JTE HCMUTE
ISSN: 2615-9740

JOURNAL OF TECHNICAL EDUCATION SCIENCE
Ho Chi Minh City University of Technology and Education
Website: https://jte.edu.vn
Email: jte@hcmute.edu.vn

Numerical Investigation on the Performance of Cyclone Separators

Thanh Tinh Tran*, Dang Khoa Nguyen
Flow Science Lab, Department of Thermal Engineering, Faculty of Vehicle and Energy Engineering, Ho Chi Minh City University of Technology and Education, Vietnam

*Corresponding author. *Email:* tinhtt@hcmute.edu.vn

ARTICLE INFO

Received:	10/08/2023
Revised:	09/09/2023
Accepted:	27/09/2023
Published:	28/10/2023

KEYWORDS

Cyclone separator;
Computational fluid dynamics;
Discrete phase model;
Collection efficiency;
Pressure drop.

ABSTRACT

Cyclone separators are commonly employed in the solid particle separation process due to their simplicity and low-cost manufacture. Collection efficiency and pressure drop are two of the most important factors. Small-size cyclone separator has outperformed collection efficiency. However, due to its small size, it's hard to detailed flow investigation experimentally. So, the target of this work is to perform a numerical investigation of the flow characteristics as well as the collection efficiency and pressure drop of small cyclone separators, using Ansys Fluent software. Discrete Phase Model is included to simulate the particle-flow interaction. Different values of velocity inlet and particle size are considered. Typical flow characteristics of cyclone separators with a strong vortex are revealed. Results also show that the larger particle size or the stronger inlet velocity provides higher collection efficiency. Particularly, for a typical particle diameter of 2.96 micrometers, the collection efficiency of the cyclone separator is always higher than 91% when the inlet velocity surpasses 15 m/s. All these results provide a valuable framework for the operation of cyclone separators.

Doi: https://doi.org/10.54644/jte.79.2023.1441

1. Introduction

Cyclone separators are gas-solid or liquid-solid separation devices that use centrifugal force produced by rotating gas or liquid to separate solid particles from gas or liquid. These kinds of devices are largely used for the separation of dispersed particles from their carrying fluid flow in industrial processes such as chemical, coal, petroleum, electric power, metallurgy, to name a few. Their advantages over other equipment include simple design, low-cost manufacture, low maintenance costs, and applicability for a wide range of operating conditions [1].

A conventional cyclone separator has a simple construction which consists of a cylinder with a tangential inlet, a cone, and a vortex finder, as shown in Fig. 1a. A mixture of particle-fluid, such as dust-containing gas enters the tangential inlet and forms spiral downwards inside the cyclone separator. The centrifugal force from the swirl motion drives particles in the mixture to move toward the outer wall. Particles are collected at the bottom of the cyclone separator. At the same time, due to the low-pressure zone in the center of the cyclone separator as well as the contraction of the cone, the remain of the mixture such as the cleaned gas moves upward and discharges from the vortex finder at the top.

Against the simplicity of construction and operating principle, the flow characteristics of the cyclone separator are complicated. The cyclone separator performance, which is mainly based on collection efficiency and pressure drop, is highly sensitive to any change in configuration and operating parameters [2]-[9]. The small-size cyclone separator is reported to have higher collection efficiency than the bigger one [10]. However, due to limited space, it's hard to give detailed complex flow characteristics of the small cyclone separator experimentally. Computational Fluid Dynamics (CFD) simulation is an alternative way to estimate the performance and to understand the effect of geometry and operating parameters on the flow field [11]-[14].

The main objective of this study is to perform a numerical investigation of flow inside a small cyclone separator. The steady state of the governing equations is solved by using Ansys Fluent 19.2 software.

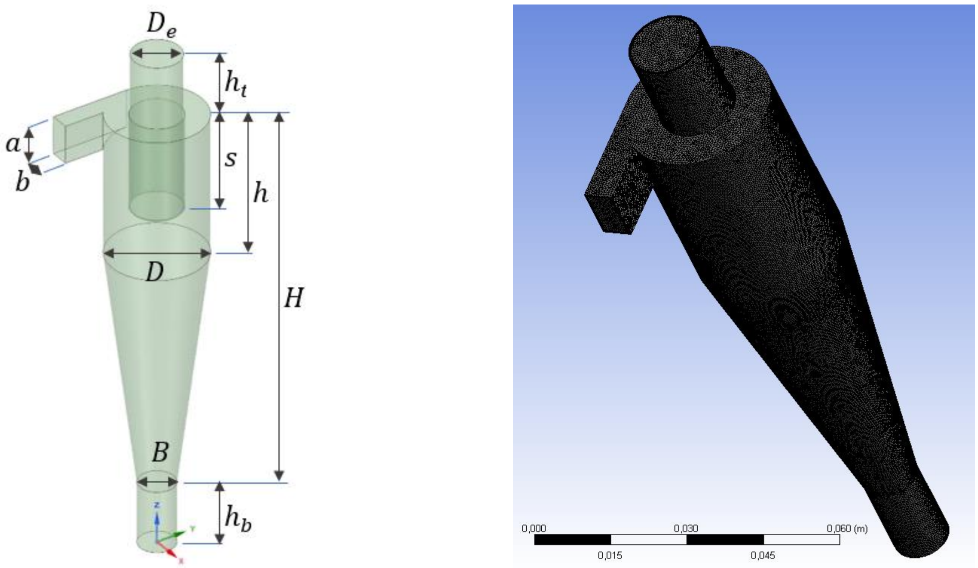
While particles are tracked through the continuum fluid by using the Discrete Phase Model – a Lagrangian approach in Ansys Fluent.

2. Model and numerical simulation

2.1. Model description

Figure 1a shows the schematic diagram of a cyclone separator which includes a tangential inlet of height a and width b; a cylinder of height h and diameter D; a cone of bottom diameter B, and a vortex finder of height s and diameter D_e. The coordinate origin is set at the center of the bottom surface, and the z-axis is positive upward. Detailed dimensions are listed in Table 1.

Unstructured meshes are generated by using Meshing package in Ansys Workbench. Meshes are fine near the wall as shown in Figure 1b. Detailed mesh size and mesh selection will be mentioned in the section of mesh convergence check.



(a) (b)

Figure 1. *Cyclone separator: (a) geometry and (b) computational mesh*

Table 1. *Geometrical dimensions of the cyclone separator*

Dimensions	Symbols	Value (mm)
Inlet height	a	12.2
Inlet width	b	6.1
Cylinder height	h	45.7
Cyclone height	H	122
Cyclone body diameter	D	30.5
Cone-tip diameter	B	11.4
Top outlet diameter	D_e	15.2
Top outlet height	h_t	20
Bottom outlet height	h_b	20
Vortex finder length	s	30.5

2.2. Governing equations

Governing equations for the three-dimensional, steady, incompressible flow are [15]:

$$\nabla \cdot \vec{u} = 0 \quad (1)$$

$$(\vec{u} \cdot \nabla)\vec{u} = -\frac{1}{\rho}\nabla p + \frac{\mu}{\rho}\nabla^2\vec{u} \quad (2)$$

Where μ and ρ are the dynamic viscosity and density of the fluid, respectively. This work uses the RNG $k - \varepsilon$ turbulence model.

Discrete phase model (DPM) [15] is used for tracking particles:

$$\frac{du_p}{d_t} = F_D(u - u_p) + \frac{v_p^2}{D/2} \quad (3)$$

$$\frac{dv_p}{d_t} = F_D(v - v_p) + \frac{u_p v_p}{D/2} \quad (4)$$

$$\frac{dw_p}{d_t} = F_D(w - w_p) - g \quad (5)$$

Where (u, v, w) are three components of fluid velocity, (u_p, v_p, w_p) are three components of particle velocity. The last term in the right-hand side of equations (3,4) are $\frac{v_p^2}{r_0}$ and $\frac{u_p v_p}{r_0}$ representing for centrifugal and Coriolis acceleration of particles. The first terms on right hand side of equations (3-5) are drag force components, and F_D is defined as $F_D = \frac{18\mu}{\rho_p d_p^2}\frac{C_D Re_p}{24}$, with the relative Reynolds number $Re_p = \frac{\rho d_p |u_p - u|}{\mu}$, and the drag coefficient $C_D = a_1 + \frac{a_2}{Re_p} + \frac{a_3}{Re_p}$. Where a_1, a_2, and a_3 are constants given by Morsi and Alexander [16].

2.3. Boundary conditions and simulation parameters

Air is used as working fluid in the simulations. Particles with uniform size and with particle phase volume fraction less than 10% are employed. Details physical parameters of working fluid and particle are given in Table 2. A non-slip boundary wall is applied on the cyclone and vortex finder wall surfaces. For liquid phase, this work used the inlet velocity from 5 m/s to 25 m/s. Pressure outlet boundary condition with zero value of gauge pressure is prescribed on both top and bottom outlets. For solid phase, particles are released at inlet with the concentration of $C_{in} = 10^{-3} kg/m^3$. They are reflected at wall, absorbed at the bottom outlet.

Table 2. *Physical parameters of working fluid and particle*

Physical parameters	Symbols	Value
Air density	ρ	1.225 kg/m^3
Air dynamic viscosity	μ	$1.789 \cdot 10^{-5}$ $Pa \cdot s$
Particle diameter	d_p	2.22, 2.96, 3.70 μm
Particle density	ρ_p	1050 kg/m^3
Particle concentration	C_{in}	10^{-3} kg/m^3

2.4. Mesh convergence

To check mesh convergence, three different meshes named M1 (1,019,337 elements), M2 (1,573275 elements), and M3 (2,092,874 elements) are employed. We perform simulations with the inlet velocity 15 m/s, and particle diameter 2.96 μm. Other parameters can be consulted Table 2. Velocity magnitude at the same point (x, y, z) = (0, 10, 100) for the three meshes are monitored as shown in Figure 2. Simulations are converged after 3000 iterations. The mesh M1 gives the converged result at 12.5 m/s, while both meshes M2 and M3 provide almost the same value of 11.5 m/s. It means that the results almost do not change if we use a finer mesh than M2. To compromise the accuracy of results and computing resources, the mesh M2 with 1.57 million elements is used for the remain of this work.

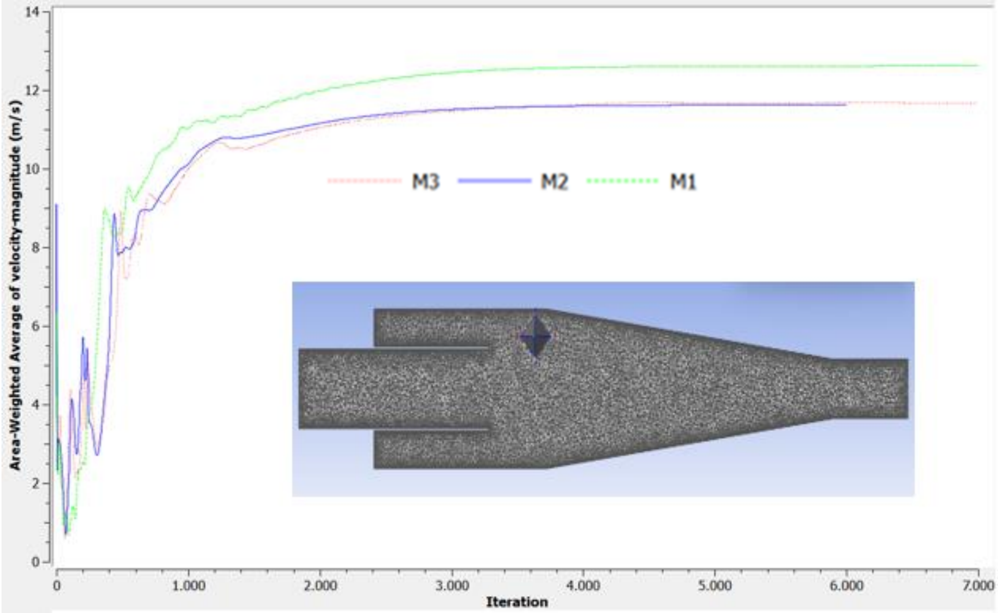


Figure 2. *Monitors of velocity magnitude for three meshes M1, M2, M3*

3. Results and Discussion

In this section, we first present flow characteristics in a cyclone separator with the typical operating parameters. Then, the effects of particle size and velocity inlet on the cyclone performance including collection efficiency and pressure drop are detailed. For convenience of result interpretation, we use different cross sections through cyclone separator as shown in Figure 3. There are five z-constant surfaces Z1 (z = 20 mm), Z2 (z = 50.6 mm), Z3 (z = 96.3 mm), Z4 (z = 129.8), Z5 (z = 135.9 mm); one y-constant surface Y (y = 0 mm); and one line L which is the interaction between surfaces Z3 and Y.

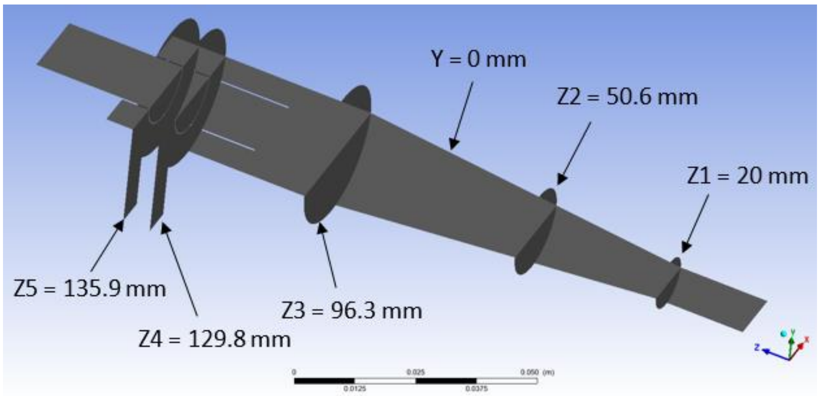


Figure 3. *Different surfaces used for result presentation*

3.1. Flow characteristics

Velocity inlet of 15 m/s and particle diameter of 2.96 μm are typical parameters of the cyclone separator [2]. These inputs are used in simulation in this section to investigate flow characteristics. Figure 4 shows vector field colored by velocity magnitude at the surfaces Z4, Z5. Due to forced flow from the tangential inlet, a strong swirl motion is formed. It holds a strong tangential component compared with a weaker axial component at the surfaces Z4, Z5. Other z-constant surfaces also have the same characteristics.

Velocity contours at the surfaces Z5 and Y in Figure 5 also reveal that a larger velocity magnitude up to 19 m/s is observed near the cyclone wall. While the zone of velocity magnitude least than 1 m/s

JTE
HCMUTE
ISSN: 2615-9740

JOURNAL OF TECHNICAL EDUCATION SCIENCE
Ho Chi Minh City University of Technology and Education
Website: https://jte.edu.vn
Email: jte@hcmute.edu.vn

locates near the center axis of the cyclone separator. Figure 6 presents the pressure contours at the same surfaces. The vicinity of center line regions is dominant with vacuum pressure (negative static pressure). Interestingly, these results align with the working principle of the cyclone separator mentioned in the previous section.

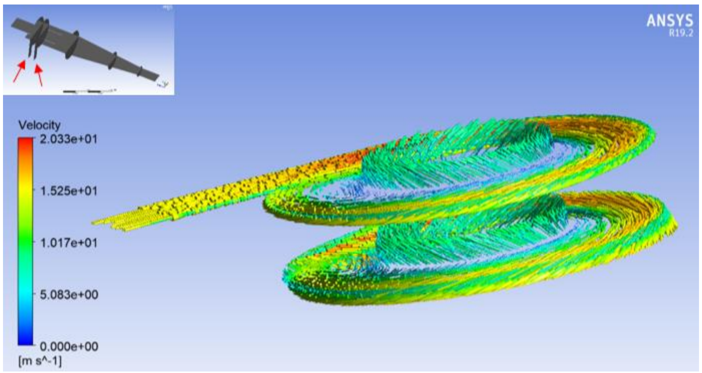
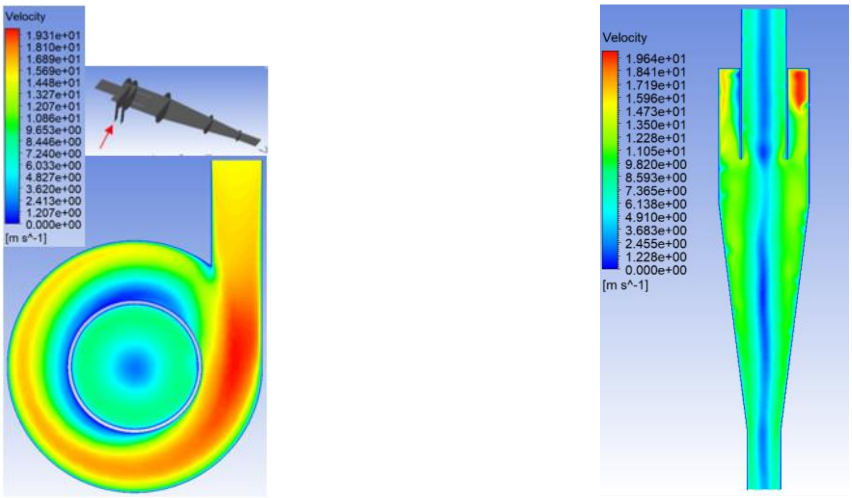
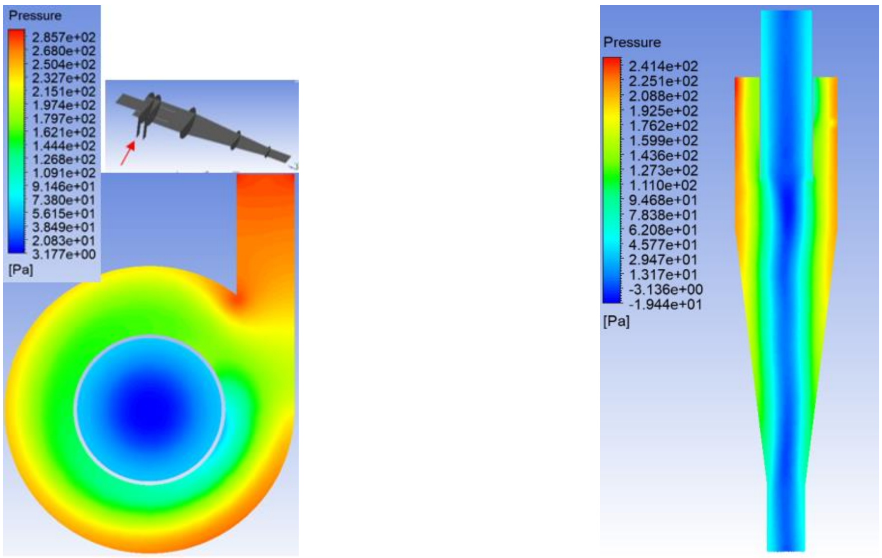


Figure 4. *Vector field colored by velocity magnitude at the surfaces Z4 = 129.8 mm, Z5 = 135.9 mm*



(a) (b)

Figure 5. *Velocity contour at the surfaces: (a) Z5, and (b) Y*



(a) (b)

Figure 6. *Pressure contour at the surfaces: (a) Z5, and (b) Y*

JTE
HCMUTE
ISSN: 2615-9740

JOURNAL OF TECHNICAL EDUCATION SCIENCE
Ho Chi Minh City University of Technology and Education
Website: https://jte.edu.vn
Email: jte@hcmute.edu.vn

Figure 7 reports profile of the axial velocity component along the line L. This profile is asymmetry due to asymmetry swirl motion which originates from only tangential inlet configuration. Around two-thirds distance from the cyclone center, the axial velocity direction is upward. While the remaining one-third outer rim has a downward velocity direction.

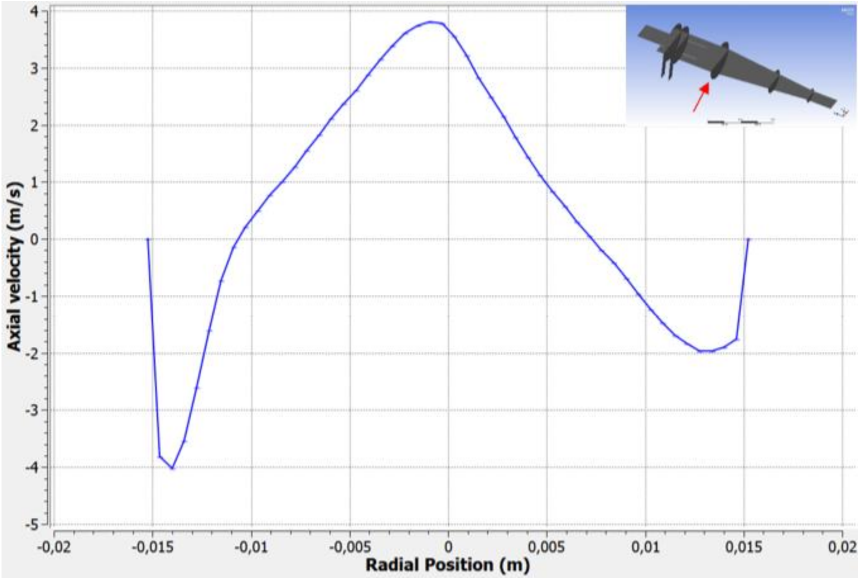


Figure 7. *Axial velocity at the line L*

The tangential velocity profile at the line L, again is asymmetry, as shown in Figure 8. It starts from zero at cyclone walls and quickly increases in the inner vicinity. The maximum tangential component in Figure 7 is more than four times larger than the axial component in Figure 8. It again emphasizes the existence of a strong vortex flow. The local value of the tangential velocity at the center is found not equal to zero. The reason could be the asymmetry characteristic of the flow which pushes the vortex center apart from the cyclone center. More importantly, there could be limits of the RNG $k-\varepsilon$ turbulence model. A similar limitation is also reported in the work of Azadi et al. [2] using RNG $k-\varepsilon$ or even with RSM turbulence models.

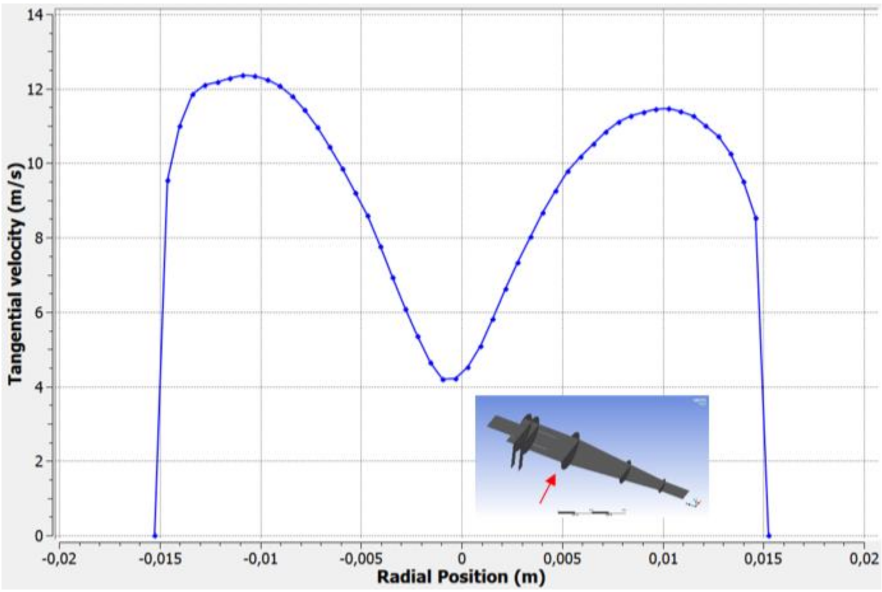
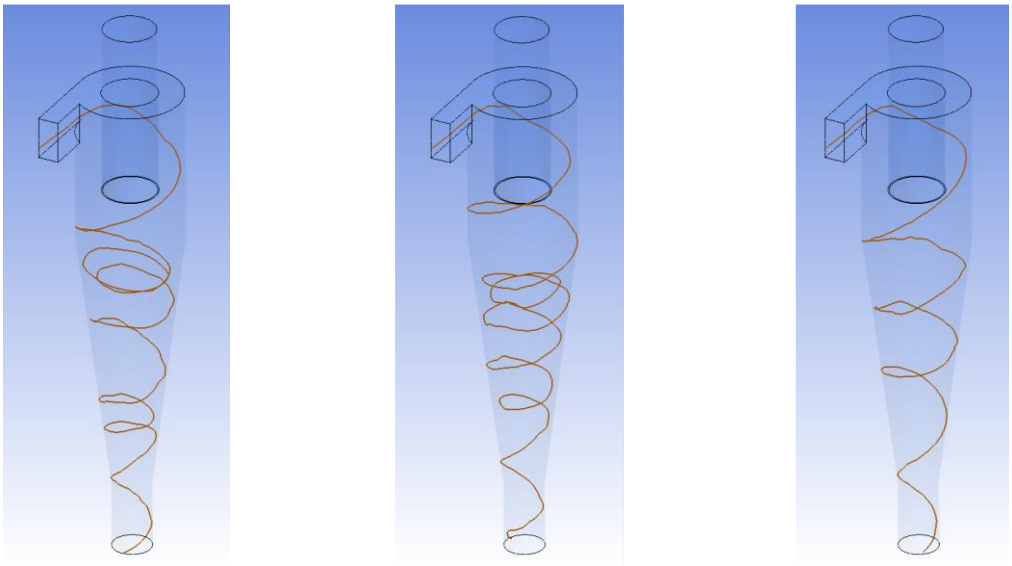


Figure 8. *Tangential velocity at the line L*

JTE
HCMUTE
ISSN: 2615-9740
JOURNAL OF TECHNICAL EDUCATION SCIENCE
Ho Chi Minh City University of Technology and Education
Website: https://jte.edu.vn
Email: jte@hcmute.edu.vn

3.2. Collection efficiency for different particle sizes

We perform simulations with a fixed inlet velocity of 15 m/s, with the same particle concentration, but with different particle diameters $d_p = 2.22\ \mu m$, 2.96 μm, and 3.70 μm. Particle tracking in Figure 9 shows that the bigger particles have shorter path lines. Larger means heavier particles that stand larger centrifugal force. So, it quickly approaches the cyclone wall. These above arguments could be used to explain the increased trend of the collection efficiency η for these three cases which is summarized in Table 3. Here, the collection efficiency η is defined as the ratio between the number of captured particles at the cyclone bottom to the total number of injected particles at the inlet. The case of the particle diameter 2.22 μm has $\eta = 76.73\%$. The remaining two cases with larger particle sizes have $\eta > 91\%$.



(a) (b) (c)

Figure 9. *Particle tracking for different particle diameters d_p: (a) 2.22 μm, (b) 2.96 μm, (c) 3.70 μm*

Table 3. *Collection efficiency for different particle sizes at the inlet velocity of* 15 m/s

Particle diameters d_p (μm)	**Collection efficiency η (%)**
2.22	76.73
2.96	91.58
3.70	95.99

3.3. Flow characteristics and collection efficiency for different inlet velocities

In this section, numerical simulations are carried out at the same particle diameter of 2.96 μm, but with different inlet velocities 5, 10, 15, 20, and 25 m/s. Figure 10 shows pressure contours at the surface Y for different inlet velocities. Vacuum pressure zones around the center line of the cyclone separator are expanded as inlet velocity increases. Due to larger vacuum pressure, one may expect that a higher inlet velocity case gives higher collection efficiency.

The above conjecture is confirmed when the collection efficiency and pressure drop of each case are calculated and reported in Table 4 and Figure 11. Higher inlet velocity gives higher collection efficiency but also leads to higher pressure drop. For the inlet velocity equal or superior to 15 m/s, the collection efficiency surpasses 91%. The increased trend of the pressure drop is more likely faster than a quadratic behavior. In addition, all these values of the pressure drop are smaller than the recommended value of 50,000 Pa [17].

JTE HCMUTE
ISSN: 2615-9740

JOURNAL OF TECHNICAL EDUCATION SCIENCE
Ho Chi Minh City University of Technology and Education
Website: https://jte.edu.vn
Email: jte@hcmute.edu.vn

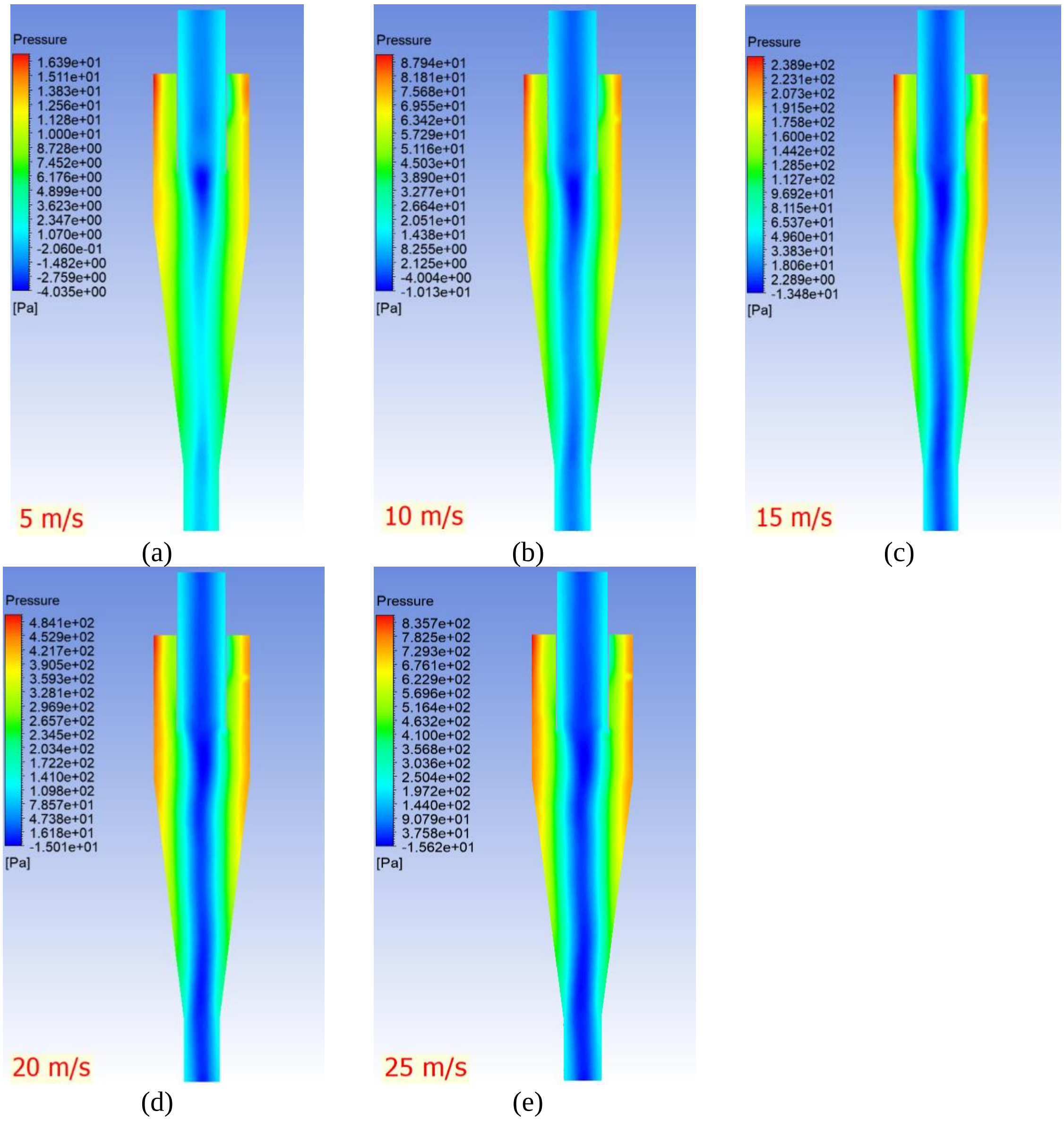


Figure 10. *Pressure contours at the surface Y for different inlet velocities: (a)* $5\ m/s$*, (b)* $10\ m/s$*, (c)* $15\ m/s$*, (d)* $20\ m/s$*, (e)* $25\ m/s$

Table 4. *Collection efficiency for different inlet velocity with particle sizes of* $2.96\ \mu m$

Inlet velocity u_{in} (m/s)	Pressure drop Δp (Pa)	Collection efficiency η (%)
5	21.01	65.24
10	90.17	77.41
15	218.38	91.58
20	418.84	98.26
25	700.58	99.86

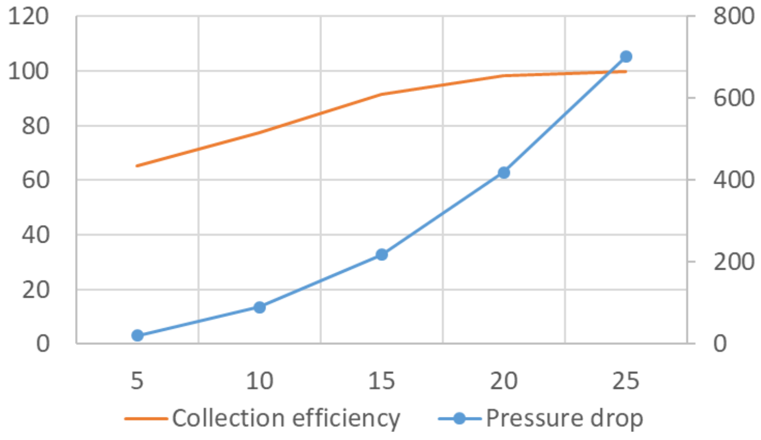


Figure 11. *Tangential velocity at the line L*

4. Conclusions

This study has performed numerical simulations of two-phase flow in a small cyclone separator. Detailed flow characteristics with a strong swirl motion and with vacuum pressure at the center line regions of the cyclone have been reported.

In addition, we assessed the influence of particle size and inlet velocity on cyclone performance. Our results indicate that the larger particle diameter has better collection efficiency. Larger inlet velocity effectively increases collection efficiency, but also accompanies with larger pressure drop.

To sum up, this investigation offers a crucial understanding of the flow characteristics and performance of cyclone separators. However, a more complicated turbulence model to better capture tangential velocity should be considered. It is also necessary to find out a more overall indicator that involved pressure drop and collection efficiency.

Acknowledgments

We acknowledge the supports from Ho Chi Minh City University of Technology and Education. Contributions from Nguyen Le Phat Dat and Nguyen Trieu Quoc Bao are highly appreciated.

REFERENCES

[1] J. Duan *et al.*, "Study and optimization of flow field in a novel cyclone separator with inner cylinder," *Advanced Powder Technology*, vol. 31, no. 10, pp. 4166-4179, 2020.
[2] M. Azadi *et al.*, "A CFD study of the effect of cyclone size on its performance parameters," *J. Hazardous Materials*, vol. 182, pp. 835-841, 2010.
[3] K. S. Lim *et al.*, "Characteristics of the collection efficiency for a double inlet cyclone with clean air," *J. Aerosol Sci.*, vol. 34, pp. 1085-1095, 2003.
[4] B. Zhao *et al.*, "Development of a symmetrical spiral inlet to improve cyclone separator performance," *Power Technol.*, vol. 145, pp. 47-50, 2004.
[5] T. C. Hsiao *et al.*, "Effects of the geometry configuration on cyclone performance," *J. Aerosol Sci.*, vol. 86, pp. 1-12, 2015.
[6] J. C. Kim and K. W. Lee, "Experimental study of particle collection by small cyclones," *Aerosol Sci. Technol.*, vol. 12, pp. 1003-1015, 1990.
[7] R. Xiang *et al.*, "Effects of cone dimension on cyclone performance," *J. Aerosol Sci.*, vol. 32, pp. 549-561, 2001.
[8] P. A. Patterson and R. J. Munz, "Gas and particle flow patterns in cyclones at room and elevated temperatures," *Can. J. Chem. Eng.*, vol. 74, pp. 213-221, 1996.
[9] Y. Zhu and K. W. Lee, "Experimental study on small cyclones operating at high flow rates," *J. Aerosol Sci.*, vol. 30, pp. 1303-1315, 1999.
[10] T. G. Chual *et al.*, "A CFD study of the effect of cone dimensions on sampling aerocyclones performance and hydrodynamics," *Powder Technol.*, vol. 162, pp. 126-132, 2006.
[11] J. Gimbun *et al.*, "The influence of temperature and inlet velocity on cyclone pressure drop: a CFD study," *Chem. Eng. Process*, vol. 44, pp. 7-12, 2005.
[12] J. J. Derksen *et al.*, "Two-way coupled large-eddy simulations of the gas-solid flow in cyclone separators," *AIChE J.*, vol. 54, pp. 872-885.
[13] S. K. Shukla *et al.*, "Evaluation of numerical schemes using different simulation methods for the continuous phase modeling of cyclone separators," *Adv. Powder Technol.*, vol. 22, pp. 209-219, 2010.
[14] S. K. Shukla *et al.*, "The effect of modeling of velocity fluctuations on prediction of collection efficiency of cyclone separators," *Appl. Math. Model.*, vol. 37, pp. 5774-5789, 2013.
[15] *ANSYS Fluent Theory Guide*, ANSYS Inc., 2019.

[16] S. A. Morsi and A. J. Alexander, "An investigation of particle trajectories in two-phase flow systems," *J. Fluid Mech.*, vol. 55, pp. 193-208, 1972.

[17] Z. Xiong *et al.*, "Development of a cyclone separator with high efficiency and low pressure drop in axial inlet cyclones," *Powder Technol.*, vol. 253, pp. 644-649, 2014.

Tran Thanh Tinh received B.Eng. in Aerospace Engineering from Ho Chi Minh City University of Technology, Vietnam, in 2008; and Ph.D. in Fluid Mechanics from École Nationale Supérieure de Mécanique et d'Aérotechnique, France, in 2013.
He worked as a Postdoctoral Researcher and a Graduate Lecturer at the Gyeongsang National University, South Korea (2013-2016); and a Postdoctoral Researcher (2016-2019) and a Research Scientist (2019-2021) at the Korea Institute of Fusion Energy, South Korea. Currently, he serves as a Lecturer at the Department of Thermal Engineering, Ho Chi Minh City University of Technology and Education, Vietnam.
Email address: tinhtt@hcmute.edu.vn.

Nguyen Dang Khoa is currently the last year undergraduate student in thermal engineering at the Ho Chi Minh City University of Technology and Education, Vietnam.
Email address: 19147116@student.hcmute.edu.vn.