Available online at www.sciencedirect.com
ScienceDirect
Procedia Engineering 102 (2015) 1316 – 1325
Procedia Engineering
www.elsevier.com/locate/procedia

The 7th World Congress on Particle Technology (WCPT7)

Effects of wall roughness on the flow field and vortex length of cyclone

Hui Ci[a], Guogang Sun[a,*]

[a]*State Key Laboratory of Heavy Oil Processing, China University of Petroleum, Beijing 102249, China*

Abstract

Cyclone separators are widely used in gas solid separation process. Their dimensions and configurations have a great influence on the performance, and have been widely studied. However, for a long-time used cyclone, the surface roughness may increase due to erosion and solid collision, and also is an important factor regarding the cyclone performance. Unfortunately, the study of roughness has been ignored for a long time. In this paper, the effects of wall roughness on flow field of cyclone are investigated. The three-dimensional flow field is simulated with Reynolds Stress turbulence model using Fluent. The simulation results indicate that wall roughness has a great influence on flow field. The increased roughness leads to a decrease in vortex length, and this reduces the separation efficiency a lot. The tangential velocity also decreased along the axial position, but there is no obvious trend for the axial velocity. However, a lower pressure drop can be achieved when the wall roughness is increased. In conclusion, the increased wall roughness has its pros and cons, on one hand, it leads to lower pressure drop; on the other hand, the separation efficiency is also decreased. For a particular cyclone, there must be a compromise condition between pressure drop and separation efficiency, which is in coincidence with Pareto optimality, and will be studied further.

Selection and peer-review under responsibility of Chinese Society of Particuology, Institute of Process Engineering, Chinese Academy of Sciences (CAS)

Keywords: Numerical simulation； Wall roughness； Pressure drop； Vortex length

1. Introduction

Cyclone separators are widely used in gas solid separation, such as Fluidized Catalytic Cracking of hydrocarbons (FCC) in oil and gas industry, Circling Fluidized Bed(CFB) in electricity plant [1] . The cyclone works by using the

---

* Corresponding author. Tel.: +86-010-89734820.
E-mail address: ggsunbj@163.com

Selection and peer-review under responsibility of Chinese Society of Particuology, Institute of Process Engineering, Chinese Academy of Sciences (CAS)
doi:10.1016/j.proeng.2015.01.262

density difference between gas phase and solid phase. A strong rotation of gas is induced in the cyclone and the centrifugal forces are exerted on the particles, separating them from the gas flow toward the cyclone wall [2] . In general, the flow field inside a cyclone is characterized with strong swirling vortex, and it is critical to the performance of cyclone separator. It's obvious that, there are many factors that have a big influence on the flow field, such as the geometry of cyclone and the operating condition [3] . Fortunately, these two factors has been investigated by many researchers using both experiment and simulation [4] . Besides them, the wall surface roughness has been ignored for a long time. However we found that it can also have a significant influence on the flow field of cyclone separator, which will be discussed in this paper.

Usually, the wall surface roughness changes not only in the manufacturing process but also during its service period. For a long time used cyclone, the wall surface roughness is changed by corrosion, abrasion and impact of particles on the inner wall. The research of Fuat Kaya [5] showed that the relative roughness has a big influence on the separation efficiency and pressure drop, and this influence is become larger with the increase of the inlet gas velocity. The wall roughness has a significant effect on the tangential velocity, thereby affecting the separation efficiency and pressure drop. Inside a gas-solid cyclone, the outer vortex flow weakens and changes its direction at a certain axial distance from the vortex finder. The position where the outer vortex turns upwards is known as the vortex end. The axial distance from the vortex finder to the vortex end is called the natural vortex length. Since the separation process mainly occurs in the region of the vortex flow in a cyclone, the vortex length is very important for cyclone performance [6] . If the vortex does not fill the cyclone volume and does not attain the cone apex, vortex structure is deformed and separation efficiency decreases. Due to the unstable characteristics of the vortex, it can also been affected by wall roughness. Therefore, it is very important to study the influence of wall roughness on vortex length.

In this paper, the effect of wall roughness on the flow field and vortex length of a typical cyclone is investigated by CFD. The numerical model will be introduced in Sec.2. The cyclone geometry and simulation grids are shown in Sec.3. The simulation results are discussed in Sec.4 and the main conclusions are drawn and further work in this direction is proposed in Sec.5.

Nomenclature

d_p	particle diameter
$\mathbf{F}$	acceleration force
F_D	drag coefficient
ks	wall roughness
P	gas pressure
Re_p	particle Reynolds number
$\mathbf{u}$	gas velocity
$\mathbf{v}_p$	particle velocity
z	axial position
ρ	gas density
ρ_p	particle density
μ	gas viscosity
τ	stress

2. Numerical model

The dilute two phase flow inside a cyclone is simulated using an Euler-Lagrange hybrid method. The gas phase is governed by the Navier-Stokes equations:

$$\frac{\partial \rho}{\partial t}+(\nabla \cdot \rho \mathbf{u})=0 \tag{1}$$

$$\frac{\partial(\rho \mathbf{u})}{\partial t}+(\nabla \cdot \rho \mathbf{u u})=-\nabla p-(\nabla \cdot \tau)+\rho \mathbf{g} \tag{2}$$

Where ρ is gas density, u is gas velocity, p is pressure and τ is the stress. The particles are tracked by Newton's law of motion:

$$\frac{d \mathbf{v}_p}{d t}=F_D(\mathbf{u}-\mathbf{v}_p)+\mathbf{g}(\frac{\rho_p-\rho}{\rho_p})+\mathbf{F} \tag{3}$$

Where $\mathbf{v}_\mathrm{p}$ is particle velocity, ρ is particle density, **F** is an additional acceleration force. $\mathbf{F}_\mathrm{D}$ is drag coefficient and is calculated by:

$$F_D=\frac{18 \mu}{\rho_p d_p^2} \frac{C_D R e_p}{24} \tag{4}$$

where C_D is calculated by an empirical equation [7] . μ is the gas viscosity, d_p is particle diameter and Re_p is particle Reynolds number given by:

$$Re_\mathrm{p}=\frac{\rho\left|\mathbf{u}-\mathbf{v}_\mathrm{p}\right| d_\mathrm{p}}{\mu} \tag{5}$$

Since the flow field inside a cyclone is characterized with strong swirling turbulence. A proper turbulent model is of critical importance for the simulation of cyclone. As state of the art, the Reynolds stress model (RSM) is capable of predicting the combined vortex and the successful applications of this model in the studies on cyclone separators have been reported by many researchers [8] . Therefore, RSM embedded in the fluent software is applied in this work to compute the gas flow field in cyclones. The pressure equation was discretized by PRESTO (Pressure Staggering Option) scheme, which is the most suitable for high-speed rotating flows. The SIMPLEC (semi-implicit method for pressure linked equations consistent) algorithm is used for pressure–velocity coupling, while for momentum, turbulent kinetic energy and dissipation rate, as well as for Reynolds stresses, QUICK (quadratic upwind interpolation of convective kinematics) scheme is selected for the sake of second order accuracy.

3. Cyclone geometry and simulation grid

The cylinder of the simulated cyclone is 300 mm diameter. Its entrance area ratio is 5.5 and the length of the feed leg is 3000mm with a 180° sealed shell. Its geometry dimensions and computation grids are shown in Fig.1. The mesh is generated by Gambit, totally, 314,534 hexahedral structured grids are used in the simulation. The upper section of the cylinder is set as the reference plane (z = 0 mm) and the midpoint of the riser entrance is set to be origin point of the global coordinate.

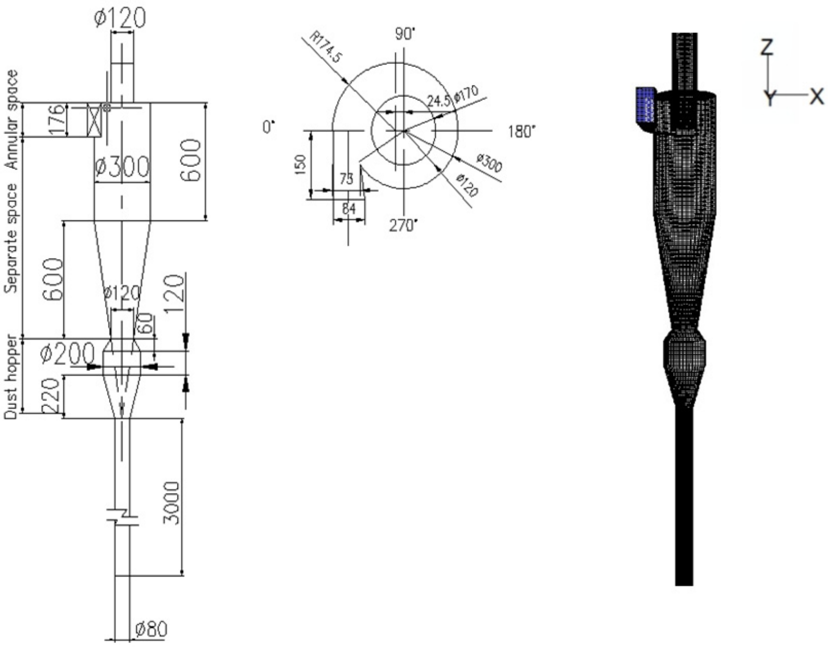
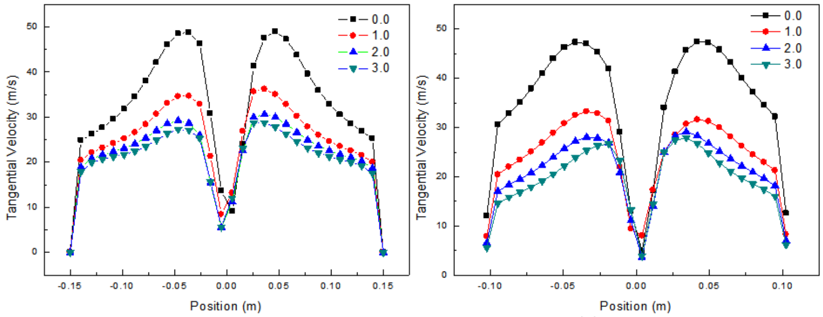


Fig. 1 geometry and computation grid of the cyclone

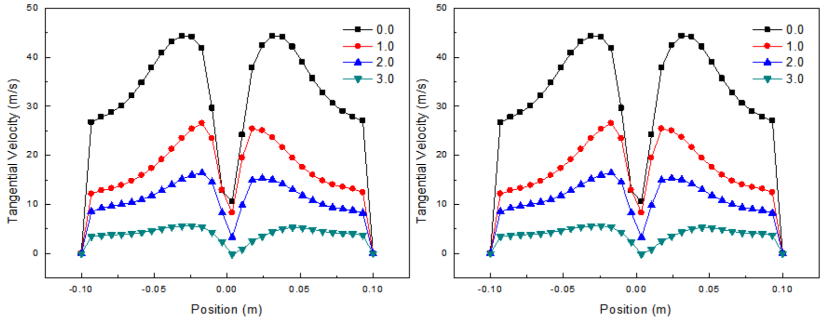
4. Results and discussion

To analyze the velocity and pressure distribution, four typical planes are chosen: one is in the cylinder, the second one is in the cone, the third one is in the dust collector and the last one is in the leg. The tangential and axial velocities at different planes with different wall roughness are analyzed, so does the static pressure.

The tangential velocity at four typical planes under different wall roughness is shown in Fig.2. The tangential velocity profiles at first three planes clearly show that the swirling flow inside the cyclone chamber consists of two parts, an outer free vortex and an inner solid rotation in the center. The maximum tangential velocity decreases with the increase of wall roughness, however the changing point between inner and outer swirling keeps constant. This phenomenon is clearly shown in (a), when ks=0.0mm the maximum tangential velocity is about 2.83 times of the inlet velocity. When ks increases to 3.0mm the maximum tangential velocity is reduced 58.62%. This is mainly caused by the wall friction. As the increase of wall roughness, the friction between gas and wall becomes larger and this reduces the swirling flow and resulting in the decrease of tangential velocity. From (c) and (d) it is shown that, the tangential velocity almost changed to a straight line especially at a position near the leg. The wall roughness also strengthen this trend, with a larger roughness this position is higher.

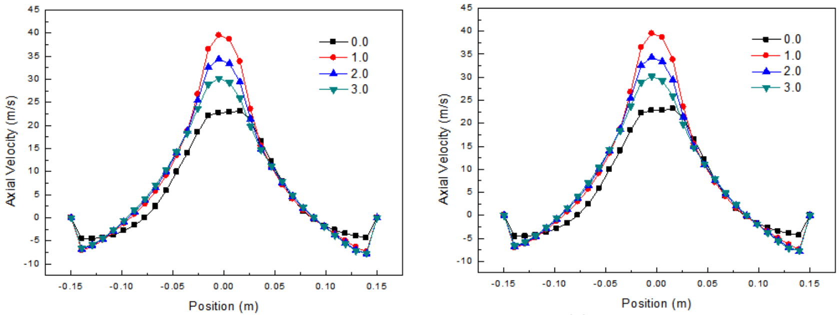


(a) z=-300mm (b) z=-900mm

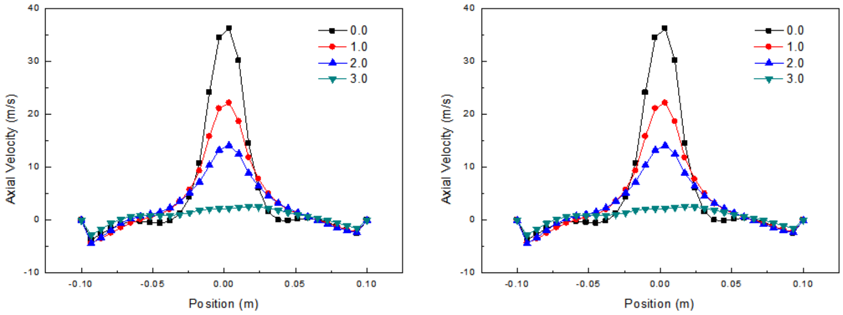


(c) z=-1320mm (d) z=-1800mm

Fig. 2 Tangential velocity of cyclone separator under different wall roughness



(a) z=-300mm (b) z=-900mm

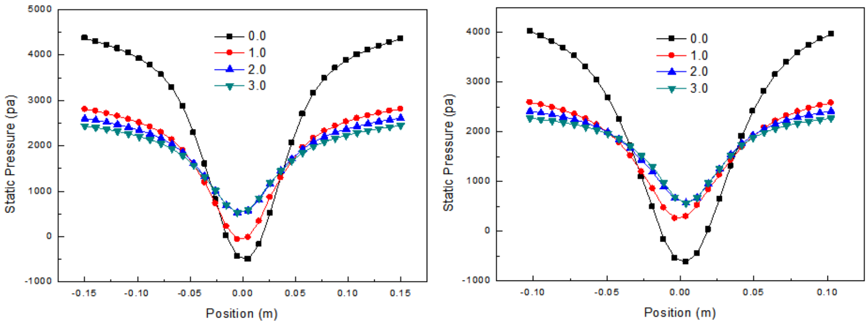


(c) z=-1320mm (d) z=-1800mm

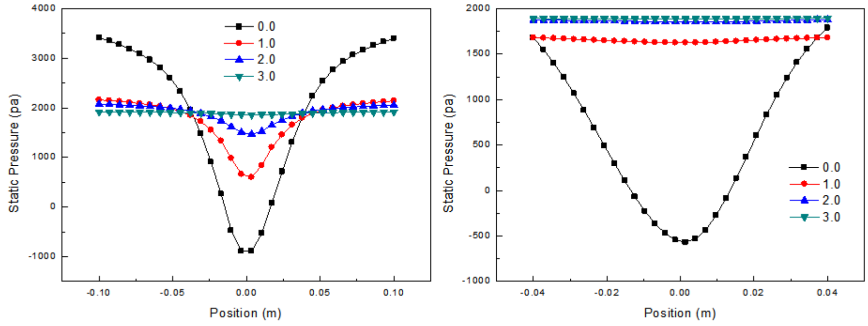
Fig. 3 Axial velocity of cyclone separator under different wall roughness

The axial velocity at four typical planes under different wall roughness is shown in Fig.3. The flow profile under different wall roughness is unchanged and axially symmetrical distributed. The gas flows downward near the wall and moves upward at the center, this is the most common phenomenon in reverse flow cyclone. Clearly, there is a reverse position, at which the axial velocity is zero. On the outer side of that position, the downward flow magnitude increase as the increase of the swirling radius; on the inner side of that position the upward flow increase as the decrease of the swirling radius, and a maximum axial velocity is achieved at the swirling center. The wall roughness has little influence on the downward flow, but its influence on the upward flow is large. At the center region of cylinder area (a), the axial velocity is reduced with the increase of roughness except for ks=0.0mm which is non-axial symmetrical distribution due to the affection of the retention area. For the cone area, due to the strong vortex swing, the axial velocity is chaos and there are no obvious laws to predict the flow pattern. As to the dust-collector area, the axial velocity is reduced with the increase of roughness, for ks=3.0mm, the profile is almost a straight line near zero. The same phenomenon can also be found in the leg region.

The static pressure distributions at different plane under different wall roughness are shown in Fig.4. With the increase of wall roughness the radial difference of pressure is lower. In the dust-collector region and leg region, the profiles almost changes to be straight lines, which means the vortex is dismissed in that region.



(a) z=-300mm (b) z=-900mm



(c) z=-1320mm (d) z=-1800mm

Fig. 4 Static pressure of cyclone separator under different wall roughness

Generally, the pressure drop over a cyclone is the difference of static pressure between the inlet and the outlet. Larger pressure drop usually means larger energy dissipation during the separation process, so reducing pressure drop is of critical importance for the design and operation of cyclone separator. In this simulation, the results show that increase wall roughness can reduce the pressure drop a lot, as shown in Fig.5. This is understandable because the friction between gas and wall reduced tangential velocity a lot, thus, the energy dissipation between gas phase and solid phase is reduced, resulting in a lower pressure drop.

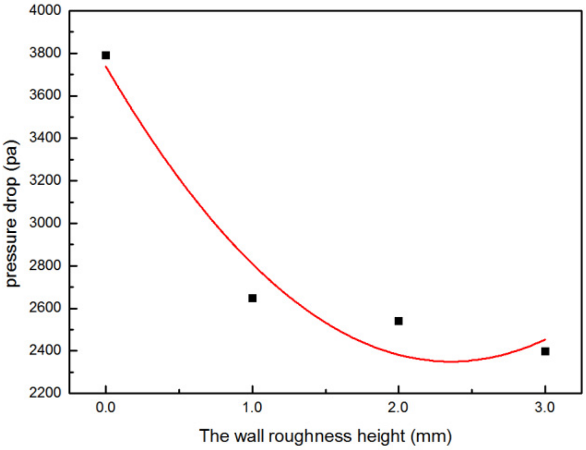


Fig. 5 Pressure drop of cyclone separator under different wall roughness

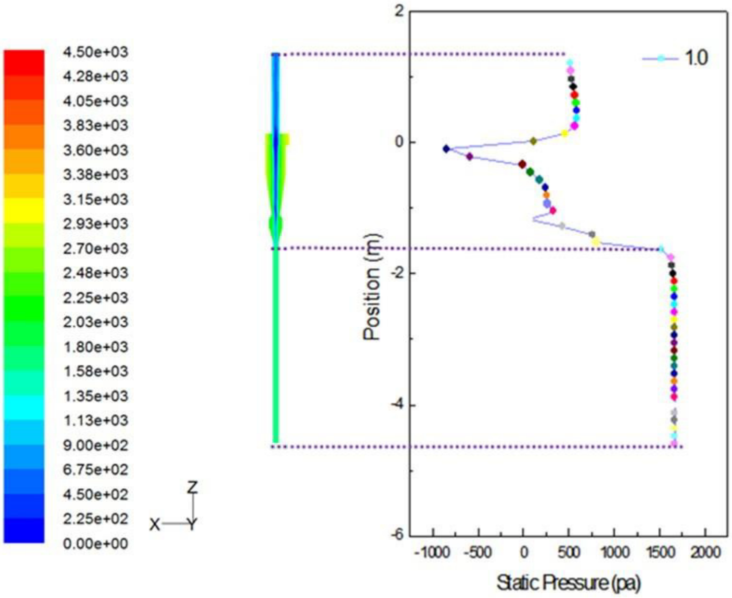
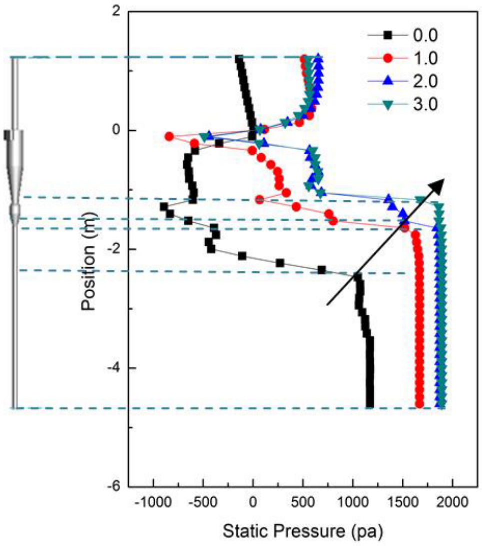


Fig. 6 Axial static pressure profile of cyclone separator with 1mm wall roughness

As discussed in introduction, the main separation domain is from the vortex finder to the end of vortex, so vortex length is critical for separation efficiency. The influence of wall roughness on vortex length is investigated in the research. Firstly, the vortex length is defined as the axial length from cyclone outlet to the end of vortex. At the end of vortex, there is a sharp change of static pressure and this is used to determine the vortex end. As shown in Fig.6, the static pressure has a sharp change at z=-1500mm~-2000mm and this is the axial position of the vortex end.



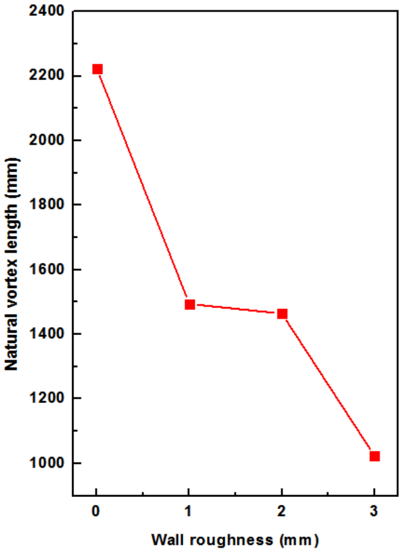
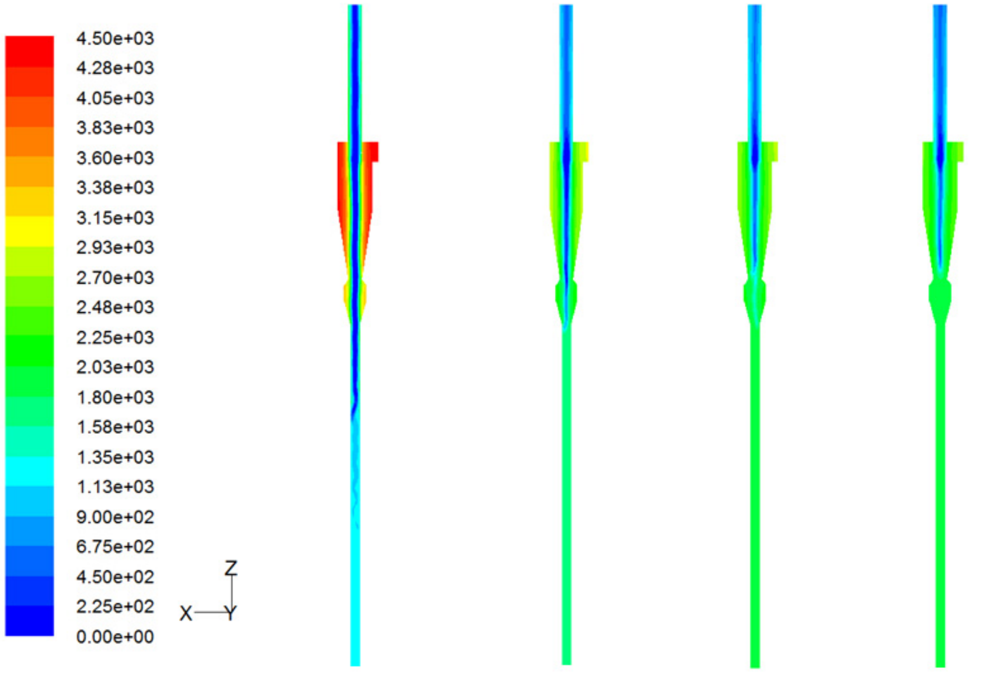


Fig. 7 Axial static pressure profile of cyclone separator with different wall roughness

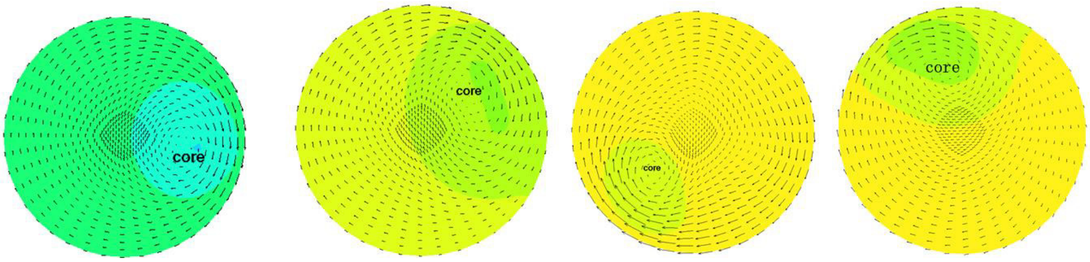
Fig. 8 Natural vortex length under different wall roughness

For other wall roughness, the static pressure profile in axial direction is compared in Fig.7. The sharp changing point increase as the wall roughness increase which means the vortex length is reduced gradually. From Fig. 8, it shows that the natural vortex length is reduced a lot with the increase of wall roughness. This is will reduce separation efficiency due to the decrease of effective separation domain. With a rougher wall, the friction between swirling gas and static wall is increased and this reduces gas swirling resulting in a reduce of vortex length. This is also clearly showed by the contour of static pressure under different wall roughness in Fig.9. The tangential velocity at vortex end is also shown in Fig.10; the swirling center is swigged to the wall with a lower velocity.



ks=0.0mm ks=1.0mm ks=2.0mm ks=3.0mm

Fig. 9 Axial static pressure profile of cyclone separator with different wall roughness



ks=0.0mm ks=1.0mm ks=2.0mm ks=3.0mm

Fig. 10 contour of tangential velocity at cyclone end

The influence on separation efficiency is also investigated. As discussed above and shown in Fig.11, with increased wall roughness the separation efficiency is reduced a lot especially for small particles. For the particles with a diameter larger than 8μm, the influence of wall roughness is negligible.

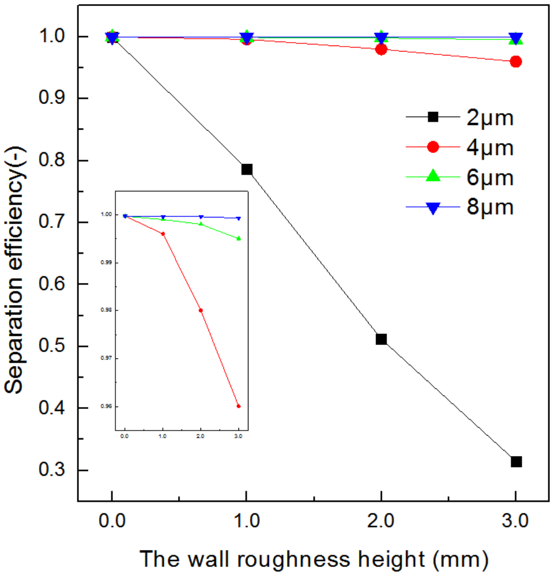


Fig. 11 under different diameters separation efficiency of cyclone with different wall roughness

5. Conclusion

The simulation results indicate that wall roughness has a great influence in cyclone flow field. With an increased wall roughness, the tangential velocity is reduced a lot and the vortex length is also reduced. This leads to two-side effects on cyclone performance: on one side, the pressure drop it reduced which is preferred; one the other side, the separation efficiency is also reduced which is not preferred. For a particular cyclone under particular working load, there must be a compromise condition between pressure drop and separation efficiency, which is in coincidence with Pareto optimality, and will be studied further.

Acknowledgements

The authors gratefully acknowledge the financial assistance from National Natural Science Foundation (No. 21276274) and the National Key Project of Basic Research of the Ministry for Science and Technology of China (No. 2014CB744304).

References

[1] Frazier, G.C., C. Mason, and E.J. Badin, Thermogravimetry-combustion characteristics of high-carbon fluidized-bed cyclone ashes. Fuel,.63(1984) 499-504.
[2] He, Y.B. and J.S. Laskowski, Effect of dense medium properties on the separation performance of a dense medium cyclone. Minerals Engineering, 7(1994) 209-221.
[3] Yang, J., G. Sun, and C. Gao, Effect of the inlet dimensions on the maximum-efficiency cyclone height. Separation and Purification Technology, 105(2013) 15-23.
[4] Surmen, A., A. Avci, and M.I. Karamangil, Prediction of the maximum-efficiency cyclone length for a cyclone with a tangential entry. Powder Technology, 207(2011) 1-8.
[5] Kaya, F., I. Karagoz, and A. Avci, Effects of Surface Roughness on the Performance of Tangential Inlet Cyclone Separators. Aerosol Science and Technology, 45(2011) 988-995.
[6] Elsayed, K. and C. Lacor, The effect of cyclone vortex finder dimensions on the flow pattern and performance using LES. Computers & Fluids, 71(2013) 224-239.
[7] S.A. Morsi and A.J. Alexander. "An Investigation of Particle Trajectories in Two-Phase Flow Systems". J. Fluid Mech. 55(1972) 193–208.
[8] Leschziner, M.A. and S. Hogg, Computation of highly swirling confined flow with a Reynolds stress turbulence model. AIAA Journal, 27(1989) 57-63.