International Journal Of Engineering Research And Development
e- ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 14, Issue 9 (September Ver. II 2018), PP.53-57

Design and Development of Tangential Cyclone Dust Collector.

Rahul Panchal[1], Zheng (Jeremy) Li[2]
M.S. Student, Department of Mechanical Engineering, University of Bridgeport, Bridgeport, USA. [1]
Assistant Professor, Department of Mechanical Engineering, University of Bridgeport, Bridgeport, USA [2]
Corresponding Author: Rahul Panchal

ABSTRACT*: Cyclone dust collector is a stationary mechanical device that is used to filter solid particles from a carrier gas. Performance of cyclone is affected by two vital parameters, namely collection efficiency and pressure drop. Both parameters are strongly influenced by the geometry of the device. The geometric parameters that affect its performance are mainly height of conical section and cylinder, vortex finder length, vortex diameter, cone dip diameter, barrel diameter etc. This paper mainly focuses on the geometric parameters of a cyclone and analyzing its effect on the collection efficiency. In this paper, three different size cyclone dust collector models were used. The effect of cyclone size on performance parameters such as cut off diameter, pressure drop, and collection efficiency was investigated. The CFD analysis of three cyclone dust collectors was performed. Collection efficiency obtained from the analysis was then used to select the best model.*

KEYWORDS*: Cyclone dust collector, Collection efficiency, pressure drop, Cut point diameter, Solidworks Flow Simulation*

Date of Submission: 18-12-2018 Date of acceptance: 31-12-2018

I. INTRODUCTION

Cyclone dust collector belongs to the type of centrifugal separator. It is a stationary mechanical device that utilizes centrifugal force to separate solid or liquid particles from a carrier gas. It is a method of removing particulates from gas or liquid stream without the use of filter through vortex separation. The centrifugal force created by the circular flow throws the dust particles towards the wall of the cyclone. As the particle reaches stagnant boundary layer at the wall, they leave the flowing gas stream and presumably slide down the wall, although some particles may be re-entrained as they bounce off the wall back into the gas stream. As the gas loses energy in the swirling vortex, it starts spinning inside the vortex and exit at the top [2]. Cyclones are designed to be effective for PM less than or equal to 10μm and less than or equal to 2.5μm (PM_{10} and $PM_{2.5}$). Cyclone dust collector advantages are it has low capital investment and maintenance costs, can be used under extreme processing conditions, high pressure and temperature. But still, cyclone have limitation such as unable to handle sticky or tacky material and relatively low PM collection efficiencies, particularly PM less than 10μm in size [1]. Cyclones are used as pre-cleaners to reduce dust loading and remove larger, abrasive particles.

The main performance characteristics of cyclone separators are collection efficiency, cut point diameter and pressure drop. Considering the complex hydrodynamics of cyclones, the mathematical approach is required to study and predict the flow behaviour and various geometrical design of cyclone. However, Computational Fluid Dynamics (CFD) is a promising tool to study cyclone size effect on performance parameters [3]. The main objective of this study was to understand the effect of cyclone size on performance parameters and flow field behaviour.

From agriculture industry, 2D2D (Shepherd and Lapple, 1939) and 1D3D (Parnell and Davis, 1979) are commonly used to control particulate matter. Simpson and Parnell (1995) introduced a new low-pressure cyclone, called 1D2D cyclone [6]. The previous research represents mathematical calculation to calculate Cut point diameter, pressure drop and collection efficiency to predict cyclone performance and hydrodynamics. However, CFD representation will help identify the weakness and strength of each model configuration.

II. BASIC STRUCTURE OF CYCLONIC DUST COLLECTOR

Cyclone Dust Collector consists of five components namely, Inlet Duct, Cylindrical body known as a barrel, Cone, Dust outlet and clean gas outlet. Cyclone collectors use centrifugal action to separate dust particles from the gas stream. Dust gas enters the barrel from inlet tangentially forcing the flow into a spiral movement. The centrifugal force created by the circular flow throws the dust particles toward the wall of the cyclone. After striking the wall, particles lose momentum and fall into a hopper located underneath.

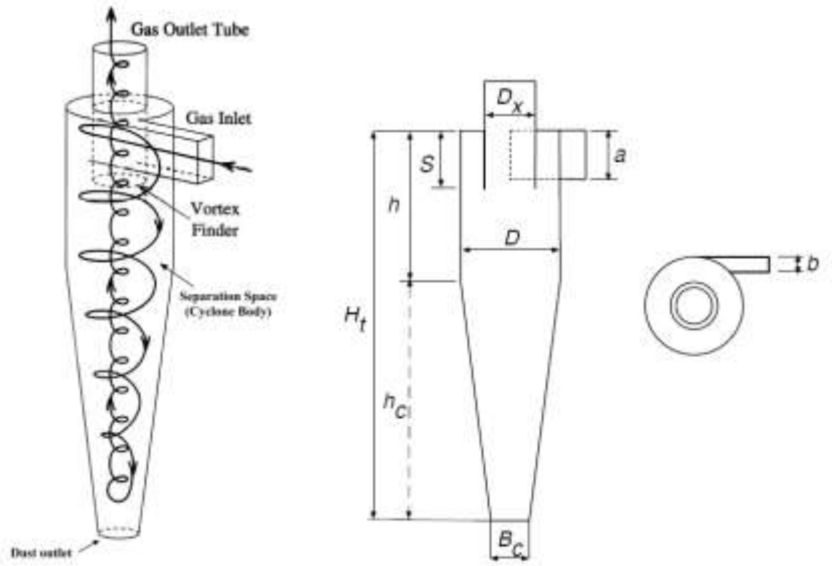

Figure 1: Tangential Inlet Cyclone Dust Collector. Geometrical Notation is indicated in right sketch.

III. MATHEMATICAL MODEL

Continuity Equation for mean motion is [3]:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

For steady state and incompressible fluid flow in cyclones, the Reynolds-averaged Navier-Strokes equations can be expressed as:

$$\rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

Reynolds stress tensor represents the effects of the turbulent fluctuation on the fluid flow and is represented as:

$$\tau_{ij} = \rho_g \overline{u_i u_j} \tag{3}$$

Nomenclature:
Greek letters
μ = Dynamic Viscosity
ρ = Density of Fluid
P = Pressure
u = Fluid velocity

IV. GEOMETRY

The actual models of cyclone separator were designed as a CAD model in Solidworks software as per dimensions provided in table 1.

Table 1: Dimension of Cyclones

Cyclone		1D3D	2D2D	1D2D
Inlet Height	a	0.100m	0.100m	0.100m
Inlet Width	b	0.05m	0.05m	0.05m
Gas Exit Diameter	D_X	0.100m	0.100m	0.125m
Body Length	h	0.200m	0.400m	0.200m
Cone Length	L_c	0.600m	0.400m	0.400m
Vortex Finder	S	0.025m	0.025m	0.125m
Dust Outlet diameter	B_C	0.05m	0.05m	0.100m

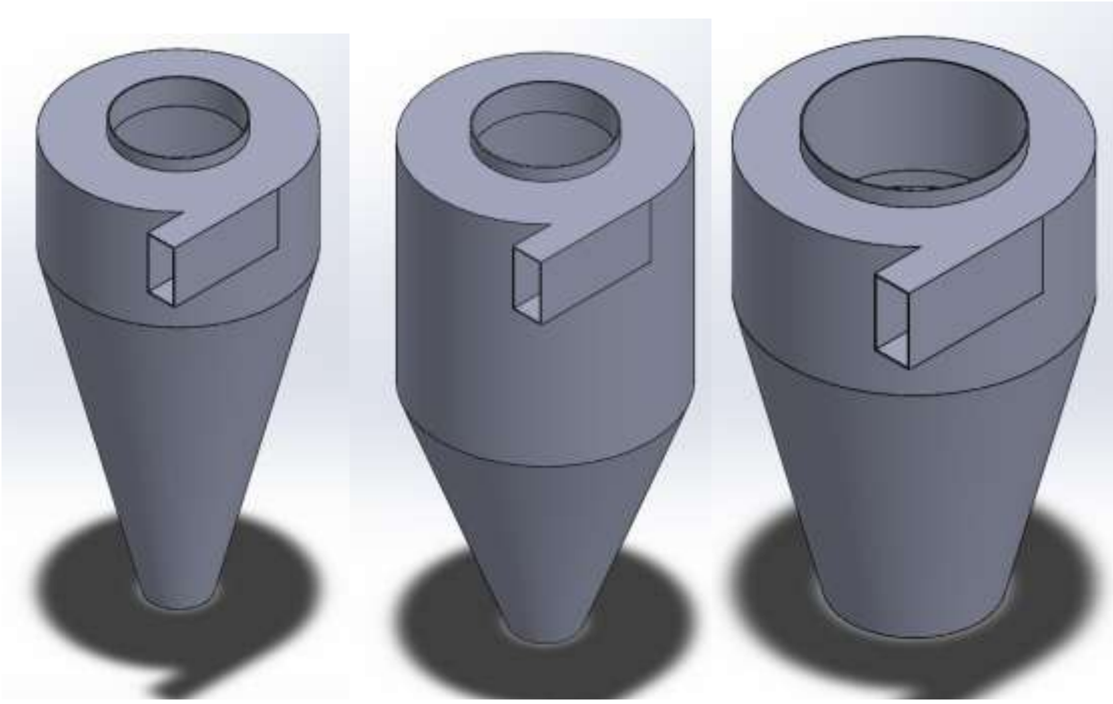

Figure 2: CAD model of cyclone 1D3D, cyclone 2D2D and cyclone 1D2D

V. FLOW SIMULATION BOUNDARY CONDITIONS

First models were assigned domain namely inlet and outlet 1 and 2. The inlet duct is set with Inlet conditions and dust outlet and Gas exit is set as an outlet. For simulation fluid is air. The boundary condition for Inlet duct is set with $\dot{Q}$Inlet Volume Flow rate of 0.0373 (m^3/s). Air flow was set laminar. For Dust outlet and gas exit outlet is set with environmental pressure condition with thermodynamic parameters of temperature set at 273.2k at pressure of 101325Pa.All the three models experienced same boundary conditions during simulation. First, system run the flow analysis with contour plot. For further understanding, flow trajectories were obtained. All the three cyclones were compared

Table 2: Boundary Conditions

Boundary	Boundary Conditions	
	Category	Values
Inlet	Velocity Inlet	$\dot{Q} = 0.0373m^3/s$
Gas Exit Outlet	Static Pressure Opening	P=101325 Pa; T=293.2 K
Dust Outlet	Static Pressure Opening	P=101325 Pa; T=293.2 K
Wall	Wall	No Slip

VI. DISCUSSION

Maximum Tangential Velocity increases with decreasing vortex finder diameter. The high of vortex finder does not have significant effect on tangential velocity. With increasing cyclone inlet width and height Maximum tangential velocity decreases. This also leads to decreasing pressure drop with increasing cut-off diameter. Inlet width have significant effect on cut-off diameter as compared to inlet height. Maximum tangential velocity decreases with increasing cyclone height (Barrel and Cone). Both cut-off diameter and pressure drop decreasing with increasing cone height. The effect of barrel height is less significant on performance of flow pattern as compared to changing cone height.

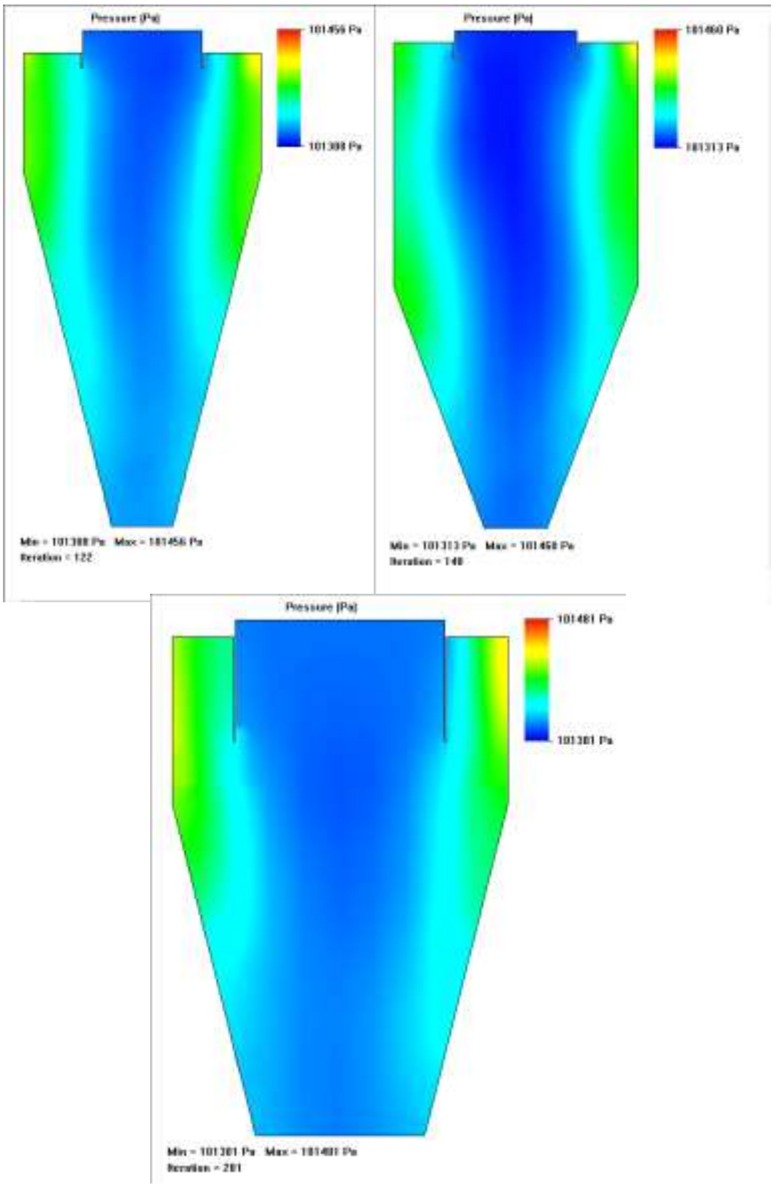

Figure 3: Pressure representation of cyclones 1D3D, 2D2D AND 1D2D

Velocity magnitude in the inner flow along the cyclone centreline increases from the bottom to top. As one can see in the figure 3, the maximum pressure variation domain belongs to cyclone 1D2D, confirming the fact that, the pressure drops in cyclones varies directly with cyclone size. For all the cases low-pressure zone is at the entrance of the exit tube, in the centre of cyclone.

Since the flow is highly swirling, the tangential component of velocity is more important than other components. Velocity vector is approximately tangential in this region, and its magnitude is increasing due to the rotational motion of fluid compared to the cyclone inlet velocity, for all the cases.

From three cyclone comparison, cyclone 1D3D is high efficiency cyclone. Cyclone 2D2D acts as standard cyclone and cyclone 1D2D is high throughput cyclone. Fine particle can be filter by 1D3D cyclone where as medium size particle can be filter by 2D2D cyclone and heavy and large size particle can be filter by 1D2D cyclone.

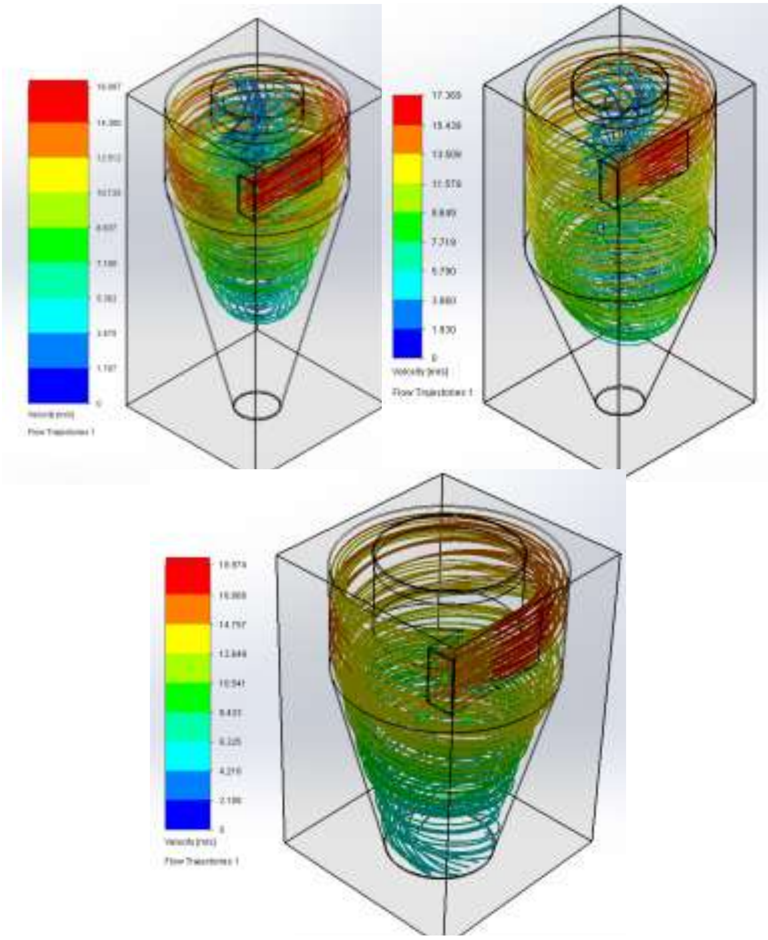

Figure 4: Velocity representation of cyclones 1D3D, 2D2D and 1D2D

VII. CONCLUSION

The effect of cyclone size was numerically investigated to study the performance parameters and hydrodynamic characteristics of the flow. The developed CFD model represented the vortex and flow trajectories of fluid. The simulation results revealed that with increasing the cyclone size the cyclone cut off diameter and pressure drop will increase. On the other hand, the higher inlet velocity leads to the lower cut-off size at a certain operating condition. Moreover, the modelling results proved that CFD model in Solidworks software is a very promising tool to understand the flow behaviour and the effect of cyclone size on its performance parameters.

REFERENCES

[1]. (n.d.). Air Pollution Control Technology Fact Sheet. Environmental Protection Agency United States.
[2]. Bashir, K. (2015). Design and Fabrication of Cyclone Separator.
[3]. Hoekstra, A. J. (2000). Gas Flow Field And Collection Efficiency Of Cyclone Separators.
[4]. Karl B. Schnelle, C. A. (2002). Cyclone Design. In Air Pollution Control Technology Handbook. CRC Press LLC.
[5]. Mahdi Azadi, M. A. (2010). A CFD study of the Effect of Cyclone size on its performance parameters. Journal of Hazardous Materials.
[6]. Prachi K. Ithape, S. B. (2017). Effect of Geometric Parameters on the Performance of Cyclone Separator using CFD. International Journal of Current Engineering And Technology.
[7]. Wang, Lingjuan. Theoretical study of cyclone design. Diss. Texas A&M University, 2005.

Rahul Panchal. "Design and Development of Tangential Cyclone Dust Collector." International Journal Of Engineering Research And Development , vol. 14, no. 09, 2018, pp 53-57