CFD Analysis of Symmetrical Tangential Inlet Cyclone Separator

Abhijeet Gayakwad[1] , Dr.Shivarudraiah[2]

[1]*PG Student Department Of Mechanical Engineering, University Visvesvaraya College Of Engineering Bangalore, India*
[2]*Professor Department Of Mechanical Engineering, University Visvesvaraya College Of Engineering Bangalore, India*

***

Abstract - *Many industries are using the cyclone dust separators without any modifications for more than a century. The gas solid cyclone separator is widely used in industries. For predicting the cyclone performance in terms of velocity and pressure variation, a large number of computational study was conducted due to its much application in the industrial area. The simulation is carried out using computational fluid dynamics (CFD) for gas-particle flow with cyclone separator in one of the approaches. Most of the attention is focused on improving the cyclone performance parameters. Recently most of the studies focused on the geometric effect on the cyclone performance. In this paper the geometric effect on cyclone separator is studied with the creation of symmetrical tangential inlet cyclone separator and compared it with the classical cyclone separator. The results showed that the new geometric modification to the cyclone improves the performance.*

Key Words : **Cyclone, CFD, Particle Flow, Symmetric inlet Separator.**

1. INTRODUCTION

The cyclone separators are widely used in the removal of dust particles from the gas flow in industrial processes. Cyclone dust separators have been used in industries to separate dust particles from the gas solid flow and intern reduce the air pollution occurring due to the chimney smoke in the chemical plant (Ogawa 1997). The cyclone separators are the most robust dust separators. In a classical cyclone separator the dust gas enters tangentially and forces the flow to follow spiral motion. Thus the created centrifugal force forces the dust particle towards the wall of the cyclone. After striking the wall of cyclone, the particles fell down and separated. In the new engineering applications the cyclone separators are used as reactors, dryers in petroleum industries to remove catalyst from the gases. The cyclone geometry is the most important parameter that affects the cyclone performance [8]. The solid dust particles are immediately bifurcated into two layers as soon it enters the cyclone due to the eddy currents generated in the coaxial space between the cyclone body and exit pipe. One of them goes upper surface around the coaxial space and rates with the gas flow around the exit pipe. The other layer rotates and descends down along the surface of the cyclone. Then in the cone surface the dust layers are pressed by the centrifugal force and descend down due to gravitational forces in the boundary layer. Lastly these dust layers are separated. However some of the dust rolls up due to the secondary air flow in the boundary and flows through the exit pipe. The centrifugal effect, which is responsible for separating the dust particles, depends on tangential velocity of particles. Therefore the tangential velocity must be increased to increase the cyclone efficiency because it relates to the pressure drop.

2. Cyclone Design

2.1 Geometry

The cyclone geometry is constructed by using Stairmand's high efficiency cyclone design method. Stairmand conducted so many experiments on the cyclone separator and finally developed the optimized geometrical ratios. By considering this geometric ratio's the modelling of the cyclone done in catia V5.

Take Dc=0.30 meter, which is comparatively safe as it is close to standard size diameter of 0.203 meter Thus, the dimension of the design cyclone is as under Stairmand's design,

Dc=diameter of cyclone=0.30 m
Height of inlet duct
Hi=0.5 Dc=0.5x0.3=0.15m
Width of inlet duct=wi=0.2
Dc =0.2x0.30=0.06 m
Diameter of out let duct=D0=0.5
Dc =0.5x0.3=0.15m
Diameter of dust outlet
Dd=0.375xDc =0.375x0.3=0.11m
Length of cyclone main body
(1.5Dc) =1.5x0.3=L1=0.45m
Length of cyclone hopper
= 2.5Dc=2.5x0.3=L2=0.75m
Total length of cyclone=L1+L2=1.2m

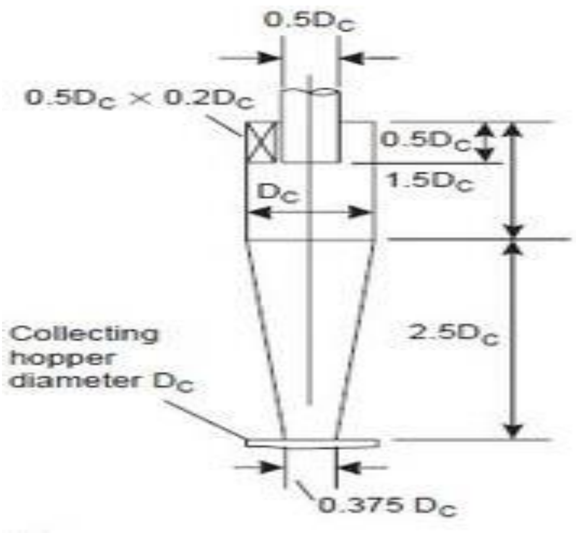


Fig. 2.1.1 Cyclone Design

2.2 Geometry modification

In order to increase the cyclone performance, tangential velocity must be increased. So in order to increase the cyclone tangential velocity an extra inlet symmetric to the standard cyclone design is added. With following design dimensions,
Height of inlet duct
Hi=0.5 Dc=0.5x0.3=0.15m
Width of inlet duct
wi=0.2 Dc =0.2x0.30=0.06 m

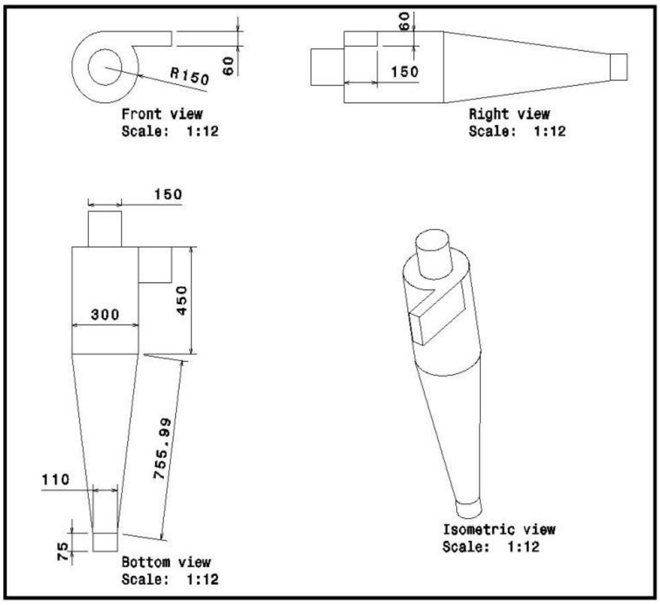


Fig. 2.2.1 Single inlet cyclone design

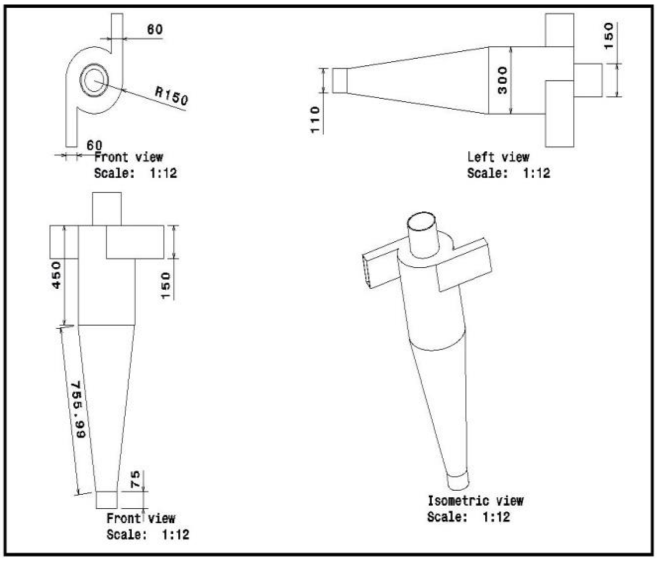


Fig. 2.2.2 Symmetrical inlet cyclone design

3. CFD ANALYSIS

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 modular and save the project.

3.1 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 and fig . 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.

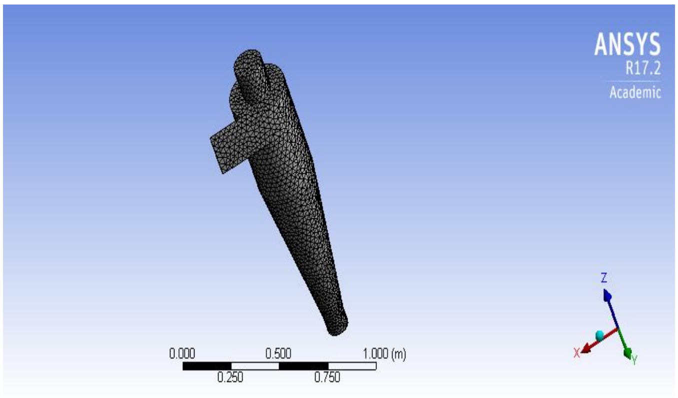


Fig.3.1.1 Meshed Single cyclone separator

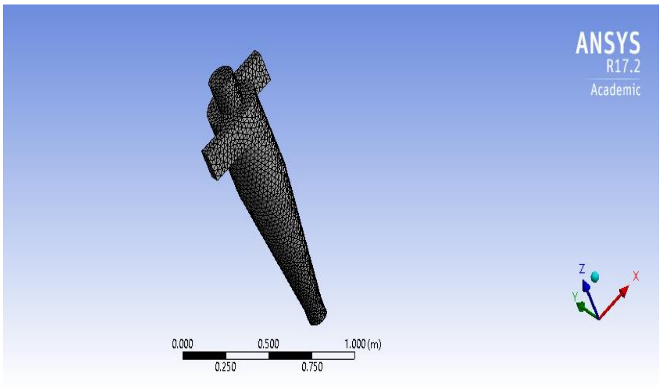


Fig.3.1.2 Meshed Symmetric cyclone separator

Mesh Properties

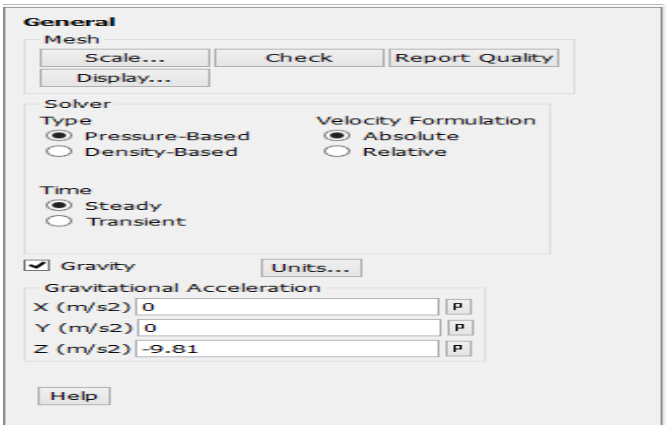
Tabel 1. Mesh properties

	Single Inlet	Symmetrical Inlet
Nodes	5426	5484
Elements	25809	25791

3.2 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 colour scheme must be enabled. Enable the double precision and serial processing options. Then click ok to open the fluent.

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



Solver
Type
● Pressure-Based
○ Density-Based

Velocity Formulation
● Absolute
○ Relative

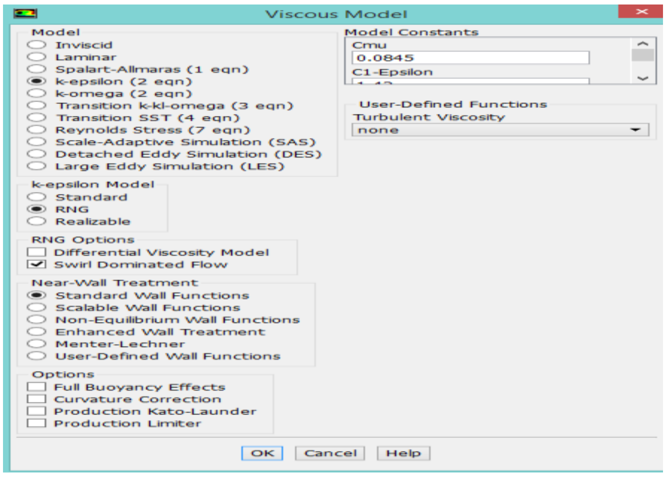
Time
● Steady
○ Transient

☑ Gravity

Gravitational Acceleration
X (m/s2) 0
Y (m/s2) 0
Z (m/s2) -9.81

Fig.3.2.1 General Conditions

STEP 2: In models select the realizable k-epsilon (2eqn) Model and RNG model with swirl dominated flow and standard wall functions.



Model
○ Inviscid
○ Laminar
○ Spalart-Allmaras (1 eqn)
● k-epsilon (2 eqn)
○ k-omega (2 eqn)
○ Transition k-kl-omega (3 eqn)
○ Transition SST (4 eqn)
○ Reynolds Stress (7 eqn)
○ Scale-Adaptive Simulation (SAS)
○ Detached Eddy Simulation (DES)
○ Large Eddy Simulation (LES)

k-epsilon Model
○ Standard
● RNG
○ Realizable

RNG Options
☐ Differential Viscosity Model
☑ Swirl Dominated Flow

Near-Wall Treatment
● Standard Wall Functions
○ Scalable Wall Functions
○ Non-Equilibrium Wall Functions
○ Enhanced Wall Treatment
○ Menter-Lechner
○ User-Defined Wall Functions

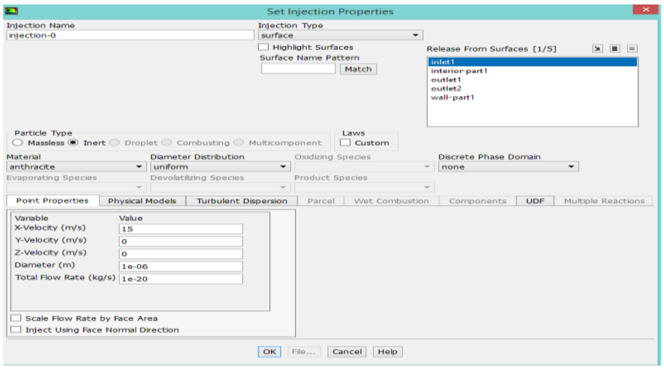
Options
☐ Full Buoyancy Effects
☐ Curvature Correction
☐ Production Kato-Launder
☐ Production Limiter

Model Constants
Cmu
0.0845
C1-Epsilon

User-Defined Functions
Turbulent Viscosity
none

Fig.3.2.2 Defining the models

Step 3 – DPM is set to on and create new injection for both the cyclones. The particles will enter from the inlet surface with 15m/s.



Injection Name
injection-0

Injection Type
surface

☐ Highlight Surfaces
Surface Name Pattern

Release From Surfaces [1/5]
inlet1
interior-part1
outlet1
outlet2
wall-part1

Particle Type
○ Massless ● Inert ○ Droplet ○ Combusting ○ Multicomponent

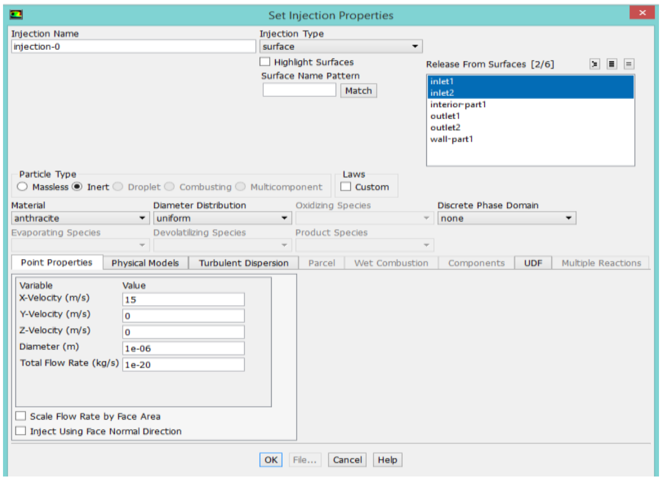
Laws
☐ Custom

Material: anthracite
Diameter Distribution: uniform
Oxidizing Species
Discrete Phase Domain: none
Evaporating Species
Devolatilizing Species
Product Species

Variable	Value
X-Velocity (m/s)	15
Y-Velocity (m/s)	0
Z-Velocity (m/s)	0
Diameter (m)	1e-06
Total Flow Rate (kg/s)	1e-20

☐ Scale Flow Rate by Face Area
☐ Inject Using Face Normal Direction

Fig. 3.2.3 Defining the models



Injection Name
injection-0

Injection Type
surface

☐ Highlight Surfaces
Surface Name Pattern

Release From Surfaces [2/6]
inlet1
inlet2
interior-part1
outlet1
outlet2
wall-part1

Particle Type
○ Massless ● Inert ○ Droplet ○ Combusting ○ Multicomponent

Laws
☐ Custom

Material: anthracite
Diameter Distribution: uniform
Oxidizing Species
Discrete Phase Domain: none
Evaporating Species
Devolatilizing Species
Product Species

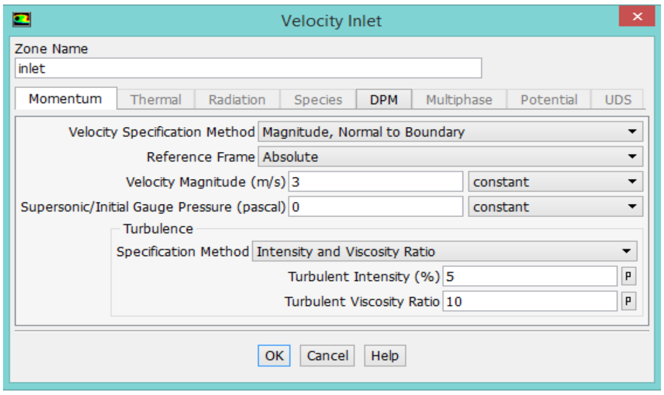
Variable	Value
X-Velocity (m/s)	15
Y-Velocity (m/s)	0
Z-Velocity (m/s)	0
Diameter (m)	1e-06
Total Flow Rate (kg/s)	1e-20

☐ Scale Flow Rate by Face Area
☐ Inject Using Face Normal Direction

Fig.3.2.4 Injection details

Creating injections

STEP 4: Boundary condition Velocity inlet>
x-velocity=15m/s

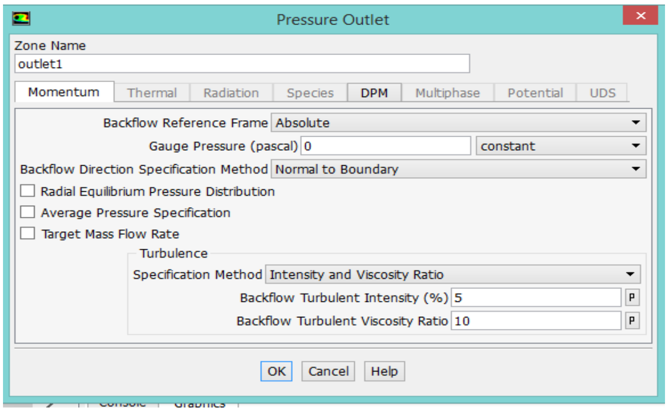


Zone Name: inlet
Velocity Specification Method: Magnitude, Normal to Boundary
Reference Frame: Absolute
Velocity Magnitude (m/s): 3 constant
Supersonic/Initial Gauge Pressure (pascal): 0 constant
Turbulence
Specification Method: Intensity and Viscosity Ratio
Turbulent Intensity (%): 5
Turbulent Viscosity Ratio: 10

Fig.3.2.5 Inlet velocity boundary conditions

Outlet

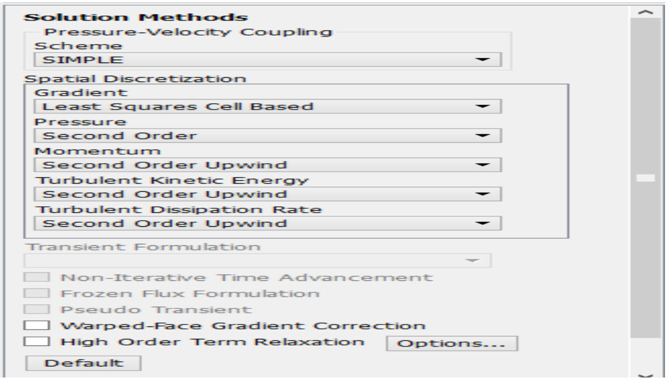
Turbulence; specification method> Intensity and Viscosity ratio Backflow Turbulence kinetic energy= 5 m2/s2 Backflow Turbulence dissipation rate=10 m2/s3.



Zone Name: outlet1
Backflow Reference Frame: Absolute
Gauge Pressure (pascal): 0 constant
Backflow Direction Specification Method: Normal to Boundary
☐ Radial Equilibrium Pressure Distribution
☐ Average Pressure Specification
☐ Target Mass Flow Rate
Turbulence
Specification Method: Intensity and Viscosity Ratio
Backflow Turbulent Intensity (%): 5
Backflow Turbulent Viscosity Ratio: 10

Fig.3.2.6 Outlet boundary conditions

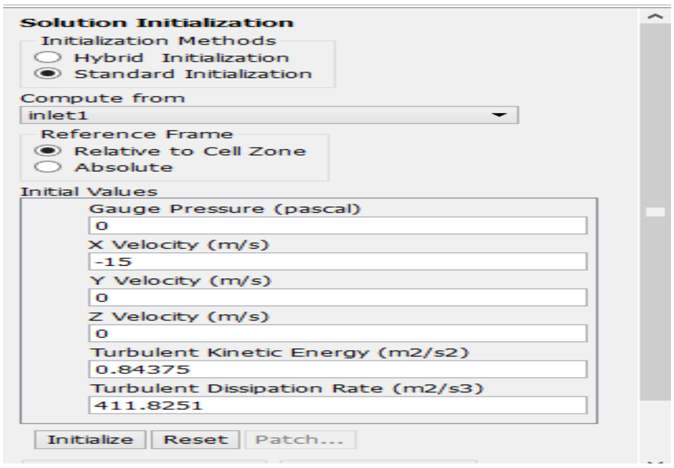
3.3 Solution Methods



Solution Methods
Pressure-Velocity Coupling
Scheme: SIMPLE
Spatial Discretization
Gradient: Least Squares Cell Based
Pressure: Second Order
Momentum: Second Order Upwind
Turbulent Kinetic Energy: Second Order Upwind
Turbulent Dissipation Rate: Second Order Upwind
Transient Formulation
☐ Non-Iterative Time Advancement
☐ Frozen Flux Formulation
☐ Pseudo Transient
☐ Warped-Face Gradient Correction
☐ High Order Term Relaxation

Fig.3.3.1 Details of solution methods

STEP 5: Initialization: Select standard initialization and compute from inlet velocity



Solution Initialization
Initialization Methods
○ Hybrid Initialization
● Standard Initialization
Compute from: inlet1
Reference Frame
● Relative to Cell Zone
○ Absolute
Initial Values
Gauge Pressure (pascal): 0
X Velocity (m/s): -15
Y Velocity (m/s): 0
Z Velocity (m/s): 0
Turbulent Kinetic Energy (m2/s2): 0.84375
Turbulent Dissipation Rate (m2/s3): 411.8251

Fig.3.3.2 Initialization conditions

STEP 6: RUN> Check case>close
Time step size(s) =1; Number of time steps =50; Max. Iterations = 555/ time step = 20 > calculate

4. SOLUTION

Residuals

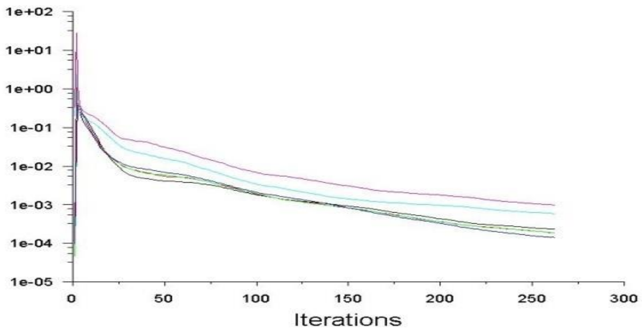


Fig.4.1 Single Inlet cyclone separator Residual Graph

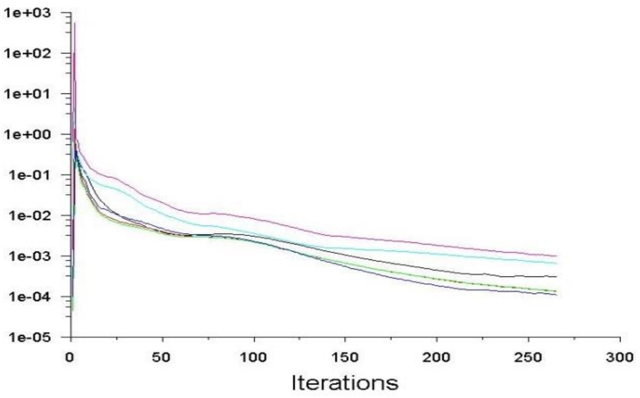


Fig.4.2 Symmetrical Inlet Cyclone Separator Residual Graph

The solution is converged at the 255th iteration. The vector fig.4.1 and fig.4.2 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 colour 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.

5.RESULTS AND DISCUSSIONS

5.1 CONTOUR RESULTS

Pressure contours

Pressure contours are plotted and it is observing that non-Dimensionalized static pressures are in the range of -22.98 to 248 respectively for single inlet cyclone. Static pressure is increasing from centre to wall surface but along the vertical section pressure is not uniform, it is decreasing at the bottom of the cone section of the cyclone as in case of single inlet cyclone separator.

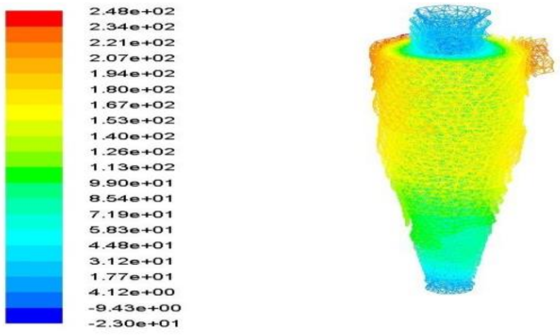


Fig.5.1.1 Static pressure contour of single inlet cyclone

Pressure contours are plotted and it is observing that non-Dimensionalized static pressures are in the range of -133 to 567 respectively for symmetric inlet cyclone. The static pressure is increasing from centre to the wall. The static pressure in uniform throughout the cyclone body as compared to the single inlet cyclone separator.

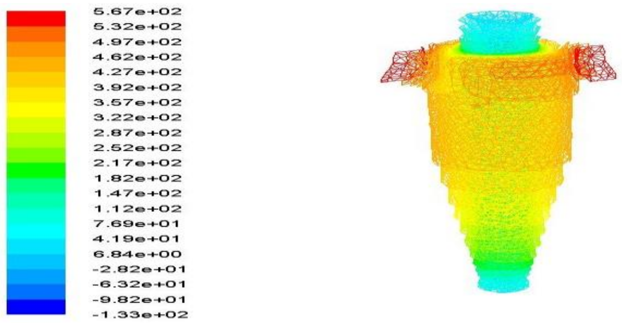


Fig.5.1.2 Static pressure contour for symmetrical inlet cyclone

5.2 VELOCITY VECTORS

Table. 2 Velocity magnitude

	Velocity Magnitude (m/s)		Tangential Velocity(m/s)	
	Min	Max	Min	Max
Single Inlet Cyclone	0.071	19.61	-2.156	18.83
Symmetrical Inlet Cyclone	0.3086	24.00	-1.2926	23.88

Velocity magnitudes

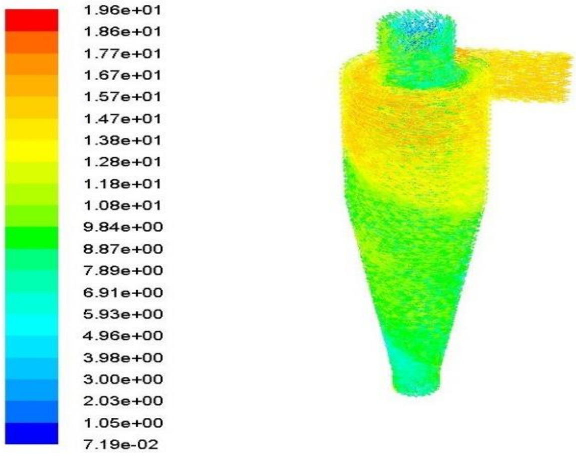


Fig.5.2.1 Velocity Magnitude Single Inlet

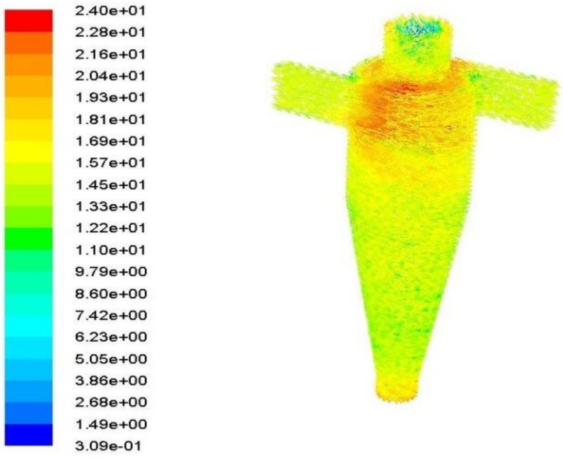


Fig.5.2.2 Velocity Magnitude Symmetrical Inlet

As we can see from the Fig.5.2.1 and Fig.5.2.2 the velocity magnitude is increasing from centre to the wall of the cyclone in both geometry. But the velocity is decreasing from top to the bottom of the cyclone in single inlet cyclone. The velocity magnitude is uniform almost uniform from the top to bottom of the cyclone in symmetrical inlet cyclone. Which gives better collection efficiency to the cyclone.

Tangential velocity vector

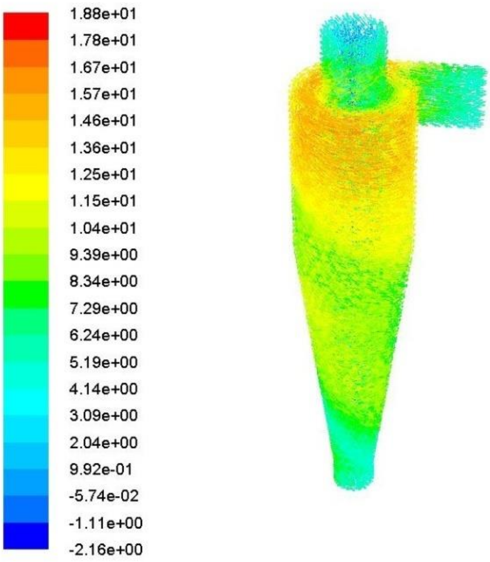


Fig. 5.2.3 Tangential Velocity Single Inlet Cyclone

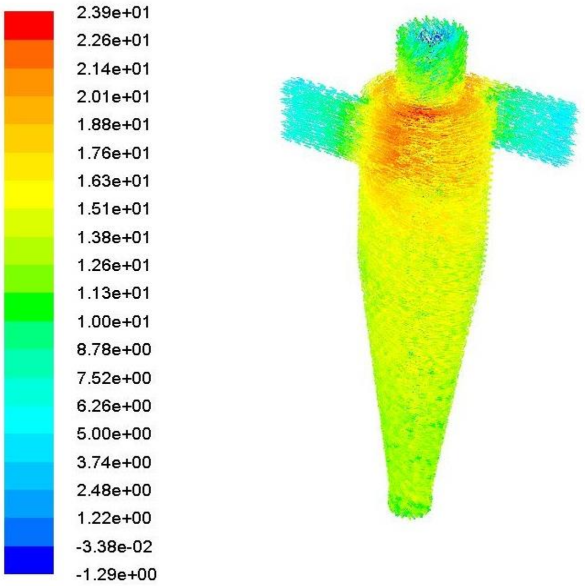


Fig.5.2.4 Tangential Velocity Symmetrical Inlet Cyclone

As we know that the tangential velocity of the gas flow, which relates to the pressure drop, must be increased in order to increase cyclone efficiency. From the data we can see that there is an increase in the tangential velocity in the symmetrical inlet cyclone as compared to single inlet cyclone.

Particle traces coloured by particle residence time for Single Inlet Cyclone

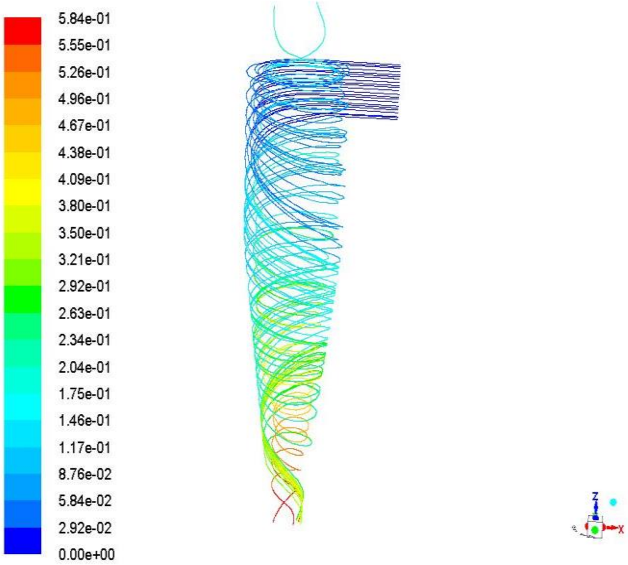


Fig.5.2.5 For single inlet cyclone

Particle traces coloured by particle residence time for Symmetrical Inlet Cyclone –

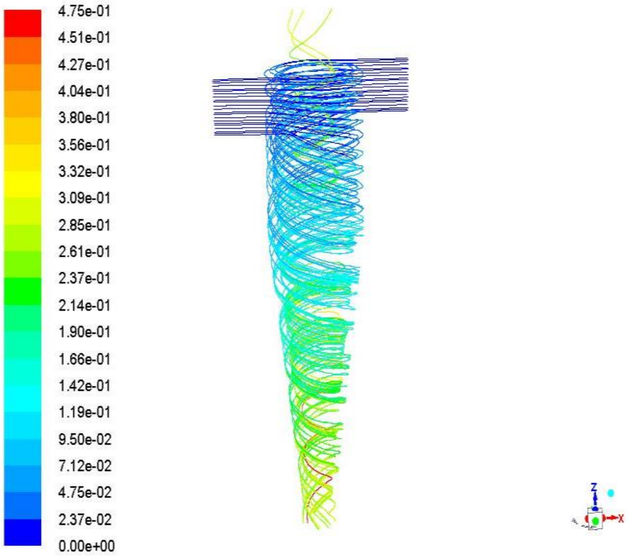


Fig.5.2.6 For symmetric cyclone

6. CONCLUSION

The Computation Fluid Dynamics (CFD) analysis is carried out for both single inlet cyclone and symmetrical inlet cyclone separator under the same condition of inlet velocity, flow rate and particle diameter. The results showed that the pressure distribution is uniform on the symmetrical inlet cyclone, the tangential velocity which relates to the pressure drop, will be increased which accounts for the better cyclone efficiency. The collection efficiency for the modified cyclone geometry is more than the standard cyclone design.

1. The results of pressure contour show uniform distribution of pressure throughout the cyclone body as compared to the standard design.
2. The results of tangential velocity vector show an increase in the tangential velocity in the cyclone body. Maximum tangential velocity for the single cyclone design is 18 m/s, and for symmetrical cyclone design is 23 m/s. This shows an increase in the tangential velocity.

7. References

1. Ogawa, A. (1997). Mechanical Separation Process and Flow Patterns of Cyclone Dust Collectors. Ind. Applied Mech. Ver. , vol. 50, n. 3, p. 97.

2. Ogawa, A. and Hikichi, T. (1981). Theory of CutSize of a Rotary Flow Dust Collector, Bulletin JSME, vol. 24, n. 188, p. 340.

3. Ogawa, A. (1984). Estimation of the Collection Efficiencies of the Three Types of the Cyclones Dust Collectors from the Standpoint of the Flow Patterns in the Cylindrical Cyclone Dust Collectors. Bulletin of JSME, vol. 27, n. 223, p. 64.

4. Stairmand, C.J. (1951).The Design and Performance of Cyclone Separators. Trans. Inst. Chem. Eng., vol. 29, p. 356.

5. Zhao, B., Shen, H. and Kang, Y. (2004). Development of a Symmetrical Spiral Inlet to Improve Cyclone Separator Performance. Powder Technology, vol. 145, issue 1, pp. 4750.

6. Cyclone Collection Efficiency: Comparison of Experimental Results with Theoretical Predictions By John Dirgo & David Leith

7. Bernardo S, Peres A P and Mori M (2005),"Computational Study of Cyclone Flow Fluid Dynamics using a Different Inlet Section Angle", Thermal Engineering (RETERM).

8. Bernardo S, Mori M and W P Martignoni (2006), "Evaluation of Cyclone Geometry and its Influence on its Performance Parameters by CFD", Powder Technology.

9. Lapple C E (1951), "Process Use Many Collector Types", Chemical Engineering.