CFD Study of Cyclone Performance: Effect of Inlet Section Angle and Particle Size Distribution

Ratchanon Piemjaiswang[a], Kongpob Ratanathammapan[b], Prapan Kunchonthara[a,c], Pornpote Piumsomboon[a,c], Benjapon Chalermsinsuwan[a,c,*]

[a]Department of Chemical Technology, Faculty of Science, Chulalongkorn University, 254 Phayathai Road, Patumwan, Bangkok 10330, Thailand
[b]SCG Chemicals Co., Ltd., 10 I-1 Road, Map Ta Phut Industrial Estate, Muang, Rayong 21150, Thailand
[c]Center of Excellence on Petrochemical and Materials Technology, Chulalongkorn University, 254 Phayathai Road, Patumwan, Bangkok 10330, Thailand

*Corresponding author
benjapon.c@chula.ac.th

Graphical abstract
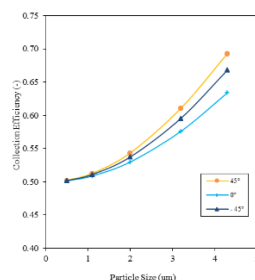
Abstract

A numerical simulation technique was employed to model the two phase flow in cyclones using computational fluid dynamics (CFD). Three different inlet angles of cyclone, including 45, 0 and -45 degrees were compared to describe the efficiency of the conventional cyclone with the modified inlet angle ones. The results showed that the interaction between solid particles in dilute system could be neglected. The pressure drop was decreased when the inlet angle of the cyclone increased. The cyclone with 45 degrees inlet angle tended to have the lowest pressure drop. The collection efficiency was improved with 45 degrees inlet angle due to high swirling motion of gas flow.

Keywords: CFD, collection efficiency, pressure drop, cyclone, inlet section angle

1.0 INTRODUCTION

Cyclone is one of the most popular devices used in many industrial processes in order to reduce solid emission by capturing and separating the solid particles using centrifugal and gravity forces since it is compact, low capital cost and easy to maintenance [1]. Adding cyclone to the circulating fluidized bed reactor leads to higher overall process chemical reaction due to the unreacted solid particles is able to re-enter the reactor. As a fundamental of cyclone, gas and solid particles are transported into cyclone inlet and moved toward the wall in swirling motion. The particular size of the solid particles will be separated. The solid particles hit the wall, lost their energy and fall into the dust bin. The gas then moves out the cyclone at the top [2].

Two parameters that most researchers used to assess the performance of cyclone are pressure drop and collection efficiency. The collection efficiency is identified as the portion of specific solid particle size that is captured by the cyclone [3] while the pressure drop is more likely related to energy consumption with representing the operation cost of cyclone.

Computational fluid dynamic is problem solving method using numerical method [4]. The calculation domain is divided into small control volumes. The calculation is based on conservation equations of momentum, mass, and energy. The flow hydrodynamics is derived based on Navier-Stokes equations [5, 6]. The employed differential equations then discretize into algebraic equations and solve using high performance computer.

From previous literature studies, Yang *et al.* [7] suggested that the vortex length played important roles in cyclone efficiency. By increasing the cone height, the vortex length was extended resulting in the increase of collection efficiency. In contrast, if

vortex length was exceed over to the cone space, the efficiency tended to decrease. Wang *et al*. [8] composed that the position of the feed point gave different collection efficiency. In addition, a solid particle with the size above critical diameter tended to stagnate to the wall of cyclone cone. Bernardo *et al*. [9] studied the three-dimensional computation fluid dynamics of cyclone by varying three different inlet angles, 30, 45, and 60 degrees. The computation showed that the overall collection cyclone efficiency increased from 54.4% to 77.2% under the same operation condition. Qian and Wu [10] purposed a numerical method of flow behavior of talcum powder inside cyclone with inlet angle of 15, 30 and 45 degrees. The results illustrated that the velocity in tangential direction at the cyclone center was slower than the outer vortex, which then drove the solid particles to flow toward the wall. This enhanced the cyclone performance and reduced the cyclone pressure drop.

In this study, the three-dimensional model for two phase flow inside cyclone that has different inlet angle and particle size distribution using Euler-Euler approaches was developed to describe the efficiency of the conventional cyclone comparing with the modified inlet angle ones. The CFD simulations, which will describe in the next section, were used to explore the flow of the gas-solid particle.
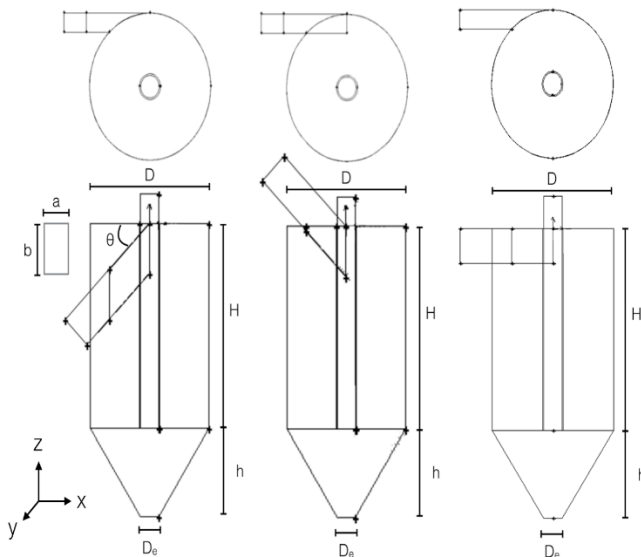
designing values. The cyclone diameter and height were 46 mm and 99 mm, respectively. The cyclone inlet was modified to be inclined with the tangent line. The computational domain was divided into grid (mesh) with the sizes in the range of 22,000 to 190,000 cells. A grid examination turned out that when the mesh is too coarse, the mesh can have a remarkable influence on the simulation results as shown by the pressure drop results in Figure 2. Therefore, it can be concluded that the mesh with 49,000 cells was adequate and the change in the result was very small when the number of cells was increased. This suggested that computational results were independent of the mesh size and it required less computational resources than it was at 190,000 cells.

Table 1 Dimensions of cyclone design

Parameter	Value
Inlet height, a (mm)	6
Inlet width, b (mm)	12
Inlet angle, θ	-45°, 0°, 45°
Outlet diameter, d (mm)	7
Vortex finder diameter, D_e (mm)	7
Vortex finder length (mm)	69
Cyclone diameter, D (mm)	46
Cylinder height, H (mm)	69
Cone height, h (mm)	30

Figure 1 Cyclone design notations and dimensions.

Figure 2 Grid independent test

2.0 EXPERIMENTAL

2.1 Computational Domain

The cyclone design was originally based on Lim *et al*. double cyclone [11]. Figure 1 illustrates the drawing and the notations of the cyclone. Table 1 shows their
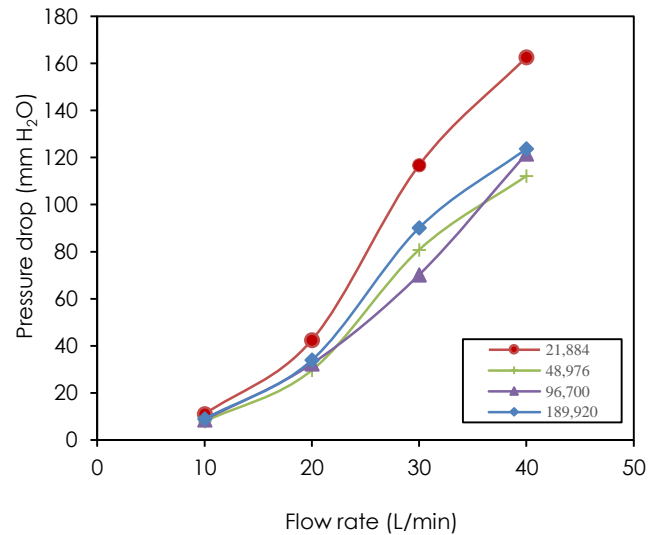
2.2 Mathematical Model

ANSYS FLUENT 14.0 was used in this study to simulate the model. The numerical model was based on conservation equations and constitutive (from the kinetic theory of granular flow) equations of Euler-Euler multiphase flow model. Correspondingly, each phase was solved separately. The drag force between solid particle and gas was calculated using

Wen-Yu drag model as the system was diluted [12]. This model is similar to Schiller-Naumann model with the modification of Reynolds number as following equations [13].

$$C_D = \frac{24}{Re_p(1-\epsilon_s)}[1 + 0.15\left((1-\epsilon_s)Re_p\right)^{0.687} \tag{1}$$

$$\beta = \frac{3}{4}C_D \frac{(1-\epsilon_s)\epsilon_s\rho_g|v_g-v_s|}{d_s}(1-\epsilon_s)^{-2.65} \tag{2}$$

Where,

$$Re_p = \frac{d_s\rho_g|v_g-v_s|}{\mu_g} \tag{3}$$

To simulate turbulence flow, the Reynolds stress model (RSM) with standard wall function was considered to be used as suggested by Wang *et al.* [8]. This model appeared to be in good agreement between the experiment and calculation, which was verified to use for predicting the flow pattern of solid particles inside cyclone reactor. The Reynolds stress model is described below:

$$\frac{\partial}{\partial t}\left(\rho\overline{u_i'u_j'}\right) + \frac{\partial}{\partial x_k}\left(\rho u_k\overline{u_i'u_j'}\right) = D_{ij} + P_{ij} + \Pi_{ij} + \varepsilon_{ij} \tag{4}$$

The stress diffusion term

$$D_{ij} = -\frac{\partial}{\partial x_k}[\rho\overline{u_i'u_j'u_k'} + \overline{(p'u_j')}\delta_{ik} + \overline{(p'u_i')}\delta_{jk} + \mu(\frac{\partial}{\partial x_k}\overline{u_i'u_j'})] \tag{5}$$

The shear production term

$$P_{ij} = -\rho[\overline{u_i'u_k'}\frac{\partial u_j}{\partial x_k} + \overline{u_j'u_k'}\frac{\partial u_i}{\partial x_k}] \tag{6}$$

The pressure-strain term

$$\Pi_{ij} = p\overline{(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i})} \tag{7}$$

The dissipation term

$$\varepsilon_{ij} = -2\mu\overline{\frac{\partial u_i'}{\partial x_k}\frac{\partial u_j'}{\partial x_k}} \tag{8}$$

The notation of each symbol in the model can be found in ANSYS FLUENT's user guide [14].

2.3 Boundary Conditions

All the simulation was operated in a three-dimensional space. The no-slip wall boundary condition was specified. Initially, a cyclone was filled with small amount of solid particles with uniform solid volume fraction for preventing the divergence problem at the simulation beginning. The gas then flowed in the system with the flow rate of 10, 20, 30 and 40 L/min, respectively, with an equal solid

volume fraction of solid particles. Polystyrene latex solid particles with a size of 0.5, 1.1, 2.0, 3.2 and 4.3 µm and density of 1,050 kg/m³ were used in this simulation. The solid particle size distribution is shown in Table 2. The solid particles were supposed to be elastically reflected by the wall. The inlet was set as velocity inlet and both outlets were set as outflow. The summary of the simulation conditions are illustrated in Table 3.

Table 2 Particle size distribution

Diameter (µm)	0.5	1.1	2.0	3.2	4.3
Solid weight percent (%)	8.72	18.27	32.09	27.35	8.73

Table 3 Simulation conditions

Parameter	Value
Multiphase model	Euler-Euler
Turbulence model	Reynolds stress model
Calculation scheme	Phase coupled SIMPLE
Spatial discretization	Second order upwind
Gas-solid drag model	Wen-Yu
Solid-solid drag model	Syamlal-Obrien-Symmetric
Inlet flow rate	10, 20, 30, and 40 L/min

2.4 Overview of Experiment

The experiments were initially performed to investigate the effect of solid-solid interaction between each particle size. The multi-size particles with equal volume fraction were fed into cyclone. Then, the mass flow rate of each size was recorded and calculated to obtain the collection efficiency. Afterward, the solid particle with normal size distribution and mono-size particle were studied. As mention above, the simulation results and numerical models were validated with the experimental result from Lim *et al.* [11]. Finally, three different cyclone inlet angles was explored.

The CFD simulations were performed with 4th Generation Intel® Core™ i7 4.00 GHz processor with 24 GB of memory. The iteration time was varied between 8-12 hours for each case.

3.0 RESULTS AND DISCUSSION

3.1 The Effect of Particle Size Distribution

Figure 3 shows overall collection efficiency of cyclone when fed with multi-size particles, normal distributed-size particles, and mono-size particle. The result indicated that the collection efficiency for

each solid particle size distribution was very similar. It can be implied that in dilute system, the interaction between solid particles is negligible. This is because the interaction between the two phases is remarkably dominant. Therefore, the rest of the study was simulated based on multi-size solid particles.

Figure 4 shows the tangential velocity contour of gas inside the cyclone after the system reached steady state. As the gas mixture entered the annulus cylindrical region, the vortex was built up in a swirling motion in the cylindrical part of cyclone downward toward the solids outlet. The main vortex in cyclone body was flowed with high velocity from inner wall approaching the outer wall which will be described in the next section. A small vortex with the moderate velocity was formed between the end of vortex finder and the cone. This vortex then drove the solid particles reversely up toward the gas outlet.

Figure 5 shows the solid volume fraction contour of solid particle inside the cyclone after the flow was reached steady state. The solid particles were flown under the influences of centrifugal and frictional forces. The larger solid particles moved along the vortex and collide with the wall. The smaller solid particles were substantial entrained. The solid particles with the size above 2.0 μm were seemed to be accumulated in the cone of cyclone as shown in Figure 5 [15, 16], due to the gravitational and frictional forces between the solid particles and cyclone wall are higher than gas-solid particle drag forces.
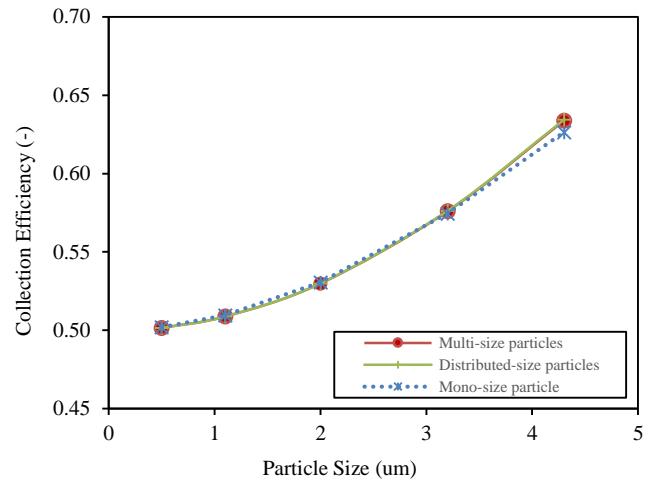

Figure 3 Cyclone collection efficiency with (a) multi-size particles feed, (b) normal distributed-size particles feed and (c) mono-size particle feed
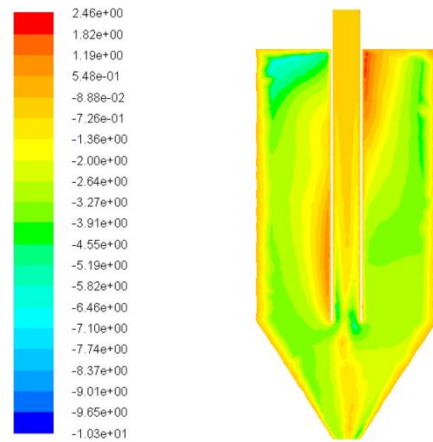

Figure 4 Tangential velocity contour of gas phase in cyclone with multi-size particles feed
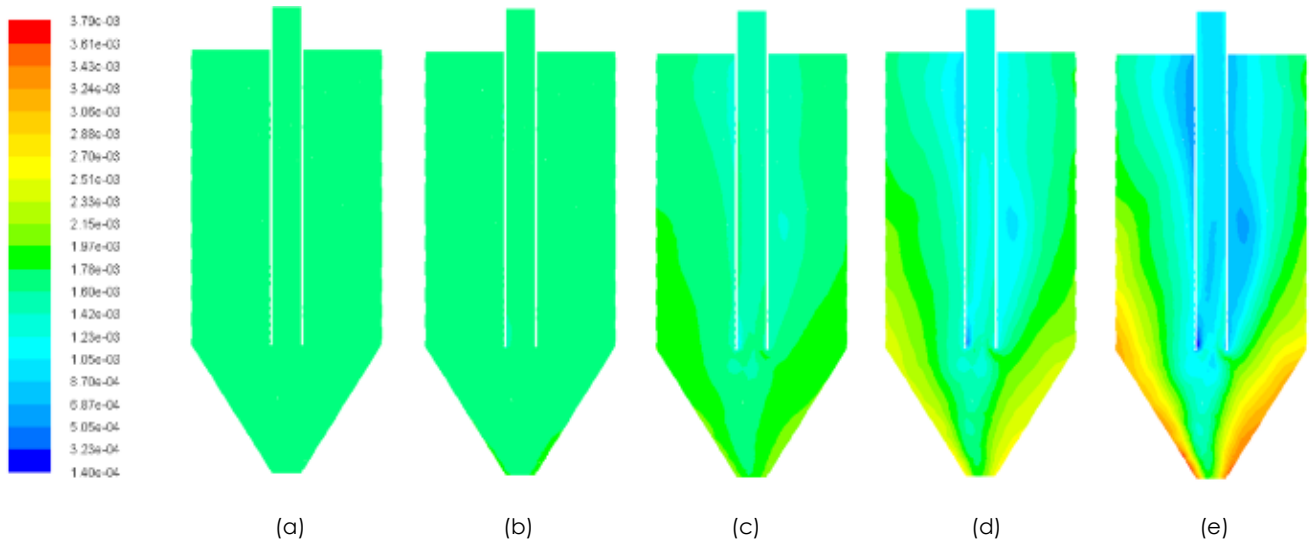

(a)　　　　(b)　　　　(c)　　　　(d)　　　　(e)

Figure 5 Solid volume fraction contour of solid particles in cyclone with different particle sizes, (a) 0.5 μm, (b) 1.1 μm, (c) 2.0 μm, (d) 3.2 μm and (e) 4.3 μm

3.2 The Effect of Inlet Angle

3.2.1 Cyclone Pressure Drop

Figure 6 displays the relation between gas flow rate and pressure drop. The changing of absolute pressure between cyclone inlet and outlet was calculated as the pressure drop. As of Figure 6, the cyclone pressure drop with -45° inlet angle using flow rate of 10 L/min could not be determined because gas velocity was too low to push the solid particles toward an inclined plane, which caused the solid particles to flow in reverse direction. The pressure drops were increased with the increase of flow rate as expected [17]. The cyclone pressure drop with the inlet angle of 45° was slightly lower than that of the cyclone with conventional inlet due to the solid particles with this cyclone geometry were driving towards the cyclone wall with the greater centrifugal force. Overall, the result trend clearly showed that, with the change of the inlet, the pressure drop was considerably decreased. The cyclone pressure drop is occurred because of the conveyance of gas and solid particles. The cyclone pressure drop is greatly increased corresponding to gas flow rate and solid particle loading.

3.2.2 Cyclone Collection Efficiency

This study result was found to be related with the flow study by Qian and Zhang [18]. Figure 7 displays the comparison of tangential velocity at the plane of Z = -69 mm and Y = 0 mm. The increasing inlet angle would seem to improve the collection efficiency. At higher inlet angle, the velocity in tangential direction was clearly noticeable. In case of positive inlet angle, the velocity in tangential direction near the inner vortex finder wall was slightly increasing toward the outer vortex. The high tangential velocity in the outer vortex drives the solid particles to proceed against the cyclone wall and lost their energy leads to the higher collection efficiency [10]. Figure 8 showed that the cyclone inlet angle could improve overall cyclone collection efficiency. For the cyclone with positive angle value, the solid particle flows reclined with the higher velocity as the higher gravity force and the lower frictional force between fluid and walls led to the higher tangential velocity as shown in Figure 7. For the cyclone with negative angle value, the gas phase moves slower because the air hit against the top wall of cyclone first and lost its energy caused the velocity of the flow to drop. In conclusion, cyclone inlet angle improved overall efficiency of the solid particle with the size of 4.3 μm by over 9% and slightly deceased as the solid particle size get smaller. However, both inclined and declined inlet angles improve the cyclone collection efficiency compared with conventional inlet section.
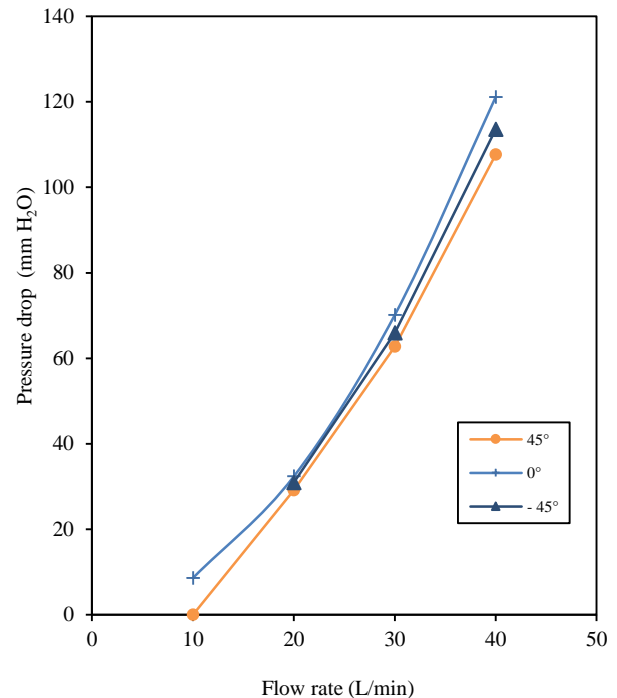

Figure 6 Cyclone pressure drop with different inlet angle, 45° inlet angle (●), 0° inlet angle (+) and -45° inlet angle (▲) at flow rate of 10 to 40 L/min
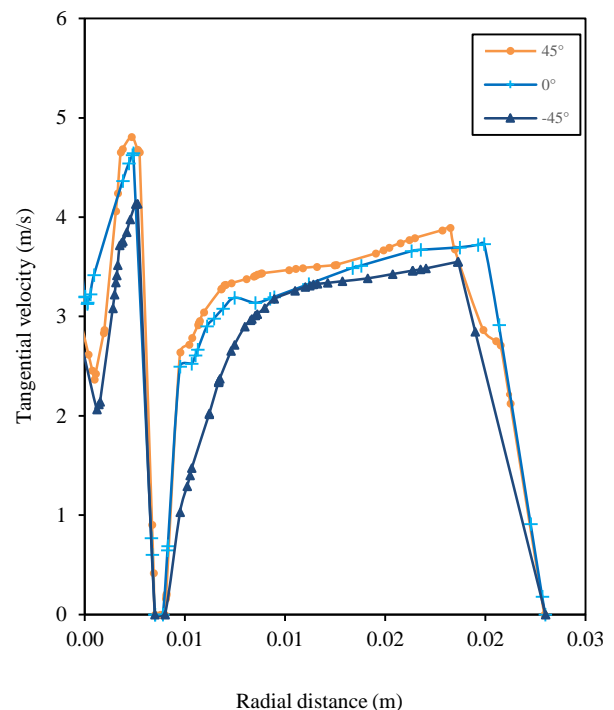

Figure 7 The tangential velocity profile, 45° inlet angle (●), 0° inlet angle (+) and -45° inlet angle (▲) at flow rate of 40 L/min
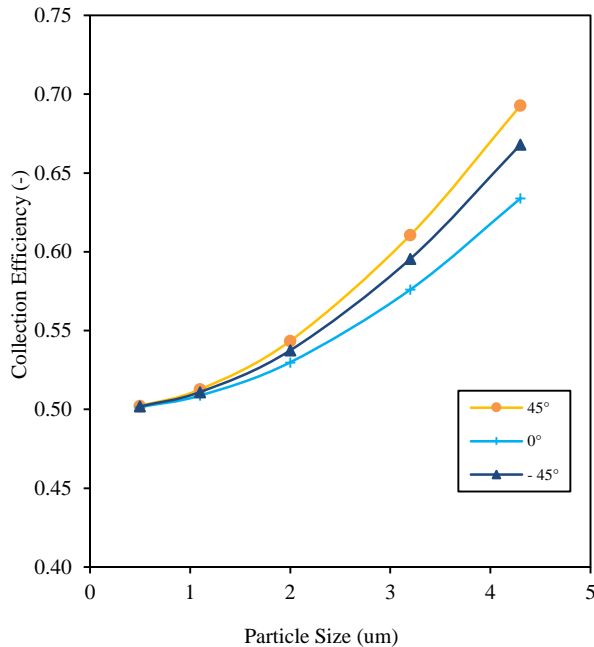
Figure 8 Overall collection efficiency of cyclone with different inlet angle, 45° inlet angle (●), 0° inlet angle (+) and -45° inlet angle (▲) at flow rate of 40 L/min

4.0 CONCLUSION

A computational fluid dynamics was used to model the flow behavior in cyclones. The employed cyclones with five different inlet angles, 45, 0 and -45 degrees were compared to determine which ones give better cyclone performance parameters. The obtained information then can be used for designing the cyclone. The results acquired from the study can be concluded as

1) With the increasing of cyclone inlet angle, the cyclone pressure drop decreased. The cyclone with 45 degrees inlet angle tended to have the lowest pressure drop.
2) The collection efficiencies of cyclone were significantly improved when the inlet angle of the cyclone was increased especially for the solid particles with the size greater than 2 μm. The cyclone with 45° inlet section angle was found to be the most desirable result for large particle size. The appropriate force balance inside the system is the reason for this obtained phenomenon.
3) The higher inlet angle, the tangential velocity tended to be higher across the section, which leads to the greater overall collection efficiency.
4) The solid particle interaction in dilute system was negligible. This is because the interaction between gas phase and solid particle phase was dominant.

References

[1] Lhairy, E. and Chris, L. 2011. The Effect Of Cyclone Inlet Dimensions On The Flow Pattern And Performance. *Applied Mathematical Modelling*. 35: 1952-1968.
[2] Wen, C. Y. 2003. *Handbook of Fluidization and Fluid-Particle Systems*. CRC Press. 601-602.
[3] Leith, D., Dirgo, J. A. and Davis, W. T. 1986. 6 - Control Devices: Application, Centrifugal Force and Gravity, Filtration, and Dry Flue Gas Scrubbing, in Air Pollution. Third Edition. A.C. Stern, Editor., Academic Press: San Diego. 233-266.
[4] Versteeg, H. K. and Malalasekera, W. 1995. *An Introduction To Computational Fluid Dynamics*. Addison Wesley Longman Limited.
[5] Kashyap, M. and Gidaspow, D. 2010. Computation And Measurements Of Mass Transfer And Dispersion Coefficients In Fluidized Beds. *Powder Technology*. 203(1): 40-56.
[6] Demir, S. 2014. A Practical Model For Estimating Pressure Drop In Cyclone Separators: An Experimental Study. *Powder Technology*. 268: 329-338.
[7] Yang, J., Liu, C., Li, S., Sun, B., Xiao, J. and Jin, Y. 2015. A New Pressure Drop Model Of Gas–Liquid Cyclone With Innovative Operation Mode. *Chemical Engineering and Processing: Process Intensification*. 95: 256-266.
[8] Wang, B., Xu, D. L., Chu, K. W. and Yu, A. B. 2006. Numerical Study Of Gas–Solid Flow In A Cyclone Separator. *Applied Mathematical Modelling*. 30(11): 1326-1342.
[9] Bernardo, S., Mori, M., Peres, A. P. and Dionisio, R. P. 2006. 3-D Computational Fluid Dynamics For Gas And Gas-Particle Flows In A Cyclone With Different Inlet Section Angles. *Powder Technology*. 162(3): 190-200.
[10] Qian, F. and Y. Wu, 2009. Effects Of The Inlet Section Angle On The Separation Performance Of A Cyclone. *Chemical Engineering Research and Design*. 87(12): 1567-1572.
[11] Lim, K. S., H. S. Kim, and K. W. Lee, 2004. Comparative Performances Of Conventional Cyclones And A Double Cyclone With And Without An Electric Field. *Journal of Aerosol Science*. 35(1): 103-116.
[12] B. G. M. van Wachem, J. C. S., and van den Bleek, C. M. 2001. Comparative Analysis of CFD Models of Dense Gas–Solid Systems. *AIChE Journal*. 47.
[13] Milad Mottaghi, H. A. and Torabzadegan, M. 2012. Assess Proper Drag Coefficient Models Predicting Fluidized Beds Of CLC Reactor By Utilizing Computational Fluid Dynamic Simulation.
[14] ANSYS Fluent Theory Guide retrieved July, 8, 2015 from http://orange.engr.ucdavis.edu/Documentation12.0/120/FLUENT/flth.pdf.
[15] Vascellari, M., Roberts, D. G. Hla, S. S., Harris, D. J. and Hasse, C. 2015. From Laboratory-Scale Experiments To Industrial-Scale CFD Simulations Of Entrained Flow Coal Gasification. *Fuel*. 152: 58-73.
[16] Wan, G., Sun, G., Xue, X. and Shi, M. 2008. Solids Concentration Simulation Of Different Size Particles In A Cyclone Separator. *Powder Technology*. 183(1): 94-104.
[17] Shephered C. B. and Lapple C. E. 1939. Flow Pattern And Pressure Drop In Cyclone Dust Collectors. *Industrial and Engineering Chemistry*. 31(8): 972-984

[18] Qian, F. P. and Zhang, M. Y. 2007. Effects of the Inlet Section Angle on the Flow Field of a Cyclone. *Chemical Engineering* & Technology. 30(11): 1564-1570.