

Computational Fluid Analysis(CFD) of Dust Collector using ANSYS

J.Srinivas^{*1}, J.Murugesan², M.Saravanan³, M. Raghupathi⁴

^{1,2,3,4}Department of Mechanical Engineering, Panimalar Engineering College, Chennai, Tamilnadu
jayaramansrinivas@gmail.com^{*1}, murugesan_kpm@yahoo.com², saravanmech2005@gmail.com³, ragu_ya@yahoo.co.in⁴

Abstract:

Cyclone separator is a machine for removing dust particles from an air or liquid stream, without the use of filters, through vortex separation. Rotational effects and gravity are used to separate mixtures of solids and fluids. This machine can also be used to separate fine droplets of liquid from a gaseous stream. The cyclone dust collector set up is to be modelled & same to be analysed by CFD technique. The gas-solids cyclone separator is industrial equipment that has been widely used. Due to its industrial relevance, a large number of computational studies have been reported in the literature aimed at understanding and predicting the performance of cyclones in terms of pressure and velocity variation. One of the approaches is to simulate the gas-particle flow field in a cyclone by computational fluid dynamics (CFD). Cyclones have often been regarded as low-efficiency collectors. However, efficiency varies greatly with particle size and cyclone design. Advanced design work has greatly improved cyclone performance. This paper have discussed the design parameters required to construct a high performing cyclone through the application of the classical cyclone design, However, the pressure drop in this design does not consider any vertical dimensions as contributing to pressure drop, This is a misleading in that a tall cyclone would have the same pressure drop as a short one as long as cyclone inlets and outlets dimensions and inlet velocities are the same.

Keywords: Dust collector, CFD, Cyclone separator

1. INTRODUCTION

A dust collector is a system used to enhance the quality of air released from industrial and commercial processes by collecting dust and other impurities from air or gas. Designed to handle high-volume dust loads, a dust collector system consists of a blower, dust filter, a filter-cleaning system, and a dust receptacle or dust removal system. It is distinguished from air cleaners, which use disposable filters to remove dust. A high speed rotating (air) flow is established within a cylindrical or conical container called a cyclone. Air flows in a helical pattern, beginning at the top (wide end) of the cyclone and ending at the bottom (narrow) end before exiting the cyclone in a straight stream through the center of the cyclone and out the top. Larger (denser) particles in the rotating stream have too much inertia to follow the tight curve of the stream, and strike the outside wall, then fall to the bottom of the cyclone where they can be removed. In a conical system, as the rotating flow moves towards the narrow end of the cyclone, the rotational radius of the stream is reduced, thus separating smaller and smaller particles. The cyclone geometry together with volumetric flow rate. Defines the cut point of the cyclone. This is the size of particle that will be removed from the stream with 50% efficiency. Particles larger than the cut point will be removed with a greater efficiency and smaller particles with a lower efficiency as they separate with more difficulty or can be subject to re-entrainment when the air vortex reverses direction to move in direction of the outlet.

Kashan Bashir (2015), To design a cyclone abatement system for particulate control, it is necessary to accurately estimate cyclone performance. In this cyclone study, new theoretical methods for computing travel distance, numbers of turns and cyclone pressure drop have been developed. The flow pattern and cyclone dimensions determine the travel distance in a cyclone. The number of turns was calculated based on this

travel distance. The new theoretical analysis of cyclone pressure drop was tested against measured data at different inlet velocities and gave excellent agreement. The results show that cyclone pressure drop varies with the inlet velocity, but not with cyclone diameter. Cyclone cut-points for different dusts were traced from measured cyclone overall collection efficiencies and the theoretical model for calculating cyclone overall efficiency. The cut-point correction models 2D2D cyclones were developed through regression fit from traced and theoretical cut-points Diameter. Experimental results indicate that optimal cyclone design velocities, which are for 2D cyclones, should be determined based on standard air density. It is important to consider the air density effect on cyclone performance in the design of cyclone abatement systems. The tangential inlet generates the swirling motion of the gas stream, which forces particles toward the outer wall where they spiral in the downward direction. Eventually the particles are collected in the dustbin located at the bottom of the conical section of the cyclone body. The cleaned gas leaves through the exit pipe at the top.

In Pakistan it has been installed in many industries its Main modes of operation is similar to collect the particles but the collection of particles and method of collection are Distinguish. Cement Sector is one of advance sector3 application for Cyclone Separator. DG cement in one of those Cement Manufacturing Plant where Cyclone working efficiently.

Muhammad I. Taiwo et al (2016), Design and analysis of cyclone dust separator, American Journal of Engineering Research, Cyclones have often been regarded as low-efficiency collectors. However, efficiency varies greatly with particle size and cyclone design. Advanced design work has greatly improved cyclone performance. This paper have discussed the design parameters required to construct a high performing cyclone through the application of the classical cyclone design,

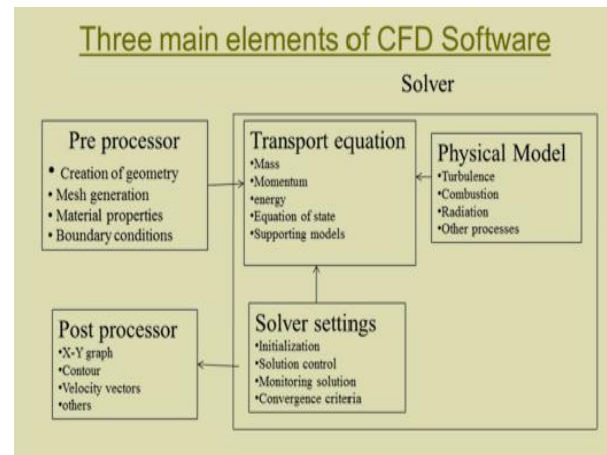
However, the pressure drop in this design does not consider any vertical dimensions as contributing to pressure drop, This is a misleading in that a tall cyclone would have the same pressure drop as a short one as long as cyclone inlets and outlets dimensions and inlet velocities are the same.

Texas cyclone design model was used to obtain an accurate pressure drop and sizing of cyclone, The Texas cyclone approach to design cyclones was to initially determine optimum inlet velocities (design velocities) for different cyclone designs, hence using the inlets velocity a cyclone dimension can be determined. • DevalPandya, M.S (2010), A low cost micro scale cyclone separator-design and computational ,fluid dynamics analysis ,Micro particle separation process has a variety of application varying from application in biological and biomedical industries for analysis and diagnosis, in biogas manufacturing to separate phases as well as in defence sector for detection of biological weapons like anthrax. Available electrical, magnetic, acoustic and various other methods are either very costly or not portable. The proposed design of micro scale cyclone separator is low cost as well as portable and easy to manufacture. Huge cyclone separators 4 are widely used in various industries since decades but due to lack of research in micro scale cyclones no direct and sufficient data is available.

This research attempts to develop a micro scale cyclone separator and study the effect of parameters like inlet velocity on pressure drop and collection efficiency in a micro scale cyclone separator. It further studies the effect of particle size on collection efficiency through Computational Fluid Dynamics (CFD) approach. CFD analysis has been proved very efficient for v calculations in larger cyclones and hence is used as a tool in this study as well, though experimental verification is recommended. Computational experiments were performed using FLUENT. The results obtained are compared with various empirical relations developed for huge cyclone separators and similarities and dissimilarities in trends are analysed. Finally a multi-cyclone model is proposed to obtain higher collection efficiency

2. COMPUTATIONAL FLUID DYNAMICS APPROACH

CFD is a method to numerically calculate heat transfer and fluid flow. Currently, its main application is as an engineering method, to provide data that is complementary to theoretical and experimental data. This is mainly the domain of commercially available codes and in-house codes at large companies. CFD can also be used for purely scientific studies, e.g. into the fundamentals of turbulence. This is more common in academic institutions and government research laboratories. Codes are usually developed to specifically study a certain problem. Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. The result of CFD analyses is relevant engineering data used in Conceptual studies of new designs, detailed product development, Troubleshooting, Redesign.



CFD analysis complements testing and experimentation. Reduces the total effort required in the laboratory. During this process you will amongst other things clean the CAD model and create the mesh. A mesh is a cluster of small cells representing an object. For Finite Element Analysis, you will usually create a mesh of the actual model that you want to simulate. In Computational Fluid Dynamics simulations however, you will most likely create a mesh around the actual model since you want to, for example analyse the aerodynamic behaviour of a race car. Mastering the art of pre-processing is the first step towards becoming an expert in engineering simulation. We at Sim Scale make this step easy for you with the following resources! Read them and get a Michelin star in engineering simulation.

2.1 PRE-PROCESSING:

Starting with something new is not always easy. Especially, if it is something as complex as engineering simulation. As always, before starting with the actual task (in this case the simulation) it is crucial to prepare for the task in order to achieve the result you aimed for.

2.2 MODELING:

MODELLING OF WING USING CATIA: Aerofoil coordinate data's are collected from UIUC aerofoil database. This aerofoil coordinates are inserted in MS Excel. This MS-Excel coordinates are imported to CATIA using macro's option to create a aerofoil section. The pressure-based coupled solver (PBCS) is applicable for most single phase flows, and yields superior performance to the standard pressure-based solver.

2.3 MESHING:

The partial differential equations that govern fluid flow and heat transfer are not usually amenable to analytical solutions, except for very simple cases. Therefore, in order to analyze fluid flows, flow domains are split into smaller sub-domains (made up of geometric primitives like hexahedra and tetrahedral in 3D and quadrilaterals and triangles in 2D). The governing equations are then discretized and solved inside each of these sub-domains.

2.3.1 MESH QUALITY CONSIDERATIONS FOR FLUENT:

FLUENT requires high quality mesh to avoid numerical diffusion. Several Mesh Quality Metrics are involved in order to quantify the quality; however the skewness is the primary metric however the skewness is the primary metric. The aspect ratio and cell size change mesh metrics are also very important. In worst scenarios and depending on the solver used (density based or Pressure based) FLUENT can tolerate poor mesh quality. However some Applications may require higher mesh quality, resolution and good mesh Distribution. The location of poor quality elements helps determine their effect. Some overall mesh quality metrics may be obtained in ANSYS Meshing under the Statistics object. Additional mesh quality metrics may

be retrieved in FLUENT GUI under Mesh/Info/Quality from the menu, or using the TUI commands 'mesh/ quality '

2.3.2 MESH QUALITY REQUIREMENTS FOR FLUENT:

The most important mesh metrics for Fluent are, Skew ness, Aspect Ratio Cell Size Change (not implemented in ANSYS Meshing). For all/most applications, Poor mesh quality may lead to inaccurate For all/most applications, For Skew ness: For Hexa, Tri and Quad it should be less than 0.8 For tetrahedral: it should be less than 0.9 lead to inaccurate solution and/or slow convergence For tetrahedral: it should be less than 0.9. For Aspect Ratio: It should be less than 40, but this depends on the flow characteristics. Some applications may require even lower Skew ness than the suggested value the flow characteristics More than 50 may be tolerated at the inflation layers. For Cell Size Change: suggested value A-8 It should be between 1 and 2.

2.3.3 HIGH SKEWNESS VALUES ARE NOT RECOMMENDED:

Generally try to keep maximum skew ness of volume mesh < 0.95. However this value is strongly related to type of physics and the location however this value is strongly related to type of physics and the location of the cell. FLUENT reports negative cell volumes if volume mesh contains degenerate cells. Classification of the mesh quality metrics based on skew ness:

* In some circumstances the pressure based solver in Fluent can handle meshes containing a small percentage of cells with skew ness ~0.98

2.3.4 MESHING USING ANSYS MESH TOOL (ICEM CFD):

Now create a mesh using ICEM CFD software. Unlike the structured approach for meshing, the unstructured meshing algorithm is more or less an optimization problem, wherein, it is required to fill-in a given space (with curvilinear boundaries) with standard shapes (e.g., triangle, quadrilaterals - 2D; tetrahedral, hexahedra's, polyhedral, prisms, pyramids - 3D) which have constraints on their size. The basic algorithms employed for doing unstructured meshing are:

- Octree (easiest from the user's perspective; robust but least control over the final cell count which is usually the highest)
- Delaunay (better control over the final cell count but may have sudden jumps in the size of the elements)
- Advancing front (performs very smooth transition of the element sizes and may result in quite accurate but high cell count) Using Octree,
- Perform volume meshing.
- Improve the quality of the volume mesh using Edit Mesh options.
- Create prism layers for boundary layer near the walls.
- Improve the total mesh quality using Edit Mesh options.
- And creating a named section for inlet, outlet, wing wall and test section wall.
- Creating a prism layer near the wing wall to capture the boundary layer effect using the inflation command.

2.4 TURBULENCE MODELS:

Turbulent flows are characterised by velocity fields which fluctuate rapidly both in space and time. Since these fluctuations occur over several orders of magnitude it is computationally very expensive to construct a grid which directly simulates both the small scale and high frequency fluctuations for problems of practical engineering significance. Two methods can be used to eliminate the need to resolve these small scales and high frequencies: Reynolds Averaging and Filtering.

2.4.1 K-Omega Model:

One formulation is the near wall treatment for ω of the advantages of the k- low-Reynolds number computations Here "low-Reynolds" refers to the turbulent Reynolds number, which is low in the viscous sub-layer, not the device Reynolds number In other words "low-Reynolds number computations" means the near wall mesh is fine enough to resolve the laminar (viscous) part of the boundary layer model only requires ω which is very close to the wall A low-Reynolds number k- $y^+ \leq 2$ If a low-Re k- ω model were available, it would require a much small y^+ In industrial flows, even $y^+ \leq 2$ cannot be guaranteed in most applications and for this reason, a new automatic near wall treatment was models ω models The Automatic wall treatment for the k- ω developed for the k- switches between a low-Reynolds number formulation (i.e. direct resolution of the boundary layer) at low y^+ values and a wall function approach at higher y^+ values This lets you take advantage of a fine near-wall mesh when present

2.4.2 SST MODEL:

Shear Stress Transport (SST) Model:

model and has the ω The SST model is based on the k- same automatic wall treatment It accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount ω of flow separation This is a good default choice When using the SST or k- models y^+ should be <300 so that the wall function approach is valid This will not take advantage of the low-Reynolds formulation, which is necessary for accurate separation prediction However, the model can still be used on these coarser near-wall mesh and produce valid results, within the limitations of the wall functions

2.5 SOLVER SETTINGS:

FLUENT Solver can be used for this project the flow chart give the brief explanation to use this solver

2.5.1 PRESSURE-BASED SOLVER:

The pressure-based solver is applicable for a wide range of flow regimes from low speed incompressible flow to high-speed compressible flow. Requires less memory (storage). Allows flexibility in the solution procedure allows flexibility in the solution procedure. The pressure-based coupled solver (PBCS) is applicable for most single phase flows, and yields superior performance to the standard pressure-based solver. Not available for multiphase (Eulerian), periodic mass-flow and NITA cases. Requires 1.5–2 times more memory than the segregated solver. The density-based coupled solver (DBCS) is applicable when there is a strong the density-based coupled solver (DBCS) is applicable when there is a strong coupling, or interdependence, between density, energy, momentum, and/or species. High speed compressible flow with combustion, hypersonic flows, shock interactions. The implicit option is generally preferred over explicit since it has a very strict limit on time step size The explicit approach is used for cases where the characteristic time scale of the flow is on the same order as the acoustic time scale (e.g. propagation of high-Ma shockwaves)

2.5.2 STANDARD INITIALIZATION:

The solver works in an iterative manner. Therefore before the very first iteration, a value must exist for every quantity in every grid cell. Setting this value is called 'Initialization' The more realistic the value, the better (quicker) convergence will be (quicker) convergence will be. It is possible to mark some cells (using the adaption tools or having different cell adaption tools, or having different cell zones) and patch specific values into certain cells. Free jet flows (high velocity for jet) Combustion problems (high temperature region to initialize reaction)

2.5.3 CONVERGENCE:

The solver should be given sufficient iterations such that the problem is converged at convergence, the following should be satisfied the solution no longer changes with subsequent iterations. Overall mass, momentum, energy, and scalar balances are achieved. All equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance Monitoring convergence using residual history generally; a decrease in residuals by three orders of magnitude indicates at least qualitative convergence. At this point, the major flow features should be established. Scaled energy residual should decrease to 10^{-6} (for the pressure-based solver). Scaled species residual may need to decrease to 10^{-5} to achieve species balance. Tightening the Convergence Tolerance If solution monitors indicate that the solution is converged, but the solution is still changing or has a large mass/heat imbalance, this clearly indicates the solution is not yet converged. In this case, you need to reduce values of Convergence Criterion or disable Check Convergence in the Residual Monitors panel. Continue iterations until the solution converges. Selecting Non Convergence Criterion disables convergence checking for all equations. Convergence Difficulties Sometimes running for further iterations is not the answer: Either the solution is diverging (aka “blowing up”) Or the residuals are ‘stuck’ with a large imbalance still remaining. Troubleshooting Continuity equation convergence trouble affects convergence of all equations. Ensure that the problem is well-posed.

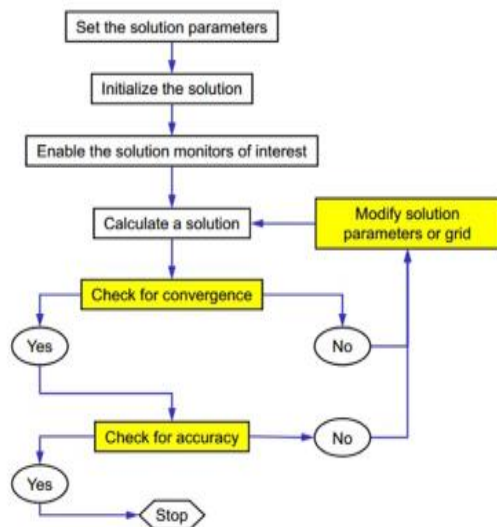


Fig.1 Solver setting

Compute an initial solution using a first-order discretisation scheme. Alter the under-relaxation or Courant Numbers Check the mesh quality. It can only take one very skewed grid cell to prevent the entire solution converging [This is why you should ALWAYS check the mesh quality before spending time with the solver] As a numerical representation of a physical system, the CFD solution imbalances will never be exactly zero. However, the imbalances should be sufficiently small before considering the solution converged. As a good practice, aim for solution imbalances of less than 1% as a starting point. Note that more sensitive applications may require tighter convergence. In a steady state analysis, the solution field should not change iteration to iteration for an analysis to be deemed converged. Monitoring integrated quantities such as force, drag, or average temperature can help the user judge when his or her analysis has reached this point.

3. FABRICATION OF BODY:

- Aluminium sheet metal of length 0.65 m and breadth 0.35 m is rolled along the length to form a cylindrical surface with sleeves.
- The joints of this cylinder are riveted along the sleeve.
- To this cylinder a conical surface is attached to the bottom portion.
- A circular cover accommodating the suction duct is placed on the top portion.
- Sealing agents like M-seal TM is used to seal-off the small gaps formed.

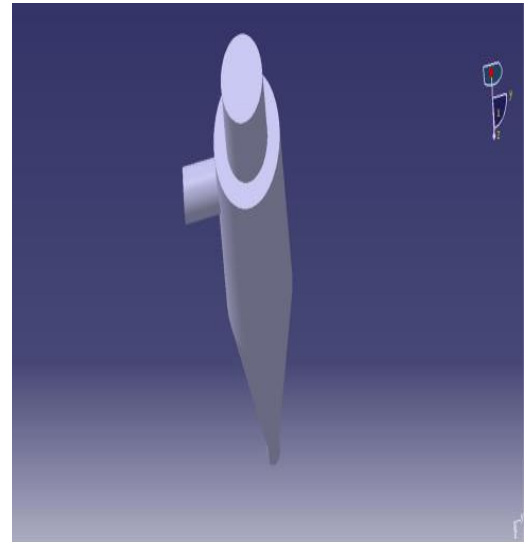


Fig.2 3D model of cyclone separator with CFD domain

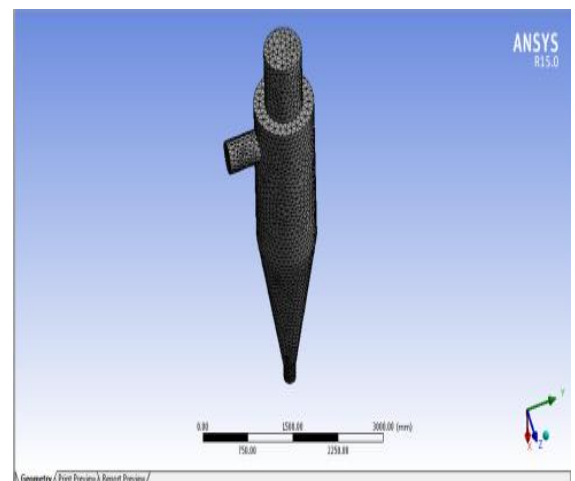


Fig.3 Mesh image of cyclone separator in CFD domain

3.1 FABRICATION OF HOUSING SETUP:

- A stand with cuboidal skeleton is prepared using rectangular MS- rods.
- The rods are welded using arc welding.
- Required number of holes are drilled on the rims.
- On these the vacuum pump is mounted using nuts & bolts.

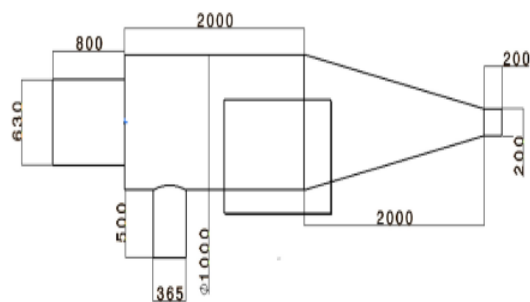


Fig.4 Fabrication of housing set-up



Fig.5 Fabrication of body

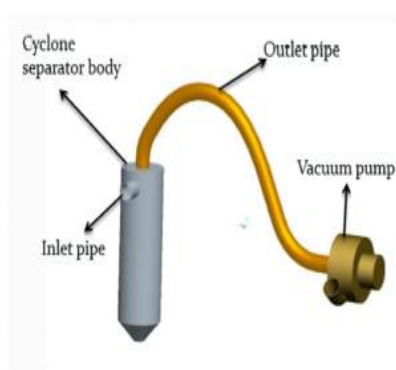


Fig.6 Set-up diagram

3.2 DESIGN OF CYCLONE SEPARATOR

According to Lapple's expression,

$$d_p = \text{particle cut size} = 1.5 \times 10^{-6} \text{ m.}$$

$$\mu = \text{dynamic viscosity of air} = 18.63 \times 10^{-6} \text{ N-s/m}^2.$$

$$N_t = \text{effective number of turns made by gas in cyclone} = 10 \text{ (assume).}$$

$$C = \text{cyclone inlet velocity} = 20 \text{ m/s.}$$

$$\rho_p = \text{density of sand particles} = 1442 \text{ kg/m}^3.$$

$$B = \text{inlet width in m.}$$

On substitution of values,

$$B \approx 0.0525 \text{ m}$$

The radial force exerted on the particles is,

$$F_r = \text{separating force in N.}$$

$$m_p = \text{mass of the particles} = 1.6 \times 10^{-3} \text{ kg.}$$

$$c_t = \text{tangential velocity} = 30 \text{ m/s.}$$

$$R = \text{radius of rotation} = \text{radius of cyclone body} = 0.063 \text{ m.}$$

On substitution of values,

$$F_r \approx 22.85 \text{ N}$$

The frictional resistance shown by the particle is given by,

Where,

$$F_f = \text{the frictional resistance to flow in N.}$$

$$K = \text{proportionality constant (dimensionless)} = 100 \text{ (assume).}$$

$$\mu = \text{gas viscosity} = 18.63 \times 10^{-6} \text{ N-s/m.}$$

$$d = \text{diameter of particle} = 1.5 \times 10^{-6} \text{ m.}$$

$$u = \text{particle velocity with respect to gas} = 20 \text{ m/s.}$$

On substitution of values,

$$F_f \approx 5.58 \times 10^{-25} \text{ N}$$

TABLE. 1 VACUUM SPECIFICATION

Type	: rotary vane pump
No. of stages	: 1
Air discharge rate	: 120 m ³ /hr.
Inlet diameter	: 40 mm.
Outlet diameter	: 46 mm.
Speed	: 1875 rpm.
Voltage	: 240 V, 50 Hz

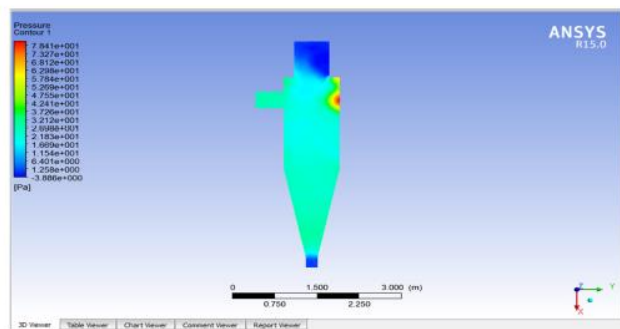
3.4 SOLVER SET-UP

SOLVER TYPE	Pressure based
TURBULANCE MODEL	SST K-OMEGA
MATERIAL	Incompressible air
VELOCITY INLET	Velocity inlet- 33.33 M/S Normal To Boundary
PRESSURE OUTLET	pressure value set as zero
WING WALL	No slip condition and stationary wall
WALL	Specified shear is zero in all components
CELL ZONE CONDITION	Air material is choose for calculation
REFERENCE VALUE	Compute fluid from inlet zone
SOLUTION METHOD	SIMPLE(Semi-Implicit Method for Pressure-Linked Equations)
MONITORS	PLOT RESIDUALS LIFT AND DRAG
INITIALIZATION	Standard initialization (compute from inlet)

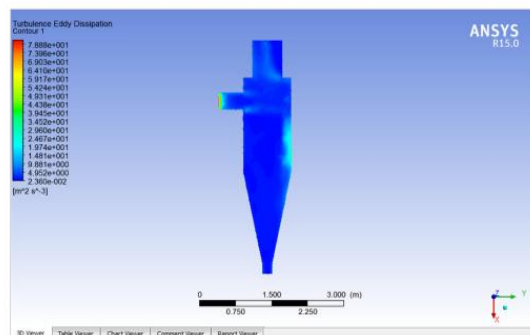
4. RESULTS AND DISCUSSION

4.1 PRESSURE CONTOUR

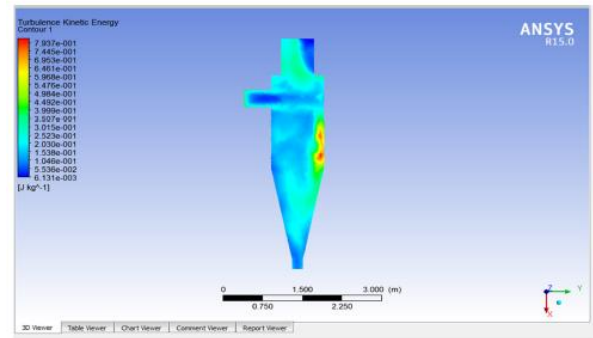
PRESSURE CONTOUR:



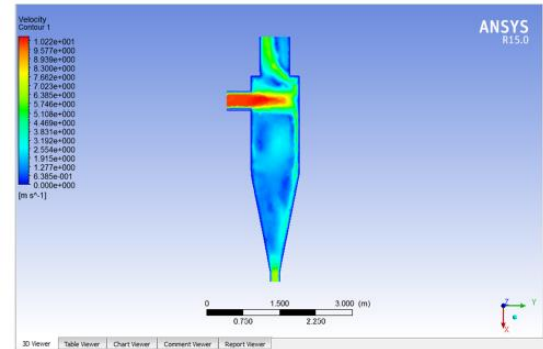
TURBULENCE EDDY DISSIPATION:



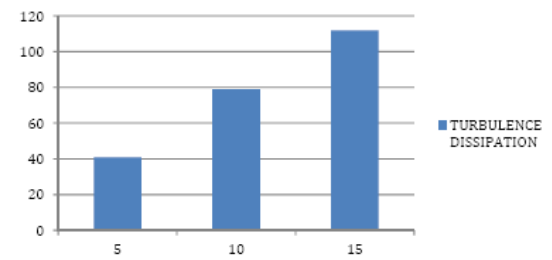
TURBULENCE KINETIC ENERGY:



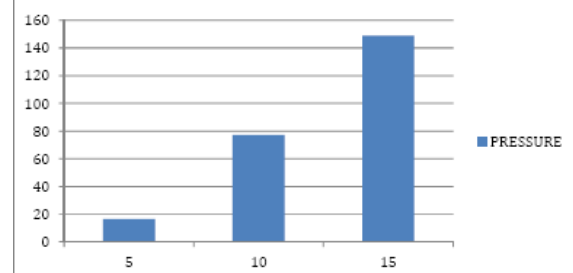
VELOCITY CONTOUR:



TURBULENCE DISSIPATION



PRESSURE



TURBULENCE DISSIPATION VALUES

INLET VELOCITY(m/s)	TURBULENCE DISSIPATION(m ² s ⁻³)
5	40.77
10	78.88
15	111.9

PRESSURE CONTOUR VALUES

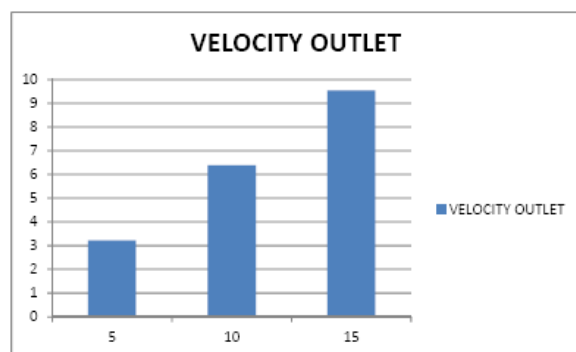
INLET VELOCITY(m/s)	PRESSURE(Pa)
5	16.3965
10	77.152
15	148.897

TURBULENT KINETIC ENERGY VALUES

INLET VELOCITY(m/s)	TURBULENCE KINETIC ENERGY(J/Kg)
5	0.97
10	0.7939
15	3.438

VELOCITY OUTLET VALUES

INLET VELOCITY(m/s)	VELOCITY OUTLET(m/s)
5	3.2111
10	6.38
15	9.52



5. CONCLUSION

The cyclone separator which was designed and fabricated has been working properly. It can separate particles up to size of 1.5×10^{-6} m diameter effectively with required amount of pressure created by the vacuum pump. The use of cyclonic separator has been demonstrated to be a cheap and reasonable effective method for controlling the particulate emission of a diesel engine without loss of fuel economy. In the current study, we have designed a cyclonic separator by optimizing conditions like collection efficiency, low pressure drop across the cyclone and vortex finder size and tested its efficiency in removing the particulate matter.

REFERENCES

- Manual of Steel Construction, 8th Edition, 2nd revised printing, American Institute of Steel Construction, 1987, chapter 1 page 1-5.
- "A Text Book of Strength of Materials", 4th Edition, Dr. R.K.BANSAL, Laxmi publications Ltd.
- "A Text Book of Manufacturing technology", 6th Edition, K.HAJRA CHOUDARY.
- Design and development of a cyclonic device for diesel engine by P.SyedaNooriBanu, International Journal of Engineering Science and Technology (IJEST)
- Combustion Modeling in CI Engine with Biodiesel as an Alternative Fuel - Study: A Review, AA M.V.Natarajan, S.Manikandan, K.V.Nagarajan, J.Srinivas, Journal of Applied Science and Computations 6 (4), 254-259
- Modeling the Correlation Between Cutting and Process Parameters in Machining of Ni Based UDIMET Alloy Using Artificial Neural Network, PS T. Mayavan, S. Thamizh Selvan, J. Srinivas, International Journal of Engineering Research & Technology (IJERT) 6 (2), 1-5
- Mohan Gift MD, Selvakumar J, John Alexis S, 'Determination of Decisive Parameters in the Crack Propagation Analysis of an Adhesive Joint', (2015), Indian Journal of Science and Technology, vol.8, pp.35-38.
- Mohan Gift MD, Selvakumar J, John Alexis S, 'Fracture studies of an Adhesive Joint involving composition alteration', (2015), Journal of Chemical and Pharmaceutical Sciences, pp.269-273.
- Mohan Gift MD, Selvakumar J, John Alexis S, 'A review on the Cohesive Zone Models for crack propagation analysis', (2013), Elixir Mechanical Engineering, vol.55, pp.12760-12763.
- Mohan Gift MD, Selvakumar J, John Alexis S, 'Fracture Analysis of an Adhesive Joint using Double Cantilever Beam(DCB) test method', (2014), International Journal of Applied Engineering Research, vol.9(22), pp. 17719-17730.