Simulation and Comparative performance analysis of Modified Cyclone Dust Separator

Suveg Bhade\textsuperscript{1}\textsuperscript{*} and Sriharsha Hegde\textsuperscript{1}

\textsuperscript{1}Department of Aeronautical and Automobile Engineering, MIT, MAHE, Manipal, Karnataka, India

Abstract. The cyclone separator is industrial equipment that is in use for a long time. Because of its industrial importance, a lot of extensive research work has been done. This paper discusses the efficiency of cyclones by considering the pressure loss and collection efficiency (cutoff diameter). The simulation of fluid-dust flow field in a cyclone is carried out using computational studies. The gas flow rate and temperature and geometrical parameters governing the efficiency of cyclone are the focus of this study. In this paper, the influence of the geometry on flow field pattern and performance of the tangential inlet cyclone separators are analyzed using computational work. The current work is carried out in three steps. First phase is dedicated to study the Stairmand’s optimized design to appreciate the pressure and speed fluctuations. This study is followed by examining the cyclone separator by varying the inlet width and height; and the cyclone total height. Finally, mathematical study to determine the efficiency of the dust separator is done.

1. Introduction

Inertial separators, fabric collectors, wet scrubbers, and electrostatic precipitators are the typical industrial dust collectors. The inertial separators separate dust using gas streams by combining centrifugal, gravitational, and inertial forces. The dust particles are pushed to a section where the power exerted by the gas stream is minimal. The separated dust is temporarily stored into a hopper. Types of inertial separators which are settling chambers, baffle chambers, and centrifugal collectors (e.g., cyclone separator) [1]. Cyclone dust separators (CDS) uses the centrifugal force of a spinning gas stream to separate particles from the carrier gas. The design simplicity coupled with economical and almost care-free operation makes them suitable for use as pre-cleaners for expensive final control devices. [2]. Cyclone separator don’t have any moving parts and so it causes very low upkeep costs. Besides, they consume very less energy as separation occurs due to the natural forces action and swirl motion of fluid. Hence, cyclone separator, with its simple design, fluid-only type of detachment, and low cost, becomes an obvious option for experimentation [3]. Due to reliability and low cost, cyclones are widely utilized as gas-solid and fluid-solid separators. Their operation depends on pressure drop and collection efficiency. Studies have indicated that the complexity increases due to confined vertical flow inherent to cyclones. The computational fluid dynamics (CFD) method is widely applied to model flows and solve them numerically. This has shown promising results in dealing with vortex breakdown, reversal flow, and high turbulence intensity [4].

The velocity of the air at input modifies the fan energy consumption which in turn influence the efficiency. The cyclone efficiency is calculated by relation developed by
Leith and Litch (1972) [5]. The influence of the dimensions of the solid particles and the inlet velocity into the cyclone on the collection efficiency can be theoretically demonstrated. If a laminarizer installed at the ingress of the cyclone, the flow field at the cyclone entrance was expected to show a laminar flow regime with the laminarizer. CFD simulation and experimental method was used to investigate the effects of placing the particle distribution at the entrance, partial separation efficiency, laminarizer on the gas flow field, and pressure drop of the cyclone separator [6]. The working principle of a cyclone chamber operator under extreme temperatures are not explored very well. These are analyzed using commercial CFD code FLUENT in which the analysis is carried using a Reynolds stress model (RSM) for turbulence and discrete phase model (DPM) for particle trajectories [7].

2. Computational Approach

2.1. Stairmand Optimized Design

Stairmand developed two types of standard designs for gas-solid CDS; a high-efficiency CDS and a high throughput design. The performance curves for these designs are obtained experimentally under standard conditions. These curves can be utilized for different sized CDS and standard ratios for the different parameters can be obtained [8], which are shown in table 1 [9].

![Figure 1. Cyclone Geometry](image)

**Table 1.** Cyclone Geometry used in this simulation (Stairmand’s Optimized Design)

| Geometry                  | a/D | b/D | D_x/D | S/D | h/D | H/D | B/D |
|---------------------------|-----|-----|-------|-----|-----|-----|-----|
| Stairmand’s High Efficiency | 0.5 | 0.2 | 0.5   | 0.5 | 1.5 | 4   | 0.375 |

2.1.2. Mesh

In the current study, medium smoothing condition and medium relevance is chosen. This resulted in the reduction of elements thus reducing the time required for solving. The generated mesh contains 16381 elements and 3540 nodes.

2.1.3. Boundary Conditions

The boundary conditions are applied to the geometry, in such a way that only velocity is
considered for the analysis. The boundary conditions were determined as follows:

In the model, the realizable k-epsilon (2-equation) model was selected and the enhanced wall treatment was enabled to account for different walls in the current design. The inlet velocity boundary condition was given as 20 m/s. Stationary wall was chosen with no slip shear condition. The number iterations were limited to 200 to get quick results.

- Significance of $k$-$\varepsilon$ equation: To simulate the turbulent cases in Computational Fluid Dynamics (CFD) the most common model used is K-epsilon ($k$-$\varepsilon$) turbulence model. This model contains two-equation which provide the overall explanation of turbulence by means of two transport equations (PDEs). The actual use for the K-epsilon model is to find a substitute to algebraically proposing turbulent length scales in modest to high difficult flows as well as to improve the mixing-length model. The primary transported equation finds the energy in the turbulence which is called turbulent kinetic energy ($k$). The secondary transported equation is the turbulent dissipation ($\varepsilon$) which finds the rate of dissipation of the turbulent kinetic energy.

2.1.4. Solution
The solution converged in the 200th iteration. The residual graph is generated during the CFD solution of CDS.

The outcome of the analysis is depicted below in the table 2. The fluctuation in the pressure and velocity within the cyclone is displayed in the figure 2. along the velocity contour of CDS in figure 3. The time taken for the solution to converge 200th iteration was 30 seconds. The speed of the particles in initially increases radially and then decreases from the center of the wall. This reveals the reduction of velocity at the midpoint and at the wall. Peak velocity was observed at the center of the marrow and along the wall. Peak velocity was observed at the center of the marrow and along the wall.

| Parameter                  | Values   |
|----------------------------|----------|
| Pressure (Max) (Pa)        | 364.0472 |
| Pressure (Min) (Pa)        | -46.79   |
| Velocity (Max) (m/s)       | 24.29    |
| Velocity (Min) (m/s)       | 0        |

Table 2. Pressure and Velocity reading of S.O.D.A.

Figure 2a. Vectors of Velocity of S.O.D.A.  Figure 2b. Vectors of Pressure of S.O.D.A.

Figure 3. The velocity contours of S.O.D.A.

2.2. Evaluating the influence of different parameters on CDS

In the current analysis, the inlet geometry, the diameter, cone height and cylinder height are considered. The boundary conditions employed are similar to the Stairmand’s analysis.
The analysis with the minimum value of maximum pressure is chosen for the modified design of CDS.

2.2.1. Inlet Geometry

The inlet geometry is one of the most important parameters influencing the performance of the separator. The analysis of various inlet geometries on the performance are analyzed in this work. The inlet geometry can be varied by varying the cross section. 6 different cyclones are considered by varying the inlet geometries are simulated in this paper. Pressure and velocity contour plots are shown in figure 4.

| Parameter                  | a=50, b=50 | a=60, b=50 | a=80, b=40 | a=100, b=40 | a=100, b=60 | a=120, b=60 |
|----------------------------|------------|------------|------------|-------------|-------------|-------------|
| Pressure (Max) (Pa)        | 231.9      | 276.9      | 282.23     | 365.05      | 473.24      | 733.77      |
| Pressure (Min) (Pa)        | -16.48     | -24.27     | -26.25     | -49.79      | -72.23      | -162.94     |
| Velocity (Max) (m/s)       | 22.38      | 22.66      | 23.55      | 24.29       | 25.39       | 31.9        |
| Velocity (Min) (m/s)       | 0          | 0          | 0          | 0           | 0           | 0           |

**Table 3. Influence of different inlet dimension**

**Figure 4a.** Pressure contours of 50*50  
**Figure 4b.** Velocity Vector of 50*50

2.2.1. Cylinder Diameter Analysis

In this analysis, the flow fields at various cylinder diameters is carried out in detail. The simulation of the cyclone is conducted for 5 different cylinder diameters. The fluctuations are recorded and compared. The effect of the cylinder diameter is justified which is shown in figure no. 5

| Parameter                  | D = 180   | D = 190   | D = 200   | D = 210   | D = 220   |
|----------------------------|-----------|-----------|-----------|-----------|-----------|
| Pressure (Max) (Pa)        | 303.32    | 288.61    | 365.04    | 258.39    | 255.30    |
| Pressure (Min) (Pa)        | -36.89    | -28.62    | -46.79    | -19.54    | -18.79    |
| Velocity (Max) (m/s)       | 23.98     | 23.06     | 24.29     | 22.23     | 22.28     |
| Velocity (Min) (m/s)       | 0         | 0         | 0         | 0         | 0         |

**Table 4. Influence of different cylinder diameter**

**Figure 5a.** Pressure contours of 200mm  
**Figure 5b.** Velocity Vector of 200mm

2.2.2. Cylinder height analysis

To estimate the effect of cylinder height, computational investigations are carried to analyze the effect of variation in cylinder height on the pressure drop and cut-off diameter.
The study is extended to analyze the details near the flow field pattern and velocity profiles which are shown in figure 6.

**Table 5.** Influence of different cylinder height

| Cylinder Height (h) | Pressure (Max) (Pa) | Pressure (Min) (Pa) | Velocity (Max) (m/s) | Velocity (Min) (m/s) |
|---------------------|---------------------|---------------------|----------------------|----------------------|
| 200                 | 375.55              | -39.09              | 24.44                | 0                    |
| 250                 | 379.80              | -43.5               | 24.64                | 0                    |
| 300                 | 365.05              | -46.8               | 24.3                 | 0                    |
| 350                 | 364.36              | -42.06              | 24.28                | 0                    |
| 400                 | 362.05              | -47.35              | 25.25                | 0                    |

**Figure 6a.** Pressure contours of 400mm  
**Figure 6b.** Velocity Vector of 400mm

**2.2.3. Cone Height Analysis**

This computational analysis is performed to understand the influence of variation in the cone height on pressure drop and cutoff diameter. Rest of the parameters were kept constant.

**Table 6.** Influence of different cone height

| Cone Height (hc) | Max pressure (pa) | Mini pressure (pa) | Max velocity (m/s) | Min velocity (m/s) |
|------------------|-------------------|--------------------|--------------------|--------------------|
| 400              | 366.16            | -34.72             | 24.9               | 0                  |
| 450              | 367.8             | -37.2              | 24.6               | 0                  |
| 500              | 365.04            | -46.7              | 24.3               | 0                  |
| 550              | 359.8             | -35.23             | 25.3               | 0                  |
| 600              | 360.2             | -42.37             | 24.8               | 0                  |

**2.2.4. Results and Discussions**

In CFD analysis, the parameter which is showing the minimum value of maximum pressure is considered to be the most efficient. Thus, on the basis of this, the following observations are made.

(i) Reduction of the inlet dimension results in pressure drop and it can be concluded that with minimum pressure drop is working better than other dimensions which was observed for the cyclone with the inlet cross section of 50mmx50mm.

(ii) By modifying the cylinder diameter results in huge variation in the pressure and velocity. For every 10mm increase in the cylinder diameter, a 10% decrease in the pressure was observed.

(iii) Small change in the velocity was observed when the cylinder height was increased. The drop-in pressure is diminished. Consequence of modifying the cylinder height is not as serious on the functioning and the flow design in comparison with the cylinder diameter.

(iv) Variation of height of the cyclone cone does not cause a major variation in pressure drop and in cut-off diameter. It can be concluded that cyclone cone height does not make any variation in cyclone efficiency as compared to other parameters.

**3. Mathematical Approach**

The detachment in a cyclone increases the sedimentation effect due to centrifugal force. This is ensured by introduction of tangential suspension. The cyclone separation efficiency is higher than dusting rooms since it has a centrifugal force field which increases the separation efficiency. The dimensional calculation was employed to derive the cyclone
dimensions used in theoretical research. For dimensional calculation, an input speed of 20 m/s of impure gas in the inlet and a volume flow of 400 m$^3$/s was chosen. With this data, the size of the cyclone parts was ascertained according to geometric similarity reports of a chosen type of cyclone. The tangential entry of gases into the cyclone was observed for this geometry. The relationship developed by Leith and Licht (1972) was employed to calculate the cyclone efficiency. In this equation a term natural length of CDS is employed, which is given by Alexander (1949) [10]. The model predicts the grade efficiencies and the Leith and Licht relation is given below,

$$\eta = 1 - \exp[-2((\frac{\pi d^2}{4} + \frac{\pi D^2}{4}(h - s) + \frac{\pi D^2}{4} \frac{(\ln+s-h)}{3} (1 + \frac{d}{D} + \frac{d^2}{D^2}) - \frac{\pi D^2 \ln D}{4}) \frac{2}{a,b} - (\frac{\rho v^2}{18 \mu D})(\frac{D}{H})^{2.5})]$$

where: $\eta$ - the cyclone efficiency;

$D$ - diameter cylindrical body; $h$ - main cylinder height;

$s$ - depth of penetration of outlet pipe; $\rho$ - solid particle density = 1110 kg/m$^3$;

$a$ - height of the cyclone inlet; $b$ - width of the cyclone inlet;

$\mu$ - gas mixture viscosity; $H$ - Height of cyclone separator;

$D_0$ - outlet pipe diameter; $v$ – the inlet velocity, m/s

$ln$ - natural length of cyclone;

$$ln = 2.3 * D_4 \left(\frac{D^2}{a^2} \right)^{\frac{1}{3}}; \text{ eq. } 2$$

d - cone diameter at natural length

$$d = D - (\frac{D - D_4}{H - h})(s - ln - h) \text{; eq. } 3$$

$$\mu = (37.4 + 0.560T) * 10^{-5}, T = 293 \text{ K; eq. } 4$$

$d_p$ - dimensions of particles, particle size of 0.00004 meters;

3.1. Collection Efficiency of Stairmand Optimized Design

The dimensions of the standard Stairmand optimized design is used to obtain the efficiency by Leith and Licht (1972) relations. The parameters are as follows: -

$D$ - 200 mm; $h$ - 300 mm; $s$ - 100 mm; $ln$ - 495.5 mm; $d$ - 140.9 mm; $H$ - 800 mm

$D_4$ - 100 mm; $\rho$ - 1110 kg/m$^3$; $a$ - 100 mm; $b$ - 40 mm; $K = 189.2 * 10^{-5}$ $d_p$ - 0.00004 m;

$v$ - 20 m/s $\mu$ - (37.4 + 0.560T) * 10^{-5}, T = 293

The efficiency obtained by this standard dimension is, $\eta = 58.43\%$

3.2. Collection Efficiency for Different Dimension of CDS

The theoretical analysis of the influence of a particular parameter dimension is carried out by varying the dimension of that parameter and keeping all other dimensions constant. The relationship developed by Leith and Licht (1972) is used to obtain collective efficiency.

3.2.1. Inlet Dimension ($a*b$)

Table 7 shows the recorded values for six different values of the cyclone inlet section and holding all other dimension constant.
Table 7. Collection efficiency in cyclone

| S. No. | Inlet Dimension (mm) | Efficiency |
|--------|----------------------|------------|
| 1.     | 50*50                | 64.65%     |
| 2.     | 60*50                | 62.35%     |
| 3.     | 80*40                | 61.44%     |
| 4.     | 100*40               | 58.43%     |
| 5.     | 100*50               | 55.33%     |
| 6.     | 120*60               | 50.24%     |

It is observed that the collection efficiency is best when the inlet section is 50*50 mm (mentioned above) in the current study.

3.2.2. Cylinder Diameter (D)
The table 8 shows the collective efficiency for five different cylindrical diameters.

Table 8. Collection efficiency in cyclone

| S. No. | Cylinder Diameter (mm) | Efficiency |
|--------|------------------------|------------|
| 1.     | 180                    | 57.08%     |
| 2.     | 190                    | 57.81%     |
| 3.     | 200                    | 58.43%     |
| 4.     | 210                    | 58.93%     |
| 5.     | 220                    | 59.33%     |

In the current work, the highest efficiency is observed for a cylindrical diameter of 220mm.

3.2.3. Cylinder Height (h)
The table 9 shows the collective efficiency for five different cylindrical heights.

Table 9. Collection efficiency in cyclone

| S. No. | Cylinder Height (mm) | Efficiency |
|--------|----------------------|------------|
| 1.     | 200                  | 51.02%     |
| 2.     | 250                  | 55.06%     |
| 3.     | 300                  | 58.45%     |
| 4.     | 350                  | 61.26%     |
| 5.     | 400                  | 63.63%     |

A cylindrical height of 400mm showed the best performance among the analyzed dimensions in the current research.

3.2.4. Outlet Pipe Height (s)
The table 10 shows the collective efficiency for five different outlet pipe heights.

Table 10. Collection efficiency depending on the outlet pipe height in cyclone

| S. No. | Outlet Pipe Height (mm) | Efficiency |
|--------|-------------------------|------------|
| 1.     | 150                     | 58.35%     |
| 2.     | 100                     | 59.26%     |
| 3.     | 200                     | 57.64%     |
| 4.     | 250                     | 55.51%     |
| 5.     | 300                     | 53.69%     |

The best efficiency was observed for an outlet pipe height of 100mm.

3.2.5. Cone Height (Hc)
The table 11 shows the collective efficiency for five different cone heights.
Table 11. Collection efficiency depending on the cone height in cyclone

| S. No. | Cone Height (mm) | Efficiency |
|--------|-----------------|------------|
| 1      | 400             | 57.72%     |
| 2      | 450             | 58.10%     |
| 3      | 500             | 58.43%     |
| 4      | 550             | 58.68%     |
| 5      | 600             | 58.93%     |

The cone height has the least influence on the collective efficiency of the dust separator.

3.3. Result and Discussion

The table 12 shows the comparison of the standard dimension of dust separator with the modified dimension obtained from the mathematical approach as well as from the CFD analysis.

Table 12. Changes in Standard Dimension that are considered above

| S. No. | Parameter                                      | Symbol | Old Dimension (mm) | New Dimension (mm) |
|--------|-----------------------------------------------|--------|--------------------|--------------------|
| 1      | Diameter Cylindrical Body                     | D      | 200                | 220                |
| 2      | Cylinder Height                               | H      | 300                | 400                |
| 3      | Depth of Penetration of Outlet Pipe           | S      | 100                | 50                 |
| 4      | Natural Length of Cyclone                     | L<sub>n</sub> | 495.5 (calculated) | 617.5 (calculated) |
| 5      | Outlet Pipe Diameter                          | D<sub>x</sub> | 100               | 100                |
| 6      | Solid Particle Density                        | P      | 1110 kg/m<sup>3</sup> | 1110 kg/m<sup>3</sup> |
| 7      | Height of the Cyclone Inlet                   | A      | 100                | 50                 |
| 8      | Width of the Cyclone Inlet                    | B      | 40                 | 50                 |
| 9      | Gas Mixture viscosity                         | M      | 189.2*10<sup>-5</sup> | 189.2*10<sup>-5</sup> |
| 10     | Dimensions of Particles                       | D<sub>p</sub> | 0.04              | 0.04               |
| 11     | Inlet Velocity, m/s                           | V      | 20m/s              | 20m/s              |
| 12     | Cone Diameter at Natural Length               | D      | 140.9 (calculated) | 155.8 (calculated) |
| 13     | Height of Cyclone Separator                   | H      | 800                | 900                |

3.4. Mathematical calculation of Modified Dimension

By substituting all the new dimension of CDS parameter (mentioned above) in collection efficiency formula of Leith and Litch (1972) (given in equation 1), we get efficiency as

\[ \eta = 72.04 \% \]

4. CFD Analysis of modified dimension

The CFD analysis of the modified dimension of the dust separator is done and this section analyses the results of the same.

4.1. Solution :- Residual

The solution converged after 200 iterations. The figure 8 depicts the path followed by the fluid. The swirl conditions as explained in the rule of the cyclone separation was observed. The velocity ranges are displayed by the left sidebar and by the color codes in the vector plot at different levels. The velocity values are investigated from the left side plate. The contour of velocity is shown in figure 9 which reveals the complete flow variation in the cyclone separator.
4.2. Result and Discussion

The optimized solutions of Stairmand’s design are shown in figures 10 and 11. The pressure and velocity variation inside the cyclone is shown in the figures 10 & 11 along direction Vector. The solution converged the 200th iteration and the total time taken for the analysis was 30s.

5. Conclusions

The geometry of a cyclone separator affects the current area and operation parameters significantly. Seven influencing parameters can be sorted into four categories: the inlet dimensions (width and height), the vortex finder dimensions (length and diameter), the hit of the cyclone (cone and barrel) and the cone-tip diameter. The inlet cross section is the most influential on the CDS collective efficiency and the cone height is the least or no influence on the efficiency. For this study two different methods are used. The results obtained by CFD analysis and mathematical formulation shows similar outcomes.

The collection efficiency of the modified CDS calculated shows an improvement of 13.6% when compared with the Stairmand’s design.
Acknowledgment

I, Suveg Bhade owe lot to many people who supported me during the write up. I thank Dr. Srijnsha Hegde, who guided me and assisted me in correcting document with attention care. I also thank Mr. Ajay Kawade, Assistant Professor, Rungta College of Engineering & Technology, for assisting me during the course of this paper and for comments that greatly improved the manuscript. I would also thank our institution and our faculty members without whom, this work would never have been accomplished. I also extend the heartfelt thanks to all well-wishers, friends, and family for their encouragement and cooperation.

References

[1] K. Elsayed, October. 2011 Analysis and Optimization of Cyclone Separators Geometry Using RANS and LES Methodologies Khairy Elsayed.
[2] J. Gimbun, T. G. Chuah, A. Fakhru’l-Razi, and T. S. Y. Choong, 2005 “The influence of temperature and inlet velocity on cyclone pressure drop: A CFD study,” Chem. Eng. Process. Process Intensif., vol. 44, no. 1, pp. 7–12.
[3] P. Tapkir and D. Kamble, 2015 “Analysis of Cyclone Separator Using Empirical Models and CFD For Variation Of Dimensions,” vol. 2, no. 6, pp. 225–233.
[4] D. de Brito Dias, M. Mori, and W. Martignoni, 2008 “Study of Different Approaches for Modeling Cyclones Using CFD,” 2008 Annu. Meet.
[5] M. Marinuc and F. Rus, 2011 “the Effect of Particle Size and Input Velocity on Cyclone Separation Process,” Bull. Transilv. Univ. Brasov, vol. 4, no. 2.
[6] A. N. Huang et al., 2017 “Influence of a laminarizer at the inlet on the classification performance of a cyclone separator,” Sep. Purif. Technol., vol. 174, pp. 408–416.
[7] J. Gimbun, 2008 “CFD Simulation of Aerocyclone Hydrodynamics and Performance at Extreme Temperature,” Eng. Appl. Comput. Fluid Mech., vol. 2, no. 1, pp. 22–29.
[8] N. Gopani and A. Bhargava, 2011 “Design of High Efficiency Cyclone for Tiny Cement Industry,” Int. J. Environ. Sci. Dev., vol. 2, no. 5, pp. 350–354.
[9] B. Raj, R. Dere, G. M. Babu, and S. R. Rao, 2014 “Design and Analysis of Cyclone Separator,” vol. 3, no. 8, pp. 12–20.
[10] J. Dirgo and D. Leith, 1985 “Cyclone collection efficiency: comparison of experimental results with theoretical predictions,” Aerosol Sci. Technol., vol. 4, no. 4, pp. 401–415.
[11] D. Pandya, August 2010 “a Low Cost Micro Scale Cyclone Separator Design and Computational Fluid Dynamics Analysis.”, pp. 1–73.
[12] B. Breiderhoff et al., 2013 “Simulation and Optimization of Cyclone Dust Separators,” Proc. 23. Work. Comput. Intell., no. March 2017, pp. 177–195.