International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 8, August - 2014

Design and Analysis of Cyclone Seperator

Bharath Raj Reddy Dere[1]
[1]Mechanical department
Chaitanya Bharathi Institute Of Technology
Hyderabad,India.

G. Mahesh Babu[2],
[2]Mechanical department
Chaitanya Bharathi Institute Of Technology
Hyderabad,India

A . Divya Sree[3],
[3]Mechanical department
Chaitanya Bharathi Institute Of Technology
Hyderabad,India.
Email

S. Rajiv Rao[4]
[4]Mechanical department
Chaitanya Bharathi Institute Of Technology
Hyderabad,India.

***ABSTRACT:* 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).**

The cyclone performance parameters are governed by many operational parameters (e.g., the gas flow rate and temperature) and geometrical parameters. This study focuses only on the effect of the geometrical parameters on the flow field pattern and performance of the tangential inlet cyclone separators using, CFD approach. The objective of this study is three-fold. First, to study the optimized stairmand's design by understanding the pressure and velocity variations. Second, to study the performance of a cyclone separator by varying its geometrical parameters. Third, to study the performance of a cyclone separator by varying its flow temperatures. Finally to study the performance of a cyclone separator with collector and compare to that without collector. Four geometrical factors have significant effects on the cyclone performance viz., the inlet width, the inlet height and the cyclone total height. There are strong interactions between the effect of inlet dimensions and the vortex finder diameter on the cyclone performance. In this study, inlet geometry, cone- tip diameter, cone height is taken for analysis.

The modeling of the cyclonic flow by computational fluid dynamics (CFD) simulation has been reported. The effect of the cone tip-diameter, cone height and inlet geometry in the flow field and performance of cyclone separators was investigated because of the discrepancies and uncertainties in the literature about its influence. The flow field pattern has been simulated and analyzed with the aid of velocity components and static pressure contour plots. The CFD model was used to predict the pressure and velocity variations of cyclone geometries based on Stairmand's high efficiency design.

1. MODELLING

1.1 THE STAIRMAND OPTIMIZED DESIGN

Stairmand's conducted so many experiments on the cyclone separator and finally developed the optimized geometrical ratios. By considering this geometric ratio's the modeling of the cyclone done in solid works.

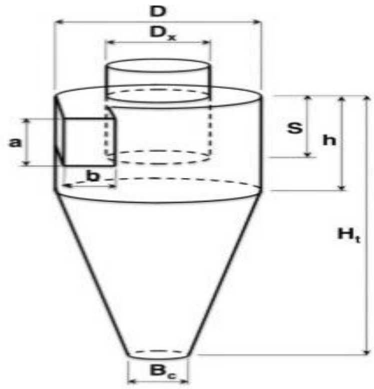


Figure 1.1.1: cyclone geometry in solid works

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

1.2. MODELLING IN SOLID WORKS

To design the cyclone the diameter (D) is considered as 20 mm.

Step 1: Draw the sketch according to the stairmand's ratios.

Step 2: specify the dimensions.

Step 3: Revolve the sketch 360 degrees and give thickness as 0.1mm.

Step 4: to get the inlet, draw a rectangle on the part which is developed by revolving.

Step 5: extrude the rectangle along the Z-axis (L=D).

Step 6: draw a rectangle on the extruded plan to get the hallow inlet. Select the rectangle and give extrude cut throughout the extruded rectangle.

Step 7: save the geometry.

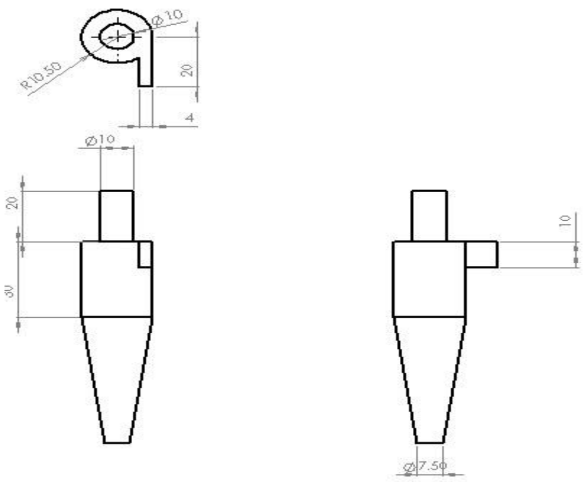


Fig 1.2.1 shows the dimensions of the cyclone separator

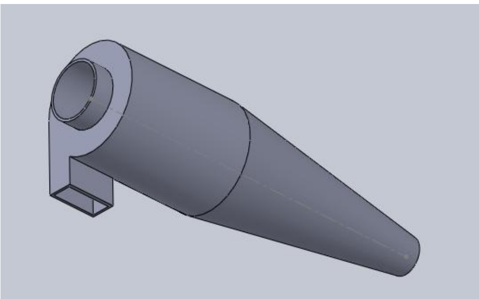


Fig.1.2.2 cyclone separator design.

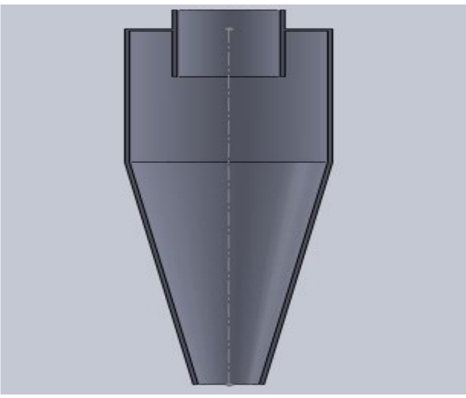


Fig.1.2.3 Half sectional view of cyclone.

2. GOVERNING EQUATIONS

2.1 CONTINUITY EQUATION

The continuity equation describes the conservation of mass.

Consider a differential control volume

Let ‘ρ’ be the density of fluid, ‘u’ be the x-component, ‘m’ be the mass of fluid element

$$[m = \partial/\partial x(\rho\, dx\, dy\, dz)]$$

a= ρ (density) b=u (velocity)

a’= $\rho + (\partial \rho / \partial x) dx$ b’=u+ $(\partial u / \partial x) dx$

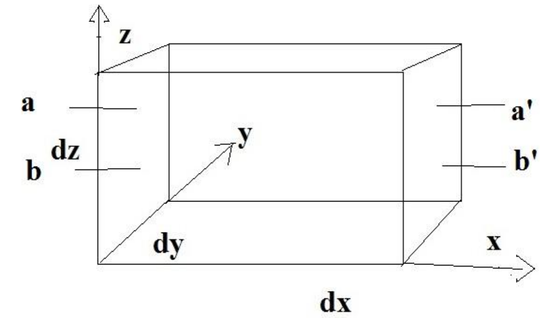


fig.4.1.1 fluid particle differential control volume

Rate of change of mass differential volume

$= \partial/\partial t(\rho\, dx\, dy\, dz)$

$$\text{Mass efflux in x-direction} = \partial/\partial x(\rho u)\, dx\, dy\, dz$$

$$\text{Mass efflux in x-direction} = \partial/\partial y(\rho v)\, dx\, dy\, dz$$

$$\text{Mass efflux in x-direction} = \partial/\partial z(\rho w)\, dx\, dy\, dz$$

$$\text{Since, Rate of accumulation} + \text{Mass efflux} = 0$$

$$\partial/\partial t(\rho\, dx\, dy\, dz) + \partial/\partial x(\rho u)\, dx\, dy\, dz +$$

$$\partial/\partial y(\rho v)\, dx\, dy\, dz + \partial/\partial z(\rho w)\, dx\, dy\, dz = 0$$

$$\partial \rho/\partial t + \partial/\partial x(\rho u) + \partial/\partial y(\rho v) + \partial/\partial z(\rho w) = 0$$

$$[\partial \rho/\partial t + \nabla.(\rho \overline{V}) = 0]$$

2.2 ENERGY EQUATION

The energy equation can be derived by applyin g I - law of thermodynamics to a moving fluid element.

Rate of change of energy inside the fluid element = Net flux of heat into fluid element + rate of work done on fluid element due to body and surface forces

$dE/dt = dQ/dt + dW/dt$

$[\dot{E} = Q + \dot{W}]$

$\dot{E}$ = Total energy of fluid per unit mass

$= e + 1/2\,(u^2 + v^2)$

Rate of change total energy of fluid element per unit mass = $D/Dt\,[e + 1/2\,(u^2 + v^2)]$

Rate of change total energy of fluid element

$= D/Dt\,[e + 1/2\,(u^2 + v^2)]?dxdy$

Considering heat transfer across surface due temperature gradient (conduction)

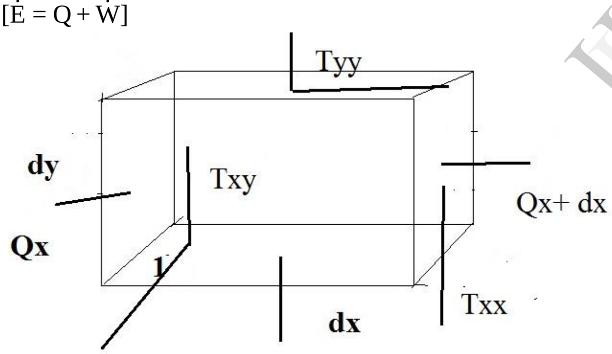
$$Q = K[\partial^2T/\partial x^2 + \partial^2T/?\partial y^2]\,dxdy$$

Rate of work done = Force x velocity

Total work done = (work done by body forces) + (work done by surface forces)

Total rate of work done $(\dot{W}) = [ufx + vfy + \partial/\partial x(u\sigma xx + v\sigma xy) + \partial/\partial y(v\sigma yy + u\sigma yx)]\,dxdy$

$[\dot{E} = Q + \dot{W}]$



$$\rho[De/Dt + 1/2\ D/Dt\,(u^2 + v^2)]dxdy = K[\partial^2T/\partial x^2 + \partial^2T/\partial y^2]\,dxdy + [ufx + vfy + \partial/\partial x(u\sigma xx + v\sigma xy) + \partial/\partial y(v\sigma yy + u\sigma yx)]dxdy$$

$$\rho De/Dt == K[\partial^2T/\partial x^2 + \partial^2T/\partial y^2] + \partial u/\partial x\ \sigma xx + \partial v/\partial y\ \sigma yy + \partial v/\partial x\ \sigma xy + \partial u/\partial y\ \sigma yx$$

We have $\sigma xx = \mu\ \partial u/\partial x$

$\sigma yy = \mu\,\partial v/\partial y$

$\tau xy = \tau yx = \mu(\ \partial u/\partial x + \partial v/\partial y)$

$$\rho C_p DT/Dt = K[\partial^2T/\partial x^2 + \partial^2T/\partial y^2]$$

$$\rho C_p\,[\frac{\partial T}{\partial t}\,u\,\frac{\partial T}{\partial x} + v\,\partial T/\partial y] = K[\partial^2T/\partial x^2 + \partial^2T/\partial y^2]$$

2.3 Significance of k-ε equation

K-epsilon (k-ε) turbulence model is the most common model used in Computational Fluid Dynamics (CFD) to simulate turbulent conditions. It is a two equation model which gives a general description of turbulence by means of two transport equations (PDEs). The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows. The first transported variable determines the energy in the turbulence and is called turbulent kinetic energy (k). The second transported variable is the turbulent dissipation (ϵ) which determines the rate of dissipation of the turbulent kinetic energy.

3 .CFD ANALYSIS

3.1 STAIRMAND'S OPTIMIZED DESIGN ANALYSIS: Open Ansys work bench and Select fluent analysis system for the current analysis.

Fig.3.1.1. fluent project schematic

3.1.1 CYCLONE GEOMETRY

Import the cyclone design from the solid works. open the design modeler. Click on generate the imported geometry appears. Select the part body in the tree outline .select the body click on the screen. Change the solid body into the fluid body. Close the design modeler and save the project.

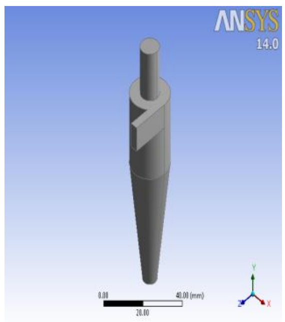


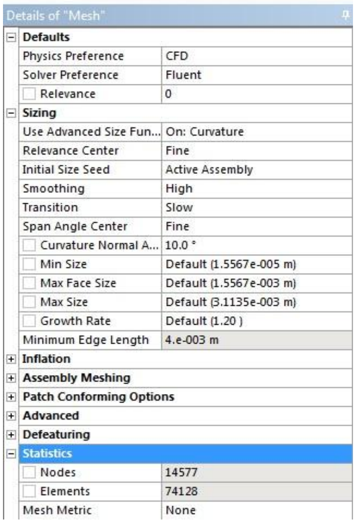
Fig.3.1.2: solid cyclone geometry for the simulation.

3.1.2 MESH

Open mesh.>create named sections

1. Select the inlet face.name it as velocity inlet
2. Select the outlet face and name it as pressure outlet.
3. Select the rest of the faces and name them as wall.

Select mesh in tree outline. In mesh details default conditions are set to be CFD and FLUENT solver as shown in the fig 5.1.3. Give high smoothing condition and fine relevance. And change the transition slow to fast to reduce the no. of elements. Select mesh and click generate mesh to obtain mesh. The generated mesh contains 74128 tetrahedron elements and 14577 nodes. Close the mesh and update the mesh in project schematic.



Details of "Mesh"

Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Sizing	
Use Advanced Size Fun...	On: Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Span Angle Center	Fine
Curvature Normal A...	10.0 °
Min Size	Default (1.5567e-005 m)
Max Face Size	Default (1.5567e-003 m)
Max Size	Default (3.1135e-003 m)
Growth Rate	Default (1.20)
Minimum Edge Length	4.e-003 m
Inflation	
Assembly Meshing	
Patch Conforming Options	
Advanced	
Defeaturing	
Statistics	
Nodes	14577
Elements	74128
Mesh Metric	None

Fig.3.1.3: mesh details and no of elements

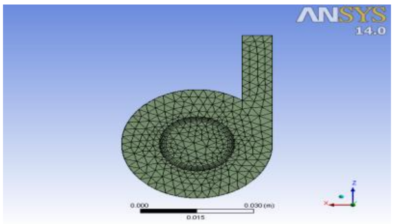


Fig3.1.4: mesh top view

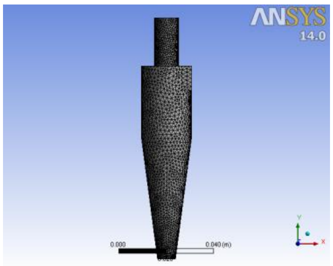


Fig 3.1.5: mesh front view

3.1.3 SETUP

Double click on the fluent set up to set the simulation conditions. The software automatically recognizes the 3d dimension. The display mesh after reading, embed graphics windows and work bench color scheme must be enabled. Enable the double precision and serial processing options. Then click ok to open the fluent.

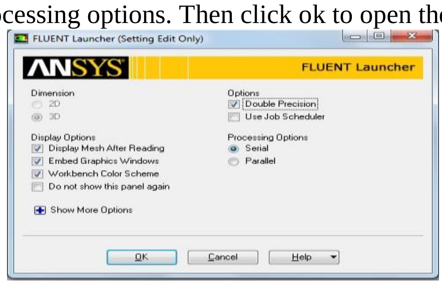


Fig. 3.1.6: fluent launcher

STEP 1: General > check mesh (To verify the mesh is correct or not) Enable pressure based type, absolute velocity formulation and transient time steps.

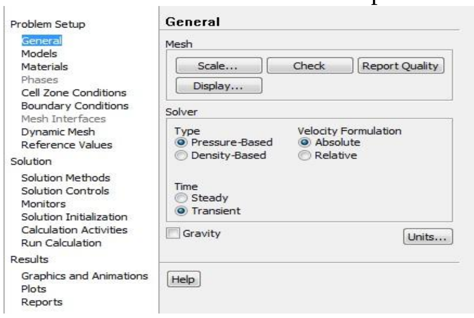


Fig.3.1.7: general conditions

STEP 2: In models select the realizable k-epsilon (2eqn) model and enable the enhanced wall treatment because the design contains many walls.

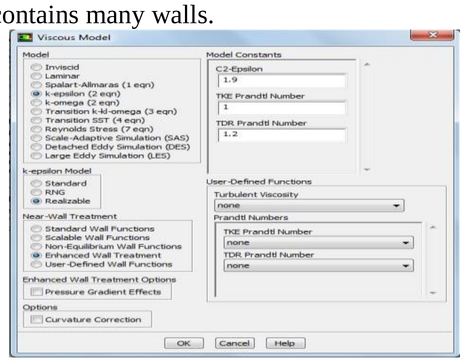


Fig.3.1.8: defining the models

STEP 3: Boundary condition
3.1. Velocity inlet> z-velocity =20 m/s
Turbulence: specification method> k-ε model

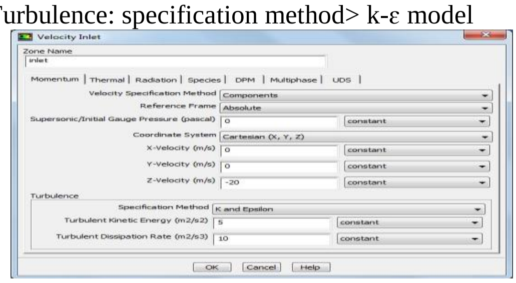


Fig.3.1.9: inlet velocity boundary conditions.

3.2. Outlet
Turbulence: specification method> k-ε model
Backflow Turbulence kinetic energy= 5 m2/s2
Backflow Turbulence dissipation rate=10 m2/s3

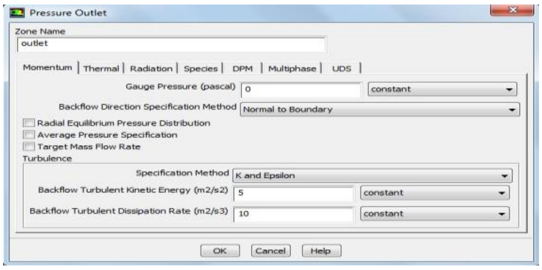


Fig.3.1.10: outlet boundary conditions.

3.3. Wall

Wall motion> stationary wall

Shear condition> no slip

STEP 4: Solution methods

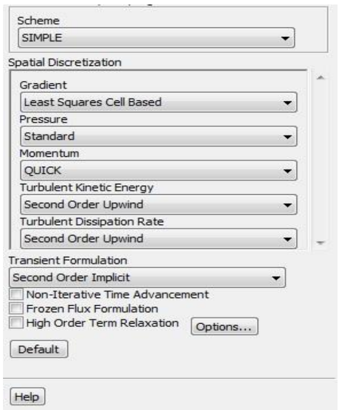
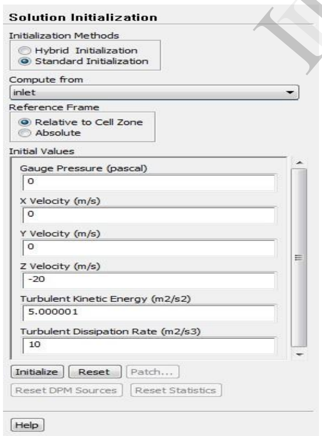


Fig.3.1.11: details of solution methods

STEP 5: Initialization: Select standard initialization and



compute from inlet velocity

Fig.3.1.12: initialization conditions

STEP 6: RUN> Check case>close

Time step size(s) =1; Number of time steps =50; Max. Iterations / time step = 20 > calculate

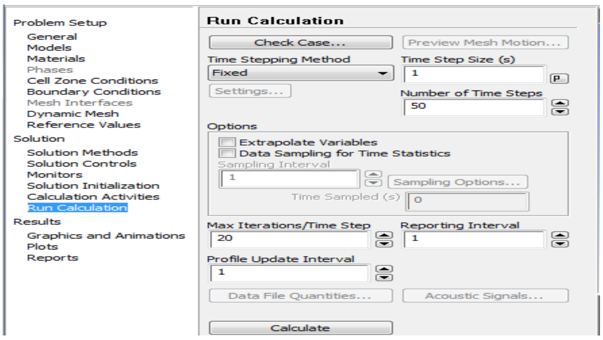


Fig.3.1.13: set up for the calculation

3.1.4 SOLUTION

Residuals

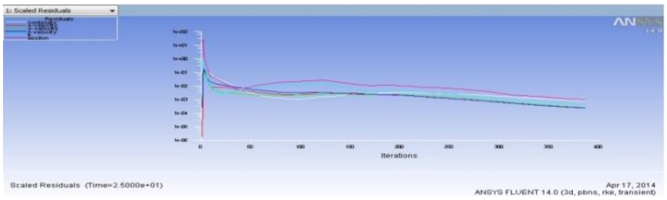


Fig.3.1.14 residual graph

The solution is converged at the 355th iteration. The vector fig 5.15 shows that the path followed by the fluid inside the cyclone. The flow follows swirl flow conditions as explained in the principle of the cyclone separator. The left side bar shows various velocity ranges. The color obtained in the vectors show the variation of velocity at the different sections. The values of the velocity can be studied from the left side scale. To understand the complete flow inside the cyclone the contours at the z=0 is computed. The velocity and pressure contours are shown in fig 5.1.16. which gives complete flow variation in the cyclone separator.

Vectors

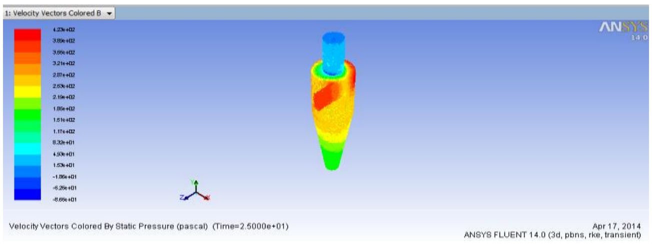


Fig.3.1.15: vectors of velocity

Pressure- velocity contours

The pressure in the cyclone separator increases from the center to wall. The maximum and minimum static pressures are 4.122e+002 (pa) and -6.44e+001 (pa) respectively. The velocity first increases then decreases from center to towards wall. The maximum and minimum velocity magnitudes are 23.62 m/s and 0 m/s respectively.

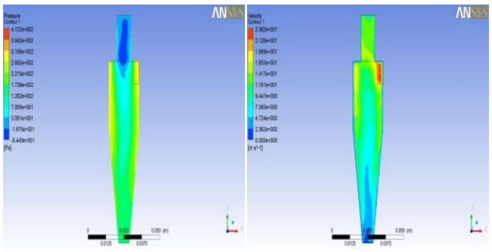


Fig.3.1.16: the pressure and velocity contours at the section Z=0 .

Pressure-velocity charts

The following charts shows the pressure and velocity variation along the y-axis at different sections. The graphs are plotted with y-axis as pressure/velocity and x-axis as radial distance along the x-axis.

Table 2: The sections

SERIES 1	AT Y=0
SERIES 2	AT Y=0.01
SERIES 3	AT Y=0.02
SERIES 4	AT Y= -0.01
SERIES 5	AT Y= -0.02

Graphs: The graphs are drawn by taking y-axis as pressure/velocity and the x-axis as radial distance along X.

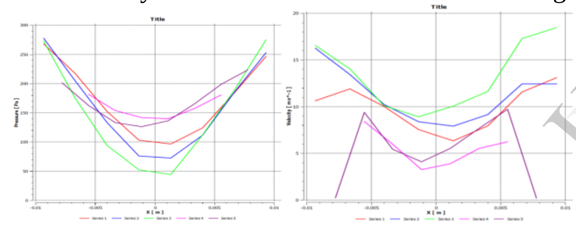


fig.3.1.17: the pressure and velocity graphs at different section along the y-axis.

3.1.5 RESULTS AND DISCUSSIONS

The stairmand's optimized design results are shown in above graphs. To study the variations in pressure and velocity 5 different sections are created along the y-axis as shown in the table 5.1.1. The variation in the pressure and velocity inside the cyclone is shown in the fig 5.1.16 & fig 5.1.17. The solution got converged at the 15th time step, total time is 15s.

The pressure inside the cyclone along the radial distance (x-axis) increases from centre to the ends. The pressure variations are also shown as a graph in fig5.1.17. The shape of the graph is U which shows the fall down of pressure at the centre of the cyclone (x=0).

The velocity in the cyclone along the radial direction first increases and then decreases from centre to the wall. The shape of the graph is inverted W which shows the fall down of velocity at the centre and at the wall. The high velocity obtains in the middle portion of the centre and the wall.

3.2 TEMPERATURE ANALYSIS

This analysis involves various flow studies at various temperatures. The stairmand's design is used for the simulation in fluent. The same set up is used for the temperature analysis as the stairmand's design analysis. Additional to that energy equation is activated to start the temperature analysis. In the velocity inlet boundary conditions the temperature of the inlet flow is added. This study involves in the simulation of the cyclone at 4 different temperatures. The variations in the pressure and velocity are noted and compared. The effect of the temperature is justified.

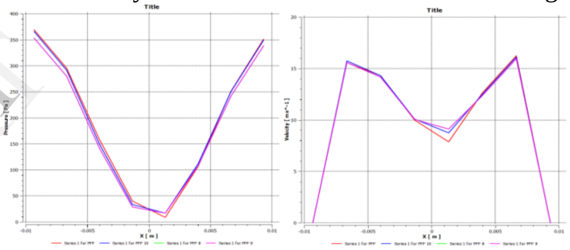
The analysis is done at the temperatures 290,300,310 & 320 (k).

3.2.1 SOLUTION

Table 3: pressure and velocity readings at different temperatures

Temp (k)	290	300	310	320
Maxpressure (pa)	533.5	535.35	522.15	522.6
Mini pressure(pa)	-185	-192.7	-194.7	194.7
Maxvelocity (m/s)	25.25	25.22	25.12	25.12
Minivelocity (m/s)	0	0	0	0

Graphs: The graphs are drawn by taking y-axis as pressure/velocity and the x-axis as radial distance along X.



Graph 3.2.1 The variation of pressure velocity in cyclone and at section Z=0 .

`Pressure contours

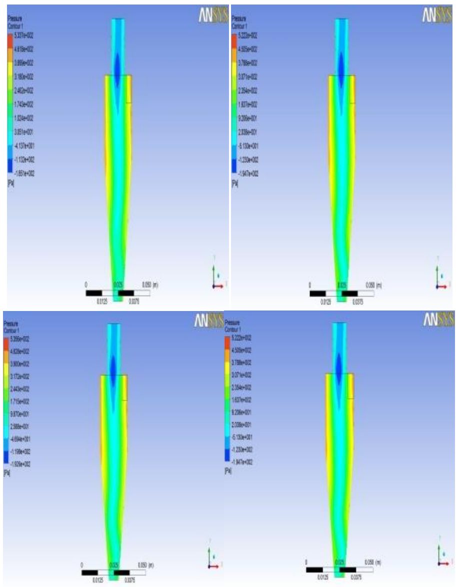


Fig 3.2.1 variation of pressure along the radial distance (x-axis) at z=0.

velocity contours

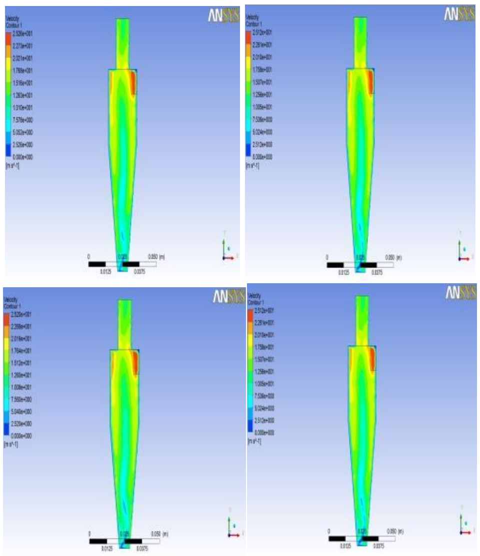


Fig 3.2.2 variation of velocity along the radial distance (x-axis) at z=0.

3.2.2 RESULTS AND DISCUSSIONS

The results are concluded that the cone height has significant effect on the performance of the cyclone. The pressure in the cyclone varies along the X-axis as shown in the contours. The pressure first decreases and then increases. The minimum pressure occurs at the mid section (x=0). The graph shows the variation in the pressure along the radial direction. The curve is in U shape explains the decrease and increase of pressure. The velocity in the cyclone first increase from the centre and then decreases at the wall. The curve will be in M shape or reversed W shape. The velocity is high at the middle portion of the center and the wall.

The variation in the flow temperature slightly varies the pressure for every 20k. The velocity of the flow doesn't vary with the temperature. So we can say that the temperature cannot affect the performance of the cyclone because of the slight variations we can neglect the effect of temperature.

3.3 THE ANALYSIS OF STAIRMAND'S DESIGN WITH COLLECTOR

The collector is the attachment to the cyclone where the particles are collected. Collector is of any shape (ex: cube, cylindrical). It locates at the end of the cone tip and it prevents the re-entertainment of particles. In this analysis a cube shaped collector is provided at the bottom (with dimensions 15mm*15mm*15mm). The same set up is used to simulate the cyclone with collector.

3.3.1CYCLONE GEOMETRY and MESH WITH COLLECTOR

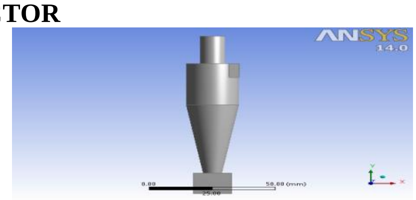


Fig 3.3.1 geometry of cyclone with collector

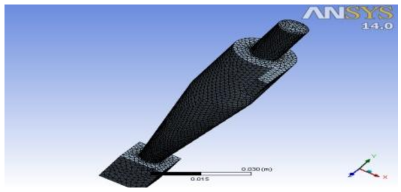


Fig 3.3.2 mesh of the cyclone with collector

3.3.2 SET UP IS SAME AS THE STARMAND'S DESIGN ANALYSIS

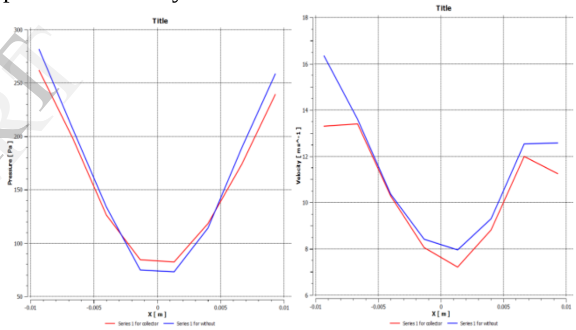
3.3.3 SOLUTION

The solution of the simulation of the cyclone with collector is compared with the cyclone without collector. The variations in pressure and velocity are noted and compared & the cyclone performance is justified.

Table 4: The pressure and velocity readings.

	Without collector	With collector
Max pressure (pa)	418.5	403.4
Mini pressure (pa)	-66.28	-55.11
Maxvelocity (m/s)	23.65	23.5
Minivelocit (m/s)	0	0

Graphs: The graphs are drawn by taking the y-axis as pressure/velocity and x-axis as radial distance along X



Graph 3.3.1: variation of pressure and velocity.

Pressure contours

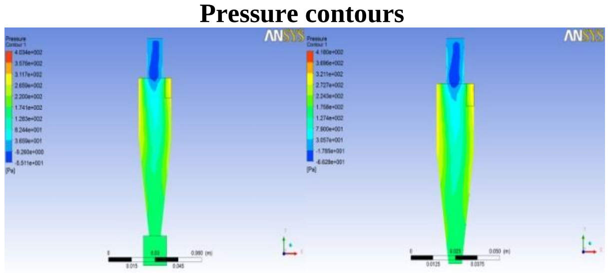


Fig 3.3.3 variation of pressure along the radial distance (x-axis) at z=0.

velocity contours

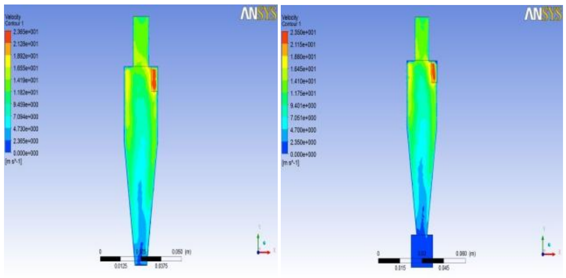


Fig 3.3.4 variation of velocity along the radial distance (x-axis) at z=0.

3.3.5 RESULTS AND DISCUSSIONS

The results are concluded that the cone height has significant effect on the performance of the cyclone. The pressure in the cyclone varies along the X-axis as shown in the contours. The pressure first decreases and then increases. The minimum pressure occurs at the mid section (x=0). The graph shows the variation in the pressure along the radial direction. The curve is in U shape explains the decrease and increase of pressure. The velocity in the cyclone first increase from the centre and then decreases at the wall. The curve will be in M shape or reversed W shape. The velocity is high at the middle portion of the center and the wall.The study shows that the collector doesn't have much effect on the cyclone performance. There are slight variations in the pressure and velocity which can be neglected.

3.4 INLET GEOMETRY

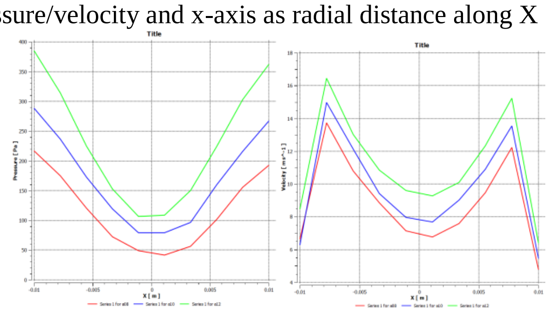
The inlet geometry is one of most important parameter which affects the cyclone performance. This study involves with various inlet geometries. By varying the height and width of inlet the inlet geometry varies. The analysis of 3 different cyclones with different inlet geometry is simulated in the fluent. By comparing all the 3 cyclones th ne better design is concluded. The same set up is used for the simulation as stairmand's design analysis.

3.4.1 SOLUTION

Table 5: pressure and velocity readings

	a=12, b=6	a=10 , b=5	a=8, b=4
Max pressure (pa)	513.5	415.7	334.8
Mini pressure (pa)	-102.9	-76.28	-46.68
Maxvelocity (m/s)	24.25	23.84	22.89
Minivelocity (m/s)	0	0	0

Graphs: The graphs are drawn by taking the y-axis as pressure/velocity and x-axis as radial distance along X



Graphs 3.4.1 the pressure and velocity plots

Pressure contours

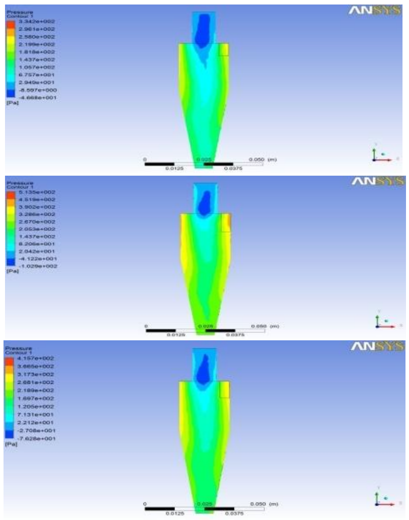


Fig 3.4.1 variation of pressure along the radial distance (x-axis) at z=0.

Velocity contours

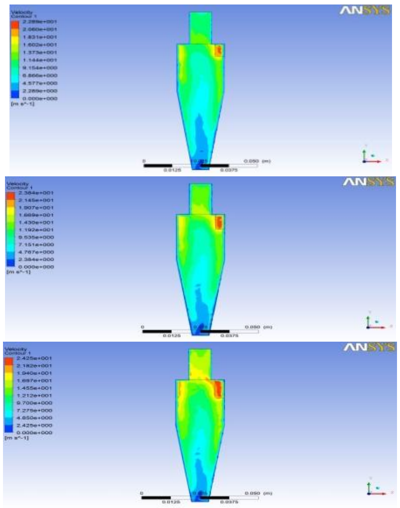


Fig 3.4.2 variation of velocity along the radial distance (x-axis) at z=0.

3.4.2 RESULTS AND DISCUSSIONS

The results are concluded that the cone height has significant effect on the performance of the cyclone. The pressure in the cyclone varies along the X-axis as shown in the contours. The pressure first decreases and then increases. The minimum pressure occurs at the mid section (x=0). The graph shows the variation in the pressure along the radial direction. The curve is in U shape explains the decrease and increase of pressure. The velocity in the cyclone first increase from the centre and then decreases at the wall. The curve will be in M shape or reversed W shape. The velocity is high at the middle portion of the center and the wall.

The inlet geometry is the most important geometrical parameter of the cyclone design. By varying the inlet dimensions there is a huge variation in the pressure and velocity. The maximum pressure in the cyclone falls drastically by the decrease in the inlet dimensions. For every 2mm decrease in the inlet height and 1mm decrease in width gives 20% decrease in the pressure. By the decrease of this inlet dimensions the pressure drop also decreases and we can say that the one with minimum pressure drop is works more efficiently. So the inlet dimension shows a large effect on the performance of the cyclone. The velocity also decreases by the increase in the inlet geometry, if velocity decreases the collection efficiency decreases so the inlet dimensions must be high. So the cyclone with the inlet height 10mm and inlet width 5mm is better design.

3. CONCLUSIONS

After studying the existing literature and performing analysis on certain cyclone parameters, the following conclusion can be drawn:

- The separation mechanism inside cyclone separators is not well understood yet, and needs more investigations.
- Nearly all published articles have no systematic and complete study for the effect of geometrical parameters on the flow field and performance.
- In all operating conditions and cyclone types the FLUENT CFD was found to be much closer to the experimental measurement.
- This project is done taking into consideration single parameter at a time, the results may vary if multiple parameters are considered at a time.
- Some parameters have less interest compared with others like the effect of inlet conditions and cyclone height.
  - Ex: 1. A cone is not an essential part for cyclone operation, whereas it serves the practical purpose of delivering collect particles to the central discharge point.
  - Ex: 2.Comparison between cyclone with and without dustbin was done and observed that a negligible effect of the dustbin on the performance

REFERENCES

[1] Khairy Elsayed 2011, PhD thesis on Analysis and Optimization of Cyclone Separators Geometry using RANS and LES Methodologies. Pages from 20-160.
[2] John Anderson 2011, A Text Book on Computational Fluid Dynamics, vol. 1