International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 – 5161

Available at http://inpressco.com/category/ijcet

Research Article

Effect of Geometric Parameters on the Performance of Cyclone Separator using CFD

Prachi K. Ithape[#*], S. B. Barve[#], S. S. Pande[&] and A. R. Nadgire[#]

[2]Department of Mechanical Engineering, MAEER's MITCOE, Kothrud, Pune, Maharashtra, India
[&]Department of Mechanical Engineering, Dr. D.Y.Patil COE, Akurdi, Pune, Maharashtra, India

Accepted 12 March 2017, Available online 16 March 2017, **Special Issue-7 (March 2017)**

Abstract

Cyclone separator is a type of mechanical dust collector that is used to filter solid particles from the incoming gas flow. The two vital parameters that affect the performance of a cyclone are the collection efficiency and pressure drop through the system. Both collection efficiency and pressure drop are strongly influenced by the geometry of cyclone separator. The geometric parameters that affect its performance are mainly cone height, cylinder height, dip tube height, inlet section etc. The main aim is to maximize collection efficiency and minimize pressure loss. This paper mainly focuses on variations of geometric parameters of a cyclone and analyzing its effect on the collection efficiency. In this paper the actual cyclone separator model was first validated by using experimental results and results obtained from the computations performed in ANSYS CFX. The actual model was then modified by changing its geometrical parameters like cylinder height, cone height, dip tube height etc. The CFD analysis of these modified cyclone separators was performed. Collection efficiency obtained from the analysis was then used as a means to select the final design of the cyclone separator. The model with maximum collection efficiency is then selected.

Keywords: *Cyclone separator, collection efficiency, pressure drop, ANSYS CFX, single-phase flow, multi-phase flow, CFD*

Introduction

A gas cyclone is industrial equipment that belongs to the centrifugal type mechanical separators. The gas cyclone is inert or stable type mechanical device that makes use of centrifugal force to separate solid or liquid particulates from incoming gas flow. It mainly consists of three parts: an upper cylindrical part called as barrel, lower conical part called as cone and the inlet section. The working principle of cyclones is to convert the inertia force of gas particle flows to centrifugal force by means of a vortex generated in the cyclone body by keeping minimum pressure loss in the system. It is relatively easy to manufacture with low operation cost and low maintenance. The cyclone separator is one of the most efficient and robust dust separators. The only major drawback that these devices face is the large pressure drop associated with them.

Concept of Computational Fluid Dynamics (CFD) is the simulation of fluids engineering systems by the use of modelling and numerical methods. To use CFD, firstly we require a fluid problem. To solve this problem, we should have knowledge of the physical properties of fluid (Fluid Mechanics of system). Then by the use of mathematical equations we describe these physical properties of the system.

*Corresponding author: **Prachi K. Ithape**

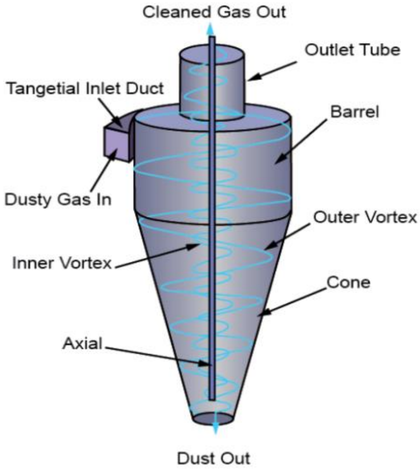

Fig. 1 Schematic Diagram of Cyclone Separator

This mathematical equation is the Navier-Stokes Equation which forms the governing equation of CFD. The Navier-Stokes Equation is analytical equation, but solving them analytically is a very difficult task. So these equations are usually solved on computers. In order to solve them we first have to convert it to the discretized form. The discretization methods include- Finite Difference method, Finite Element method, and Finite Volume method. The next step is to divide the entire problem into small domains or parts which nothing but the discretization of the system. Then, we write programs to solve them. The commonly used languages are FORTRAN and C. These programs are usually run on supercomputers. At the end, we can get our simulation results. We can compare and analyse the simulation results with experimental results and the real problem. If the results are not sufficient to solve the problem, we have to repeat the process until we get satisfied solution.

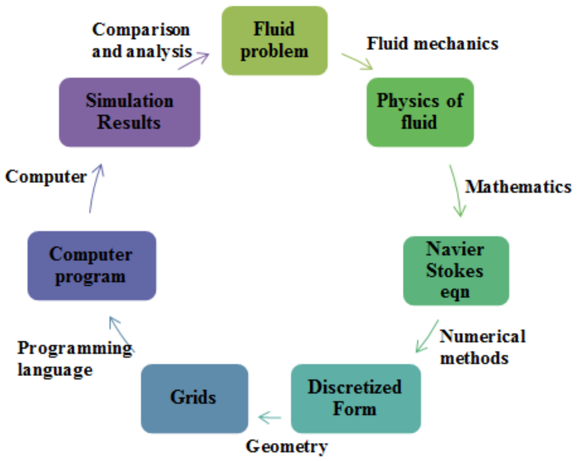

Fig.2 Flow Diagram of CFD

Objectives and Aim

The research focuses to analyze the cyclone separator model and modify its *geometric parameters only*with the aim to increase its *collectionefficiency* by using CFD and thus selecting the optimum model. The objectives of the study are-

- To study the various parameters that affects the collection efficiency of the cyclone.
- To design and optimize the model so as to get maximum efficiency and minimum pressure loss.
- Perform CFD analysis of the model for the following cases:
  a. Single phase (gas only) analysis.
  b. Two-phase or multi-phase (gas and liquid) analysis.

Table 1: Cyclone separator performance parameters

Cyclone Dimensions	Particle Properties	Gas properties
Cyclone diameter	Density	Density
Inlet height and inlet width	Diameter distribution	Velocity
Cylinder height		
Cone height	Shape	Temperature
Cyclone total height		
Cone tip diameter	Mass loading	Pressure
Vortex finder diameter		

There are various factors that affect the performance of a cyclone, particularly the cyclone dimensions, particle properties and gas properties. But this research study concentrates only on the cyclone dimensions i.e. the geometric parameters of the cyclone. These geometric parameters include cylinder height, cone height, inlet dimensions, dip tube height, outlet tube diameter etc. In particular the various modifications that were made in the actual model were- dip tube height increased and decreased, cylinder height increased and decreased, cone height increased and decreased, outlet tube diameter increased, and decreased, and a few combinations of these.

Table 2 Input parameters

Sr. No.	Parameter	Value	Unit
1.	Input gas flow rate	2,88,033.25	m^3/hr
2.	Temperature	460	°C
3.	Dust particle flow rate	123120	kg/hr
4.	Dust particle density	2700	kg/ m^3
5.	Operating pressure	-715	mm of H_2O

The collection efficiency of the cyclone separator in its working condition was 52% and the aim of the study was to increase this efficiency keeping in mind that the pressure drop does not increase much.

Geometry and Meshing

CAD Modeling of Cyclone Separator

The actual model of cyclone separator was designed as a CAD model in CATIA software as per dimensions provided. In CATIA, the model was designed in part modeling workbench. After preparing the CAD model, it was then exported to *.iges* file for further meshing process. Following is the draft corresponding CAD model.

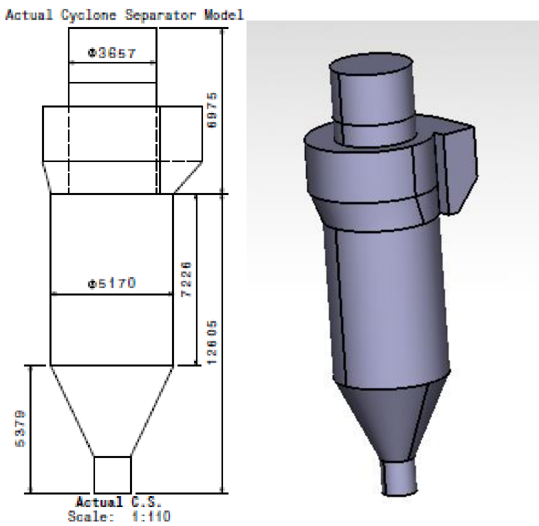

Fig.3 CAD Model of Actual Cyclone Separator

Meshing

Then the geometry was imported in ANSYS workbench by exporting the CAT product into *.iges* file. All the bodies were then combined into a multi-body part in order to produce a single mesh that contains all the bodies of cyclone separator. In order to generate the mesh, meshing parameters were specified, namely-physics preference, solver preference, inflation, sizing and method used for meshing. Thus the mesh was generated. The statistics representing number of nodes and number of elements of the meshed component denotes the quality of mesh.

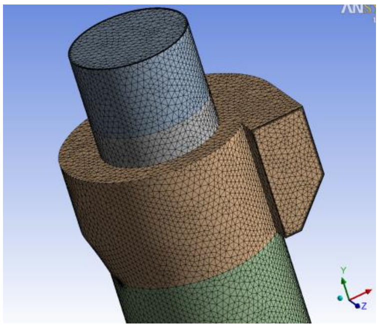

Fig.4 Meshed Model of Cyclone Separator

Single phase flow analysis

Pre- processing of single phase flow

Initially the analysis was performed by considering only single phase i.e. considering only air as the incoming fluid. The analysis was begun with importing of the meshing file in CFX-pre as *.msh* file. After importing this file each part was categorized into three domains namely-inlet, outlet and wall domain. Accordingly the boundary conditions like mass flow rate of incoming gas (36.546 kg/s) and total pressure(-6470.2Pa) were applied. The run was then defined by specifying the required parameters i.e. Number of iterations, specified blend factor etc. Specified Blend Factor allows us to set a blend factor between 0.0 and 1.0 for the advection scheme. A value of 0.0 is equivalent to using the first order advection scheme and is the most robust option whereas a value of 1.0 uses second order differencing for the advection terms scheme.

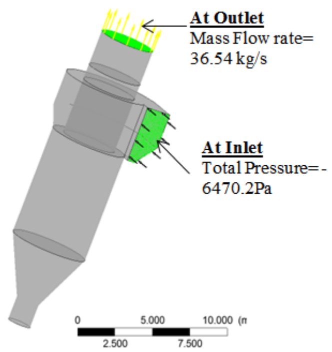

Fig.5 Boundary Conditions Applied on Actual Model in ANSYS CFX

Solver of single phase flow

To simulate the flow of fluid in cyclone separator, CFX solver was used. In CFX-solver, the imported file *.def* from CFX-pre which contains boundary conditions that were given previously was solved and the newly created result file was saved as *.res* file.

Post- processing of single phase flow

The results thus obtained from CFX-Solver were used to calculate the pressure drop. The results were analyzed in the form of pressure and velocity contours and velocity vectors. These results were plotted by taking vertical plane of cyclone separator. CFX–post also gives us the behavior of fluid inside cyclone separator as streamline flow.

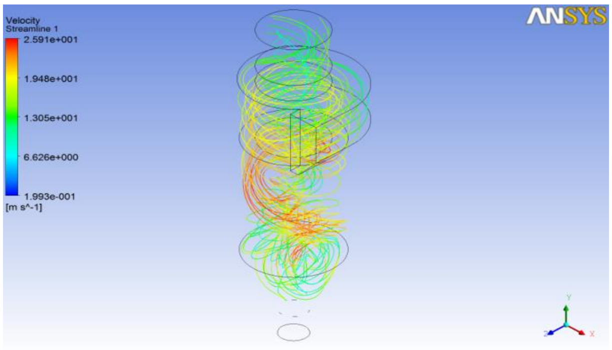

Fig.6 Streamline Flow of Actual Cyclone Separator

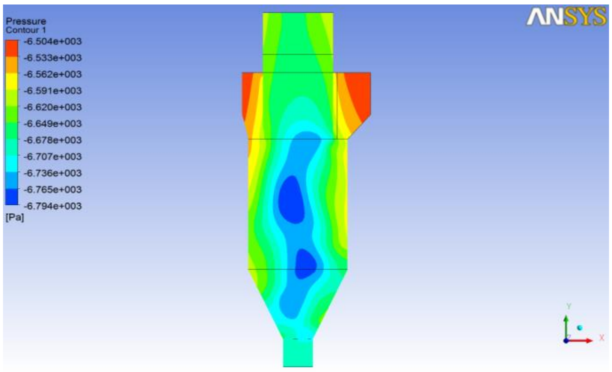

Fig.7 Pressure Contour of Actual Cyclone Separator

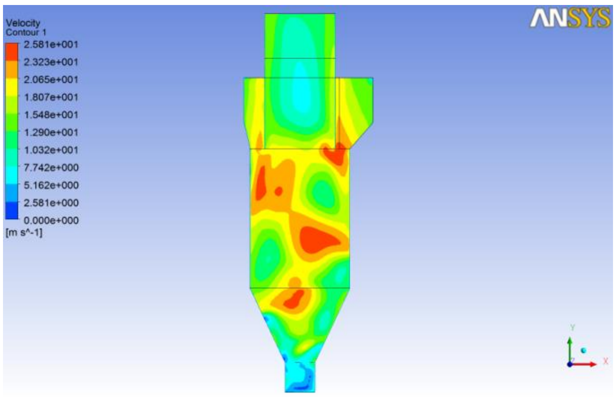

Fig.8 Velocity Contour of Actual Cyclone Separator

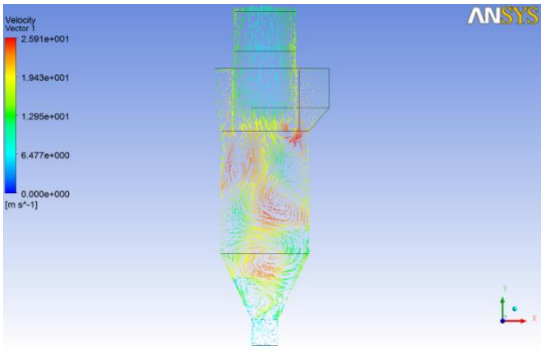

Fig.9 Velocity Vector of Actual Cyclone Separator

Multi-phase flow analysis

Pre- processing of multi-phase flow-

In this analysis, initially material properties of dust particles were specified like density, particle size, mass fraction etc. In multiphase both air and dust particles were considered. The mass flow rate was taken as 34.2 kg/sec. The morphology used was 'Particle transport solid'. The total number of dust particles entering the domain was taken as 5000. The restitution coefficients in parallel and perpendicular directions for walls were given. The maximum tracking time and distance were specified. Finally run was defined using the initial result file of single phase cyclone separator.

Solver of multi-phase flow-

The procedure of CFX-solver for multiphase is same as single phase.

Post- processing of multi-phase flow-

The results obtained were used to determine the dust collection efficiency of the cyclone separator. The number of dust particles leaving the domain was noted down. The dust collection efficiency was calculated using the expression as given below:

Efficiency

Efficiency can be calculated by

Collection Efficiency =

$$\frac{\left(\begin{array}{c}\text{Total No. of Particles Entered Domain} - \\ \text{No. of Particles Left Domain}\end{array}\right)}{\text{Total No. of Particles Entered Domain}}$$

The collection efficiency of cyclone separator is defined as the fraction of particles of a given size collected in the cyclone, compared to those of that size going into the cyclone. So, in this analysis total no. of particles entered domain are 5000 and in the result no. of particles which left domain are 2345. Therefore, collection efficiency is calculated as –

Collection Efficiency

$$= \frac{(5000-2345)}{5000} * 100 = 53.1\%$$

Pressure Drop

The pressure drop is obtained by taking difference of pressure present at inlet and pressure obtained at outlet.

Pressure Drop = Pressure at inlet – Pressure at outlet

Pressure Drop in this case = 105.971Pa

Experimental Validation

The validation estimates how accurately the user can apply the CFD code in simulating an environment problem in the real world. It gives the user the confirmation to use the CFD code. A CFD code may have solved the physical models that the user selects to describe the real world; however, the results may not be exact because the selected models do not represent the physical reality.

In this study, validation makes it possible to confirm that the assumed method can be successfully applied to visualize flow of fluid and solid blends in cyclones. The efficiency of actual cyclone separator by CFD analysis came out to be 53.1 % whereas the experimental value of it was 52 %. Thus, the method used is validated.

Modifications in the existing models and its analysis

Various modifications in the geometric parameters of the actual model were carried out. The same procedure of analysis was carried out on these modifications. The modifications carried out were-

a) Dip tube height increase and decrease by 30%.
b) Cylinder height decrease and increase by 25% keeping overall height constant.
c) Outlet tube dia. decrease by 30%.
d) Cylinder height decrease and increase by 25% & keeping cone dimensions constant.
e) Cylinder height increase and decrease by 25% & dip tube height increase by 30%.
f) Cone height increase and decrease by 25% & cylinder height constant.

Following are the modifications and corresponding are their results obtained on analysis.

Table 3 Summary of all results

Model No	Modifications	Pressure Drop (Pa)	Collection efficiency (%)
	Actual Cyclone Model	105.971	53.1
Modifications			
1.	Dip Tube Ht. Increase by 30%	103.46	56.22
2.	Dip Tube Ht. Decrease by 30%	91.09	48.86
3.	Cylinder Ht. Decrease by 25% Keeping Overall Height Constant	122.78	70.86
4.	**Cylinder Height Increased by 25% Keeping Overall Height Constant**	**134.90**	**77.26**
5.	Outlet Tube Dia. Decrease by	191.93	60.96

	30%		
6.	Cylinder Ht. Decrease by 25% &constant Cone Dimensions	132.20	70.06
7.	Cylinder Ht. Increase by 25% & constant Cone Dimensions	103.20	55.70
8.	Cone Ht. Increase by 25% &Cylinder Ht. Constant	100.20	65.96
9.	Cone Ht. Decrease by 25% &Cylinder Ht. Constant	125.00	53.94
10.	Cylinder Ht. Increase by 25% &Dip tube Ht. Increase by 30%	152.74	69.60
11.	Cylinder Ht. Decrease By 25% &Dip tube Ht. Increase By 30%	127.40	65.14

Results and discussion

From the results, we conclude that due to the shape of cyclone separator low pressure area is created inside the cyclone which forms the inner vortex. Due to this, a pressure difference is created between inlet and air outlet of cyclone. This enables the clean air to flow outside the cyclone separator.

The predictions of velocity field, pressure drop and particle separation efficiency from the CFD model were critically compared for 11 different modifications done on the cyclone separator. Out of these 11 modifications, the one having the highest efficiency i.e. the model with “cylinder height increased by 25% keeping overall height constant” was selected. The efficiency of the selected model is 77.26%and that of actual separator is 53.1%.It can be seen that there is about 24.16 % increase in the efficiency. Following is the bar graph representing the different modifications of the cyclone model and their corresponding collection efficiencies.it can be seen from the graph that model 4 i.e. the model with cylinder height increased by 25% keeping overall height constant shows the highest efficiency of 77.26%.

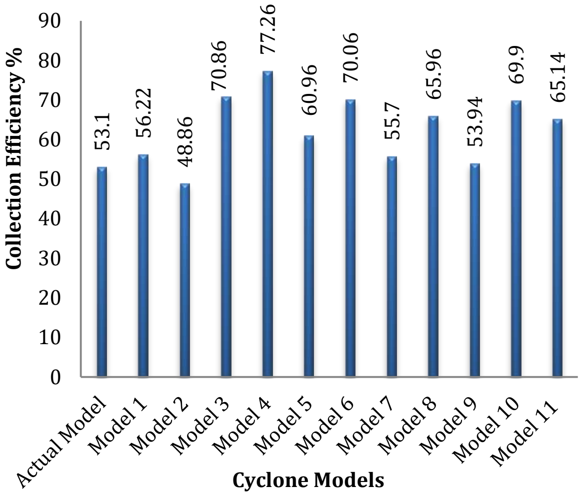

Fig.10 Bar Graph of Various Models and Their Efficiencies

Conclusions

1) Cyclone separators are used to separate dust particulates from air, gas or liquid stream. Among all the dust collectors, cyclone separator has proved to be the most efficient device. The overall performance of cyclone separator depends largely on the inner and outer vortex formed by air and dust particles respectively. This has necessitated efforts to concentrate on exploring the basic flow physics in the cyclone separator. The results indicate that the efficiency of cyclone separator depends upon collection of dust particles.
2) The completed validation makes it possible to confirm that the assumed method can be successfully applied to visualize flow of fluid and solid blends in cyclones. The efficiency of actual cyclone separator by CFD analysis came out to be 53.1 % whereas the experimental value of it was 52 %. Thus, the method used is validated.
3) Thus a promising conclusion can be drawn from these research is that CFD analysis could be used to predict the overall performance of the cyclone separator, as a cost effective tool in order to obtain clean air at the outlet of the separator.

References

Bharath Raj Reddy Dere, G. Mahesh Babu, A .Divya Sree and S. Rajiv Rao, Design and Analysis of Cyclone Separator, International Journal of Engineering Research & Technology (IJERT) Vol. 3 Issue 8, August - 2014 IJERT ISSN: 2278-0181.

Chen, Q. and Srebric, J. 2002. “A procedure for verification, validation, and reporting of indoor environment CFD analyses,” HVAC&R Research, 8(2), 201-216.

John D. Anderson, Book on Computational Fluid Dynamics-The Basics with Applications.

Jolius Gimbun, Thomas S. Y. Choong, T. G. Chuah, and A. Fakhru’l-Razi, Paper onA CFD Study on the Prediction of Cyclone Collection Efficiency.

Marek WASILEWSKI, Jerzy DUDA, Application of Computational Fluid Dynamics to optimization of cyclone dust separators operated in the cement industry, XIV Conference Environmental CHEMIK 2013, 67, 10, 985–994.

Pandya Devel, Report on A Low Cost Micro Scale Cyclone Separator- Design And Computational Fluid Dynamics Analysis, The University Of Texas At Arlington, August 2010.

Shepherd, C. B., and C. E. Lapple. 1939. Flow patterns and pressure drop in cyclone dust collectors, *Ind. and Eng. Chem.,* Vol. 31(8): 972-984.

W. B. Faulkner, B. W. Shaw, Efficiency And Pressure Drop Of Cyclones Across Range Of Inlet Velocities, 2006 *American Society of Agricultural and Biological Engineers* ISSN 0883–8542, Vol. 22(1): 155-161.