Research on Sand-dust Separation Technology Based on Gas-solid Two-phase Numerical Simulation

Lei Zhang¹, Junwei Wang^{1*} and Guohua Li¹

¹Beijing Institute of Spacecraft Environment Engineering, Beijing, 10094, China

Abstract. The separation technology in the large-scale sand-dust environment ground simulation test system applicable to the environmental adaptability and reliability verification of aerospace electromechanical products is studied. The gas-solid two-phase numerical simulation method is adopted, and the possible cyclone separation, inertial separation methods are used to study the separation efficiency and regularity technology, which provides a basis for the separation design and test of the sand-dust environment simulation of large electromechanical products.

1 Introduction

In the ground simulation test system of large sand-dust environment, after the sand-dust particles are injected by means of gravity or gas-solid jet, the floating and migration of sand-dust are realized by the wind tunnel. The migration and distribution of sand-dust in the circulating wind are important factors influencing the performance of simulating system. In order to ensure the concentration control accuracy when the sand airflow hits the testing product, it is necessary to recover the sand-dust particles in the airflow downstream of the injection section and the test zone, to avoid the concentration increase caused by the dust particle circulation operation, and to reduce the erosion of internal moving parts such as fans by dust particle. So the space separation in the testing system is critical. Because it is difficult to realize the design and optimization of the separation method by means of direct solution, the gas-solid two-phase numerical simulation is used to study the separation efficiency and regularity of various separation methods that may be used.

2 Gas-solid two-phase mathematical model

There are generally two methods for solving multiphase flow: the Euler-Eulerian method and the Euler-Lagrangian method. Since the volume fraction of the particle phase in the gas stream is extremely small, the influence of the sand-dust on the air flow field is extremely small, so the DPM (Discrete Phase) model is used in this calculation. The gas (air) is calculated as a continuous medium, and its flow law is described by the two-phase coupled Navies-Stokes equation. The particles (sand-dust) are used as discrete media to describe the

motion orbit of each single particle in the Lagrange coordinate system [1-2].

(1) Motion control equation of single particle

The forces that a single particle moves in the sand-dust separation flow field are: drag force, pressure gradient force, additional mass force, Basset force, Saffman lift, Magnus force and gravity. The equation of motion of a single particle can be directly from Newton's second law:

$$m_p \frac{d\vec{V}_p}{dt} = \vec{F}_D + \vec{F}_P + \vec{F}_A + \vec{F}_B + \vec{F}_S + \vec{F}_M + \vec{F}_G + \vec{F}_C \quad (1)$$

In the above formulas, m_p is the particle phase mass. Although the forces acting on the particles are quite complex, not all forces are equally important. In the gas-solid two-phase flow, since the density of the gas is much smaller than the density of the particles, the buoyancy, pressure gradient force, and additional mass force are small in magnitude compared with the inertial force of the particles themselves, which can be ignored.

(2) Particle random orbit model

The key of multiphase turbulence simulation is to determine the particle phase model. Due to the influence of turbulence, the fluid has pulsation velocity in different directions at different times. The position and velocity of the particles in flow field will change due to the pulsation velocity. Therefore, it is necessary to consider the influence of turbulent pulsation and adopt random orbit model.

The random orbit model of the particle is:

$$\frac{dV_{px}}{dt} = \frac{1}{\tau_p} (U + \mu' - V_{px}) \quad (2)$$

$$\frac{dV_{py}}{dt} = \frac{1}{\tau_p} (V + v' - V_{py}) \quad (3)$$

*Corresponding author's e-mail: tobenol@126.com

$$\frac{dV_{pz}}{dt} = \frac{1}{\tau_p} (W + w' - V_{pz}) + g \quad (4)$$

When the turbulence equation is a two-equation model, μ can be obtained by turbulent kinetic energy. The calculation formula is:

$$\mu' = \xi \sqrt{\frac{2}{3}k}, \quad v' = \xi \sqrt{\frac{2}{3}k}, \quad w' = \xi \sqrt{\frac{2}{3}k} \quad (5)$$

Where ξ is a random number and $-1 \leq \xi \leq 1$, the pulsation velocity obtained by the above formula satisfies the Gaussian distribution. When the turbulence model is the Reynolds stress model, the turbulent pulsation velocity is directly obtained from the three normal stress components of the Reynolds stress.

$$\mu' = \xi \sqrt{\mu' \mu'}, \quad v' = \xi \sqrt{v' v'}, \quad w' = \xi \sqrt{w' w'} \quad (6)$$

3 Modelling and analysis of cyclone separation methods

Cyclone separation has always dominated in the machinery, metallurgy, chemical, petroleum and other industries due to its simple structure, low energy consumption and convenient maintenance. The principle is to use the centrifugal force of the airflow to pry the particles to the wall to realize the separation. In the separation process, it is necessary to consider the airflow, the characteristics of the particles themselves, and the complex interaction between the two phases, such as static electricity and adsorption. The internal flow of the cyclone separation is a complex three-dimensional strong swirl flow, which is accompanied by a local secondary flow such as longitudinal circulation, short-circuit flow, and back-mixing in addition to the main flow vortex [3-5].

3.1 Geometric Model

Table 1. The structure dimensions of cyclone separation (unit: mm)

diameter of cylinder	width of import	height of import	diameter of exhaust pipe	diameter of insertion depth	height of cylinder	height of cone	diameter of sand-dust outlet
D	$0.2D$	$0.5D$	$0.5D$	$0.5D$	$1.5D$	$2.5D$	$0.375D$
500	100	250	250	250	725	1250	187.5

The three sizes of sand-dust particle was simulated. The particle size of the sand-dust was $3\mu\text{m}$, $8\mu\text{m}$ and $80\mu\text{m}$ respectively. The inlet gas flow rate and particle phase velocity were both 18m/s, and the outlet pressure was atmospheric pressure.

3.2 Numerical simulation results

3.2.1 Track of sand-dust particle

In order to study the effect of the cyclone separation method, the particles of different diameters were simulated. The particle size of the particles was set to

$3\mu\text{m}$, $8\mu\text{m}$, and $80\mu\text{m}$, and the mass flow rate was 0.05 kg/s. It can be seen from the trajectories of three different particle sizes that when the particle size is $3\mu\text{m}$, the particle follows the gas flow better, and some particles reach the taper of sand-dust outlet and then spiral upward with the internal ascending airflow to enter the exhaust pipe, some of the particles are also directly discharged from the sand-dust outlet. When the size of the sand-dust particles is $8\mu\text{m}$ and $80\mu\text{m}$, the two large-diameter particles are smashed toward the wall due to the large centrifugal force, and are spiraled down to the sand-dust exit position along the wall surface. It can be seen the inertia of $80\mu\text{m}$ particle is larger and it is easier to maintain the original motion trajectory.

Figure 1. Different trajectories of particle using cyclone separation method

3.2.2 Distribution Cloud Map of Pressure & Velocity

The static pressure distribution of cyclone separation has good symmetry. It can be seen from the figure that the

static pressure is larger at the side wall and decreases inward. At the central position (near the axis), the pressure is lower due to the core area in the vortex, where may even be negative pressure, and the stratification is more obvious. The internal flow field of

the cyclone separation is a three-dimensional strong rotating turbulent flow field. The gas entering the cyclone separation is downward, and the external vortex

flow is gradually formed. After reaching the bottom of the cone, the airflow turns upwards to form an upward internal vortex.

Figure 2. Different pressure clouds of particle using cyclone separation method

4 Modelling and analysis of inertial separation methods

The inertial separation method mainly includes a semi-circular shape, a flat bottom U-shape, a triangular bottom

U-shape, and a semi-circular bottom U-shape structure, wherein the latter two are provided with fins and flanges, and the triangular bottom edge and the fins are formed. 45°, the width and depth of the separating elements are 40 mm, the specific structure is shown in Figure 3[6-8].

Figure 3. The structure of several inertial separation methods

4.1 Geometric Model

The inertial separation works as follows: the airflow containing sand-dust enters the separator from the left inlet, passes through separation components with multiple impact, and the energy loses during the collision process. Most of the particles fall into the lower sand-dust outlet under the influence of gravity to achieve gas-solid separation. The four inertial separation methods are named semi-circular, flat bottom U-shape, triangular bottom U-shape and semi-circular bottom U-shape according to the configuration.

4.2 Numerical simulation results

The three sizes of sand-dust particle was simulated. The particle sizes of the sand dust were 80μm, 300μm and 850μm, the particle density was 1550 kg/m³, and the particle flow rate was 1e-5 kg/s. The inlet gas flow rate

and the particle phase velocity were both 2 m/s, and the outlet pressure was atmospheric pressure.

4.2.1 Track of sand-dust particle

In order to study the inertial separation effect of different bottom U-shapes, numerical simulations were performed on particle diameters of 80μm, 300μm and 850μm respectively. The trajectories of different particles are shown in Figure 4. It can be seen that particles with size of 80μm, a part of which is blocked by the separation element, discharged from the lower outlet, and a large part of the flow enters the right end basin with the gas flow. The majority of the particles with size of 300μm and 850μm are blocked by the separating element and fall into the bottom outlet at the same flow velocity. In addition, comparing the three separation methods, it can be seen that the U-shaped separation method at the bottom of the plane has better separation effect on particles with a particle size of 300μm.

Figure 4. Different trajectories of particle using U-shaped inertial separation method

Figure 5. Different trajectories of particle using semi-circular shaped inertial separation

The semi-circular inertial separation method is used to calculate the same boundary conditions. The particle diameters are also 80μm, 300μm and 850μm, which are shown in Figure 5. It can be seen that the semi-circular inertial separation has a better separation effect on sand-dust with a particle size of 850μm while only a small fraction of the particles flow into the right region. For sand-dust with particle size of 80μm and 300μm, the

separation effect is poor, and a large part of the particles flow into the right end region with the air flow.

By comparison with the three U-type separation methods, the semi-circular separation method has a poor separation effect on small-sized sand-dust particles. The reasons for the analysis are as follows:

1) The semi-circular separator has better conductivity to the airflow than the U-type separator;

2) The semi-circular separator has a smaller entrance boundary area than the U-shaped boundary area, so a considerable part of the sand-dust particles can't enter the separation element area, so that sufficient collision cannot be performed, and the total kinetic energy loss of the particle phase is small, resulting in poor separation effect.

4.2.2 Distribution Cloud Map of Pressure & Velocity

Figure 6 below shows the horizontal section pressure (left) and velocity map (right) of the above four separation elements. It can be seen that the airflow uniformity of the U-shaped separation at the bottom of the plane is better than other inertial separation methods, and the pressure drop is smaller.

Figure 6. pressure (left) and velocity map (right) of four separation methods

The pressure in the semi-circular separator decreases from left to right, and the overall pressure loss is about 45 Pa. The pressure loss is significantly smaller compared with U-shaped. It indicates that the gas pressure loss of the semi-circular separator is smaller, and the flow conductivity is better. However, the uniformity of the velocity distribution is worse than that of the flat bottom slot type U-shaped separator.

5 Conclusion

The several separation techniques applicable to the aerospace electromechanical sand-dust ground environment simulation system are simulated. The separation efficiency and law under different separation conditions are compared and analyzed. The results show

that the cyclone separation method can be used to separate sand with smaller particle size. The diameter of the particles suitable for separation is $>8\mu\text{m}$. The U-shaped and semi-circular inertial separation methods can be used to separate sand-dust with larger particle size. The diameter of the U-shaped separation suitable for separation is $>300\mu\text{m}$, and the semi-circular separation is suitable for the separation with the diameter of $>850\mu\text{m}$. Compared with the three bottom-side U-shaped inertial separation methods, the U-shaped separation with flat bottom type is better, while the semi-circular separator has better diversion effect and the overall pressure drop is smaller. The research results can provide technical support for the further development of large-scale sand-dust ground environment simulation system.

References

1. Zhou Lixing. Research Progress and Demand of Discrete Turbulent Multiphase Flows[J]. *Advances in Mechanics*, 2008, 38(5): 610-622.
2. Tan Xiaojun, Chen Lihua. Numerical Simulation of Two-Phase Flow in Diffusion Gas-Solid Separator[J]. *Journal of Engineering Thermophysics*, 2007, V28(1): 74-76.
3. Long Shuyu, Liu Genfan, Mao Rui, et al. Numerical simulation of large eddy of cyclone vortex core in cyclone separator[J]. *ACTA PETROLEI SINICA SINICA (PETROLEUM PROCESSING TECHNOLOGY)*, 2016, 32(4): 734-740.
4. Azizi S, Mowla D, Ahmadi G. Numerical evaluation of turbulence models for dense to dilute gas-solid flows in vertical conveyor[J]. *Journal of Granules (English Edition)*, 2012 (5): 553-561.
5. Liu Lin. Numerical simulation and stability study of gas-solid two-phase flow in cyclone [D]. 2016.
6. Li Hongwei, Han Haibo, Wang Wei, et al. Research on performance of U-shaped inertial separator for large sand dust test system[J]. 2009, 32(2): 225-229.
7. Xu Feng. Experimental study and numerical simulation of small-scale particle gas-solid flow in inertial separator [D]. Shanghai Jiaotong University, 2010.
8. Li Hongwei, Liu Jingjiang, Han Haibo, et al. Performance Study of Inertial Separation Components in Pneumatic Sand Transport Test System[J]. *Journal of Computer and Applied Chemistry*, 2013(03):99-101.