International Journal of Science and Research (IJSR)
ISSN: 2319-7064
Index Copernicus Value (2016): 79.57 | Impact Factor (2017): 7.296

A CFD Analysis of Cyclone Separator

Mayank Chetiwal[1], Dr Bharat Jhamnani[2]

[1]Master of Technology Scholar, Department of Environmental Engineering, Delhi Technological University, Shahbad Daulatpur, Main Bawana Road, Delhi, 110042, India

[2]Assistant Professor, Department of Environmental Engineering, Delhi Technological University, Shahbad Daulatpur, Main Bawana Road, Delhi, 110042, India

Abstract: ***Cyclone Separator is a type of air quality control device which uses physical phenomenon of centrifugal acceleration to separate particulate matter. The efficiency of cyclone separator to eliminate particulate matter is average as compared to other devices and techniques but its efficiency can be increased by altering physical dimensions. Pressure drop and different velocities generated through cyclone body is the most important parameter to predict its performance. Solving the flow and particle transport equations using CFD (Computational Fluid Dynamics) approach is one of the methods that can be followed with specific sets of equations and model. The present study is carried out to analyse the performance of three models of cyclone separator with different vortex finder diameter using RSM (Reynolds stress model) methodology to predict turbulent flow behaviour and DPM (Discrete phase model) to trace the particles trajectories through each cyclone model. The equations are solved computationally using the CFD based software FLUENT (v18.2). The case in the study is computed using an assumption of one-way coupling in the cyclone separator. The graphs and contours of velocities and pressure drop are analysed and compared. Collection efficiency is determined by injecting a fixed number of particles from inlet and counting the trapped particles***.

Keywords: Cyclone Separator, vortex finder, Computational Fluid Dynamics (CFD), Reynolds Stress Model (RSM), Discrete Phase Model (DPM), Collection efficiency, Pressure drop

1. Introduction

1.1 Cyclone Separator

Cyclone Separatoris one of the most widely used air pollution control technology and also known as pre-cleaners. They commonly remove large size-range of particulate matter.It has no moving parts and works on the principle of centrifugal separation to eliminate dust particles from the polluted gaseous stream or liquid stream.It is quite efficiently provides a better method of removing particulate matter from the particle laden stream at low cost and maintenance. There are various types of cyclone used in the industries but reverse flow cyclone with tangential entry is the most widely used type of cyclone separator. High efficiency can be achieved by proper designing the physical parameters of Cyclone Separator.

1.2 Working principal and governing parameters

The cyclone separators work on a simple principle of centrifugal-separation.It's quite popular device for separating particles because of its simple working and efficient collection rate. Cyclone Separator can be considered as a special type of settling chamber for discrete particles with strong centrifugal force acting on them instead of gravitational force.Cyclone performance is usually conveyed in terms of the overall pressure drop and collection efficiency through the separator.It comprises of an upper cylindrical and hollow part known as barrel and a lower conical part signified to as cone which helps in the formation of vortex in the cyclone. They basically change the inertial force of gas particle to a centrifugal force by means of a vortex formation in the cyclone body. The contaminated gas stream with particles enters extraneously from inlet which is situated at the highest point of the cylindrical barrel. In the wake of entering, it descends into the conical area in spiral pattern forming an outer vortex. As the air velocity increases in the outer vortex due to geometry transition, a centrifugal force on the particles which separates them from the air stream starts acting on them. When the air finally reaches at the bottom of the conical part, it begins to flow radially inwards and out the top as clean air. The pollutants fall into the **dust collector** chamber attached to the bottom of the cyclone.

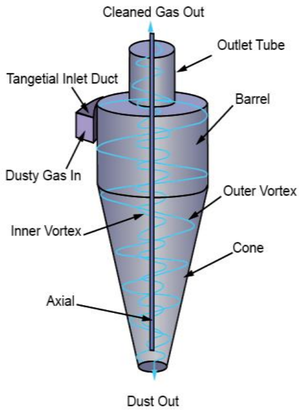


Figure 1: A schematic diagram of reverse flow cyclone separator

The flow in a cyclone separator has been studied by many researchers but there is no standard model or mathematical equation that can give the exact or pre concluded results. A number of studies and researches are already done for the optimization of the cyclone separator. However, the complexity of the flow inside a cyclone can be predicted by a new set of mathematical equations known as Computational Fluid Dynamics which is reliable and can be solved with more ease as compare to the conventional methods. There is a widespread literature on the effect of cyclone geometry on

performance, using one or more of the four main approaches of study B. Zhao.et al [7], which are:
1) Mathematical models, which can be classified into:
- Theoretical and semi-empirical models
- Statistical models

2) Experimental measurements
3) Computational fluid dynamics (CFD) simulations
4) Artificial neural networks (ANN) approach

Few of the names are Shepherd and Lapple **[1],** Alexander [**2**]**,** First [**3**]**,** Stairmand **[4],** Barth [**5**]**,** Avci and Karagoz **[6],** Zhao [**7**]**.** These models were derived from physical descriptions and mathematical equations which was a complex procedure to follow.Predictions by some models are twice of the experimental values at times as shown by P. K Swamee et al [**8**]**.**The most common mathematical models for the cut-off diameter and pressure drop calculations were developed by Stairmand model [**4**] and Shepherd and Lapple model [**1**]**.**

Dirgo and Leith **[9]** measured the collection efficiency and pressure drop experimentally for the Stairmand high efficiency cyclone [**4**] only at variable rates of flow. Hoekstra et al. [**10**] measured the mean and fluctuating velocity components for gas cyclones with different geometric swirl numbers by means of the laser doppler anemometry technique. Boysan et al. [**12**] presented the first CFD investigation in the field of cyclone separators. From that time, the CFD technique becomes a widely used approach for the flow simulation and performance estimation for cyclone separators. For example, Griffiths and Boysan computationally investigated three cyclone samplers. Ravi et al. [**12**] carried out a multi-objective optimizationof a set of N identical reverse-flow cyclone separators in parallel byusing the non-dominated sorting genetic algorithm (NSGA). They reported that the CFD predicted pressure drops are in excellent agreement with the measured data. TheCFD modeling approach is also able to predict the features of the cycloneflow field in great details, which providing a better understanding of thefluid dynamics in cyclone separators. K. Elsayed **[13-14]** has also done a significant work on optimizing the cyclone separator geometry using CFD.

The newest significant CFD application on cyclone separator was shown by El-Batshet al. [**15**] from which my work is influenced the most. Consequently, CFD approach isa reliable and relatively inexpensive method of examining the effects of anumber of design changes. Moreover, this makes the CFD methods representa cost-effective route for geometry optimization in comparison withthe experimental approach.

The objectives of my study will be following:
1) Construction of standard model by referring the **Standard Stairmand design [4].**
2) Validation of the CFD model created in the present work by comparing the results with that of published literature of **Hoekstra [10].**
3) To study the various parameters like pressure drop and velocity profiles with variable diameters of vortex finder.
4) Calculation and comparison of collection efficiencies for particulate matter of different geometries of cyclone.

2. Problem Descriptions

Standard Stairmand design [4] is the standard cyclone model which was utilized as a part of this examination with comparative cyclone geometrical parameters. Three cyclone models were generated in this study using different diameters of the vortex finder which is given in the Table 1. These computational cyclone separator models have different dia. of the vortex finder i.e., D_x= 0.4D, 0.5D and 0.6D. CFD terminology was utilized to forecast the stream pattern field and total drop in pressure. The tangential & axial velocities are calculated from the flow simulation for the standard cyclone design by Stairmand and then differentiated with the literature published by Hoekstra. The results obtained from the standard Stairmand design were then subjected to the optimization.Table 1 gives an overview of the dimensional parameters of the models used in this study.

Table 1: Dimensional parameters of the cyclone models

Part Name	*Dimensions*
Cyclone diameter, D	290 mm
Inlet height, a	0.5 D
Inlet width, b	0.2 D
Cone-tip diameter, Dc	0.37 D
Exit pipe diameter, Dx	0.6 D, 0.5 D(**Standard**), 0.4 D
Cylindrical part height, h	1.5 D
Cone part height, hc	2.5 D
Length of inlet section, Li	1.38 D
Length of cylindrical barrel top, Le	D
Exit pipe length, S	0.5 D

In this study, a standard cyclone geometry based on Stairmand design is first designed in the computer and simulated in Ansys Fluent 18.2. The obtained data for standard cyclone is then compared with the work of Hosketra to validate the model. A grid is generated of specific cells and nodes for the computation. The description about the grid is given further in this further.In this study of cyclone separator, only the effect of different vortex finder diameters are taken into account so all the other dimensions will remain constant. Only vortex finder diameter i.e., D_x will vary in all the designs. The objective will be to study the effects on the pressure drop, velocity profiles and collection efficiency of particulate matter with rendering the D_x. For all the evaluation and calculation, a reference section is considered at -0.25 D from the origin to compare the results obtained as shown in Figure 2.

International Journal of Science and Research (IJSR)
ISSN: 2319-7064
Index Copernicus Value (2016): 79.57 | Impact Factor (2017): 7.296

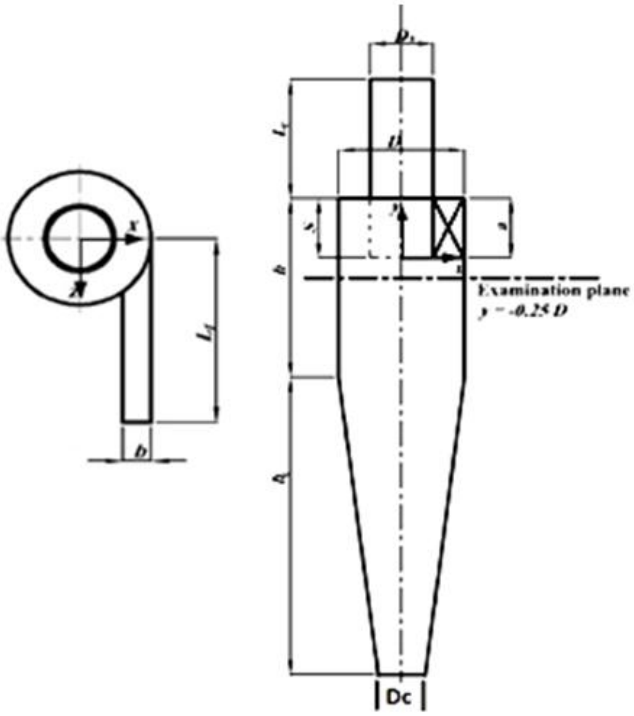


Figure 2: Schematic diagram of Standard Cyclone Geometry. Front and top view

2.2 Grid generation and dependency study

a) Adopted grid: A computational grid is constructed using 3-D blocks unstructured method. The grid or mesh is generated using meshing module of **Ansys Workbench** called **ICEM**. Due to the complicated geometry of cyclones, **Hexahedral meshing** is employed which is a collection of cubes. Hex meshing is the most complex meshing to create numerical grid blocks but on the other hand it is the most accurate for complicated geometries like cyclone separator. Table 2 shows the details of grid or mesh used for the analysis.

Table 2: Details of the grid used for the standard Stairmand cyclone model

Element	Number
Nodes	**579379**
Quads	**28846**
Hexas	**565404**

b) Grid dependency study: The response of the numerical simulation to the grid sizing and elements are tested for standard Stairmand geometry. Two types of computational meshes are used with elements around 580000 and around 370000 cells. The obtained pressure drops using these grids are then tested against different velocities, as shown in Table 3. The maximum difference between the investigated results obtained by using coarse and the fine meshing is less than ±0.4% which is clearly proved by the table. This is why; the finer grid with 580000 cells is the assigned grid which is used in all calculations of standard geometry. This grid dependency study is carried out to be sure that the obtained results are grid independent. The hexahedral computational grids were generated using Ansys ICEM 18.2 mesh generator and the simulations is carried out in Ansys Fluent 18.2

Table 3: Grid dependency results

Inlet velocity (m/s)	Pressure drop (Pa)		Error (%)
	580000 cells (Finer Grid)	370000 cells (Coarse Grid)	
15	542.66	541.82	0.155 %
20	970.5	969.75	0.103 %
25	1535.74	1537.63	-0.123 %
30	2205.74	2214.33	-0.389 %

Figure 4 shows the final geometries which were used in this study. Similarly, grid for two other models were developed also using same steps and process. Figure 3 shows the hex meshing which was used as a grid for the cyclone models.

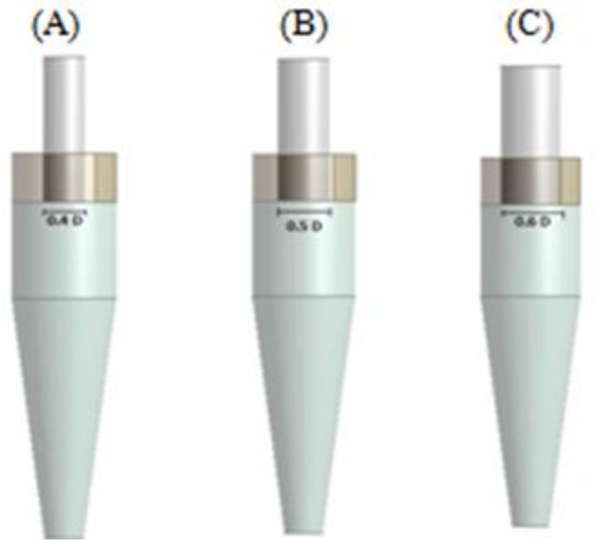


Figure 4: Picture of three cyclonemodels which are tested with different vortex finder diameter

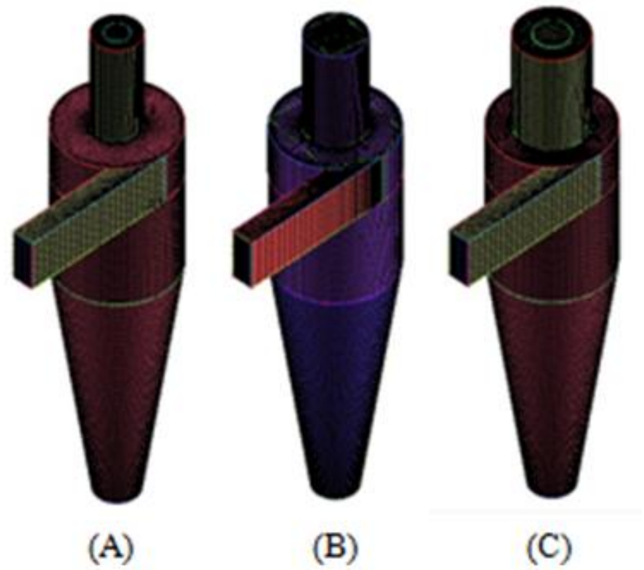


Figure 5: Picture explaining the detailed grid generated for all the three tested cyclone models

3. Methodology

Numerical calculations are solved for the gas and solid interactive flow using **CFD approach**. The flow is assumed is assumed to be steady, turbulent, incompressible and isothermal. The gas flow fields are attained by solving together the continuity and the momentum equations. The **Reynolds Stress Model** (RSM) is used to represent the turbulent flow and the **Discrete Phase Model** (DPM) is used to forecast particle trajectories and collection efficiency. The particle equations of motion are solved to track the particles trajectories in the flow field and also the relevant forces acting on each particle are taking into account. The 3-D gas and particle interactive flow in cyclone separators is defined by the assumption that the particulate phase is in dilute state and particle loading rate is low as compared to the air flow. That is why, the interaction between particles and flow is kind of negligible. Plus, the particles do not distress the gas flow field through the flow. So the case in the study is computed using an assumption of **one-way coupling** in the cyclone separator. The contributing terms in the Navier-Stokes equations due to gas-solid momentum exchange are neglected due to this assumption.

3.1 Transport equations

The 3-D mass conservation time-averaged equations for turbulent and incompressible behaviour flow and neglected the other forces developed by body:

$$\frac{\partial \bar{u}_i}{\partial x_j} = 0,$$
$$\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\right] + \frac{\partial \tau_{ij}}{\partial x_j}, \quad \text{...... (1)}$$

Where $\bar{u}_i$ and $\bar{u}_j$ the mean velocitiesin i and j direction, respectively. x_j is the position, mean pressure is P , τ_{ij} is called Reynolds stresses and it is given by:

$$\tau_{ij} = -\rho u_1' u' \quad \text{.........(2)}$$

The CFD numerical prediction's success rate intensely hinge on the accurate description of turbulent behaviour. Recentstudies on numerical models of cyclones separators have shown that selection of turbulence model has substantial effect on the flow field pattern in cyclone separators. Studies have also suggested that the Reynolds Stress Model (RSM) can be responsible for the high unsteady swirl in cyclone separators.

RSM solves the transportation equation for each term of Reynolds stress tensor without the necessity of isotropic turbulent viscosity field. For steady and incompressible flows, RSM is given by the following equation:

$$u_k \frac{\partial}{\partial x} k(\overline{u_1' u'}) = Dij + Pij + \phi ij - \grave{o} ij \quad \text{.......... (3)}$$

D_{ij} acknowledges the diffusive transport term which is as follows:

$$D_{ij} = -\frac{\partial}{\partial x_k}\left(\frac{\mu_t}{\sigma_k}\frac{\partial \overline{u_1' u'}}{\partial x_k}\right). \quad \text{.......... (4)}$$

P_{ij},represents the stress is calculated as:

$$P_{ij} = -\rho\left(\overline{u_i' u_k'}\frac{\partial u_j}{\partial x_k} + \overline{u' u_k'}\frac{\partial u_i}{\partial x_k}\right). \quad \text{........ (5)}$$

ϕ_{ij} is the pressure strain

$$\phi_{ij} = \overline{P\left(\frac{\partial u_i'}{\partial x} + \frac{\partial u'}{\partial x_i}\right)}. \quad \text{........ (6)}$$

The dissipative ϵ_{ij}

$$\grave{o}_{ij} = 2\mu \overline{\frac{\partial u_i'}{\partial x_k}\frac{\partial u'}{\partial x_k}}. \quad \text{........ (7)}$$

3.2 Discrete Phase Model (DPM)

The discrete phase model is employedto calculate particle trajectories in the flow and to track every particle through the volume of fluid. It is based on the Lagrangian approach. In Cartesian coordinates, the equation of motion for any particle in the control volume is as follows:

$$\frac{d\vec{U}_p}{dt} = \frac{18\mu}{\rho_p d_p^2}\frac{C_D \operatorname{Re}_p}{24}(\vec{U} - \vec{U}_p) + \frac{\vec{g}(\rho_p - \rho)}{\rho_p} \quad \text{.......... (8)}$$

Here, the fluid and particle velocity vectors are $\bar{\mathbf{U}}_\mathbf{p}$ and $\bar{\mathbf{U}}$, respectively. Fluid and particle densities are represented by $\boldsymbol{\rho}$ and $\boldsymbol{\rho}_\mathbf{p}$ and **dp** is the particle diameter,C_D represents drag coefficient and the acceleration of gravity vector is **g**. $\mathbf{R_{ep}}$ is the particle relative Reynolds number which can be definedas:

$$\operatorname{Re}_p = \frac{\rho d_p |\vec{U}_p - \vec{U}|}{\mu} \quad \text{..............(9)}$$

At the point when a particle contacts the wall of the cyclone, it bounce back and loses some quantity of its kinetic energy. The proportion of particle rebound velocity and particle impact velocity is known as the coefficient of restitution. The estimations in this examination are performed utilizing a diverse coefficients of restitution 'e' in the scope of 0.6 to 1 which demonstrated that collection efficiency has not an indispensable association with it. In this way, the computations are done in this investigation by taking a presumption of elastic collisions of particles and wall with coefficient of restitution 'e' being 1 through. Stochastic method is utilized to represent to the impact of turbulent collision on the particle trajectory. In this technique the instantaneous stream velocity is considered as the entirety of the normal velocity and the fluctuating velocity which was figured from the stream turbulent kinetic energy

4. Results and Discussions

The present study is intended to computationally solve the flow equations through the cyclone to study the effect of increasing the exit pipe diameter $\mathbf{D_x}$ on the pressure drop and collection efficiency for particulate matter and to gain more details about the flow pattern and velocity profiles using the Reynolds Stress Model (RSM) methodology.

4.1 Model validation

The flow field is computed by comparing the results obtained for velocity profiles in the cyclone with the experimental results obtained by of Hoekstra [] for the standard Stairmand design using LDA technique. Theprofiles of tangential and axial velocitiesare examined at an axial position of**y = 0.25 D** below the vortex finder. The origin of the model is where the cover plate of vortex finder starts as displayed in Figure3.1 in the previous chapter. The position of y= 0.25D is said to be the examination plane for this study. The flow is analyzed for incoming velocity of **U_o=15** m/s corresponding to Reynolds number of cyclone Re = 2.8 x 10^5.The comparison ofnumerical obtained results in the present study and the experimental results of the published literature for axialand tangential velocity profiles are shown in figure 4.1 and figure 4.2. The velocities are shown as dimensionless using the ratio of instantaneous velocity at a point to incoming velocity **U/U_o**.

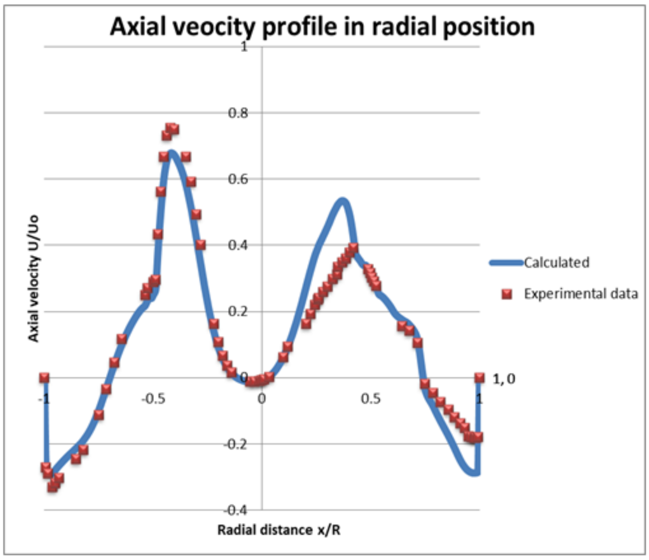


Figure 6: Graph showing variation of dimensionless axial velocity vs. radial position for calculated results and experimental data of *Hoekstra* at axial position of y = -0.25D

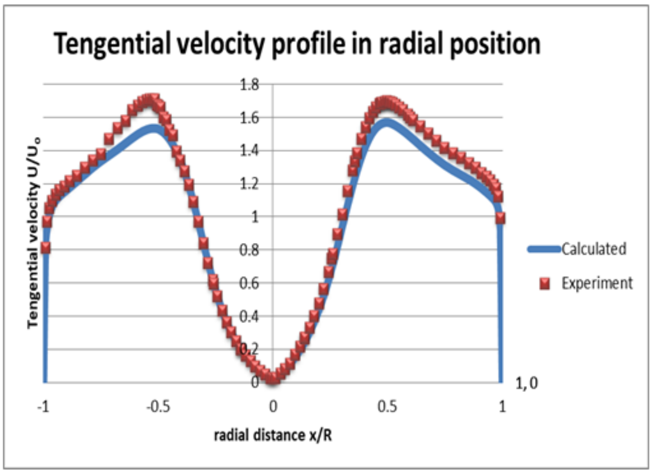


Figure 7: Graph showing tangential velocity profile with comparison of calculated and experimental data at y=0.25 D

4.2 Axial Velocity

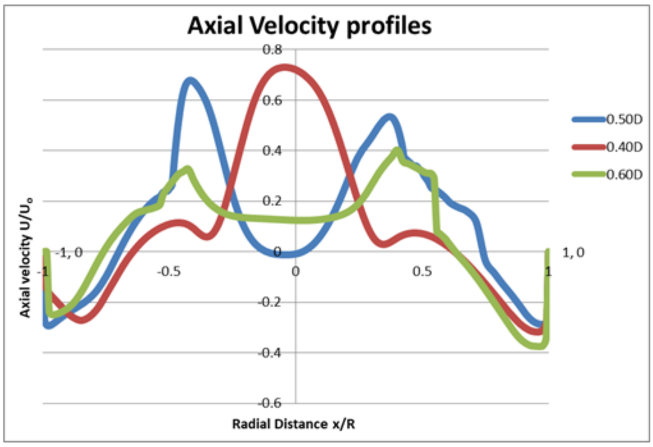


Figure 8: Graph showing variation of axial velocity profile comparison of all the three models together in radial direction at y= -0.25D

From Figure 8, the evaluation of dimensionless axial velocity profiles through cyclone model can be done. The variation profile is given at the axial position of y =0.25D and for the inlet velocity of 15 m/s. The profile shows inconsistencies from the axisymmetric profiles. This is due to the close location observation plane from the inlet.

Figure 8 indicates that with decreasing of vortex finder diameter, an increase of upward axial velocity occurs in the cyclone.The axial velocity profile changes from the inverted W to the inverted V profile as the vortex diameter decreases. This phenomenon occurs in the cyclone for adjusting the decrease in diameter of vortex finder. The graph also indicates that for the larger vortex finder diameter, two peaks are obtained in the axial profile. The spike in the velocity increases with decreasing the diameter of vortex finder. Also, thegap in between these high peak values diminish and the maximum velocities unites together into one large peak eventually for lesser vortex finder diameter that can be seen in D_x = 0.4D.From observing the contours of axial velocity, it is clear that with the increase of vortex finder diameter, the axial velocity decreases thorough the cyclone. However, with the decrease in the diameter the axial velocity increases.

4.3 Tangential Velocity

The vortex finder diameter is inversely proportional to the maximum tangential velocity attained in the cyclone. Tangential velocity is always zero at the center of the cyclone and increases progressively as moving away from the mid plane. The tangential velocity graphs using different diameters ofvortex finder can be seen in figure 4.5. This velocity is taken at the axial position of y =0.25D. The Inlet velocity is same as for obtaining axial velocity variation i.e., U_o = 15 m/s^2. The graph shows that high value of maximum tangential velocity is gained using smaller diameter. The dimensionless tangential velocity contours is shown in figure 4.6 which also indicate that by decreasing vortex finder diameter, an increases in tangential velocity occurs. Whereas, larger the diameter of vortex finder, lesser is maximum the tangential velocity attained in the cyclone. It can give a clearer view as far as the efficiency is concerned. 0.4 D geometry attains maximum tangential velocity so it is more efficient in terms of swirling speed.

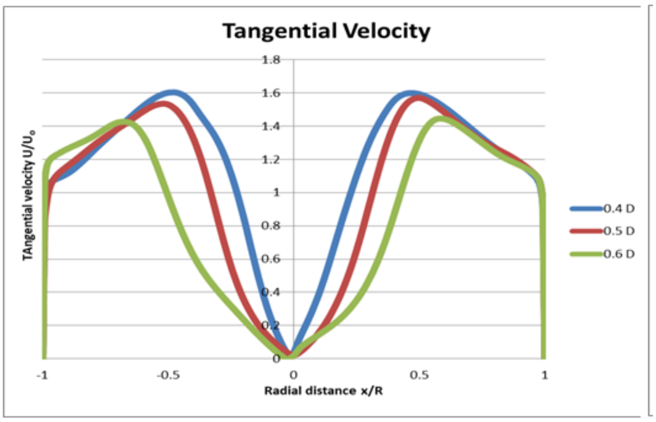


Figure 9: Graph showing variation of tangential velocity profile comparison of all the three models together in radial direction at y= -0.25D

4.4 Pressure Drop

Pressure drop across the cyclone plays a significant role in its overall performance. The total pressure drop in a cyclone is due to:

- The entry and exit loss
- Friction loss
- Kinetic energy loss

In this study, CFD methodology is used to compute the pressure drop. Two types of pressure is measured through the cyclone:

1) Static Pressure
2) Total pressure drop

Both the pressure parameters are evaluated for all the three models. Graphs and contours are obtained and compared to study the effect of vortex finder diameter on pressure drop.

a) Static Pressure: Figure 4.7 shows a graph of static pressure variation at axial position of y= 0.25 D for all the three geometries. It can be easily observed that the geometry with vortex finder diameter of 0.4 D shows the highest value of static pressure drop with 736.003 Pa near the cyclone wall whereas the geometry with vortex finder diameter of 0.6 D shows the least pressure drop near the wall with the value of 389.743 Pa. Also the 0.4 D geometry also has the highest negative static pressure at the centre with the value of -154.989 Pa. The geometry of 0.5 D and 0.6 D shows almost equal negative static pressure value of -33.50 Pa and -33.42 Pa respectively.

This clarifies the fact that with increasing the vortex finder diameter, a sharp decrease in the pressure drop occurs. On the other hand, with decreasing its diameter, pressure drop increases significantly.The energy loss in the vortex finder tube is the main contributor to the overall pressure drop in the cyclone, which mainly depends on the maximum tangential velocity in the cyclone. It can be observed in figure 4.5 and 4.6, the maximum tangential velocity decreases with increasing the vortex finder diameter.

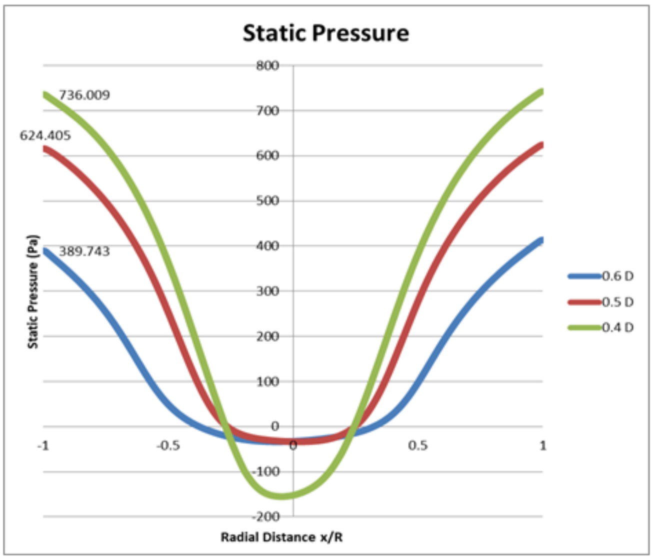


Figure 10: Graph of static pressure for all the three geometries at y=0.25D

b) Total Pressure: Total pressure gives more vivid scenario about the pressure filed changes through the cyclone. Total pressure accommodates all type of losses occurs in the cyclone. Figure 11 shows the graph of total pressure variation across the radial distance of the cyclone. The profile is generated at an axial position of y = 0.25 D with the incoming standard speed of U_o = 15 m/s. The Total pressure variation for all the tested models are shown in the graph.

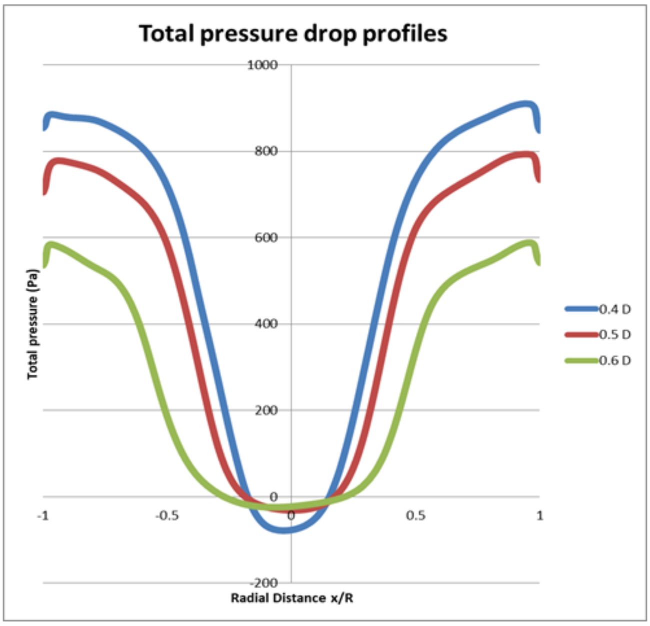


Figure 11: Graph showing Total pressure variation of the tested geometries at axial position of y= 0.25 D

It can be stated from the above graph that the total pressure drop trough the cyclone increases with decreasing the vortex finder diameter and vice versa. The profiles are in similar trend with the static pressure.

4.5 Collection efficiency

The main aim of this work is to examine the behavior of cyclone separator on particulate matter separation from the incoming particle laden gas. Cyclone separator is basically used as a pre-cleaner and less efficient with small size of particles. To examine the collection efficiency of cyclone separator towards particulate matter is a complex procedure and many hit and trial methods were conducted to study the efficiency more precisely. To simulate and track the particles through the cyclone, **discrete phase model (DPM)** is used along with RSM model. Coupling isthe phenomenon of exchange ofmass, momentum and energy between the phases. The particles have negligibleeffects on turbulence as the values ofdispersed-phase volume fraction less than 10^{-6}. This is termed as one-way coupling.The volume fraction of particulate matter we are dealing with in the present work is much less than 10^{-6} and hence one-way couplingis assumed. The collection efficiency mainly depends on 2 factors:

- Pressure drop
- Tangential velocity

More the pressure drop and tangential velocity across the cyclone, more is the collection efficiency. Strong centrifugal and pressure profiles are generated where these two factors dominates. The formula for calculating the collection efficiency in this study is taken as follows:

$$\eta = \frac{\text{Particles trapped} - \text{incomplete particles}}{\text{Total tracked particles}} \qquad (10)$$

With the help of the above formula, collection efficiency of different particles is calculated. The grade efficiency curves are given in figure 12 and 13.

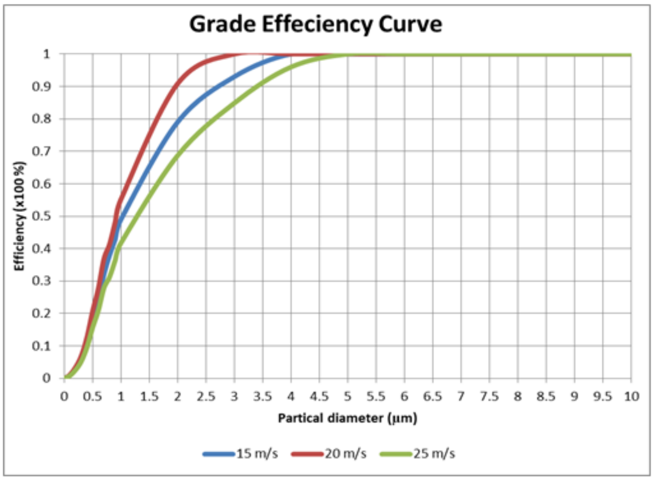


Figure 12: Graph of grade efficiency curve for standard Stairmand design (D_x= 0.5 D) at variable inlet velocities of 15, 20, 25 m/s

The mass flow rate is kept as $2x10^{-10}$ kg/m^{3}from all the tested models. The particles are dispersed uniformly at inlet and inserted with velocityequals to the inlet flow. Figure 12 shows the numerical results of grade efficiency curve for standard Stairmand design at inlet velocities of 15, 20 and 25 m/s and particle density is taken as2750 kg/m^3. The graphshows that with increasing inlet velocity, an increase in the collectionpercentageoccurs. It is caused by the increase in the centrifugal force acting on the particles due to high tangential velocity.

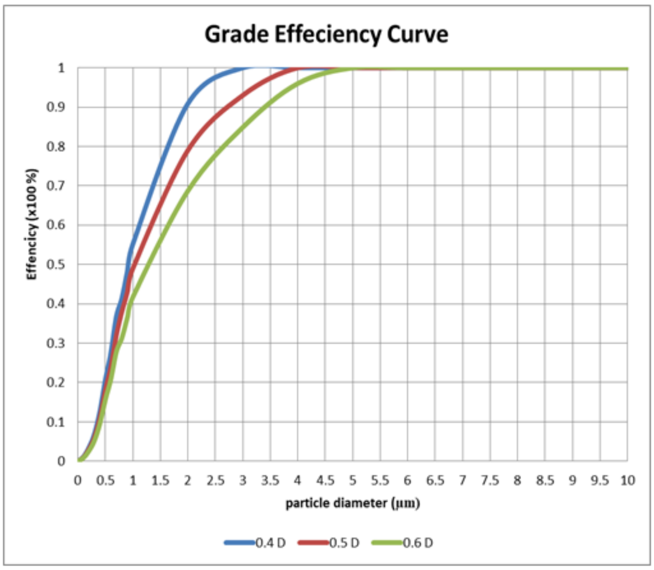
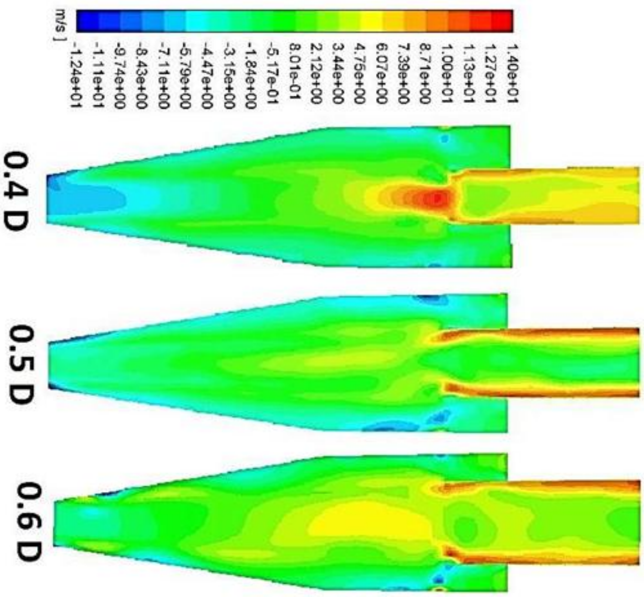


Figure 13: Graph of grade efficiency curve for all the three models (Dx =0.4 D, 0.5 D, 0.6 D)

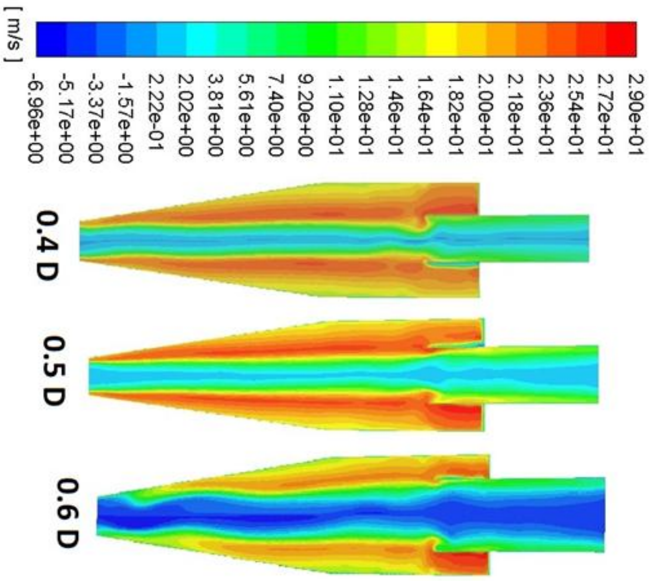
Figure 13 shows the effect of vortex finder diameter on the collection efficiency when the inlet velocity is kept constant at 15 m/s. The curve shows that for a specific particle size, decreasing the vortex finder diameter increases the cyclone collection efficiency. It is due to the fact that the increase in tangential velocity and centrifugal force with decreasing the vortex finder diameter.

4.6 Contours
a) Axial velocity

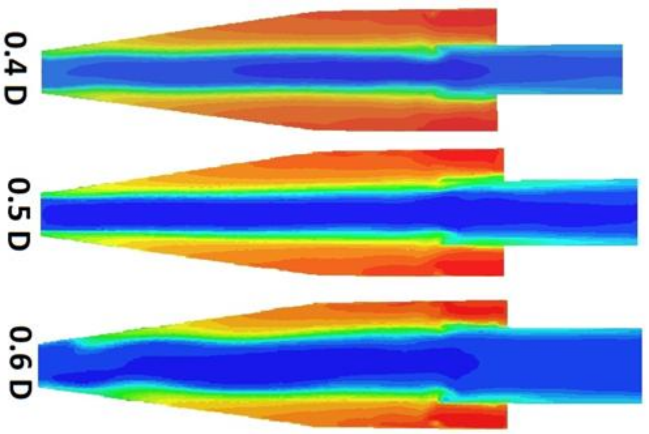


Paper ID: ART2019831 DOI: 10.21275/ART2019831

b) Tangential velocity



c) Total pressure drop



5. Conclusions

A numerical technique based on CFD approach is used in this work to study the flow of air and particles through cyclone. Reynolds Stress Model (RSM) methodology is used to predict the turbulence behaviour of the flow and Discrete Phase Model (DPM) is used to predict the particle trajectories through cyclone separator. All the dimensional parameters of cyclone is kept constant except the diameter of vortex finder. Three cyclone models with different vortex finder diameters have been simulated using RSM methodology to study the effect of vortex finder diameter on the performance and flow pattern of the cyclone separator. The most significant conclusion of this study is the numerical techniques of CFD approach can be used to predict the flow parameters through the cyclone. This fact is clarified by the similarity obtained between the results of present study and published literature. Following conclusions can be drawn from the investigation:

- The maximum tangential velocity in the cyclone increases with decreasing the vortex finder diameter and vice versa.
- Decreasing the vortex finder diameter gradually increases the axial velocity through the cyclone and vice versa. Also, the axial velocity profile changes from the inverted W to the inverted V profile.
- Increasing vortex finder diameter reduces the pressure drop through the cyclone and vice versa.
- Increasing the vortex finder diameter reduces the collection efficiency for particulate matter and vice versa.
- Increasing the inlet velocity increases the collection efficiency of cyclone separator for particulate matter.
- Optimization of cyclone separator model is possible using numerical techniques of CFD approach.

References

[1] C. B. Shepherd and C. E. Lapple. Flow pattern and pressure drop in cyclone dust collectors cyclone without Intel vane. Industrial & Engineering Chemistry, 32(9):1246–1248, September 1940.

[2] R. M. Alexander. Fundamentals of cyclone design and operation. In Proceedings of the Australian Institute of Mineral and Metallurgy, number 152, pages 203–228, 1949.

[3] M. W. First. Cyclone dust collector design.ASME Annual General Meeting, Paper No. 49A127 (1949).

[4] C. J. Stairmand. The design and performance of cyclone separators.Industrial and Engineering Chemistry, 29:356–383, 1951.

[5] B. Zhao. A theoretical approach to pressure drop across cyclone separators. Chemical Engineering Technology, 27:1105–1108, 2004.

[6] B. Zhao and Y. Su. Particle collection theory for cyclone separators: summary and comparison. Particle & Particle Systems Characterization, 23:484–488, 2006.

[7] A. Avci and I. Karagoz. Theoretical investigation of pressure losses in cyclone separators. International Communications in Heat and Mass Transfer, 28(1):107–117, 1 2001.

[8] P. K. Swamee, N. Aggarwal, and K. Bhobhiya. Optimum design of cyclone separator.American Institute of Chemical Engineers (AIChE), 55(9):2279–2283, 2009.

[9] D. L. Iozia and D. Leith. Effect of cyclone dimensions on gas flow pattern and collection efficiency.Aerosol Science and Technology, 10(3):491–500, 1989.

[10] A. J. Hoekstra. Gas flow field and collection efficiency of cyclone separators.PhD thesis, Technical University Delft, 2000.

[11] A.J. Hoekstra, J. J. Derksen, and H. E. A. Van Den Akker. An experimental and numerical study of turbulent swirling flow in gas cyclones. Chemical Engineering Science, 54:2055–2065, 1999.

[12] F. Boysan, W.H. Ayer, and J. A. Swithenbank. Fundamental mathematical-modelling approach to cyclone design.Transaction ofInstitute Chemical Engineers, 60:222–230, 1982.

[13] G. Ravi, Santosh K. Gupta, and M. B. Ray. Multiobjective optimization of cyclone separators using genetic algorithm. Ind. Eng. Chem. Res., 39:4272–4286, 2000.

[14] K. Elsayed and C. Lacor. Modeling, analysis and optimization of aircyclones using artificial neural

network, response surface methodology and CFD simulation approaches. Powder Technology, 212(1):115–133, 2011.
[15] K. Elsayed and C. Lacor. Multi-objective optimization of gas cyclone based on CFD simulation. In ECCOMAS thematic conference, CFD & Optimization, Antalya, Turkey, 23-25 May 2011.
[16] El-Batsh, Hesham. (2013). Improving cyclone performance by proper selection of the exit pipe. Applied Mathematical Modelling. 37. 5286–5303. 10.1016/j.apm.2012.10.044.

Author Profile

Mayank Chetiwal, Master of Technology Scholar, Department of Environmental Engineering, Delhi Technological University

Dr Bharat Jhamnani, Assistant Professor, Department of Environmental Engineering, Delhi Technological University