Comparison of a Classical Cyclone Separator and Protruding Surface Cyclone Separator using CFD Software

Vinay Sati1a*, Shivasheesh Kaushik1a*, Dr. Rahul Kshetri2a, Dr. Kuldeep Panwar2b, Rahul Pandey2c

1a*Assistant Professor, Department of Mechanical Engineering, Amrapali Group of Institute, Haldwani, Uttarakhand, India.

1aAssistant Professor, Department of Mechanical Engineering, Shivalik College of Engineering, Dehradun, Uttarakhand, India.

2c Scholar, B.Tech, Department of Mechanical Engineering, Amrapali Group of Institute, Haldwani, Uttarakhand, India.

2a Department of Production and Industrial Engineering, G.B. Pant University of Agriculture & Technology, Pantnagar, Uttarakhand, India.

2bAssociate Professor, Department of Mechanical & Automation Engineering, Delhi Technical Campus, Greater Noida, India.

*Corresponding author email: vinaysati379@gmail.com

Abstract. Industrial smoke contains various hazardous particles, ash namely to be one of the various substances. The cyclone separator is a specially designed device used to separate the ash particles from the stream of air. The stream of dirt and air mixture flows in two vortexes, the inner vortex and the outer vortex, this device in turns acts as a scrubber and the ash particles get collected down in the ash collector whereas the fresh air stream is released by the device. To increase the efficiency, i.e. to increase the ash collecting capacity of the device, we have tried to compare a classical cyclone separator with the one we created which has protruding surfaces internally and the comparative analysis is done using the CFD software ANSYS R16.0.

1. Introduction

1.1. CFBC Technology:
Cyclones are generally used with CFBC (Circulating Fluidized Bed Combustion) type of boilers, the coal is burnt in a bed comprising of hot particles and maintaining the flow of air through the bed. At sufficiently high air speeds, the bed will act as a fluid resulting in quick combination of the particles. This fluidised action will promote the complete combustion of coal at a relatively low temperature and will provide a channel to transfer heat generated efficiently from the bed to the steam tubes.

The boiler in which coal is burnt in an environment of high concentration of bed material (mineral matter) is derived from combustion of coal retained by using cyclone. This bed material is fluidized by primary air which is a part of combustion air.

1.1. Cyclone Separator:
Relatively coarse particle of limestone and unburnt char are captured in the cyclone and are recycled back near the base of the furnace. Finer solid residues (ash and spent sorbents) are generated during the process of combustion and desulphurization which leave the furnace, escaping through the cyclones, but they are collected by a bag-house or electrostatic precipitator which is further located downstream. A cyclone is a simple device which provides a sufficiently high degree of separation for a minimum pressure drop. Most of the solids (approx. 98%) are successfully removed by a cyclone separator when operating at designed flow rates. Cyclone Separators offer removal of particulate matter from air streams at very economical cost of operation and maintenance. A cyclone separator consists of mainly two parts namely, an upper cylindrical part referred to as the barrel and a lower conical part referred to as cone, also known as hopper. The stream of air enters tangentially at the top of the barrel from the inlet and travels downward by the action of
gravitational force into the cone forming an outer vortex. Increasing air velocity in the outer vortex causes centrifugal force on the particles, separating them from the air stream. When the air reaches the bottom of the cone, an inner vortex is created in the reverse direction and exits from the top as clean air (free of charge), while the particulates are collected at the bottom of the cyclone. The separation efficiency of the separators depends on the size of the ash particles being separated as shown in figure number 2.

![Figure Number 1. A CFBC type boiler arrangement.](image)

2. Literature Review

CFD acronym for Computational Fluid Dynamics has evolved over the past few years which enables designers to design customised piping arrangement for a required system as per the requirements of the specifications. Although, CFD offers results with great precision and accuracy, but in general, the numerical values differ by a certain factor as various external factors are present in reality. The difference in values is due to CFD being done in an ideal case solution. Most CFD solve the Navier-Stokes equation either in the Lagrangian or Eulerian approach. Apart from these methods, some solve the Boltzmann equation instead of Navier-Stokes equation. CFD can broadly be classified under two categories; namely conventional and accelerated methods.

Accelerated methods are further classified as advanced numerical methods and hardware methods. The advanced numerical methods hierarchy which is explained as hardware methods is beyond the scope of this paper. Advanced numerical methods are further classified into three categories as:

- Mesh based methods
- Mesh free methods
- Hybrid methods

Out of these three methods, the most commonly employed is the mesh-based methods in which there are certain meshed structure created on the geometry of the body. The mesh designs can vary from very fine complex meshes to simple larger meshes; as applicable by the need. Some of the common meshed based methods are listed as:

- Reduced Order Modelling (ROM)
- Marker and Cell (MAC)

In the mesh free methods, direct surface nomenclature is done by determining the direction of flow and the corresponding parametric values are applied. This is generally used as a macroscopic approach as surface analysis is done in this case. But, in Mesh-based model the microscopic analysis is done. Mesh offers variation of parameters at every instant of the mesh, which depends on how much fine meshing has been done. For in-depth analysis of flow, Mesh-based models are preferred as compared to Mesh free models. The advantage of Mesh free model is that it can provide quick results; which translates into easy adaptation in industries wherein continuous production of goods is taking place. The mesh free methods generally employed are as follows:

- Smoothed Particle Hydrodynamics (SPH)
Moving Particle Semi-Implicit Method (MPS) Our research has been carried out by the conventional methods as they offer a great degree of accuracy, although being slow as compared to accelerated methods. The most popular conventional methods used in practice are:

- Finite Element Method (FEM)
- Finite Volume Method (FVM)
- Finite Difference Method (FDM)

These methods solve the Navier-Stokes equation, which are the governing equations of the CFD Software. These equations describe the conservation of mass, momentum and energy. The method solves equation of complexity order $n^3$, where $n$ is the degree of freedom. To solve the program in stages, concepts of dynamic programming can be used.

3. Research Methodology

The research was carried out from scratch and performed in multiple stages. The primary stage of the research involved the designing of the cyclone models, which was done with the use of Solidworks 2016 software.

3.1. First Stage:
We started with the designs that needed to be checked for their integrity. This was again done in the same software by applying the geometric region tests. Since all parts of the models were perfect regions as obtained from the tests performed on them; the first step in the research process was completed as the geometric bodies had perfect regions indicating that there shall be no losses due to leakage anywhere when we perform the fluid flow simulation.

3.2. Second Stage:
In the second stage of research, we imported the design files into the CFD software ANSYS R16.0. Then, we assigned the names to the sections of the pipe geometry as inlet, body and outlet. This named selection was done so that analysis at every region of the pipe as a whole body can be performed. This was followed by fine meshing of each section was done in order to provide accurate graphs while simulation is done. Lastly in this stage, we applied the flow direction to the body as inlet>body>outlet respectively.

3.3. Third Stage:
The third stage of the research was to select the applicable equations for the meshed model generated. As mentioned, we applied the Cyclone equation and selected the K-Epsilon model to obtain the results for our geometry. These equations have been explained further in the mathematical modelling section.

3.4. Fourth Stage:
In the fourth stage, which being the most important stage of the research was to do the simulation of flow. Now using FLUENT we applied the fluid flow through the geometry designed one by one, for each of the case we obtained the velocity and the pressure contours which are illustrated as the variation of the velocity and the pressure component at each point. Also, the graphs obtained indicated the variation of velocity and the pressure within the pipe.

Multiple number of simulations were done to check the integrity of the results obtained. We, then concluded that the results obtained from the simulation was genuine and hence our research work came to its conclusion. The results obtained are of for ideal conditions and have not been practically verified. There is a possibility of a variation in the values mentioned in the paper by a certain factor.

4. Research Gap

These results were obtained by CFD software are under ideal conditions with standard wall functions. Henceforth, during practical validity we may employ material depending on various factors, some of them being GCI, PVC, SS of varying grades with each possessing different chemical properties. These may affect...
the values of fluid parameters when tested under these materials. Also, various losses like losses due to friction will be taken into account when practically performed. Various environmental conditions prevailing may also affect the calculations in general practice as this was an ideal case. Due to lack of equipment and certain research tools the practical validity of the models discussed could not be tested and hence comparison between the practical and ideal approach could not be done.

5. Assumptions made
- Gas density is negligible as compared to the density of particles
- Gravitational forces are negligible as compared to the centrifugal forces
- Tangential velocity \( (V_t) \) does not vary in the axial direction, it is neglected due to short distance being covered; thus, relative velocity is purely radial
- Particles are spherical of low size and with small enough relative velocity to apply stokes law and not disturb the natural flow around it.
- No interaction between the particles.

6. Mathematical Modelling
There are two models used for the analysis- one is the simple cyclone (SC-1) and the other one is protruding cyclone (PC-2) as shown in figure number 2 and 3.

Figure Number 2. The SC-1 solid geometry.

Figure Number 3. The PC-1 solid geometry.

The details about SC-1 are as follows:
Mass properties of simple cyclone (Part Configuration - Default)
Output coordinate System: -- default --
Density = 7800.00 kilograms per cubic meter
Mass = 5617.28 kilograms
Volume = 0.72 cubic meters
Surface area = 5.41 square meters
Centre of mass: (meters )
- \( X = 0.02 \)
- \( Y = 1.37 \)
- \( Z = 0.05 \)
Principal axes of inertia and principal moments of inertia: (kilograms * square meters )
Taken at the centre of mass,
- \( I_x = (0.03, 0.99, 0.10) \) \( P_x = 529.46 \)
- \( I_y = (0.31, -0.10, 0.95) \) \( P_y = 1277.03 \)
Iz = (0.95, -0.00, -0.31)   Pz = 1448.99
Moments of inertia: (kilograms * square meters )
Taken at the center of mass and aligned with the output coordinate system.
Lxx = 1431.88 Lxy = 25.12   Lxz = 52.76 Lyx = 25.12   Lyy = 537.35   Lyz = 72.19
Lzx = 52.76   Lzy = 72.19   Lzz = 1286.24
Moments of inertia: (kilograms * square meters )
Taken at the output coordinate system.
Ixx = 11984.49   Ixy = 155.51   Ixz = 57.39
Iyx = 155.51   Iyy = 552.28   Iyz = 446.74
Izx = 57.39   Izy = 446.74   Izz = 11827.16
The only difference between the two models was of the studs, as the SC-1 has no studs, i.e. it has no protruding surfaces at the inner wall of the separator whereas the PC-2 has studs on its surface.
The dimension of the studs is: -
 Diameter= 40 mm
 length= 50 mm

7. Data Reduction
For the sake of simplification of these above equations, we consider that the particle under observation has reached "terminal velocity", i.e., there is no component of acceleration. This kind of situation occurs in the case when the radial velocity has generated enough drag force to counter the acting centrifugal and buoyancy forces.
F_d + F_c + F_b = 0

\[ -6\pi\rho_l\mu V_r + \frac{4}{3} \pi \rho_l^2 V_r^2 - \frac{4}{3} \pi \rho_f^2 V_r^2 = 0 \]

In non-equilibrium conditions when the component of radial acceleration is not zero, then the general equation must be considered. Rearranging the terms, we get-

\[ \frac{dV_r}{dt} + \frac{9}{2} \frac{\mu}{\rho_l r^2} V_r - \left( 1 - \frac{\rho_f}{\rho_l} \right) \frac{V_r^2}{r} = 0 \]

Experimentally it has been found that the velocity component of rotational flow is directly proportional to the square of the radius(r^2), therefore

\[ V_r \propto r^2 \]

8. Input Parameters
Analysis of the flow conditions was done using ANSYS Fluent 16.0 CFD Software. These equations are solved by converting complex partial equation into simple algebraic equation. All the geometries SC-1, PC-2 are 3-D rigid solids comprising of 1 region 3 named selections as inlet, outlet, and path. These 3-D geometries were used for solving momentum and energy equations. The initial phase velocity of the flow was defined at the inlet section of the pipe upstream. Acceleration due to gravity has been taken as 9.81 m2/sec. The standard wall functions were applied to the k-\(\varepsilon\) turbulence models for solving the problems. The initial velocity given to all the geometries were 3 m/sec.

9. Results and Observations
The charts of the pressure and velocity indicating variation in their respective values were obtained as shown in figure number 4, 5, 6 and 7 also the velocity and pressure contours obtained are as shown below: -
9.1. Variation of Pressure along Cross Section:

**Figure Number 4.** Pressure observed in SC-1

**Figure Number 5.** Pressure observed in PC-2

9.2. Variation of Velocity along Cross Section:

**Figure Number 6.** Velocity observed in SC-1

**Figure Number 7.** Velocity observed in PC-2

9.3. Vortex Core Region of Cyclone Separator:

**Figure Number 8.** Vortex Core observed in SC-1

**Figure Number 9.** Vortex Core observed in PC-2
9.4. Velocity Contour of Cyclone Separator:

**Figure Number 10.** Velocity Contour of SC-1

**Figure Number 11.** Velocity Contour of PC-2

9.5. Pressure Contour of Cyclone Separator:

**Figure Number 12.** Pressure Contour of SC-1

**Figure Number 13.** Pressure Contour of PC-2

9.6. Velocity Vector of Cyclone Separator:

**Figure Number 14.** Velocity Vector of SC-1

**Figure Number 15.** Velocity Vector of PC-2
The various pressure zones are indicated in the protruding cyclone clearly depicts that the fresh air free of charge flows out with a pressure of 5.34 Pa. The 3.897E+001 Pa zone indicates the mass flow rate at the inlet of the cyclone; it is evident from the contour that the pressure at the hopper is 3.010 Pa and same is the pressure of the fresh air free of charge coming out of the cyclone as shown in figure number 12 and 13.

The velocity remains in the zone of 3.11 m/s of the fresh charge free air flowing through the device as shown in figure number 14 and 15.

1. Conclusion
Based on the above plots and graphs obtained we can infer that the protruding surface cyclone separator offers better ash collection as compared to the simple cyclone separator; which in turn increases its efficiency with minimal design alteration. This in turn confers that in industrial applications where we deal with CFBC type of boilers we may use protruding surface type cyclone separator to enhance the ash collection and make it more economical for the operations and maintenance as compared to the classical cyclone separator.

Reference
[1] Parnell, C. B. Jr. 1996. Cyclone design for air pollution abatement associated with agricultural operations. In Proc. Beltwide Cotton Production Conferences.
[2] K.S. Lim, H.S. Kim, K.W. Lee, 2004. Characteristics of the collection efficiency for a cyclone with different vortex finder shapes, J. Aerosol Sci. 35, 743–754.
[3] W.E. Conard, H.G. Conard, T. Szylowlee, Insert for a cyclone separator, US, 0008072A1, 2002.
[4] J. Casal and J.M. Martinez-Benet. A better way to calculate cyclone Pressure Drop. Chemical Engineering, 90(2):99100, 1983
[5] B. Wang, D.L. Xu, K.W. Chu, and A.B. Yu. 2006, Numerical study of gas-solid flow in separator. Applied Mathematical Modeling, 30:1326-1342.
[6] D. Benoni, C.L. Briens, T. Baron, E. Duchesne and T.M. Knowlton, 1994, "A procedure to determine particle agglomeration in a fluidized bed and its effect on entrainment", Powder Technology, 78, 33-42.
[7] Barth W. 1956, Design and layout of the cyclone separator on the basis of new investigations. Brennstoff-Warme-Kraft 8: 1-9
[8] P. Sivakumar, 2015, Development of an optimum CFBC Cyclone separator with REPDS for low pressure drop and denudation rates using CFD, International Journal of Applied Engineering Research ISSN 0973-4562 Volume 10, Number 13 (2015).