DESIGN AND CFD ANALYSIS OF SINGLE AND SYMMETRICAL INLET CYCLONE DUST SEPARATOR

Anu Priya

Apprenticeship Trainee Vibration Engineering Group, Gas Turbine Research Establishment Bangalore, India

ABSTRACT

We are using cyclone dust separators for quite a century. Gas-solid cyclone separators are the most frequently used equipment in industries. To enhance the performance of cyclone dust separators, many Computational Fluid Dynamics studies conducted for its wide variety of applications in industries. Computational Fluid Dynamics is a conventional method to forecast the flow and collection efficiency of a cyclone. Primarily three models were used in cyclone simulation K-Epsilon Model, Revnolds Stress Turbulence Model, and Algebraic Stress Model. The K-epsilon turbulence model is the foremost crucial model utilized in computational fluid dynamics analysis to simulate turbulent kinetic and dissipation conditions. Pressure drop in the cyclone separator is one of the most significant functions to be kept in mind while designing the cyclone system. For further improvement in cyclone dust separator, a comparison of the pressure drop in a single and symmetrical tangential input cyclone separator perform theoretically and computational fluid dynamics analysis using the Reynolds stress turbulence model. The result showed that the pressure drop in the symmetric inlet cyclone separator exceeds the single inlet cyclone separator. I also performed the Computational Fluid Dynamics analysis to calculate the efficiency of a cyclone separator with a dust collector using the K-Epsilon Turbulence Model. So far, we all know that the pressure drop in a cyclone separator is directly associated with the tangential velocity of the cyclone separator, which must increase to extend the efficiency of the cyclone. Cyclone efficiency will generally increase with increment in particle size or density, tangential velocity, cyclone body length, and smoothness of the inner wall of the cyclone.

Keyword: Computational Fluid Dynamics, Pressure Drop, Efficiency Calculation, Symmetrical Inlet Cyclone, Reynolds Stress Turbulence Model

1000

1. INTRODUCTION

Cyclone dust separators are pollution control devices designed to extract fine particles from what is also known as dust in an air stream. It consists of a conical shape that utilizes the role of the air vortex to collect dust particles which have proven to be a piece of better settling equipment than gravity. Cyclones do not have any moving parts and are available in many shapes and sizes. It is categorized into two types of orientation, namely vertical and horizontal, and can be set up together as a step or multi-step cyclone separator. In vertical cyclones, air penetrates the equipment tangentially and then forms a vortex as it moves along the conical section. The vortex generates the force that pushed dust particles to maneuver to the walls of the equipment and slides under the influence of gravity. During the design of cyclones, we consider particle size (particles with larger mass subject to greater force), the force exerted on the dust particles, and the time that force exerted on the particles. Discrete levels of collection efficiency and operation can achieve by varying the standard cyclone dimension. There are some limitations of the various models used in cyclone simulation. The K-Epsilon model embraces the hypothesis of isotropic turbulence, so it is unsuitable for flow in a cyclone of anisotropic turbulence. The Algebraic Stress Model cannot predict the recirculation zone and Rankine vortex in strongly swirling flow. Reynolds Stress Model forgoes the hypothesis of isotropic turbulence and solves a transport equation for every component of the Reynolds Stress. It is considered the foremost applicable turbulent model for cyclone flow. Lagrangian and Eulerian techniques are the most commonly used to predict mean particle diffusion in turbulent streams.

2. MODELING

2.1 The Stairmand's High-Efficiency Cyclone Design

In this paper, the cyclone geometry is constructing by using the reference of Stairmand's high-efficiency method. Several experiments were carried out by Stairmand on cyclone dust separator and eventually developed efficient geometrical ratios. The sketching and modeling of the cyclone perform in the design modeler of Ansys Workbench by using the geometrical ratio of Stairmand. Here, I am taking the diameter of the cyclone separator as 0.20 meters, which is close to the standard size diameter of 0.203 meters.

Sr. No.	Cyclone Data	Dimensions(m)
1.	Diameter of Cyclone (Dc)	0.20
2.	Height of Rectangular Inlet (A)	0.10
3.	Width of Rectangular Inlet (B)	0.05
4.	Diameter of Circular Outlet (De)	0.10
5.	Height of Circular Outlet (C)	0.125
6.	Diameter of Collection Bin (Db)	0.05
7.	Length of Cyclone main Body (L ₁)	0.40
8.	The total length of a cyclone (L)	0.80

Table-1: The	e Geometrical	Parameter	values fo	r Cyclone	Design
)	



Fig-1: Cyclone Design Dimensions

2.2 Geometry Modification

The performance of the cyclone separator can improve by increasing its tangential velocity. If we do slight modification and add the symmetrical inlet to the geometry of the cyclone then tangential velocity can increase. The dimension of the new inlet is shown below,

• Height of Rectangular Inlet $(A_1) = 0.10 \text{ m}$

• Width of Rectangular Inlet $(B_1) = 0.05 \text{ m}$



Fig-2: Symmetrical Inlet Cyclone Modelling

3. CFD ANALYSIS

3.1 Single and symmetrical inlet cyclone separator analysis

First, open the Ansys workbench and then drag the Fluid Flow (Fluent) Analysis system from the toolbox to the project schematic window for performing the current analysis.

▼		А			▼		А		
1		Fluid Flow (FLUENT)			1	C	Fluid Flow (FLUENT)		
2	OM	Geometry	~	4	2	OM	Geometry	× .	1
3	۲	Mesh	~	4	3	6	Mesh	× .	
4	٢	Setup	~	4	4	٢	Setup	× .	
5	G	Solution	~	4	5		Solution	× .	
6	6	Results	~	4	6	6	Results	× .	
	Sing	le Inlet Cyclone Separa	ator		 Sy	mmet	trical Inlet Cyclone Sep	arator	

Fig-3: Fluid Flow (Fluent) project schematic

3.2 Single and symmetrical inlet cyclone geometry

For creating cyclone geometry open the Design Modeler tab and select X-Y plane, select the sketch to design cyclone geometry with the given dimensions. After sketching the cyclone go to the modeling section and choose extrude, revolve command to complete the cyclone geometry is. Now, save the geometry and shut the Design Modeler window.



Fig-4: Single and symmetrical inlet cyclone separator geometry for simulation

3.3 Mesh

Double click on Mesh to open the Meshing window, create named selections

- 1) Select the inlet face and name it as velocity inlet
- 2) Select the outlet face and name it as a pressure outlet
- 3) Select the dust outlet face and name it as a collection bin
- 4) Select the remaining faces and name them like a wall.

Select the mesh in the tree outline after that in the Method option choose Tetrahedron, sizing option choose Relevance Centre: Fine, Smoothing: High and minimum size = 5.e-006 m. Select the mesh and click on generate mesh option to obtain mesh.



Fig-5: Single and symmetrical inlet cyclone meshed models

Table- 2: Mesh Statistics

Statistics	Single Inlet	Symmetrical Inlet
Nodes	7952	8739
Elements	41127	45425

3.4 Setup

Double click on the setup to open Fluent Launcher, select Double-precision, and in the processing option click on Serial, select ok.

FLUENT Launcher (Setting Edit Only)	- 🗆 ×
ANSYS	FLUENT Launcher
Dimension 2D 3D	Options Double Precision Use Job Scheduler
Display Options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme Do not show this panel again	Processing Options Serial Parallel
	ancel <u>H</u> elp ▼
Fig-6: Flue	nt Launcher

STEP 1: General > check Mesh (To confirm mesh is correct or not) then in Solver select Pressure-Based type, Absolute Velocity Formulation, Transient Time Steps and enable Gravity and put a value of Gravitational Acceleration -9.81 m/s² in Y-Axis.

Image: A:Single Inlet Cyclone S File Mesh Define Sol Image: Define Image: Define Sol Image: Define Sol Image: Define Image: Define Image: Define Sol Image: Define Sol Image: Define Image: Define Image: Define Image: Define Sol Image: Define Sol Image: Define Image: Define Image: Define Image: Define Image: Define Sol Sol Sol Sol Sol<	Separator FLUENT [3d, dp, pbns, RSM, transient] [ANS\ lve <u>A</u> dapt <u>Su</u> rface <u>D</u> isplay <u>R</u> eport Parallel <u>V</u> i S ↔ Q ⊕ 🖋 🗄 @ 🏷 📑 マ 🗆 マ
Problem Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Monitors Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	General Mesh Scale Check Report Quality Display Solver Type Velocity Formulation • Pressure-Based • Absolute • Density-Based • Relative Time • Steady • Transient Units Gravity Units Gravitational Acceleration P Y (m/s2) -9.81 P Z (m/s2) 0 P P

Fig-7: General Conditions

STEP 2: In models select the Reynolds Stress (7 eqn) and in Reynolds-Stress Model select Linear Pressure-Strain, in Reynolds-Stress Options enable Wall BC from k Equation and Wall Reflection Effects, in Near-wall Treatment enable Standard Wall Functions.

Model	Model Constants	
 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) 	Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 C1-PS 1.8	^
Reynolds-Stress Model		~
 Linear Pressure-Strain Quadratic Pressure-Strain Stress-Omega 		
Reynolds-Stress Options		
Wall BC from k Equation		
Near-Wall Treatment		
Standard Wall Functions Scalable Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment		
Options		
OK	ncel Help	

STEP 3: In Materials choose Air then in properties value of Density = 0.4 kg/m^3 , Viscosity = 0.02 kg/m-s and click on change/create option.

į	Create/Edit Ma	terials		×
	Name air Chemical Formula		Material Type fluid	Order Materials by
			air	✓ FLUENT Database
			Mixture	User-Defined Database
	Properties Density (kg/m3)	constant	► Edit	
	Viscosity (kg/m-s)	0.4 constant	✓ Edit	
		0.02		
	1	Change/Cre	eate Delete Close	Help

Fig-9: Materials

STEP 4: Boundary Conditions
A) Zone name: Inlet > Velocity magnitude = 15 m/s²
Turbulence: Specification Method > Intensity and Hydraulic Diameter
Turbulence Intensity (%) = 10, Hydraulic Diameter(m) = 0.067
B) Zone name: Outlet
Turbulence: Specification Method > Intensity and Hydraulic Diameter
Backflow Turbulence Intensity (%) = 10, Backflow Hydraulic Diameter(m) = 0.1

Velocity Inlet	\times
Zone Name	1
Inlet	
Momentum Thermal Radiation Species DPM Multiphase	UDS
Velocity Specification Method Magnitude, Normal to Be	oundary ~
Reference Frame Absolute	~
Velocity Magnitude (m/s)	constant ~
Supersonic/Initial Gauge Pressure (pascal)	constant ~
Turbulence	
Specification Method Intensity and Hydraulic Di	iameter ~
Turbulent Intensity	(%) 10 P
Hydraulic Diameter	r (m) 0.067
Reynolds-Stress Specification Method K or Turbulent Intensity	· · · · · · · · · · · · · · · · · · ·
·	
OK Cancel Help	
Fig-10: Inlet Boundary Conditi	ions
Pressure Outlet	,
Zone Name	
oduer	
Momentum Thermal Radiation Species DPM Multipha	ase UDS
Gauge Pressure (pascal)	constant \sim
Backflow Direction Specification Method	
	~
Average Pressure Specification	
Considential Mathed	
Specification Method Intensity and Hydrauli	ic Diameter 🗸 🗸
Backflow Turbulent Inten	sity (%) 10
Backflow Hydraulic Diam	eter (m) 0.1
Reynolds-Stress Specification Method	•••
K or Turbuient Intensi	v V
OK Cancel He	elp

Fig-11: Outlet Boundary Conditions

STEP 5: Solution Methods

Pressure-Velocity Coupling		
Scheme		
SIMPLE	\sim	
Spatial Discretization		
Gradient		
Least Squares Cell Based	\sim	
Pressure		
Standard	\sim	
Momentum		
Second Order Upwind	\sim	
Turbulent Kinetic Energy		
Second Order Upwind	\sim	
Turbulent Dissipation Rate		
Second Order Upwind	\sim	
Transient Formulation		
First Order Implicit	\sim	
Non-Iterative Time Advancement		
Frozen Flux Formulation	_	
High Order Term Relaxation Options		
Default		

STEP 6: Solution Initialisation: Standard Initialisation Method and compute from Inlet Initial values: Y-velocity = 9.5 m/s, Z-velocity = -15 m/s

	nitialization Methods
	Hybrid Initialization Standard Initialization
1	Compute from
Ē	Reference Frame
	Relative to Cell Zone Absolute
I	nitial Values
	Gauge Pressure (pascal)
	X Velocity (m/s)
I	Y Velocity (m/s)
I	9.5
	Z Velocity (m/s)
	Turbulent Kinetic Energy (m2/s2)
	3.375
I	Turbulent Dissipation Rate (m2/s3)

Fig-13: Details of Solution Initialisation

STEP 7: Run Calculation > check case> close Time step size(s) = 0.0001; Number of Time Steps =3, Max Iterations/Time Step = 20

Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method	Time Step Size (s)
Fixed ~	0.0001 P
Settings	Number of Time Steps
Options	
Extrapolate Variables Data Sampling for Time S Sampling Interval Time Sampled (s	Statistics Sampling Options
Max Iterations/Time Step	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Fig-14. Details of Run	Calculation

D. SOLUTION

The solution is converged at the 53^{rd} iteration. Flow time is 0.0003s and time step = 3.







Fig-16: Symmetrical inlet cyclone separator residual graph

4. THEORETICAL CALCULATION

While designing a cyclone dust separator, the pressure drop in cyclone is one of the most important parameters to keep in mind. Theoretical Pressure drop calculation:

- Inlet Velocity $(u_1) = 15 \text{ m/s}$
- Gas Density (f_f) = 0.4 kg/m³ and Viscosity of the gas = 0.02 kg/m-s
- Hydraulic diameter of the rectangular inlet = $\frac{4(a \times b)}{2(a+b)} = 0.067 m$
- Hydraulic diameter of the circular outlet = 0.1 m
- Area of the rectangular inlet $(A_1) = 0.1 \times 0.05 = 0.0005 \text{ m}^2$
- Cyclone Surface Area (As) = $\pi .200 \times (400 + 400) = 0.502400 \text{ m}^2$
- $\psi = f_c \frac{A_s}{A_1} = 0.5024$, here, f_c is taken as 0.005 r. $\frac{100 - \binom{50}{2}}{100}$
- $\frac{r_t}{r_e} = \frac{\frac{100 \left(\frac{50}{2}\right)}{50}}{50} = 1.5$

where $r_{t=}$ radius of the circle to which the

centre line of the inlet is tangential and r_e = outlet pipe radius

- Based on ψ and $\frac{rt}{r_{e}}$ value we can find out φ from figure 17.
- •
- φ = 0.9
- Outlet Pipe Area (Ae) = $\frac{\pi}{4} d^2 = 0.007850 \text{ m}^2$
- $Q = A_1 u_1 = 0.075 \text{ m}^3/\text{s}$ $Q = A_e u_{2,} \rightarrow u_2 = 9.5 \text{ m/s}$

$$\rightarrow \Delta P = \frac{\rho_f}{203} \left\{ u_1^2 \left[1 + 2\varphi^2 \left(2\frac{r_t}{r_e} - 1 \right) \right] + 21 \right\}$$
$$\rightarrow \Delta P = 2.23 \ mbar = 223 \ Pa$$



Fig-17: Radius ratio vs φ graph

5. RESULTS AND DISCUSSIONS

5.1 Contour Results

Pressure Contours

Pressure contours obtain from Fluid Flow (fluent) show that non-dimensionalized static pressure is in the range of -36.098 Pa to 833.437 Pa for a single inlet cyclone separator. Static pressure is increasing from center to wall surface but along the vertical section, pressure isn't uniform and decreasing at bottom of the conic section of the cyclone as within the case of a single inlet cyclone separator.

1122			
8.33	ie+02		
7.90	e+uz		
(.46	e+UZ		
7.03	e+02		
6.60	ie+02		
6.16	e+02		
5.73	le+02		
5.29	le+02		
4 86	ie+02		
4 4 2	e+02		
3 99	ie+02		
3.55	e+02		
3.12	e+02		
2.68	e+02		
2.25	e+02		
1.81	e+02		
1.01	e+02		
0.42	02		
9.43	e+01	10.2	
5.09	le+01	1	
7.38	e+uu	- 人	
-3.61	1e+01		

Fig-18: Static pressure contour of single inlet cyclone



Fig-19: XY plot of static pressure versus position for single inlet cyclone separator

5.2 Comparison of Results for single inlet cyclone separator

- Pressure Drop from CFD Analysis = 252.75 Pa
- Pressure Drop from Theoretical Calculation = 223 Pa
- Error % between two results = 11.77 %

Pressure contours plot and shows that non-dimensionalized static pressure is within the range of -42.703 Pa to 825.722 Pa for symmetric inlet cyclone. The static pressure is increasing from the middle to the wall surface. I observed that maximum pressure is at the inlet and minimum pressure is at the outlet of the cyclone.



Fig-20: Static pressure contour of symmetrical inlet cyclone separator



Fig-21: XY plot of static pressure versus position for symmetrical inlet cyclone separator

5.3 Comparison of Results for symmetrical inlet cyclone separator

- Pressure Drop from CFD Analysis = 259.27 Pa
- Pressure Drop from Theoretical Calculation = 223 Pa
- Error % between two results = 13.98 %

6. THE ANALYSIS OF SINGLE AND SYMMETRICAL INLET CYCLONE WITH DUST COLLECTOR

The application of dust collectors is to collect dust particles located at the endpoint of the cyclone. A collector can be of any shape (example: cubical, cylindrical). It is fixed at the endpoint of the conic tip and prevents the re-entertainment of particles. During this analysis, two cylindrical-shaped collectors attach at the endpoint of the cone tip (with dimensions 50 mm diameter, 50 mm height for the first cylindrical shape collector, and 150 mm diameter, 150 mm height for the second cylindrical shape collector). The same setup uses to simulate the cyclone with a collector.

Single and symmetrical inlet cyclone geometry with dust collector



Fig-22: Single and symmetrical inlet cyclone separator geometry for simulation

Mesh



Fig-23: Single and symmetrical inlet cyclone meshed models

Table-3: Mesh Statistics

Statistics	Single Inlet	Symmetrical Inlet
Nodes	8848	8843
Elements	42188	42204

Setup

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

2) STEP 2: In models select the realizable k-epsilon (2eqn) Model and Standard model and standard wall functions.

3) STEP 3 – Discrete Phase Model is on and create new injection (Injection-0) for both the cyclones. The particles will enter from the inlet and in injection type choose Surface, Diameter distribution: Uniform.

Olimpingial			
Laminar		Cmu Lo op	
O Spalart-Alimar	as (1 eqn)	0.09	
k-epsilon (2 eqn) k-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (7 eqn)		C1-Epsilon	
		1.44	
		C2-Epsilon	
		1.92	
O Detached Edd	v Simulation (SAS)	and the second s	
O Large Eddy Sir	nulation (LES)	TKE Prandt Number	
ensilon Model			
(Standard		User-Defined Functions	
ORNG		Turbulent Viscosity	
ORealizable		none	~
ear-Wall Treatme	nt	Prandtl Numbers	
Standard Wall	Functions	The Designation of the	1
O Scalable Wall f	unctions	none	
O Non-Equilibriur	n Wall Functions	TOP Drandil bi mbas	× 1
O User-Defined	Wall Euroctions	Depe	
Cose Denned		Tuone	~
ptions	10000		
Full Buoyancy	Effects	1	
	field and the second se		
Injection Name	Fig-24: I	Defining Models	
Injection Name Injection-0 Injection Type surface	Fig-24: I	Defining Models	
Injection Name Injection-0 Injection Type surface	Fig-24: I Release From Surfa collection_bin Interior-solid outlet	Defining Models	
Injection Name Injection-0 Injection Type surface Particle Type	Release From Surfa collection_bin interior-solid outlet	Defining Models	
Injection Name Injection-0 Injection Type surface Particle Type O Massless	Release From Surfa collection_bin interior-solid outlet Highlight Surface t Droplet	Defining Models	
Injection Name Injection-0 Injection Type surface Particle Type O Massless Iner Material	Release From Surfa collection_bin interior-solid outlet Highlight Surface t Droplet Diameter Distribution	Combusting Multicomponent Custom	hase Domain
Injection Name Injection -0 Injection Type surface Particle Type Massless	Fig-24: I Release From Surfa Collection_bin Intert Interior-solid outlet Highlight Surface t Diameter Distribution V Linform	Combusting Multicomponent Custom	hase Domain
Injection Name Injection-0 Injection Type surface Particle Type Massless Iner Material steel	Fig-24: I Release From Surfa Collection_bin Inter: Interior-solid outet Highlight Surface t Diameter Distribution Uniform	Defining Models	hase Domain
Injection Name Injection-0 Injection Type surface Particle Type Massless Iner Material steel Svapor sting Species	Fig-24: I Release From Surfa Collection_bin Interior-solid outlet Highlight Surface t Diameter Distribution Uniform Deviabilizing Species	Defining Models	hase Domain
Injection Name Injection-0 Injection Type surface Particle Type Massless	Fig-24: I Release From Surfa collection_bin Interior-solid outlet Highlight Surface t Diameter Distribution Uniform peviolabilising Species t Dispersion Wet Combustion 0	Defining Models	hase Domain
Injection Name Injection Type surface Particle Type Massless Iner Material steel Eveporating Species Point Properties Turbulen Variable	Fig-24: I Release From Surfa Release From Surfa Interior-solid outlet Highlight Surface Highlight Surface t Diameter Distribution Uniform Devolabilizing Species t Dispersion Wet Combustion 0 Value	Defining Models	hase Domain
Injection Name Injection-0 Injection Type surface Particle Type Massless Iner Material steel Evapor ating Species Point Properties Turbulen Variable Diameter (m)	Release From Surfa collection_bin Interior-solid outlet Highlight Surface t Diameter Distribution Linform Deviolatiliang Species t Dispersion Wet Combustion 0 Value Se-06	Defining Models	hase Domain
Injection Name Injection-0 Injection Type surface Particle Type Massless Iner Material steel Evaporating Species Point Properties Turbulen Variable Diameter (m)	Fig-24: I	Defining Models	hase Domain
Injection Name Injection-0 Injection-Type surface Particle Type Massless	Fig-24: I Release From Surfa Collection bin Inter Interior solid Outlet Highlight Surface t Diameter Distribution Uniform Devolabilizing Species t Dispersion Wet Combustion 0 Value Se-06 3	Defining Models	hase Domain
Injection Name Injection Type surface Particle Type Massless	Fig-24: I Release From Surfa Release From Surfa Collection_bin Intert Interior-solid outlet Interior-solid Outlet Interior-solid Intertore Distribution Uniform Devolabilizing Species t Dispersion Wet Combustion (Value Se-06 3	Combusting Multiple Reactions	hase Domain
Injection Name Injection O Injection Type surface Particle Type Massless Iner Material steel Evaporating Species Point Properties Turbulen Variable Diameter (m) Velocity Magnitude (m/s) Total Flow Rate (kg/s)	Fig-24: I Release From Surfa Collection_bin Interior-solid outlet Highlight Surface t Diameter Distribution Uniform Devolabilizing Species t Dispersion Wet Combustion 4 Value Se-06 3 0.00001	Defining Models	hase Domain
Injection Name Injection O Injection Type surface Particle Type Massless Iner Material steel Eveporating Species Point Properties Turbulen Variable Diameter (m) Velocity Magnitude (m/s) Total Flow Rate (kg/s)	Fig-24: I Release From Surfa collection_bin Interior-solid outlet Highlight Surface t Diameter Distribution Uniform Devolabilizing Species t Dispersion Wet Combustion 0 Value Se-06 3 0.00001	Defining Models	hase Domain
Injection Name Injection -0 Injection Type surface Particle Type Massless Iner Material steel Evaporating Species Point Properties Turbulen Variable Diameter (m) Velocity Magnitude (m/s) Total Flow Rate (kg/s) Viscale Flow Rate by Face	Fig-24: I Release From Surfa Collection_bin Interior-solid outlet Interior-solid Uniform Diameter Distribution Uniform Dispersion Wet Combustion 0 Value Se-06 3 0.00001 Area	Defining Models	hase Domain
Injection Name Injection O Injection Type surface Particle Type Massless Infer Massless Infer Material steel Eveporating Species Point Properties Turbulen Variable Diameter (m) Velocity Magnitude (m/s) Total Flow Rate by Face Infect Using Face Normal	Fig-24: I Release From Surfa Collection bin relet Collection bin relet Pighlight Surface t Diameter Distribution Uniform Devolabilizing Species t Dispersion Wet Combustion 0 Value Se-06 3 0.00001 Area Direction	Defining Models	hase Domain

Fig-25: Creating new Injection

Material: Steel, Density of Steel = 8030 kg/m³, Particle size = 5 μ m, Total flow rate = 0.00001 kg/s and Velocity magnitude = 3m/s.

6.2 Solution Methods

Solution Methods		
Pressure-Velocity Coupling		
Scheme		
SIMPLE	~	
Spatial Discretization		
Gradient	^	
Least Squares Cell Based	~	
Pressure		
Standard	~	
Momentum		
Second Order Upwind	~	
Turbulent Kinetic Energy		
Second Order Upwind	~	
Turbulent Dissipation Rate		
Second Order Upwind	~ ~	

Fig-26: Details of Solution Method

Initialization: Select standard initialization, compute from inlet velocity, and put the value of Z velocity -3 m/s.

Solution Initialization	
Initialization Methods	
Hybrid Initialization Standard Initialization	
Compute from	
inlet_1	-
Reference Frame	
Relative to Cell Zone Absolute	
Initial Values	
Gauge Pressure (pascal)	
0	
X Velocity (m/s)	
0	
X Velocity (m/c)	
Z Velocity (m/s)	
-3	
Turbulent Kinetic Energy (m2/s2)	
0.135	
Turbulent Dissipation Rate (m2/s3)	
1.737839	
	\sim
Initialize Reset Patch	
Reset DPM Sources Reset Statistics	

Fig-27: Solution Initialization

6.4 Residual Graphs

The solution is converged at 352^{nd} iteration for single inlet cyclone separator. Total number of iterations = 500, Reporting Interval = 1, Profile Update Interval = 1.



Fig-28: Single inlet cyclone separator residual graph



Fig-29: Symmetrical inlet cyclone separator residual graph

7 RESULTS AND DISCUSSIONS

7.1 Pressure Contours

Pressure contours obtain from Fluid flow (Fluent) observe that non-dimensionalized static pressures are within the range of -2.57 Pa to 14.64 Pa for a single inlet cyclone. Static pressure is increasing from the core of the wall surface but decreasing at bottom of the cyclone. I observed that maximum pressure is at the inlet and minimum pressure is at the outlet of the cyclone.



Fig-30: Contours of static pressure (pascal) for single inlet cyclone

Pressure contours obtain from Fluid flow (Fluent) observe non-dimensionalized static pressures are within the range of -2.81 Pa to 14.3 Pa for symmetrical inlet cyclone separator. Static pressure is increasing from the core to the wall surface but decreasing at bottom of the cyclone. I observed that maximum pressure is at the inlet and minimum pressure is at the outlet of the cyclone.



Fig-31: Contours of static pressure (pascal) for symmetrical inlet cyclone



7.2 Velocity Contours

Fig-32: Contours of velocity magnitude (m/s) for single inlet cyclone

1020+50			100+100		
1006-00		8	4.068+00		
100510			3.808+00		
3.000400			3.606+00		0
1.488480		S	3.408+00		
3.20e+00			3.208+00		
3.01e+00			3.00e+00		
280e+00			2,80e+00		
2.60e+00			260e+00		
248e+80			2 400+00		
2,20e+00			2 200+00		
100e+00		- 10 A	2,004+00		
1.80+00			1.000-00		
1.800-00		1 11	1.000/00		
1.005100			1.008+00		<u>2</u>
1.406400			1.408+00		
1_000400			1.208+00		
8.998-01			9.998-01		
7.99e-01			7.99e-01		
6.01e-01			6.00e-01		
401e-01			4.00e-01		
2.00e-01	1		2,00e-01		
0.00e+00	1		0.00e+00	2	

Fig-33: Contours of velocity magnitude for symmetrical inlet cyclone

Cyclone Types	Velocity Ma	<mark>gnitude (</mark> m/s)	Tangential Velocity (m/s)	
	Minimum	Maximum	Minimum	Maximum
Single Inlet Cyclone	0	3.98	-3.52	2.88
Symmetrical Inlet Cyclone	0	4.00	-3.68	2.92

Table-4:	Velocity	magnitude	for both	cyclone	models
Labic-4.	velocity	magintude	101 0000	cyclone	mouch

Here, input velocity is 3 m/s and we can see from the table that the velocity magnitude and tangential velocity in the symmetrical inlet cyclone are more than a single inlet cyclone.



Fig-34: Contours of tangential velocity (m/s) for single inlet cyclone

2030+81			292e+00		
3.690+00		100	259e+00		
2.36+03		- 28 A.A.	2.26e+00		
1030+00			1.93e+00		
1.60e+00			1.60e+00		
1 370+00		B. A	1.376+00		
Q 40a.01			9.40+.01		
610c01			6100-01		
2,900,01			2,904,01		
500001			-5104-07		
1910/1			.3916.01		
.7110.01			.1110.01		
1.015-00			1.046400		- N
1.045100			-1.046700		
-1.3/5700			-1.318±00		
1,100,00			-1.708*00		
-21036700			-2.038+00		
-2.308700			-2.386+00		
-2.098400	100		-2.696+00		
-3.028+00	1		-3.02e+00	4	
-3.392+00	~~		-3.35e+00	-	
-1.088+00			-3.68e+00		

Fig-35: Contours of tangential velocity (m/s) for symmetrical inlet cyclone

7.3 Particle Tracks

Collection efficiency calculation for cyclone dust separator

- Number of particles trapped = 633
- Number of particles escaped = 627
- Number of particles injected = 1260
- Number of particles incomplete = 0

$$Efficiency = \frac{(Number of particles trapped)}{(Number of particles injected - Number of particles incomplete)}$$

Efficiency % =
$$\frac{633}{(1260-0)} \times 100 = 50.24\%$$
 for the particle size of 5 μm



Fig-36: Particle traces contours for single inlet cyclone model

8. CONCLUSION

After studying the prevailing literature and performing CFD¹ analysis on single and symmetrical inlet cyclone separator, the successive conclusion extract: The CFD analysis performed for both the cyclone models under the identical condition of pressure, velocity, material properties, and total flow rate. From the theoretical calculation and CFD analysis result, I found that the pressure drop value varies with the cyclone geometry. There is more pressure drop by symmetrical inlet cyclone separator as compared to single inlet cyclone separator. The error between the theoretical calculation and CFD analysis in a single inlet cyclone is 11.77% and in a symmetrical inlet cyclone separator is 13.98%. The result allowed me to observe that the tangential velocity and the velocity magnitude for the symmetrical inlet are higher than the single inlet cyclone separator. The efficiency calculation for the cyclone with dust collector is 50.24% for the particle size of 5µm.

9. REFERENCES

[1]. Khairy Elsayed 2011, Ph.D. thesis on Analysis and Optimization of Cyclone Separators Geometry using RANS and LES Methodologies. Pages from 20-160.

[2]. Abhijit Gaikwad and Dr.Shivarudraiah "CFD Analysis of Symmetrical Tangential inlet Cyclone Separator" IRJET Vol. 04, No- 08, Aug. 2017

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

[8]. C B Shepherd and C. E. Lapple. Flow pattern and pressure drop in cyclone dust collectors. Industrial and Engineering Chemistry, 32(9): 1246-248, September 1940

[9]. Optimization of the cyclone separator geometry for minimum pressure drop using mathematical models and CFD simulations By Khairy Elsayed and Chris Lacor, Chemical Engineering Science; 2010

[10]. Numerical study of gas-solid flow in a cyclone separator By B. Wang, D.L. Xu, K.W. Chu, A.B. Yu, Applied Mathematical Modelling 30 (2006) 1326–1342

¹ CFD – Computational Fluid Dynamics