Simulation of particle dispersion behavior in an accelerative container using CFD-DEM

O. Zhao 1, O. Masahiro Takei 2

*Tong Zhao 1, Masahiro Takei 2

Abstract: In the present study, a combined two-dimensional model of computational fluid dynamics and discrete element method (CFD-DEM) was used to investigating the particle dispersion behaviors under acceleration conditions in the solid-air two-phase flow. In the CFD-DEM model, the motion of discrete particles is obtained by solving Newton's equations of motion and the flow field of air by Navier-Stokes equations. This simulation model provides the information regarding the air velocity, the particle movement and distribution under different parameter conditions, such as particle size and acceleration. The results demonstrate that the particle distribution is directly influenced by the particle size inseparably.

1. Introduction

Dispersions of dust, impurities, droplets, pollen and other finite-size particles in the incompressible flow are commonly encountered in many natural phenomena and industrial processes. Different from that of the carrier fluid, particles with a finite size and a mass density have inertia. They do not evolve as simple point-like fluid tracers and are termed ‘inertial particles’. This leads to preferential concentration, clustering, and separation of particles as observed in numerous studies. Many theoretical and numerical studies have been done on the dynamics of inertial particles, such as clustering of inertial particles in two-dimensional flow past a cylinder and similar studies considered inertial particle clustering in turbulent flows. Recently, numerical methods have been widely used to study particle–fluid flow systems. The continuum–discrete approach at a microscopic level mainly represented by the so-called combined computational fluid dynamics and discrete element method (CFD-DEM) is one of the popular mathematical models [1, 2].

In the present study, a combined two-dimensional model of computational fluid dynamics and discrete element method (CFD-DEM) was used to investigating the dispersion behaviors of particles with finite size and mass under the acceleration condition. The simulation model provides the information regarding the air velocity, the particle movement and distribution under different parameter conditions, such as particle size and acceleration.

2. Governing Equations of CFD-DEM simulation

In the simulation model, the motions of a particle at any time are determined by momentum balance, given by

$$m \frac{dv}{dt} = f_a + f_c + g \quad I \frac{d\omega}{dt} = T$$

(1)

where $m$ is the mass of the particle; $v$ and $\omega$ is the translational and rotational velocity; $I$ is the moment of inertia; $f_a$ is the force acting on particle that exerted by surrounding fluid; $f_c$ is the contact force; $g$ is the gravitational force; $T$ is the torque caused by the contact force and the moment of inertia of particle. The gas phase is treated as a continuous phase. The governing equations are the conservation of mass and momentum in terms of the local mean variables over a computational cell, given by

$$\frac{\partial \epsilon}{\partial t} + \nabla \cdot (\epsilon \mathbf{u}) = 0$$

$$\frac{\partial (\rho \epsilon \mathbf{u})}{\partial t} + \nabla \cdot (\rho \epsilon \mathbf{u} \mathbf{u}) = -\epsilon \nabla p + \sum_{i} f_{a,i} + \epsilon (\mu + \mu_i) \nabla^2 \mathbf{u}$$

(2)

where $\epsilon$ is the void fraction; $\mathbf{u}$ is the velocity vector of fluid; $p$ is the pressure of fluid; $f_a$ is the fluid drag force exerted on the particle; $\delta V$ is the volume of a computational cell, and $n$ is the number of particle inside the cell. The fluid drag force can be calculated by

$$f_{a,i} = \beta (v_p - \mathbf{u}) \delta V$$

(3)

where $v_p$ is the particle velocity. In the present study, $\beta$ was deduced based on the summarized equations in previous works [2].

3. Simulation Conditions

Figure 1 shows the calculation domain and the grid arrangements of the container. The cell size for the calculation of gas motion is 5 mm×1.5 mm. The initial particle loading region is marked by black color as shown in figure 1. Particles can only runs out of the container through the outlet. The parameter settings of the simulation are summarized in Table 1.

---

1 日大 院 機械 2 日大 敎員 機械
4. Simulation results and discussion

Figure 2 shows the distribution images of particle volume fraction obtained from the simulation mode for case 1 and case 2. The images in Figure 2(a) show that in case 1 as the air inject into the container, particles became dispersive, move toward the outlet area and then runs out of the container through the outlet. Finally, most of the particles have already gone out of the container. Figure 2(b) shows that in case 2 while the particle diameter is ten times of case 1, the particle dispersion behaviours are basically similar with that in case 1, except the particle dispersion speed seems slower than case 1. Moreover, in Figure 4, it can be found that some of the particles concentrated at the right side corner due to the additional acceleration at horizontal direction.

The particle run-out rate $R_{\text{out}}$, which means the percentage of particle run out from the outlet at each time interval, can be calculated as follow:

$$R_{\text{out}} = \frac{N'_{\text{out}}}{N_p} \times 100\%$$

(4)

where $N_p$ is the initial loading particle number; $N'_{\text{out}}$ is the number of particles run out from the outlet during the time interval $t=0.01\,$s. The calculated results in Figure 3 show that, for both case 1 and 2, the highest particle run-out rate always occurs at $t=0.02\,$s, although the highest particle run-out rate at case 1 is much higher. Moreover, as shown in Figure 3, the effective particles run-out period ($R_{\text{out}}>0.01$) in case 1 is from $t=0\,$s to $t=0.03\,$s, which is shorter than that in case 2 from $t=0\,$s to $t=0.05\,$s.

5. Conclusion

A numerical simulation for particle dispersion behaviors under acceleration condition was performed by combining the DEM and CFD in a two-dimensional domain. The distribution images of particle volume fraction were obtained by the simulation. As a result, the particle dispersion speed is inverse proportional to the particle size in the simulation. Due to the additional acceleration at horizontal direction, some of the particles concentrated at the right side corner. The highest particle run-out rate always occurs at $t=0.02\,$s for case 1 and case 2. The highest particle run-out rate at case 1 is much higher than case 2, but the effective particles run-out period ($R_{\text{out}}>0.01$) in case 1 is shorter.

6. Reference
