gradually increase. At the same position of different particles at different times, the velocity of square particles is the largest, and the velocity of ellipsoidal particles in the middle of the three particles is the smallest.

4. By studying the gas flow characteristics of particles with different shapes in the pursuit process, this paper deeply discusses the interaction mechanism of gas-solid two-phase flow, which provides a design idea for the structural optimization of fluidized bed and the transformation and upgrading of rocket engine. The research on the structure of gas-solid two-phase flow is of great significance and engineering practical value to the development of energy, electric power, chemical industry and other fields. Therefore, the catching up of particles with the same shape and different spacing and the intersection of particles with different shapes will be further studied in the future.

Reference
[1] Di,Y.T., Zhao,L.H., Mao,J., (2010) High resolution CFD DEM IBM fluid structure coupling numerical simulation method for particle sedimentation. J. Progress in water science., 32 (02): 286-294.
[2] Liu,K., You,C.F., (2013) Numerical solution of combustion process of single moving coke particle. J. Journal of Engineering Thermophysics., 34(07):1375-1378.
[3] Wachs,A., (2009)A DEM-DLM/FD method for direct numerical simulation of particulate flows: Sedimentation of polygonal isometric particles in a Newtonian fluid with collisions. J. Computers & Fluids., 38(8):1608-1628.
[4] Lv,K., Min,F.F., Zhu,J.B., Bai,X.J., (2021) Study on sedimentation drag coefficient of fine kaolinite particles in coal measures. J. PowderTechnology., 50(02):389-395.
[5] Chhabra, R.P., Agarwal, L., Sinha, N.K., (1999) Drag on non-spherical particles: an evaluation of available methods. J. Powder Technology., 101(3):288-295.

[6] Chou, Y., You, C.F., Qi, H.Y., Xu, X.C., (2006) Background integration scheme in meshless method and numerical simulation of single particle descent process. J. Computational physics., 23 (05): 525-529.

[7] You, C.F., Qi, H.Y., Xu, X.C., (2003) Numerical study on drag force of non spherical particles in gas-solid two-phase flow. J. Journal of chemical engineering., 54 (02): 188-191.

[8] Huang, H., Yang, X., Lu, X.Y., (2014) Sedimentation of an ellipsoidal particle in narrow tubes. J. Physics of Fluids, 26(5):161.

[9] Wang, K., Zhu, L.Y., Li, A.J., Wang, Z.B., Wang, G.T., (2021) Numerical simulation of particle flow characteristics in turbulent fluidized bed. J. Petrochemical equipment technology, 42 (01): 5-10.

[10] Jiang, S.T., (2006) Numerical simulation of viscous flow around vehicle body with moving boundary. D. Nanchang Institute of Aeronautical Technology.

[11] Tang, H.T., Miao, X.Q., (2018) Aerodynamic simulation of transient overtaking. J. Journal of automotive engineering., 8(02): 104-113.

[12] Noger, C., Regardin, C., Széchényi, E., (2005) Investigation of the transient aerodynamic phenomena associated with passing manoeuvres. J. Journal of Fluids & Structures., 21(3):231-241.

[13] Liu, L.N., Wang, X.S., (2018) Transient aerodynamic characteristics of vans during the accelerated overtaking process. J. Journal of Hydrodynamics., 30(02):357-364.

[14] Sae, J., (2016) Aerodynamic testing of road vehicles: testing methods and procedures. S. Sae Technical Information Report.

[15] Zhang, L.P., Deng, X.G., (2010) Review on the progress of dynamic grid generation technology and unsteady calculation methods. J. Progress in mechanics., 40 (04): 424-447.

[16] Bai, W., E.X.Q., (2003) Simple algorithm based on unstructured collocated grid. J. Journal of computational mechanics., 20 (06): 702 - 710.

[17] Yang, L., Zhang, J., Yuan, H., (2018) Analysis of the motion of small-scale ellipsoidal particles in a horizontal laminar flow field. J. Particuology., 40:44-51.