Studies of the Effect of Cylinder’s Size on Separation Performance of the Circumfluent Cyclone Separator

Sanping Zhou and Wenpeng Wang

ABSTRACT

Based on the existing studies of the gas-solid two phase separation performance and the circumfluent cyclone separator both at home and aboard, this paper used the FLUENT software to research the effect of the outer cylinder length and both the outer and internal cylinder length changed on separation performance. The results of the simulation have shown that the effect of spans is more obviously than which of the changes of length on the separation performance. The study provides an important reference for a better techniques design of the circumfluent cyclone separator.

INTRODUCTION

The circumfluent cyclone separator [1], a kind of gas-solid separation device, has the characteristics of the lower pressure reduction, smaller amplification effect and higher separation efficiency. Up to now, the device has only a short span of a decade years of development, so there’s a few the relevant references and all of these just relate to internal flow field and mathematical models, the study of effect of the size on the separation performance is still in blank state. By changing the length of the outer and internal cylinder, this paper researches the effect of the cylinders length on separation performance, which has important reference meaning for the techniques design of the circumfluent cyclone separator.

THE GROMETRY AND MESH MODELS

The circumfluent cyclone separator is composed of the gas-inlet, outer cylinder, inner cylinder, cone, gas-outlet and dust pot etc., the 3D model as shown in Fig.1, the
initial sizes as shown in Tab.1. The gas-outlet stretches into the outer cylinder 5mm, and the distance between the top of internal cylinder and outer cylinder is 10mm, the gas-inlet is tangent to the internal cylinder. For simplifying the model, the simulation and the Fig.1 take no account of the dust pot.

The mesh model is generated by the ANSYS 16.2 Workbench Mesh, except the complex boundary, there’s mainly used structured mesh, which is adaptable, and orthogonality well, benefit to deal with the region coupling in simulation[2]. After the disrelation test of the mesh, the final mesh model is shown in Fig.2, there’re 57803 nodes and 57399 Hexahedral grid cells.

| Structure   | Gas-inlet | Inner cylinder | Outer cylinder | Cone | Gas-outlet |
|-------------|-----------|----------------|----------------|------|------------|
| Size / mm   | 18×18     | Φ38×70         | Φ60×90         | Φ60×Φ14×120 | Φ20×30 |

Figure 1. The geometry model.  
Figure 2. The mesh model.

THE MATHEMATICAL MODEL AND BOUNDARY CONDITIONS

The Mathematical Model

Whatever flow form it is, it must be satisfied with the law of mass conservation, as Eq.(1), and the law of conservation of momentum, as Eq.(2)[3].

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = S_m
\]  

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i
\]

In these equations, \( \rho \) is density; \( p \) is the static pressure forced on the infinitesimal fluid; \( \tau_{ij} \) is stress tensor; \( \rho g_i \) is the component of gravity on \( i \) direction, \( F_i \) the component of body force on \( i \) direction.

In this simulation, RSM is chose as turbulence model of continuous phase, and the wall function adopts the standard wall function, PISO(Pressure-Implicit with Splitting of Operators) algorithm is used to deal with the pressure-velocity coupling,
Pressure interpolation algorithm adopts the PRESTO!, the discretization scheme uses QUICK (Quadratic upwind interpolation).

The volume fraction of the particles in fluid is 5%, so the multiphase flow model adopts the DPM[4], “distract” means that takes no account of collision between particles. The forces acting on particles are very complex, including the drag force, pressure gradient, additional body force, Basset force, Saffman lift force, and Magnus force, gravity etc. Because the gas density is more lower than the particle density, so there loses sight of the buoyancy force, pressure gradient, additional body force and so on. The fluctuating velocity caused by turbulence flow must be considered in the simulation, so in order to display particles movement condition in separator accurately, the Stochastic Tracking Model was chose.

The Boundary Condition

In the simulation, the continuous phase is air, of which the density is 1.225kg/m3, and the viscosity is 1.789×10⁻⁵. The gas-inlet is the velocity inlet, the value is 15m/s; the gas-outlet is pressure outflow, and the value is one bar. The wall adopts no slip wall boundary condition, and the roughness of the wall is 0.5mm.

The discrete phase is coal ash, the density of which is 2800 kg/m3, the flow rate of particles in gas-inlet is 2800 kg/m³; the injection type of particles chose surface injection, and the initial velocity of particles is 0. The collision between particles and walls is regarded as completely elastic, so the boundary is defined as “reflect”. When particles reach to the dust pot, the boundary is defined as “trap”, it means the particle was trap by the dust pot. When particles reach to the gas-outlet, the boundary condition is defined as “escape t”, it means the particle is escaped.

THE ANALYSIS OF THE SIMULATION RESULTS

The Effect of Analysis of the velocity field

The separation performance of the circumfluent cyclone separator is determined by the velocity field, so before researching the effect of cylinder length changed on separation performance, there shall test and analyze the velocity to judge if it’s satisfied with the principles of the circumfluent cyclone separator or not.
There’s a very complex turbulent condition which composed by tangent, radial and axial flows. The velocity contour is shown in Fig.3, the central velocity of the separator is lower, and the velocity nearly the gas-outlet is higher which affected by the convergent flow. The velocity vector is shown in Fig.4, the fluid flows into the gas-inlet then rotates from the wall to the axis, and the velocity of the fluid reduced gradually and formed a low speed swirl finally alone with the rotation. The faster flow discharged from the gas-outlet nearly the axis, which is defined as clear gas after the first time separation. The slower flow moved into the annular space between the outer and internal cylinders and closed to the bottom of device gradually, which is regarded as dusty gas flow. Because just little of fluid flows from the annulus to the cone, so the radial velocity is reduced, which benefits to conquer the back maxing phenomenon[5], therefore the de-dusting efficiency is increased.

So we can get the path of fluid in separator as following: the dusty gas enters into the first separation region by the gas-inlet, and then flows upward rotationally, when the fluid near the capping of the outer cylinder, a part of the fluid flows out the separator by the gas-outlet, another part of the fluid flows from the annulus to the cone until reaches to the bottom of the cone then turns up alone the axis and enters into the first separation region.

The Effect of the Spans

The dust particles can stay longer if increasing the cylinders’ lengths, which benefits to the separation performance and gives more chance to separate the dust particles from gas near the gas-outlet. But the dust particles which haven’t fell into the dusty pot maybe be carried into separation region again, caused a bad influence to separation performance. What’s more, the length of the circumfluent cyclone separator is limited to the space, especially for the built-in separator. Therefore it’s very important to research the effect of the cylinders’ size on separation performance.

In this paper employed five different lengths of the outer cylinder to simulate the internal flow field, they respectively are 90mm, 105mm, 120mm, 135mm and 150mm, and the lengths of 105mm and 135mm are used to increase the distance between the top of internal cylinder and the capping of outer cylinder, the lengths of
120mm and 180mm are used to increase the distance between the bottom of internal cylinder and the top of the cone. The simulation results are shown in Tab.2. It’s important to note that the “L” is the length of the outer cylinder; the “T” is the distance between the top of internal cylinder and the capping of the cylinder; “B” is the distance between the bottom of the internal cylinder and the top of the cone; “AR” is the arresting rate at the dust-outlet, the higher the arresting rate is, the better the separation performance.

Obviously, with the increase of the length of the outer cylinder, the arresting rate is falling. Comparing the dates in Tab.2, when B is 10mm constantly and T increases to 25mm, the arresting rate decreases by 8.5%; and when T is 25mm constantly and B increases to 25mm, the arresting rate decreases by 5.5%. In the same way, it’s found that with the increase of B and T, the arresting rate is falling. And T is a more important factor for separation performance, there’re two major reasons: on the one hand the increase of T makes the velocity slower than before; on the other hand the increase of T reduces the differences of velocities between the convergent flows and dusty flows. Both of them make dust particles be carried out the gas –outlet by gas, which leads to the separation performance decreased. The dust particle can stay longer when B is increased, which makes the dust particles that haven’t moved into the dust pot be carried into the separation region again and causes the back mixing phenomenon.

Table II. The effects of t and b on the separation performance.

| L/mm | 90  | 105 | 120 | 135 | 150 |
|------|-----|-----|-----|-----|-----|
| T/mm | 10  | 25  | 25  | 40  | 40  |
| B/mm | 10  | 10  | 25  | 25  | 40  |
| AR/% | 81.6| 73.1| 67.6| 63.9| 62.4|

The Effect of the Length

In order to research the effect of the cylinder length on performance, in this paper increased the lengths of inner and outer cylinders synchronously, so B and T are constant here, and the lengths of outer cylinder has introduced in last section. The lengths of inner cylinder respectively are 70mm, 85mm, 100mm, 115mm and 130mm, and the lengths of 85mm and 115mm are used to increase the distance between the top of internal cylinder and the capping of outer cylinder, the lengths of 100mm and 130mm are used to increase the distance between the bottom of internal cylinder and the top of the cone. The simulation results are shown in Fig.5.

![Figure 5. The trend of arresting rate.](image-url)
As the Fig.9 shown, with the increase of the length of the outer cylinder, the arresting rate increase firstly, and then falls down rapidly; and when the length of inner cylinder increase to 115mm, the downtrend of the arresting rate slows down. The reasons of the phenomenon introduced before as following: firstly an appropriate length of cylinders makes the dust particle stay longer which benefits to the separation, but if the cylinders are too long, the radial velocity will carry the dust particle into the rotational flow, therefore the separation efficiency is reduced. On the whole, the arresting rate only decreased by 7.5% when the lengths of cylinders increased by 60mm respectively, which imply that the change of the lengths has a little influence to the separation efficiency when B and T is constant.

CONCLUSIONS

(1) The differences of velocities between the clear fluid and dusty fluid are the major factor to separation performance,
(2) The pressure of the circumfluent cyclone separator distributes symmetrically along the axis, and increases with the increase of the radius,
(3) The pressure is lower close to the gas-outlet, In addition, the fluid moving closer to the bottom, the lower the pressure is. And the pressure in annulus is higher than it on the wall of cone,
(4) The influence of B on the separation efficiency is greater than T’s; and The influence of B and T on the separation efficiency is greater than L’s.

REFERENCES

1. Fujun Wang. CFD Analysis-The Principle and Application of software. Beijing: Tsinghua University Press, 2004, 9.
2. Haigang Wang. The application and comparison of Different turbulence model in 3D simulation of the circumfluent cyclone separator. Engineering for Thermal Energy and Power Press, 2003,18(4): 337-342.
3. Xinli Wei. The simulation of particle path in the cyclone separator. Zhengzhou University Press 2004, 25(3): 14-17.
4. Fan Jiang, Peng Huang. Fluent in advanced application and instance analysis. Beijing: Tsinghua University Press, 2008.7.
5. Chunyu Mou, Dazhi Jiang. Numerical study of air flow’s moment in cyclone’s annular space. Environmental Engineering, 2007, 1(8): 96-100.