Design & Analysis of Cyclone Separator for Spray Dryer

Mr. Bhagvat A. Wadekar[1] Prof. Mr. Rohan Katwate[2] Prof. Mr. Avinash S. Patil[3]
Ms. Punam M. Chabukswar[4]
[1]PG Student [2,3,4]Assistant Professor
[2,3]Department of Mechanical Engineering
[1,2,3]D. Y. Patil School of Engineering Academy, Ambi, India

Abstract— This Paper is about the analysis of Cyclone separator for Spray Dryer with improved design. Industrial spray dryers are used to convert the sludge product into powder form. The size of powder product is in the range of microns and it is necessary to separate the powder from process air. Cyclone, Bag filter and Scrubber are most widely used devices in particle separation process. The cost of bag filter and scrubber is approximately three times that of cyclone. Hence by improving the efficiency of cyclone, the load on Bag filter and Scrubber can be reduced. The primary objective of the present study was to analyze the cyclone separator theoretically and CFD analysis. The second objective was to propose a new design of cyclone which will have greater efficiency than the existing design cyclone. The present cyclone was analyzed in ANSYS Fluent. The results from theoretical and CFD analysis were compared. The second objective was achieved by introducing an invert cone at bottom part. CFD analysis was carried out using the same procedure as mentioned in the case of previous analysis. The trajectory of the particle, flow path of the gas and the drop in pressure were analyzed. The theoretical collection efficiency of the existing cyclone separator was found to be 94.2 %. The efficiency of the existing cyclone from CFD analysis is found to be 95.7 %. The variation in theoretical and CFD results was found to be 7%. The pressure drop in the proposed cyclone separator from CFD analysis is found to be increase by 5 mmwc compare with the CFD results of present cyclone separator. The efficiency of proposed cyclone was found to be 98%. Thus the proposed cyclone separator is 2% more efficient than present cyclone separator.

Key words: ANSYS Fluent, Collection Efficiency, Pressure Drop, Cyclone Separator, Trajectory, Invert Cone

I. INTRODUCTION

A cyclone separator is a very cheap Pollution control device used now days. From past few decades, Cyclone has been use in many industries like Food Processing Industries, Cement Industries, and Chemical Industries etc. Now days, the use of cyclone for process industries has been increased drastically due to strict pollution control norms. The Cyclone Device consist of two chambers called as Shell and Cone.

The upper part of cyclone is shell and bottom part is Cone. The air with particulate enter into the shell part from top in tangential direction. The air stream attends very high velocity at outlet due to which particle is separated from air stream. This is because of the inertial and drag force that act on particle against the air stream. Lapple was the first person who studied the working of cyclone separator and in 1951, he proposed the derivation for calculation of Efficiency and Pressure drop for cyclone separator. Stern et al. carried out many experiments on Shepherd and Lapple model and found that his result fits better with the theoretical model. Nevertheless, after few years the invention of cyclone with different configuration came out for those different configurations. The model of Lapple was not appropriate. The pressure drop values from theoretical calculation was varying with the factor of two. [9]

Many researchers worked for the formulation of single model for cyclone separator but they found it very difficult. After a long span, Iozia and Leith in 1990 came up with new method and they claimed that their method works better than the other theories. Experimental analysis of cyclone separator with all the arrangement is not an easy job and required time and money, which is not affordable for industries. Therefore, many of the experiments were performed on small-scaled prototype model. This problem can be overcome by CFD analysis. CFD analysis gives a good prediction of the physics inside the cyclone.

"Arkadiusz Kepa", [4] presents the CFD analysis using fluent for a large cyclone with core diameter of 0.7m with a cone assembly at bottom. With lots of changes in diameter and cone assembly, he was able to increase the cyclone efficiency with increase in pressure drop value. In his research, he hadn't carry out theoretical analysis for validation

Hesham M. El-Batsh, [7] studied the effect of outlet duct of cyclone on the collection efficiency of cyclone. He carried out CFD analysis using Reynolds Mean stress model, with DPM and solved particle equation in CFD. He has done the validation of CFD results with experiments, but the experimental value with which he compared his result has taken from other literature. Even though he found some interesting results with CFD analysis that, changing the diameter of outlet duct improves the collection efficiency but there was also increase in pressure drop. Hence, he optimized the diameter of outlet duct and suggested proper selection method of selection of outlet duct diameter.

II. OBJECTIVE

The main objective of the project is to improve the collection efficiency of Cyclone separator and to carry out the theoretical analysis of various important parameter such as Velocity, Pressure Drop using various equation. Another objective is to find out pressure drop in cyclone using CFD analysis and comparing the same with theoretical values.

Bag Filter is one of the Particle separation device used after cyclone to separate the particle which are not separated by cyclone. This is one of the efficient device but it cost is 3 to 4 times that of cyclone. Efforts has been made to reduce the capacity of bag filter by improving the collection efficiency of cyclone. To achieve this objective a cone in the form of vortex breaker is proposed at the bottom assembly of cyclone.

III. THEORETICAL ANALYSIS OF CYCLONE SEPARATOR

Important performance parameters of cyclone Separator are cyclone collection efficiency and pressure drop. Both the parameters are very important while designing the cyclone. These two parameters of cyclone are very sensitive to the dimensions of cyclone body. Any change in single dimension can affect the pressure drop or efficiency of cyclone.

In cyclone separator, the gas stream with particle enters tangentially into the cyclone body form top part of the cyclone. Due to strong swirling flow inside the body, two vortexes are creating inside the body, which are outer vortex and inner vortex shown in Fig. 1. The particles loaded with gas stream start to separate due to the difference between their drag force and inertial force.

Particles with heavy mass or larger diameter will collide to the cyclone wall hence is separated from gas stream. This process will continue until all the particles are separate.

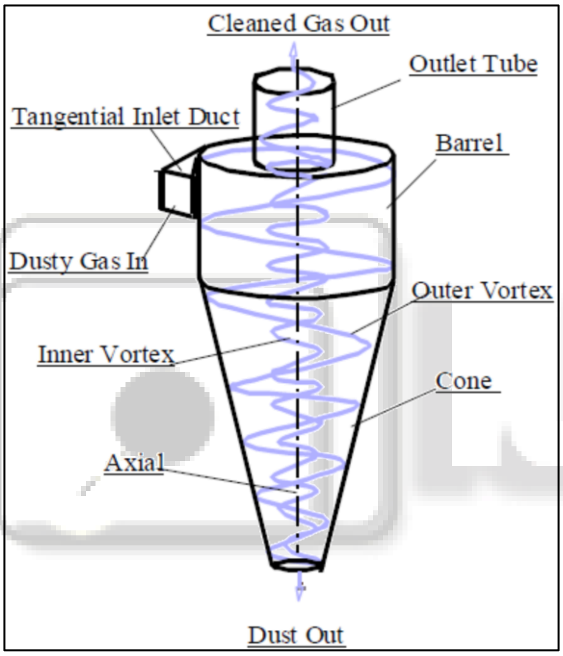

Fig. 1: Flow inside the Cyclone

Following are some important parameters, which is indirectly relate to the cyclone collection efficiency and the pressure drop.

1) Number of Effective Turns.
2) Inlet velocity of cyclone.
3) Tangential, axial, and radial velocity of cyclone.
4) Cut point diameter of cyclone.
5) Overall collection efficiency of cyclone.

A. *Number of Effective Turns*

Number of effective means the revolution of gas stream inside the vortex. If effective turn increase, then efficiency of cyclone also increases. Lapple developed the general equation to predict the number of effective turns.

$$N = \frac{1}{H}\left[L_b + \frac{L_c}{2}\right] \quad \ldots(1)$$

It has been experimentally found that the number of turns vary much with this formula. LINGJUAN WANG predicts new equation for two separate sections as Shell section and Cone section.

Number of effective turns in shell section.

$$N_{e1} = \frac{L_1}{\pi * D_c} \quad \ldots(2)$$

Number of effective turns in cone section.

$$N_{e2} = \frac{L_2}{\pi * \left(\frac{D_c + D_o}{2}\right)} \quad \ldots(3)$$

B. *Inlet Velocity of Cyclone*

Inlet velocity of cyclone is calculating from the flow rate inside the cyclone and the inlet area of cyclone as follows,

$$Q = V_i W H \quad \ldots(4)$$

Here Q is flow rate and Vi is inlet velocity and W is width and H is height of inlet.

C. *Tangential Velocity*

Tangential force is much dominated flow due since it contributes to the centrifugal force inside the cyclone body. The derivation for velocity can be given as follows,

$$V_{t2} = \frac{R}{r} * V_{in} = \frac{R * V_{in}}{r_o + Z * \tan\theta} \quad \ldots(5)$$

Here, Vt is Tangential velocity and Vin is the Inlet velocity of cyclone, R is the radius, r is the distance from centre to the required point to where the tangential velocity is to be found.

D. *Axial Velocity*

Axial velocity divides into two parts as in shell part and Cone part.

The Axial velocity inside the shell part

$$V_{z1} * \left(\frac{\pi * D_c^2}{4} - \frac{\pi * D_e^2}{4}\right) = V_{in} * \frac{D_c^2}{8} \quad \ldots(6)$$

Radial velocity in shell part is very less hence can be neglected, and it is applicable for cone part.

E. *Cut Point Diameter of Cyclone*

It is defined as the size of diameter which cyclone can separate with 50% efficiency. Hence all the particles with diameter more than cut point diameter can be separated by the cyclone.

$$d_{pc} = \left[\frac{9\mu w}{2\pi N V_i(\rho_p - \rho_a)}\right]^{\frac{1}{2}} \quad \ldots(7)$$

F. *Over All Efficiency of Cyclone*

This efficiency of cyclone is based on the size of the particles with their mass fraction in solid loading. Cyclone performs with different efficiencies for different particle size, hence by taking the mass fraction of particular particle size and considering the separated efficiency particle wise is much better approach. Further the average efficiency can be found out by individual efficiency.

$$\eta_j = \frac{1}{1 + \left(\frac{d_{pc}}{d_{pj}}\right)^2} \quad \ldots(8)$$

$$\eta = \frac{\sum n_j m_j}{M} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (9)$$

where η_j is Efficiency of cyclone for j[th] particle , n is the overall efficiency, m_j is the mass fraction of j[th] particle.

G. *Pressure Drop Inside the Cyclone Body*

Theoretical calculation of cyclone is very complex part, yet some researchers found the nearly approximate semi-empirical formula from lots of experiments. The formulation of this semi-empirical formula found to be inaccurate with slight change in any dimension of cyclone body.

A simple and traditional formula used is as follows,

$$\Delta P = K_c \rho v_i^2 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (10)$$

where K_c is Proportionality constant found experimentally by many researchers. Its value ranges from 0.0025-0.0100.

IV. COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF PRESENT CYCLONE SEPARATOR

ANSYS Fluent is one of the very powerful tool to study the fluid motion inside the cyclone body. RMS (Reynold Mean Stress) model is used for this analysis. This model can accurately predict the strong swirl flow.

Fluent basically solve seven equations which include three Momentum, mass equation and three shear stress equation in x, y and z direction.

Due to calculation of shear stress in this model the pressure drop prediction is very accurate. The analysis of Particles is done by using the concept of discrete phase model. In this case, Fluent solves the Force balance equations of particles with respect to gas stream.

Major three forces, which are considered by fluent while solving the equations, are inertial forces, Drag Forces, and Gravitational Forces.

A. *Design of Cyclone*

The most important design parameter in design of cyclone is the inlet velocity. The inlet velocity of a cyclone can be controlled by selecting the appropriate inlet dimension of the cyclone.

1) Design Considerations

1) Flow rate 14820 m3/hr
2) Air Temperature 110 ^{o} C
3) Density of DMAHCL 900 kg/m3
4) Inlet velocity (Assumed) 21 m/s

2) Calculations

1) Inlet Area

$$Q = Vi*W*H \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (11)$$

Where Q is flow rate, Vi inlet velocity, W Width of inlet, H Height of Inlet.

14820 = 21 x W x 0.7 x3600

W=14820/(21x0.7x3600)

W= 0.280m = 280mm

Other dimension of cyclone is calculated as follow.

Dimensions ratio	Value
D/W	4.2
H/W	2.5
De/W	2
S/W	5.25
Lb/W	6.42
Lc/W	10.8

Table 1: Dimension Ratio of Present Cyclone Separator

From table 1, we have,

D = 1200 mm H = 700 mm S = 1470 mm

Do = 360 mm De = 560 mm Lb = 1800 mm

Lc = 3050 mm

Following are the dimensions shown in the Fig. 2.

All dimensions are in mm.

D	H	W	D_e	S	Lb	L_c	D_d
1200	700	280	560	1470	1800	3050	450

Table 2: Dimensions of Cyclone

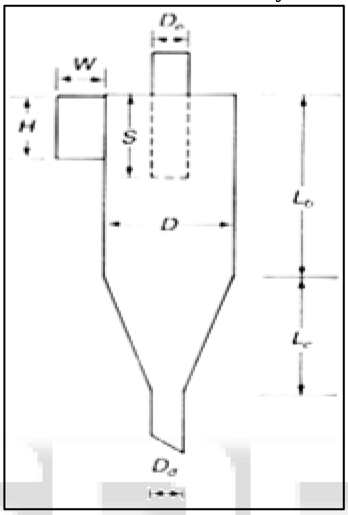

Fig. 2: Geometry of Cyclone

B. *Meshing & Boundary Conditions*

The meshing is done by using the ANSYS meshing tool. A tetra hydral element is created by using default parameter with following elements and nodes. A picture of Boundary condition and mesh view are shown in Fig. 3 & 4 respectively.

Number of elements	606065
Number of nodes	108071
Element quality	0.129

Table 3: Mesh Quality

B.C.	Fluent DMP conditions
Inlet	Escape
Air Outlet	Escape
Particle Outlet	Trapped

Table 4: Boundary Conditions

Properties	Values
Density of Air	1.12 Kg/m3
Density of DMAHCL	900 kg/m3
Viscosity of Air	1.7e-5 pa-s
Velocity at inlet	21 m/s
Flow rate	250 m3/min

Table 5: Properties Considered For Analysis

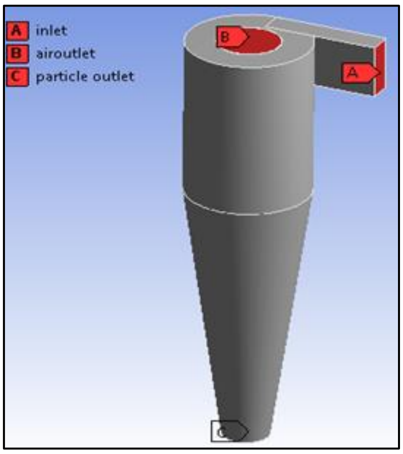

Fig. 3: Boundary Conditions

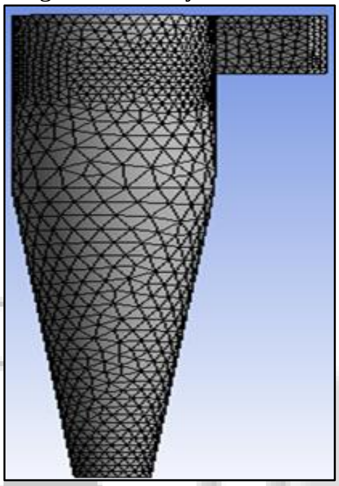

Fig. 4: Mesh View

V. RESULT & DISCUSSION

The project dealt with the design of new cyclone separator, CFD analysis is performed on both present and proposed cyclone separator. The result obtained from CFD analysis of present cyclone separator are used to design the proposed cyclone separator. The pressure contours and velocity vectors are shown in this section for present and proposed cyclone separator, which give us idea about the improvement in new design. The results of CFD analysis for present cyclone separator are compared with theoretical and experimental value. The design approach of cyclone is empirical, hence to perform theoretical analysis strong experimental results are required which is not a part of this project thus theoretical analysis is carried out only for present cyclone separator and not for the proposed cyclone separator.

A. Velocity

From equation number (5), we know that tangential velocity at shell section is same as that of inlet velocity, while for cone section it is different. Axial velocity also varies for shell and cone part. Below mentioned table shows the details of velocity values found from theoretical calculations.

Velocity	Shell	Cone
Tangential	21	13.39
Axial	4.3	3.32
Resultant	21.44	14.28

Table 6: Velocity in m/s

B. Number of Turns & Cut Point Diameter

Number of turns found from Lappel and Lingjuan Wang approach. It is found that from lapel approach i.e. from equation number (1) the number of turns found as 5.5 turns and that for Lingjuan Wang from equation number 2 & 3 it is 6.5 turns. Cut point diameter of cyclone found to me is 9.69 micron with 50% collection efficiency.

C. Efficiency & Pressure Drop

For calculation of Efficiency, we can use lapel approach which is best suited for this application using equation 7 &8.

Size range		Average size	Mass fraction	Efficiency
5.00	10.00	7.50	0.13	4.87
10.00	30.00	20.00	0.13	10.53
30.00	50.00	40.00	0.13	12.28
50.00	70.00	60.00	0.13	12.67
70.00	100.00	85.00	0.13	12.83
100.00	120.00	110.00	0.13	12.90
120.00	150.00	135.00	0.13	12.93
150.00	230.00	190.00	0.13	12.97
			Overall efficiency	91.89

Table 7: Efficiency of Cyclone

From equation (9), taking Kc proportionality constant as 0.0055, from experimental results, the Pressure Drop value inside the cyclone body was found to be 67.71mm of water column.

D. CFD Analysis Results

1) For Present Cyclone Design

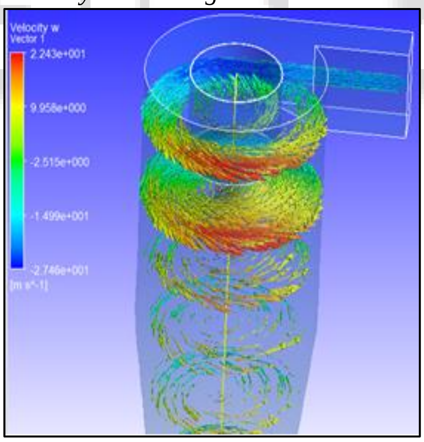

Fig. 5: Tangential Velocity Vectors of Shell Part

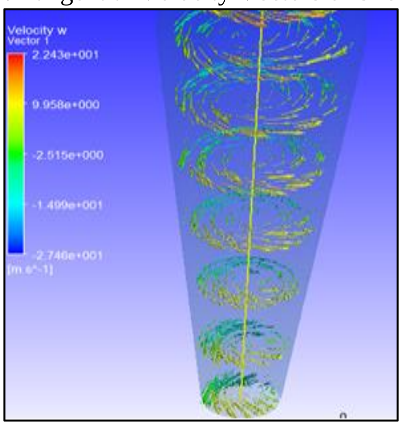

Fig. 6: Tangential Velocity Vectors of Cone Part

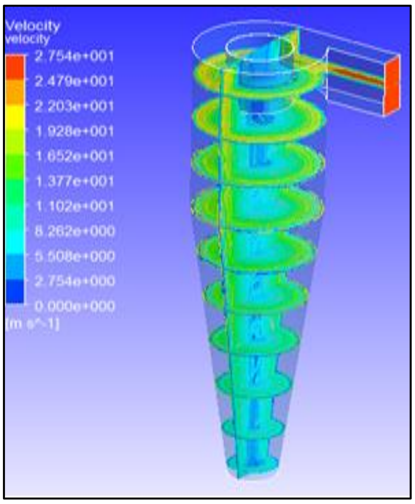

Fig. 7: Velocity Contours in Various Planes

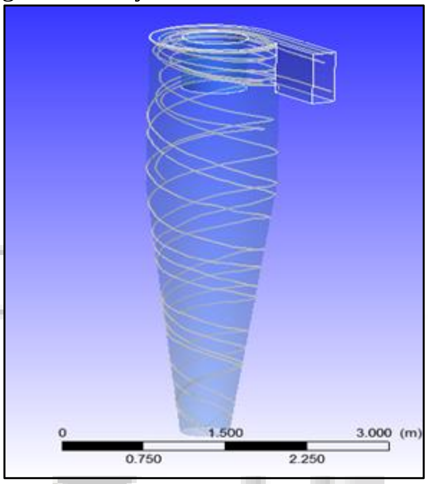

Fig. 8. Number of Turns inside the Cyclone

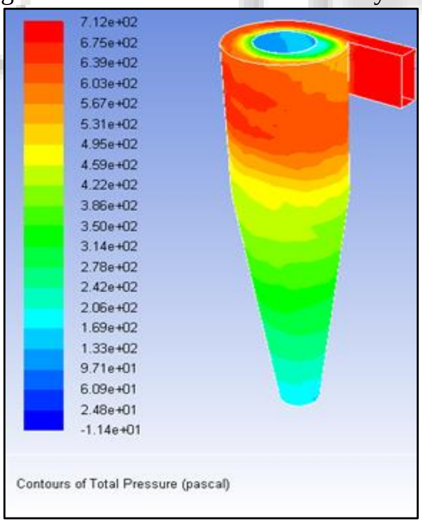

Fig. 9: Pressure Drop along the Outer Wall

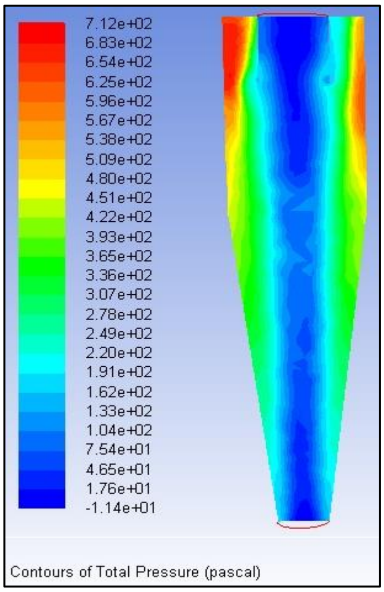

Fig. 10: Pressure Contours at Mid Plane

2) For Proposed Cyclone Design

The geometry is prepared with the same concept as mention in case of present cyclone separator. The entire dimensions are same in present and proposed cyclone, except there is slight change in proposed cyclone design. Cone shape apex is attached at the bottom cone of cyclone body as shown in Fig. 11. With the help of cone shape, inner vortex can be break as per requirement for efficiency increment by varying the height of cone. Inner vortex generation starts from the bottom of cyclone; hence the cone is placed at bottom. The dimensions of cone shape are chosen as,

R1=180mm, R2=60mm, Hc=340mm

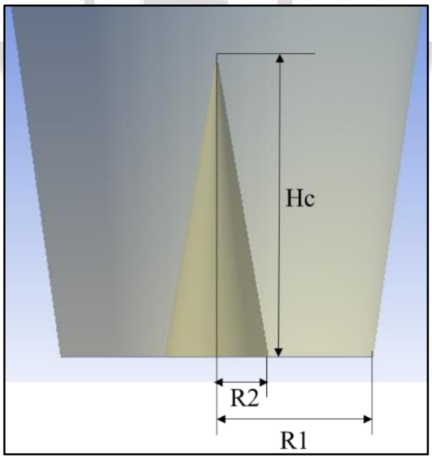

Fig. 11: Cone Dimensions of Proposed Cyclone Separator

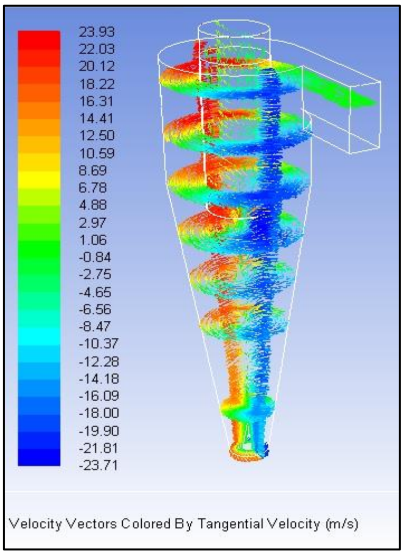

Fig. 12: Velocity Vectors of Proposed Cyclone

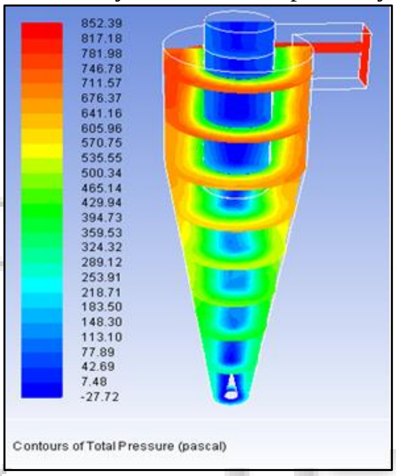

Fig. 13: Pressure Contours of Proposed Cyclone

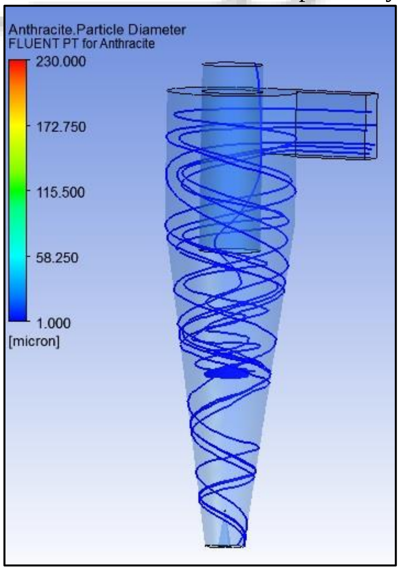

Fig. 14: Number of Turns for Proposed Cyclone

The efficiency of cyclone is found by tracing the discrete particles. For present design of cyclone separator, the efficiency found to be 94.2 % and 98 % for proposed cyclone. Number of turns is found directly in fluent by tracing the particle trajectory. It was found to be 6 turns for present cyclone and 6.5 turns for proposed cyclone.

E. Comparisons of CFD & Theoretical Results for both the cyclones

The velocity inside the cyclone body by theoretical method is an average velocity inside the cyclone body. The result obtained from CFD for velocity on various planes is not an average value hence an average value is considered which is taken from fluent. Histogram shown below in Fig. 15, 16, 17 & 18 gives a better idea about difference in theoretical and experimental values for velocity, pressure drop, Collection efficiency & number of turns respectively for present and proposed cyclone design.

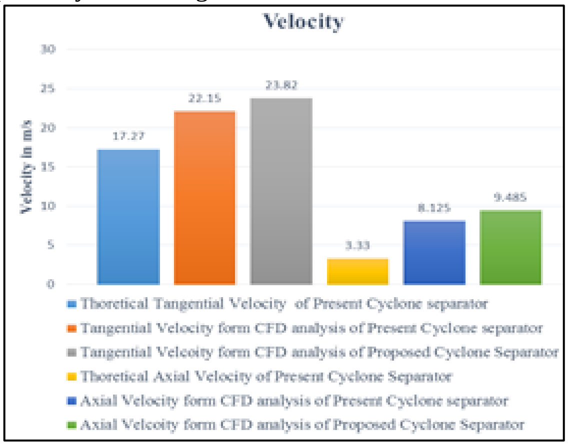

Fig. 15: Velocity Comparison

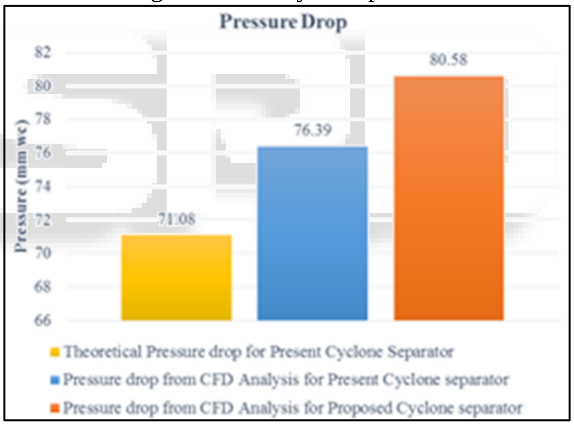

Fig. 16: Pressure Drop in Cyclone Body

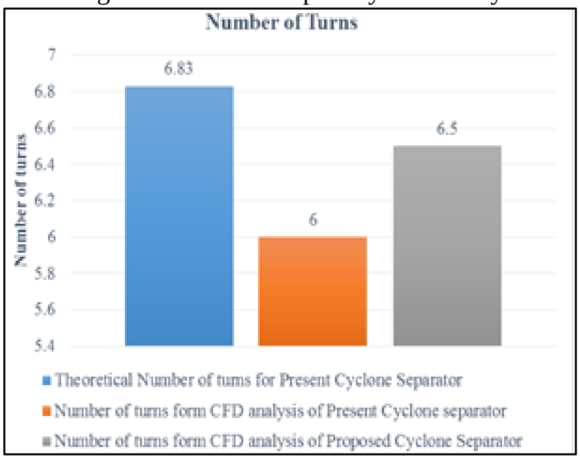

Fig. 17: Number of Turns

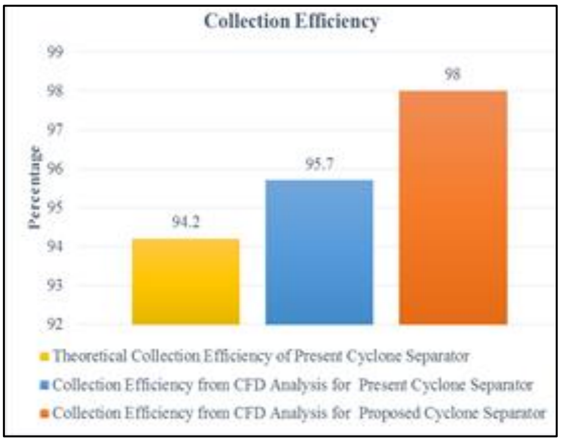

Fig. 18: Collection Efficiency of Cyclone

VI. CONCLUSION

The efficiency of proposed cyclone separator is increased approximately by 2% as compared to the Present Cyclone Separator. The pressure drop in proposed cyclone is increased by 4.19mm of water column as compared to the present cyclone. The pressure drop for present cyclone separator from theoretical calculation is 71.08mmwc and that of CFD analysis is 76.39 mmwc.

In case of proposed cyclone separator, drastic pressure variation is observed at the bottom section, this infer that there is loss of energy in bottom region. It is observed that the effective number of turns from theoretical analysis is 6.83 turns and the effective number of turns from CFD analysis is 6 for present cyclone separator and 6.5 for proposed cyclone separator.

However, the collection efficiency found from theoretical analysis is less as compare with the CFD analysis.

The possible reason for this variation might be the variation in theoretical particle distribution and CFD particle distribution. The collection efficiency from theoretical analysis and collection efficiency from CFD analysis for present cyclone separator are 94.2 % and 95.7% respectively. The possible reason for this difference is incomplete tracing of 12 particles in cyclone separator. Hence with improved design, efficiency increases by 2%, which results in reduction in size and cost of bag filter used after Cyclone separator.

ACKNOWLEDGMENT

I wish to express my deep sense of gratitude and honor towards my respected project guide Prof. R. R. Katwate, Mechanical Engineering department, DYPSOEA, Ambi, Pune for his unending, inspiring and all dimensional guidance and encouragement to complete this project work.

I am thankful to project coordinator Prof. Avinash Patil. Head of Mechanical Engineering Department Dr. Sambhaji Kadam. & all those unknown faces of DYPSOEA, Ambi, Pune who helped me in all respects to fulfil the work with effectiveness for helping me for my project work.

I also express my sincere thanks to Dr. V. N. Nitnaware, principal of DYPSOEA, Ambi, and Pune for his encouragement to complete this project.

REFERENCES

[1] H. Yoshida, Y. Kwan-Sik, K. Fukui, S. Akiyama, and S. Taniguchi, “Effect of apex cone height on particle classification performance of a cyclone separator”, Advanced Powder Technology, Volume No. 14, Issue No. 3, 2003, pp. 263–278.
[2] Dr. I. Satyanarayana, “Computational Fluid Dynamics Analysis of Cyclone Separator”, (IJIRTSE), ISSN: 2395-5619, Volume – 1, Issue – 3. May 2015.
[3] Bharath Raj Reddy Dere, et al, “Design and Analysis of Cyclone Separator”, IJERT, ISSN: 2278-0181, Vol. 3 Issue 8, August – 2014.
[4] Arkadiusz Kepa, “The efficiency improvement of a large-diameter cyclone – The CFD calculations”, ELSEVIER, Separation and Purification Technology 118 (2013) 105–111.
[5] J. Gimbun et al, “A CFD Study on the Prediction of Cyclone Collection Efficiency”, ResearchGate, International Journal for Computational Methods in Engineering Science and Mechanics, May 2005.
[6] Lingjuan Wang, “Theoretical Study of Cyclone Design, Doctoral thesis”, Texas A&M University, 2005.
[7] Hesham M. El-Batsh, “Improving cyclone performance by proper selection of the exit pipe”, ELSEVIER, Applied Mathematical Modelling 37 (2013) 5286–5303
[8] L. Wang et al, “Effect of air density on cyclone performance and system design”, Transactions of the ASAE, 2003 American Society of Agricultural Engineers ISSN 0001–2351.
[9] C.E. Lapple, “Processes use many collector types”, Chemical Engineering, Volume No. 58, 1951, pp. 144–151.
[10] John Anderson, “A Text Book on Computational Fluid Dynamics”, vol. 1, 2011.

