Performance analysis of a cyclone separator using CFD

N.N.W RATHNAYAKE

(159262V)

Degree of Master of Science in Sustainable Process Development

Department of Chemical and Process Engineering

University of Moratuwa
Sri Lanka

June 2018

DECLARATION OF THE CANDIDATE AND SUPERVISOR

"I declare that this is my own work and this thesis does not incorporate without acknowledgement any material previously submitted for a degree or diploma in any other university or institute of higher learning and to the best of my knowledge and belief it does not contain any material previously published or written by another person except where the acknowledgement is made in the text.

Also, I hereby grant to university of Moratuwa the non-exclusive right to reproduce and distribute my thesis, in whole or in part in print, electronic or other medium. I retain the right to use this content in whole or part in the future works."

………………………… ………………………..
Signature Date

The above candidate has carried out research for the Master Dissertation under my supervision.

…………………………… ……………………….
Signature of the supervisor Date
Dr. Mahinsasa Narayana
Senior Lecture Grade II

ABSTRACT

Cyclone separator is a well-recognized, cost effective procedure of particle separation which used in many industrial works. As in cement industry, this cyclone separator used in order to separate calcium carbonate ($CaCO_3$) particles from hot gas. Apart from that, it also used to pre heat $CaCO_3$ particles by cyclone riser duct and to produce calcium oxide (CaO) (calcinations). Both of these procedures take place within the cyclone separator simultaneously. The efficiency of the cyclone separator determined by many factors such as cyclone dimensions & geometry, particle diameter & density and gas velocity. In this study, we considered about the effect of following 2 parameters on the efficiency of our fabricated cyclone separator. They are, Air flow velocity (inlet velocity) and Particle diameter. Experimental data were taken from the INSEE cement plant at Puttalam. Our experimental setup was the four stage preheater cyclone zone at the INSEE cement plant. Experimental data were taken from the bottom cyclone of the, Four Stage Pre Heater Cyclone Zone at the INSEE cement plant and figured the optimum values for those parameters to enhance the efficiency of the cyclone separator. CFD (Computational Fluid Dynamics) analysis also involved in to figure the optimum values for same parameters. In CFD analysis, for two phase air & calcium carbonate dust mixture, both multiphase ((k-epsilon, RNG (Re Normalization Group), wall function)) & discrete phase models have been used. Using multiphase model, we could plot contours of velocity, volume fraction and etc, of the individual phases. The Discrete model enabled us to track particles. This helped us to study collection efficiency by changing particle diameters & inlet velocities. It appeared that the final results of the experimental data and the CFD analysis were quite similar.

AKNOWLEDGEMENTS

I am very grateful to my supervisor Dr. Mahinsasa Narayana, Senior Lecture, Department of chemical and process Engineering University of Moratuwa, for this enormous support and guidance from the initiation to the end of my research work. He always encouraged me throughout the work and guided me to get a fruitful output.

My sincere thank also goes to Prof. A Rathnasiri, Head of the Department-Department of Chemical and Process Engineering of University of Moratuwa, for teaching Computational Fluid Dynamics in an interesting manner and giving essential basic theoretical background of CFD.

My special thanks go to Mr. Gayan Harshana, Production and Process Manager of INSEE Lanka (Pvt) Ltd, for encouraging me to get knowledge about cement cyclone and for his enormous support given throughout the research to fulfill this work successfully.

My immense gratitude goes to my loving parents for guiding me to face all challenges in the life and for their encouraging words in every down step.

Finally, I would like to thank my beloved wife who has always been a part of my life, for encouraging me and for being patient during my research period. Without her valuable support this may, not be achieved.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF TABLES PAGE

LIST OF ABBREVIATIONS

Abbreviation	Description
$CaCO_3$	Calcium Carbonate
CaO	Calcium Oxide
F_c	Centrifugal force
E_{ij}	Component of rate of deformation
CFD	Computational Fluid Dynamic
ρ_f	Density of the fluid
DPM	Discrete Phase Model
ϵ	Dissipation
Y_p	Distance from point to the wall
V_r	Distance per time
F_d	Drag force
μ	Dynamic viscosity of the fluid
μ_t	Eddy viscosity
E	Empirical constant
g_i	Gravitational acceleration
LES	Large Eddy Simulation
U_p	Mean velocity of the fluid at the near- wall node
V_r	Outward radial velocity
ρ_p	Particle density
d_p	Particle diameter
PSD	Particle Size Distribution
U_{pi}	Particle velocity
RNG	Re Normalization Group
R_e	Reynolds number

RSM	Reynolds Stress Model
RSTM	Reynolds Stress Turbulence Model
rpm	Rounds per minutes
q	Specific heat consumption
V_t	Tangential velocity
k	Turbulence kinetic energy
k_p	Turbulence kinetic energy at the near- wall node
U_i	Velocity component in corresponding direction
V_p	Volume of the particle
K	Von Carman constant

LIST OF APPENDICES

1. INTRODUCTION

This description will give you a further idea about the use of cyclone separator in the cement industry and its' advantages and disadvantages, Operating principle, Mathematical model description& force analysis. Apart from that this will also give an overall idea about our work on cyclone separator. Cyclone separators are used in many industrial works as a pollution control method because of their cost effective nature.

Cyclone separator uses centrifugal force along with low amount of pressure via turning movement for particle separation. The selected substance is pushed at elevated levels into the pipe entering the device. The robust form of the filter makes the incoming substance spines into a vortex. Bigger more textured particles are swept towards the outer barriers of the cyclone. Then it spun in the air as heavy particles are pushed into another section. The thinner particles are released on the top. The process taken by the model uses a constant flow.

The cyclone separator holds a greater reputation in many industries due to its numerous advantages. Main advantages of the cyclone separator are as follows;

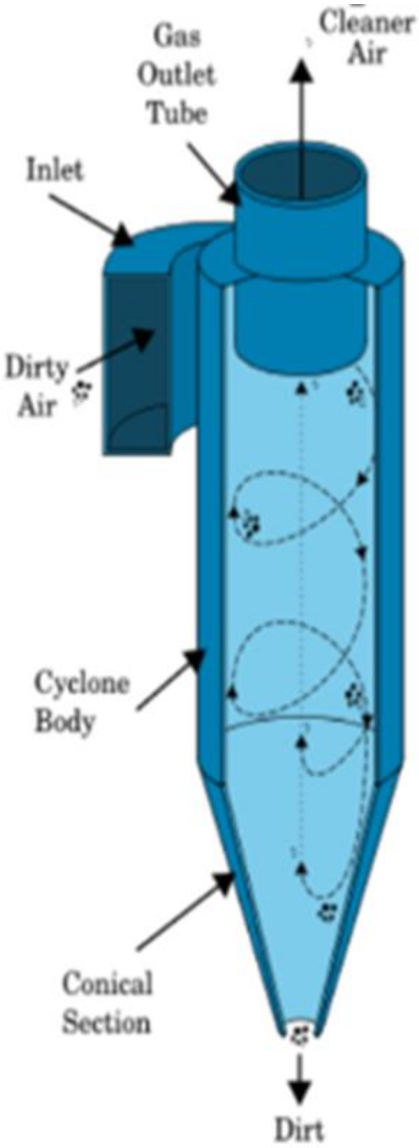
1. Need minimum maintains.
2. Ability to operate at high temperature.
3. Less affected by the climate conditions.
4. Low capital cost.
5. Easy to transport.

Low collection efficiency and high operating cost are the disadvantages of the cyclone separator.

1.1 OPERATIONAL PRINCIPLE

Cyclone dust separator is forming in centrifugal force caused by vortex movement of hot gas. The multiphase blend of gas and dust particles is supplied to top of the barrel. Vortex flow of the blend through the cyclone leads to concentration of the solid phase move towards walls of the outer cylinder & slides downward to the bottom of the device. Gas phase, is reversed and transferred upward in a smaller inner spiral. Then cleaned gas is released from the top through a vortex-finder tube.

Figure 1.1: cyclone process condition



1.2 STEADY STATE FORCE ANALYSIS

V_t = Tangential velocity
V_r= Outward Radial Velocity

Assuming stroke's law, the drag force in the outward radial direction is opposing the outward velocity on any particle in the inlet stream.

$$F_d = -6\pi r_p \mu v_r \qquad (1.1)$$

Using ρ_p as the particle density, the centrifugal component in the outward radial direction is

$$F c = \frac{m v_t^2}{r} \qquad (1.2)$$

$$= \frac{4}{3}\pi \rho_p r_p^3 \frac{v_t^2}{r} \qquad (1.3)$$

The buoyancy force component is in the inward radial direction. It is in the opposite direction to the particle's centrifugal force because it is on a volume of fluid that is missing compared to the surrounding fluid. Using ρ_f for the density of the fluid, the buoyant force is,

$$F_b = -V_p \rho_f \frac{v_t^2}{r} \qquad (1.4)$$

$$= \frac{4}{3}\pi r_p^3 \times \frac{v_t^2}{r} \rho_f \qquad (1.5)$$

In this case, V_p is equal to the volume of the particle. Determining the outward radial motion of each particle is found by setting Newton's second low of motion equal to the sum of these forces.

$$m \frac{dv_r}{dt} = F d + F c + F b \qquad (1.6)$$

To simplify this, we can assume the particle under consideration has reached a terminal velocity. In example, acceleration(dv_r/dt) is zero. This occurs, when the radial velocity has caused enough drag force to counter the centrifugal & buoyancy forces. This simplification changes our equation to,

$$F_d + F_c + F_b = 0 \tag{1.7}$$

Which, can be expanded to

$$-6\pi r_p \mu v_r + \frac{4}{3}\pi r_p^3 \frac{v_t^2}{r}\rho_p - \frac{4}{3}\pi r_p^3 \frac{v_t^2}{r}\rho_f = 0 \tag{1.8}$$

Solving for v_r we have,

$$V\,r = \frac{2}{9} \times \frac{r_p^2}{\mu} \times \frac{v_t^2}{r}(\rho_p - \rho_f) \tag{1.9}$$

That if the density of the fluid is greater than the density of the particle, the motion is (-), towards the center of rotation & if the particle is denser than the fluid the motion is (+), away from the center. In most cases, this solution is gives as a guidance in designing a separator, while actual performance is evaluated and modified empirically.

In non – equilibrium conditions when radial acceleration is not zero, the general equation from above, must be solved.

Rearranging terms we obtained,

$$\frac{dv_r}{dt} + \left(\frac{9}{2}\frac{\mu}{\rho_p r_p^2} v_r\right) - \left(1 - \frac{\rho_f}{\rho_p}\right)\frac{v_t^2}{2} = 0 \tag{1.10}$$

Since v_r is distance per time, this is a 2nd order differential equation of the form $X^{II} + C_I X^{I} + C_2 = 0$

Experimentally it is found that the velocity component of rotational flow is proportional to r^2, therefore,

$$V_t \; \alpha \; r^2 \tag{1.11}$$

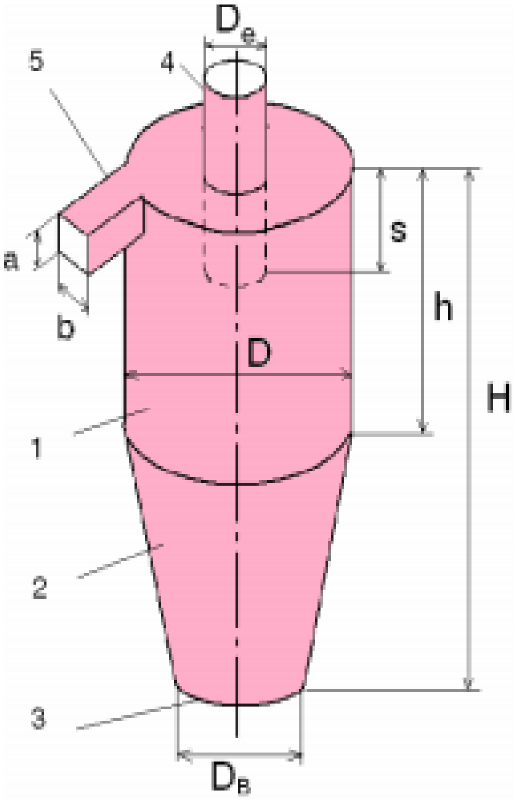
This means that the established feed velocity controls the vortex rate inside the cyclone, & the velocity at an arbitrary radius is therefore,

$$u_r = u_{in}\frac{r}{R_{in}} \tag{1.12}$$

Subsequently, given a value for V_t possibly based upon injection angle, and a cutoff radius, a characteristic particle filtering radius can be estimated, above which particle will be removed from the gas stream.

1.3 THEORETICAL AND CFD EFFICIENCY EQUATION ON CYCLONE SEPARATOR

Figure 1.2: cyclone with tangential entry



1 –cylindrical body
2 –cone shaped body
3 –outlet of solids particle
4 –outlet of gas
5 –gas supply hole doped

See Appendix -A Theoretical efficiency equation on cyclone separator

CFD FRACTIONAL EFFICIENCY EQUATION

Fractional efficiency equation=Number of particle trapped/ (number of particle injected-number of particle incomplete) (1.13)

1.4 MATHEMATICAL MODEL DESCRIPTION

1. K – epsilon; RNG, standard wall function
2. Discrete phase

1.4.1 K – Epsilon Model

For turbulence kinetic energy: - **k**

$$\frac{\partial(\rho\, k)}{\partial\, t} + \frac{\partial}{\partial\, X_i}(\rho\, k\, u_i) = \frac{\partial}{\partial\, X_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial\, k}{\partial\, X_j}\right] + 2\mu_t E_{ij} E_{ij} - \rho\, \epsilon \qquad (1.14)$$

For dissipation $\boldsymbol{\epsilon}$

$$\frac{\partial(\rho\, \epsilon)}{\partial\, t} + \frac{\partial}{\partial\, X_i}(\rho\, \epsilon\, u_i) = \frac{\partial}{\partial\, X_j}\left[\frac{\mu_t}{\sigma_\epsilon}\frac{\partial\, \epsilon}{\partial\, X_j}\right] + C_{1\epsilon}\,\frac{\epsilon}{k}\, 2\, \mu t\, E_{ij} E_{ij} - C_{2\epsilon}\,\frac{\rho\, \epsilon^2}{k} \qquad (1.15)$$

Rate of change of **k** or $\boldsymbol{\epsilon}$
\+
Transport of **k** or $\boldsymbol{\epsilon}$ by convection

=

Transport of **k** or $\boldsymbol{\epsilon}$ by diffusion
\+
Rate of production of **k** or $\boldsymbol{\epsilon}$
–
Rate of destruction of k or $\boldsymbol{\epsilon}$

(1.16)

Where;

$\boldsymbol{u_i}$**: -**Represents velocity component in corresponding direction

$\boldsymbol{E_{ij}}$**: -**Represents component of rate of deformation

$\boldsymbol{\mu_t}$**: -**Represents Eddy viscosity

$$\boldsymbol{\mu_t = \rho\, c_\mu \frac{k^2}{\epsilon}} \tag{1.17}$$

$\boldsymbol{\sigma_k, \sigma_\epsilon, c_{1\epsilon}, c_{2\epsilon}}$are constant.

1.4.2 RNG K-Epsilon Model

Transport equations

There are number of ways to write the transport equation for **k** & $\boldsymbol{\epsilon}$**.** A simple interpretation where buoyancy has neglected is;

$$\frac{\partial}{\partial\, t}(\rho\, k) + \frac{\partial}{\partial\, X_i}(\rho\, k\, u_i) = \frac{\partial}{\partial\, X_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial\, k}{\partial\, X_j}\right] + \mathrm{p_k} - \rho\epsilon \tag{1.18}$$

$$\frac{\partial(\rho\, \epsilon)}{\partial\, t} + \frac{\partial}{\partial\, X_i}(\rho\, \epsilon\, u_i) = \frac{\partial}{\partial\, X_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial\, \epsilon}{\partial\, X_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}\,\mathrm{p_k} - C_{2\epsilon}^{*}\frac{\rho\, \epsilon^2}{k} \tag{1.19}$$

Where;

$$C_{2\epsilon}^{*} = C_{2\epsilon} + \frac{c_\mu\, \eta^3\left(1-\eta/\eta_0\right)}{1+\beta\eta^3} \tag{1.20}$$

$$\eta = sk/\epsilon \tag{1.21}$$

$$s = \left(2s_{ij}s_{ij}\right)^{1/2} \tag{1.22}$$

1.4.3 Standard Wall Function

The law- of the-wall for mean velocity yield

$U^{*}=\frac{1}{k}\ln\left(E\,Y^{*}\right)$ (1.23)

Where

$$U*=\frac{U_p\,c_\mu^{1/4}k_p^{1/2}}{\tau_w/\rho} \qquad (1.24)$$

Is the dimensionless velocity

$$Y*=\frac{\rho c_\mu^{1/4}k_p^{1/2}y_p}{\mu} \qquad (1.25)$$

Is the dimensionless distance from the wall

K = Von Karman Constant

E = Empirical Constant

U_p = Mean velocity of the fluid at the near- wall node

k_p=Turbulence kinetic energy at the near- wall node

Y_p =Distance from point to the wall

μ =Dynamic viscosity of the fluid

1.4.4 Discrete Phase

Force balance equation of a single dispersed particle can be expressed as,

$$\frac{du_{pi}}{dt} = \mathrm{F_D}\,(\mathrm{U_i} - \mathrm{U_{pi}}) + \frac{(\rho_p - \rho)}{\rho_p}\mathrm{g_i} + \mathrm{F_i} \tag{1.26}$$

U_{pi}= Particle velocity

ρ_p=Particle density

g_i= Gravitational acceleration

$\mathrm{F_D}\,(\mathrm{U_i} - \mathrm{U_{pi}})$ = Drag force per unit mass due to relative slip between dispersed particle & gas

$$\mathrm{FD} = \frac{18\mu}{\rho_p d_p^2} \cdot \frac{c_D R_e}{24} \tag{1.27}$$

d_p= Particle diameter

R_e = Reynolds number

$$R_e = \frac{\rho d_p\,(u_p - u)}{\mu} \tag{1.28}$$

1.5 OBJECTIVES AND SCOPE

1.5.1 Objectives

To develop, CFD model to analyze cyclone separator efficiency. The collection efficiency of the cyclone separator used in our experimental setup found to vary between 55%-65% at stable operation conditions. The performance of the cyclone found to be below the designed value. The Original Designed value of the cyclone was 70%. Therefore, in this project our objectives happened to be as follows**,**

- To, find the effect of inlet velocity and particle diameter on collection efficiency of the cyclone.
- To, find the optimum collection efficiency of the cyclone separator.
- To, find optimum values for inlet velocity and particle diameter of the given cyclone separator to receive optimum collection efficiency.

1.5.2 Scope

Our main scope was to find the optimum inlet velocity & particle diameter for optimum collection efficiency of this fabricated cyclone separator. (See Appendix B- Flow chart of the bottom cyclone and kiln process)

1.6 RESEARCH GAP

Many research papers, which were, published regarding cyclone efficiency had only considered about the effect of cyclone geometry and the operational conditions of cyclone, on cyclone efficiency. However, this research work considered not only about the effect of cyclone geometry and the operational conditions, but also about the effect of kiln burning conditions of the cement industry.

(See appendix B -Flow chart of the bottom cyclone and kiln process)

1.7 METHODOLOGY

1.7.1 Experimental

- To find the optimum particle diameter, hot meal samples were taken from the inlet and the outlet of the cyclone and the cyclone efficiency was analyzed against hot- meal sample particle diameter.
- To find the optimum air- flow velocity, inlet air -flow velocity was changed by changing the fan rpm (rounds per minutes) and cyclone separator efficiency was analyzed against air- flow velocity.
- PSD (Particle Size Distribution) analysis was conducted using a laser diffraction instrument known as Mastersizer 3000. This equipment could analyze particle diameter between 0.3µm-3000µm by using Laser beams.

1.7.2 CFD analysis

- Modeling the separator and creating the mesh in workbench.
- Applying boundary conditions and studying the flow in fluent.
- Validating results with experimental data.
- Analysis using two phase k-epsilon and discreet phase models for air-dust mixture in cyclone.

2. LITERATURE REVIEW

Followings are the research papers, books and videos, which were referred for this research.

Collection efficiency of cyclone by using predictions of four theories, representing three different approaches. These predictions were compared with experimental results. They had received several fraction efficiency values by changing the inlet velocity and then, those results were taken in to a graph and compared with the theoretical curve. Fraction efficiency for particles smaller than 10 micrometers was taken in to account in another experiment. However finally, the theories needed to be evaluated over a range of cyclone design and operating conditions.[1]

Cyclone series in cement industry verify the performance based CFD technique. These cyclones, operated in high temperature with high solid loading capacity. The model was validated with experimental data on pressure drop and collection efficiency. The result obtained from this work had demonstrated the sensitivity of model to particle size. Thereby it showed that the cyclo code has a considerable potential to predict the collection efficiency.[2]

Performance parameter of the cyclone changing by cyclone body diameter. Numerical simulations had carried out using commercially available CFD code fluent 6.3.26 to predict the cyclone pressure drop and collection efficiency, as well as to investigate the flow field of scalars and vectors, at inlet velocity of 16.1ms^{-1}. It is found that with increase in diameter

- Pressure drop increases
- Collection efficiency increases
- Turbulence intensity increases by small amount
- Stability of flow in the core region of the cyclone[3]

Both numerical and experimental results for optimization and analysis of the separator efficiency, as well as fluid patterns in parallel vortex tube as one of the most interesting type of air separator.[4]

A mechanical device which will implement this cyclone & optimize collection efficiency by using compared experimental data and numerical simulation.

There are some positive impacts, those are as follow,

- About 20% decrease in pressure drop.
- Reduction in energy consumption.
- Positive effect on collection efficiency.

This mechanical device can be installed in any cyclone, since this is merely an adaptation of the vortex finder region.[5]

Using Reynolds Stress Model (RSM) model of turbulence, which is described in the relevant literature as the most suitable for cyclone dust separator, since it takes account of the phenomena associated with the flow of anisotropic nature. The complete analysis enabled the interpretation of measurement data and to find out the most beneficial cyclone structure design for the assumed flow of fluid and solids.[6]

CFD simulation of different scales of cyclone separators presented. The prediction of velocity field, pressure drop and particle separation efficiency from the CFD model were critically compared with different turbulent models and found the most accurate model to predict the cyclone flow, close to the experimental observation.[7]

The design and performance of hot cyclone for cleaning of particulate matter off from the hot producer gas. Particle size distribution being the key design parameter and similar trends were also observed with CFD analysis. However, the collection efficiency was much higher compared to experimental results, but matched closely with the theoretical results.[8]

The effects of inlet velocity & inlet concentration on pressure drop & emission. Cyclone performance models were developed using response surface methodology. Based on the obtained results, they had identified operating below inlet velocity to reduce pressure losses would reduce both the financial & the environmental cost of procuring electricity.[9]

A numerical approach using the Reynolds Stress Turbulence Model (RSTM) & large Eddy Simulation (LES) for turbulence closure is employed to study the effect of modeling of velocity fluctuations on prediction of collection efficiency of cyclones.[10]

The literature indicates that the cyclone efficiency is depended on the particle size, inlet velocity and particle density. The objective of this paper is to demonstrate the influence of the particle size and the input velocity of the gas into the cyclone over the collection efficiency.[11]

The literature reveals that the cyclone efficiency depends on the particle size from the mass of the mixtures heterogeneous solid-fluid. The input air velocity affects both the fan energy consumption and the dust collection efficiency. The objectives of these papers are to demonstrate theoretically the influence of the dimensions' solid particle and the input velocity into the cyclone over the collection efficiency.[12]

2.1 SUMMARY

Unlike convection application, cyclone for cement industry application is meant to process a gas of higher temperature (between 800^{o}c to 900^{o}c), containing CaO, $CaCO_3$ compounds with high dust particles. There are several theories proposed on particle collection efficiency in the cyclone by researches using different approaches and assumptions. Most of the researches had been carried out to investigate the cyclone geometry and operating conditions of the cyclone performance. However, in this research, the geometry of the cyclone was not taken in to account. This research was to find the optimum cyclone efficiency by changing parameters of the operating conditions such as particle diameter & inlet velocity. However, theories can calculate the collection efficiency & compare with our experimental data. Then we discovered optimum parameters by using CFD analysis. But CFD values were not exactly same, because the cyclone we used in this project was designed according to our plant conditions & requirements. However, we studied on many models and decided to use the K-epsilon (RNG, wall function) and Discrete Phase Model (DPM) models for our application.

3. TYPES OF CYCLONE DESIGNS

2D2D (Shepherd and Lapple, 1939), 1D3D (Parnell and Davis, 1979) cyclone designs are the most commonly used designs in industrial works. The D's in the 2D2D designation refers to the barrel diameter of the cyclone. The numbers preceding the D's relate to the length of the barrel and cone sections, respectively. Whereas the 1D3D cyclones have barrel length equal to the barrel diameter and a cone length of three times the barrel diameter. Previous research (Wang, 2000) indicated that, compared to other cyclone designs, 1D3D and 2D2D cyclones are the most efficient cyclone collectors for fine dust (particle diameters less than 100 micrometers).

Simpson and Parnell (1995) introduced a new low-pressure cyclone, called 1D2D cyclone, for the cotton ginning industry to solve the cycling-lint problem. The 1D2D Cyclone had a better design for high- lint content trash when compared with 1D3D and 2D2D cyclones.

Table 3.1: standard cyclone dimensions

	High efficiency		**Convention**		**High throughput**	
Cyclone type	1	2	3	4	5	6
Body diameter D/D	1.0	1.0	1.0	1.0	1.0	1.0
Height of inlet H/D	0.5	0.44	0.5	0.5	0.75	0.8
Width of inlet W/D	0.2	0.21	0.25	0.25	0.375	0.35
Diameter of gas inlet D_e/D	0.5	0.4	0.5	0.5	0.75	0.75
Length of vortex finder S/D	0.5	0.5	0.625	0.6	0.875	0.85
Length of body L_b/D	1.5	1.4	2.0	1.75	1.5	1.7
Length of cone L_c/D	2.5	2.5	2.0	2.0	2.5	2.0
Diameter of dust outlet D_D/D	0.375	0.4	0.25	0.4	0.375	0.4

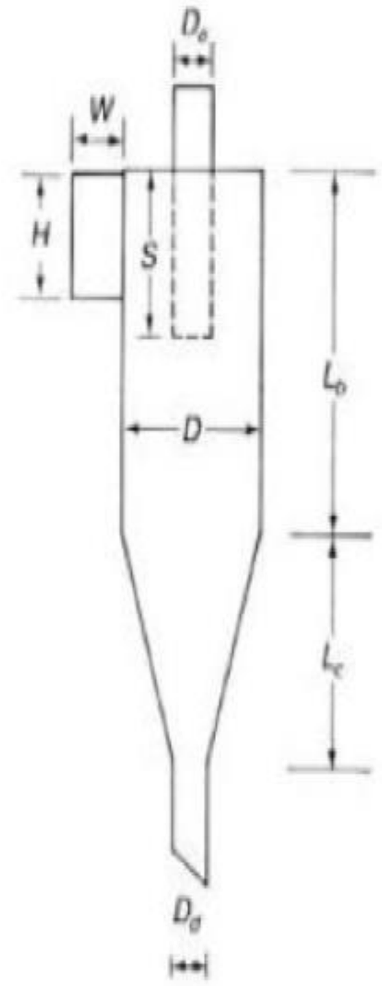


Figure 3.1: standard cyclone dimensions

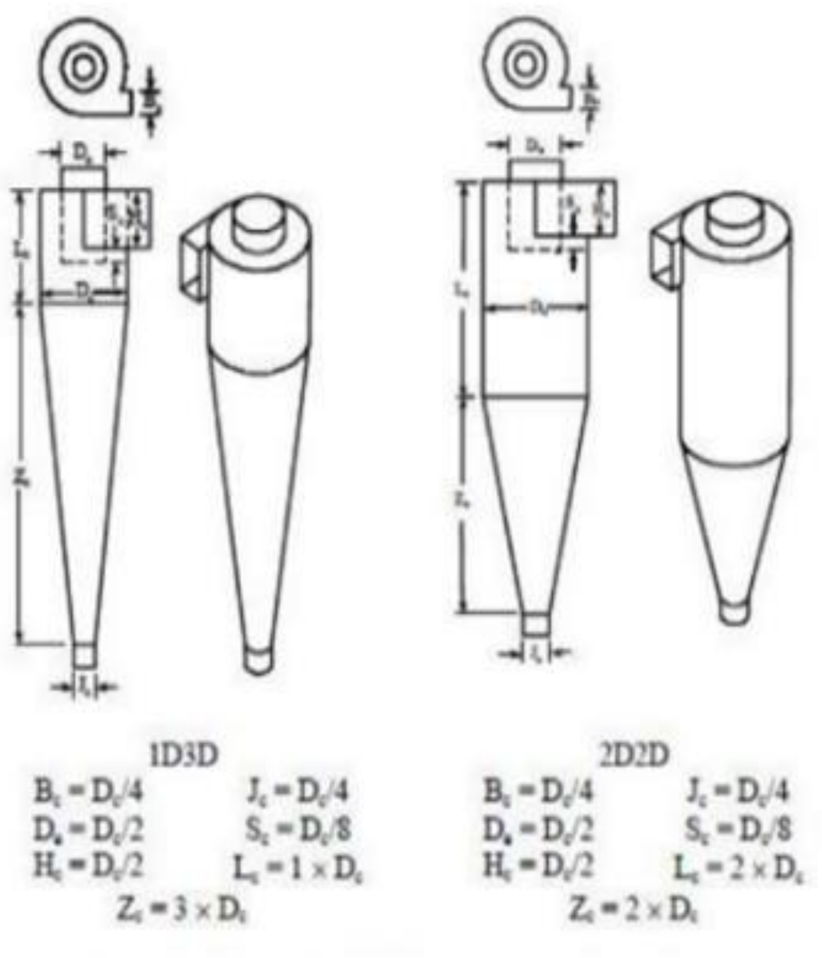


Figure 3.2: standard cyclone dimensions of 1D3D and 2D2D cyclone types

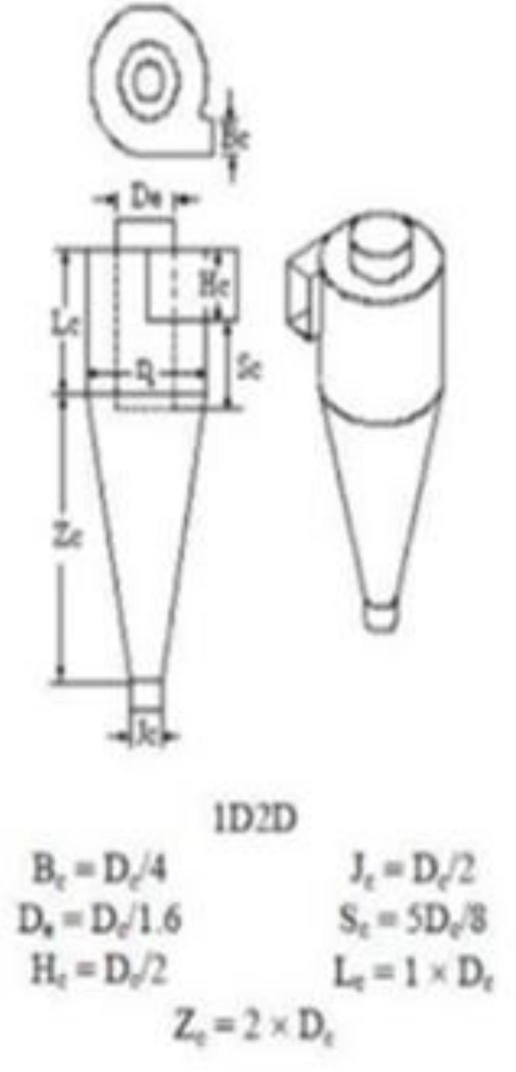


Figure 3.3: standard cyclone dimensions of 1D2D cyclone type

Source: Perry and Green[13]

3.1 DESIGN OF CYCLONE SEPARATOR

There are three types of cyclone designing classifications. These classifications are based on geometric proportions of cyclone dimensions. Those are, high efficiency cyclone, conventional efficiency cyclone & high through put cyclone. However, efficiency varies greatly with particle size and cyclone design. During the last few decades, advanced design work has greatly improved cyclone performance. Current literature from some of the cyclone manufacturers advertise cyclone that have efficiencies greater than 98% for particles lager than 5 microns, and others that routinely achieve efficiencies of 90% for particles lager than 15-20 microns.

Our cyclone's optimum values of each dimension mentioned by manufacture, the design values for the current design are summarized in the table.

3.1.1 Design condition

Parameters

Vessel diameter	4.4m
Cross section Inlet	3.8m^2
Cross section Dip tube	4.9m^2
Cross section vessel	15.3m^2
Dip tube diameter.	2.5m
Dip tube height	1.735m
Overall height	9.7m
Cylindrical height	3.7m
Meal pipe 4th cyclone to inlet chamber	10.3m

3.1.2 Design Operation condition

Inlet velocity	$18ms^{-1}$
Wall velocity	$31ms^{-1}$
Tangential velocity	$41ms^{-1}$
Kiln inlet velocity	$17ms^{-1}$
Pressure difference 4th cyclone	11mbar
Pressure difference across riser duct	3.5mbar
Pressure difference across entire cyclone	75-80mbar
Temperature	$830^{o}c$
Conical section angle	30^{0}

3.1.3 Design analysis

Inlet velocity for the bottom cyclone. (see Appendix B- Flow chart of the bottom cyclone and kiln process)

Gas flow at pre heater exit (dry kiln)

Total wet kiln exhaust gas (i.e. after full de carbonation) for a measured O_2 concentration % O_{2dry} & specific heat consumption q.

$$V_{\text{(kiln gas wet)}} [\frac{Nm^3}{kg_{cli}}] = (0.28 + \overbrace{0.28q}^{\boxed{\text{From raw meal}}}) + \underbrace{(0.27+ 0.25q)}_{\boxed{\text{From combustion}}} \overbrace{[\frac{\%o_{2dry}}{21-\%o_{2dry}}]}^{\boxed{\text{Excess air based on dry } o_2 \text{ measurement}}}$$

q $[\frac{MJ}{kg_{clin}}]$ = specific heat con m

% $O_{2\ dry}$ [%] = measured O_2 concentration of dry gas

Source: Process performance book

In our experimental data for cyclone,

Temperature = 860^{o}c + 273

=1133 k

Inlet pressure = -110 mmws

= -1.1 mbar

Normalize pressure =1atm

& temperature = 0^{o}c = 273 k

Calculation

V = (0.28 + 0.28q) + (0.27 + 0.25q) $\left(\frac{\%o_{2dry}}{21-\%o_{2dry}}\right)$

q = 3.85 (MJ/kg_{clin})

% $O_{2\ dry}$ = 2

1 hr. clinker = 45T

V= (1.358) + (1.2325) - $\frac{2}{19}$

= 1.358+0.13

=1.487

=$\frac{1.487 \times 45000}{3600}$

= 18.59 Nm^3/s

T = 1133k

P = -1.1 mbar

$$\frac{P_1 V_1}{T_1} = \frac{P_2 V_2}{T_2}$$

$$\frac{[1013+(-1.1\,)]V_1}{1133} = \frac{1013\times 18.59}{273}$$

V_1 = 77.2 m^3 /s

Inlet velocity = $\frac{77.2\ m^3/s}{3.84\ m^2}$

= 20.11 ms^{-1}

Mass flow rate in bottom cyclone

Kiln feed = 81 t/h

Kiln feed 72 t/ h ×Twin cyclone efficiency × (1-Twin cyclone calcination) ×Second cyclone efficiency × (1-Second cyclone calcination) ×Third cyclone efficiency × (1-Third cyclone calcination)

$72 \times \frac{93}{100} \times (1- \frac{1.44}{100}) \times \frac{85}{100} (1- \frac{4.78}{100}) \times \frac{75}{100} (1- \frac{11.7}{100})$

= 11.1 kg/s

3.1.4 Estimation of cyclone performance

Test 1

In this trial, we have used a constant dust load (11.1kg/s).

Data were collected by, changing inlet velocity by changing fan rpm and then recorded in following tables.

Table 3.2: Data collected when inlet velocity was17.22ms^{-1}

Stream A-Fan rpm 1150

Fluid	Gas with particle
Volumetric flow rate	66.12m^3/s
Inlet velocity	17.22ms^{-1}
Dust Load	11.1Kg/s
Mean Temperature	837 °C
Mean Pressure	1011.2mbar

Table 3.3: Data collected when inlet velocity was 16.92ms^{-1}

Stream B-Fan rpm 1120

Fluid	Gas with particle
Volumetric flow rate	64.97m^3/s
Inlet velocity	16.92ms^{-1}
Dust Load	11.1Kg/s
Mean Temperature	832 °C
Mean Pressure	1011.6mbar

Table 3.4: Data collected when inlet velocity was 16.55ms^{-1}

Stream C-Fan rpm 1100

Fluid	Gas with particle
Volumetric flow rate	63.55m^3/s
Inlet velocity	16.55ms^{-1}
Dust Load	11.1Kg/s
Mean Temperature	829 °C
Mean Pressure	1011.8mbar

In test 2 we have changed the dust load from 11.1 kg/s to 10.06 kg/s and data were collected by changing inlet velocity.

TEST 2

Table 3.5: Data collected when inlet velocity was 13.55ms^{-1}

Stream A-Fan rpm 1150

Fluid	Gas with particle
Volumetric flow rate	52.032m^3/s
Inlet velocity	13.55ms^{-1}
Dust Load	10.06Kg/s
Mean Temperature	820 °C
Mean Pressure	1011.4mbar

Table 3.6: Data collected when inlet velocity was 12.92 ms^{-1}

Stream B-Fan rpm 1120

Fluid	Gas with particle
Volumetric flow rate	49.61m^3/s
Inlet velocity	12.92ms^{-1}
Dust Load	10.06Kg/s
Mean Temperature	818°C
Mean Pressure	1011.5mbar

Table 3.7: Data collected when inlet velocity was12.45ms^{-1}

Stream C-Fan rpm 1100

Fluid	Gas with particle
Volumetric flow rate	47.808m^3/s
Inlet velocity	12.45ms^{-1}
Dust Load	10.06Kg/s
Mean Temperature	817 °C
Mean Pressure	1011.6mbar

4. EXPERIMENTAL SETUP AND RESULTS

4.1 EXPERIMENTAL SETUP

Our experimental setup was the four stage pre heater cyclone zone at the INSEE cement plant. Experimental readings were taken from the bottom cyclone in the series of the cyclone zone. Measurements of feed rate, pressure, temperature were recorded & hot meal samples taken from the bottom cyclone were analyzed. (see Appendix B-Flow chart of the bottom cyclone and kiln process)

Figure 4.1: Cyclone tower and kiln at INSEE cement plant

4.2 EXPERIMENTAL RESULTS

Hot meal samples were taken from the bottom cyclone & subjected to the PSD analysis.

TEST -01

Table 4.1: Experimentally analyzed data taken from test 1 stream A

Stream A

Particle size(µm)	PSD Analyzer (Mass fraction- cumulative value)
0.5	0.56
1	2.83
3	34.34
16	61.20
45	69.43
63	78.04
90	95.45
212	98.99

Table 4.2: Experimentally analyzed data taken from test 1 stream B

Stream B

Particle size(µm)	PSD Analyzer (Mass fraction- cumulative value)
0.5	0.40
1	2.26
3	29.91
16	57.76
45	66.19
63	75.16
90	94.23
212	98.42

Table 4.3: Experimentally analyzed data taken from

test 1 stream C

Stream C

Particle size(μm)	PSD Analyzer (Mass fraction -cumulative value)
0.5	0.40
1	2.07
3	28.43
16	56.09
45	64.61
63	73.78
90	93.51
212	97.98

TEST -02

Table 4.4: Experimentally analyzed data taken from test 2 stream A

Stream A

Particle size(μm)	PSD Analyzer (Mass fraction- cumulative value)
0.5	0.60
1	6.16
3	38.96
16	63.23
45	71.00
63	78.74
90	94.27
212	97.80

Table 4.5: Experimentally analyzed data taken from

test 2 stream B

Stream B

Particle size(µm)	PSD Analyzer (Mass fraction -cumulative value)
0.5	0.68
1	6.86
3	38.95
16	62.20
45	69.69
63	77.37
90	93.47
212	97.38

Table 4.6: Experimentally analyzed data taken from test 2 stream C

Stream C

Particle size(µm)	PSD Analyzer (Mass fraction -cumulative value)
0.5	0.80
1	6.97
3	37.01
16	59.20
45	66.61
63	74.42
90	91.88
212	96.36

PSD analyzer analyzes by using 100 g hot meal sample & feeding material content 90 µm=14+-1%, 212 µm=1.5% +-0.2 & other mass percentages were included 63 µm,45 µm,16 µm ,3µm.

The cyclone supposed to get maximum collection efficiency in the gas-solid medium, generated during the Calcination process. In separate experimental setups, using cyclone, two trials were conducted with six different inlet velocities (17.22ms^{-1}, 16.92ms^{-1}, 16.55ms^{-1},13.55ms^{-1},12.92ms^{-1},12.45ms^{-1}). During these trials, $CaCO_3$ particles were converted in to CaO (calcination process) in the cyclone. To evaluate performance parameters, the inlet velocities & particle diameters were measured. Flow rates & densities were constant throughout the trials.

The data shown in above tables indicate cyclone efficiency for different inlet velocities & particle diameters. It is evident that the collection efficiency of all cases increases as the particle size &cyclone inlet velocity increase. So we could find optimum condition considering other operation parameters (kiln conditions, burning conditions). This improvement can be attributed to the combined effect of enhanced cyclone inlet velocity & particle diameter range changes.

5. CFD ANALYSIS

See Appendix C: Ansys operation procedure for CFD Analysis**.**

Number of particle tracked-451

Number of iteration-800

TEST 1: Stream A

Table 5.1: Test 1 stream A data CFD analysis

Particle size(μm)	Particle trapped	Particle incomplete
0.5		
1		
3	0	283
16	296	155
45	0	269
63	4	447
90	7	251
212	450	1

TEST 1: Stream B

Table 5.2: Test 1 stream B data CFD analysis

Particle size(μm)	Particle trapped	Particle incomplete
0.5		
1		
3		
16	0	451
45	0	399
63	413	38
90	451	0
212	446	5

TEST 1: Stream C

Table 5.3: Test 1 stream C data CFD analysis

Particle size(μm)	Particle trapped	Particle incomplete
0.5		
1		
3		
16	14	437
45	242	209
63	0	81
90	88	363
212	0	444

TEST 2: Stream A

Table 5.4: Test 2 stream A data CFD analysis

Particle size(μm)	Particle trapped	Particle incomplete
0.5		
1		
3		
16	0	334
45	0	228
63	0	180
90	81	362
212	449	1

TEST 2: Stream B

Table 5.5: Test 2 stream B data CFD analysis

Particle size(μm)	Particle trapped	Particle incomplete
0.5		
1		
3		
16	0	451
45	0	14
63	375	76
90	0	28
212	451	0

TEST 2: Stream C

Table 5.6: Test 2 stream C data CFD analysis

Particle size(μm)	Particle trapped	Particle incomplete
0.5		
1		
3		
16	0	438
45	9	442
63	0	423
90	430	21
212	0	345

6. RESULT ANALYSIS

After running simulation, in this chapter, graphs & contours are plotted. Comparisons are made with experimental as well as simulation results of these models. Validation of experimental results are done & turbulence model for optimum operational parameters was found.CFD method tracked 451 particles for each diameter.

TEST 1: Stream A

Table 6.1: Test 1 stream A data analysis between experimental and CFD

Particle size(µm)	PSD Analyzer (Mass fraction -cumulative value)	CFD Analysis (Fractional separation efficiency)
0.5	0.56	
1	2.83	
3	34.34	0
16	61.20	100
45	69.43	0
63	**78.04**	**100**
90	95.45	3.5
212	98.99	100

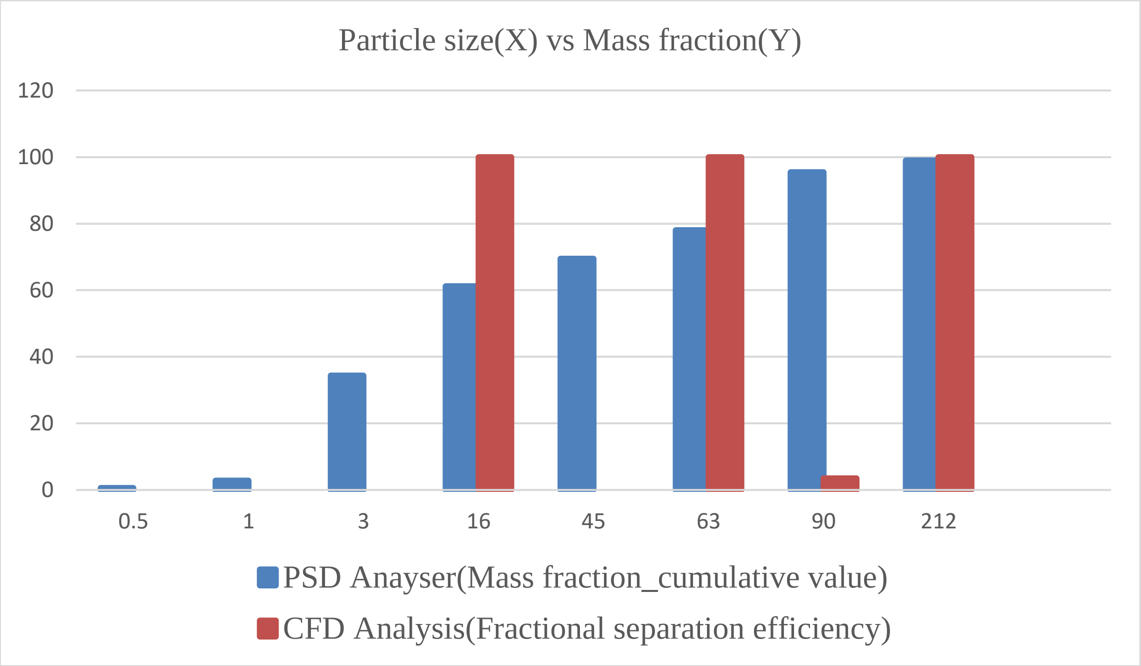


Figure 6.1: Test 1 stream A data analysis between experimental and CFD in a graph

Stream B

Table 6.2: Test 1 stream B data analysis between experimental and CFD

Particle size(μm)	PSD Analyzer (Mass fraction -cumulative value)	CFD Analysis (Fractional separation efficiency)
0.5	0.40	
1	2.26	
3	29.91	
16	57.76	0
45	66.19	0
63	75.16	100
90	94.23	100
212	98.42	100

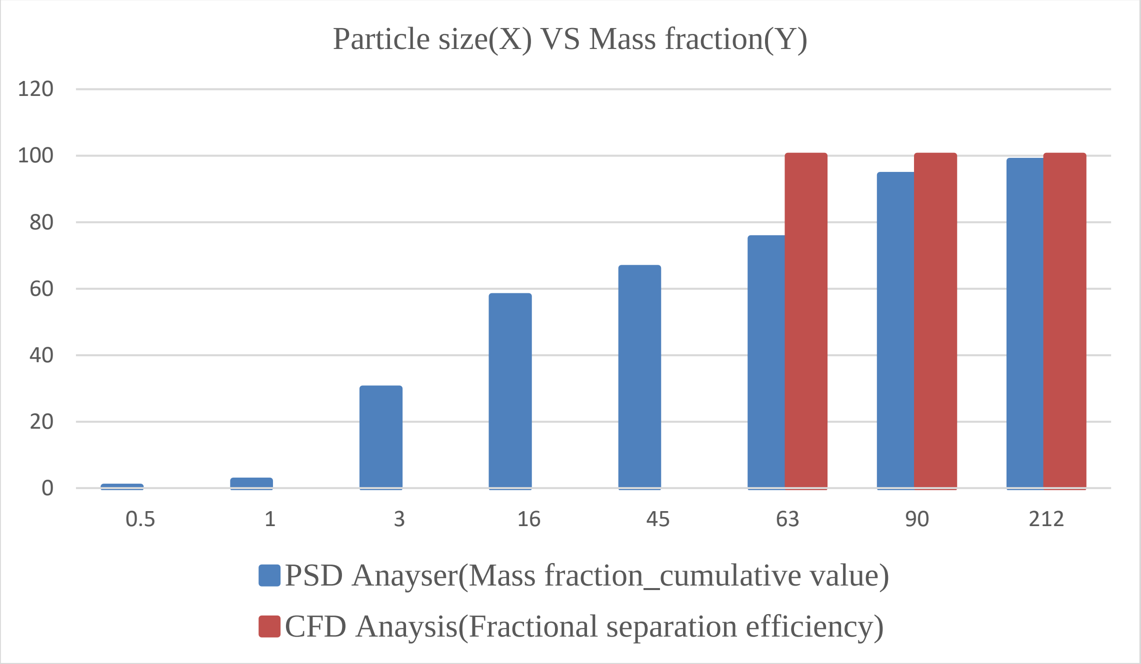


Figure 6.2: Test 1 stream B data analysis between experimental and CFD in a graph

Stream C

Table 6.3: Test 1 stream C data analysis between experimental and CFD

Particle size(μm)	PSD Analyzer (Mass fraction -cumulative value)	CFD Analysis (Fractional separation efficiency)
0.5	0.40	
1	2.07	
3	28.43	0
16	56.09	100
45	64.61	100
63	73.78	0
90	93.51	100
212	97.98	0

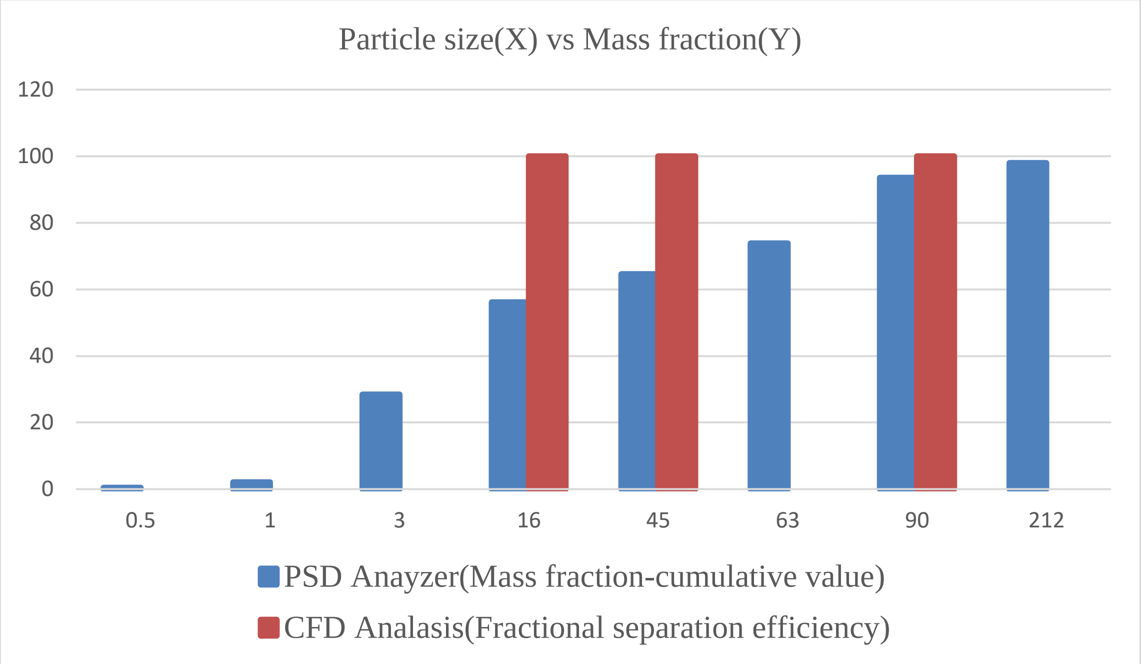


Figure 6.3: Test 1 stream C data analysis between experimental and CFD in a graph

TEST -02

Stream A

Table 6.4: Test 2 stream A data analysis between experimental and CFD

Particle size(µm)	PSD Analyzer (Mass fraction -cumulative value)	CFD Analysis (Fractional separation efficiency)
0.5	0.60	
1	6.16	
3	38.96	
16	63.23	0
45	71.00	0
63	78.74	0
90	94.27	100
212	97.80	99.7

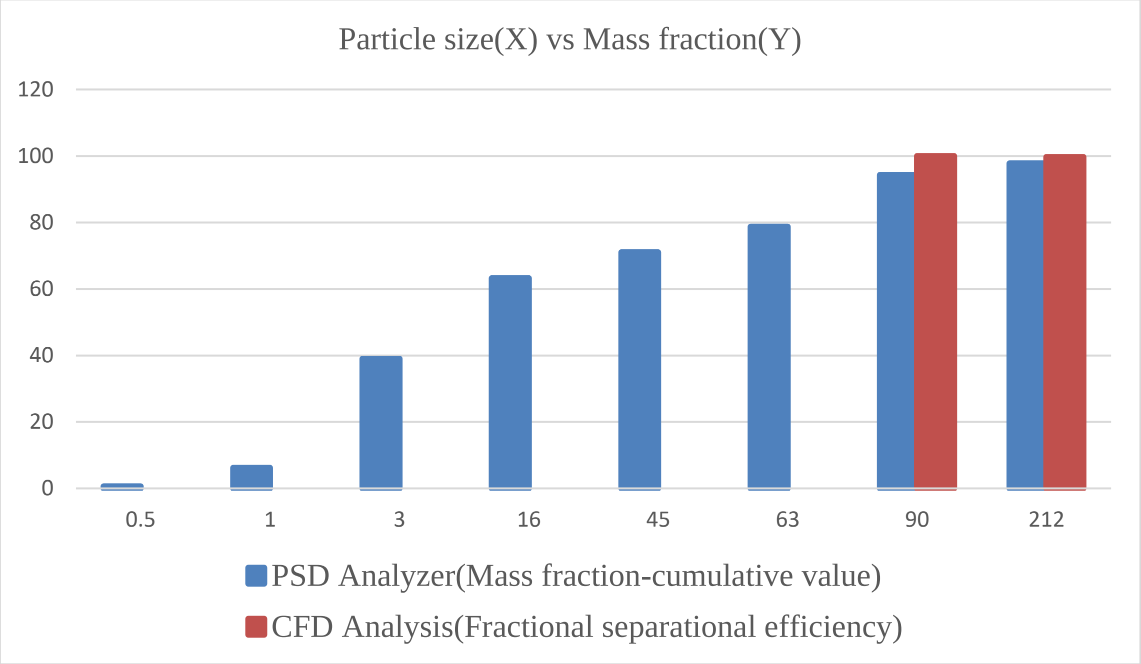


Figure 6.4: Test 2 stream A data analysis between experimental and CFD in a graph

Stream B

Table 6.5: Test 2 stream B data analysis between experimental and CFD

Particle size(µm)	PSD Analyzer (Mass fraction -cumulative value)	CFD Analysis (Particle tracked) cumulative value
0.5	0.68	
1	6.86	
3	38.95	
16	62.20	0
45	69.69	0
63	77.37	100
90	93.47	0
212	97.38	100

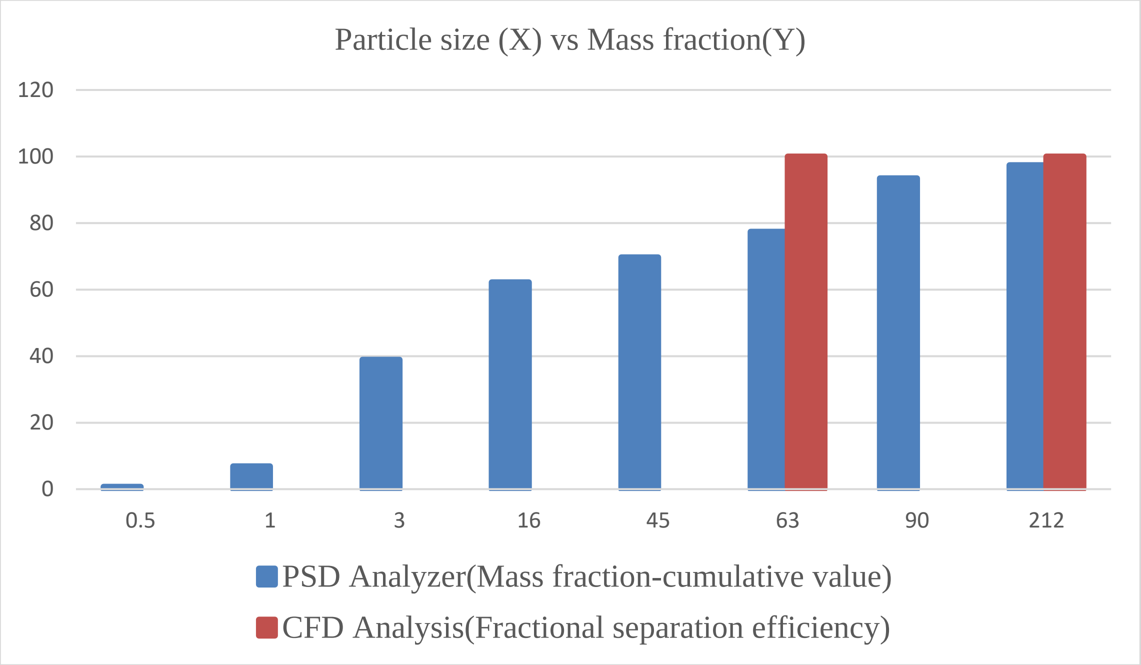


Figure 6.5: Test 2 stream B data analysis between experimental and CFD in a graph

Stream C

Table 6.6: Test 2 stream C data analysis between experimental and CFD

Particle size(µm)	PSD Analyzer (Mass fraction -cumulative value)	CFD Analysis (Particle tracked) cumulative value
0.5	0.80	
1	6.97	
3	37.01	
16	59.20	0
45	66.61	100
63	74.42	0
90	91.88	100
212	96.36	0

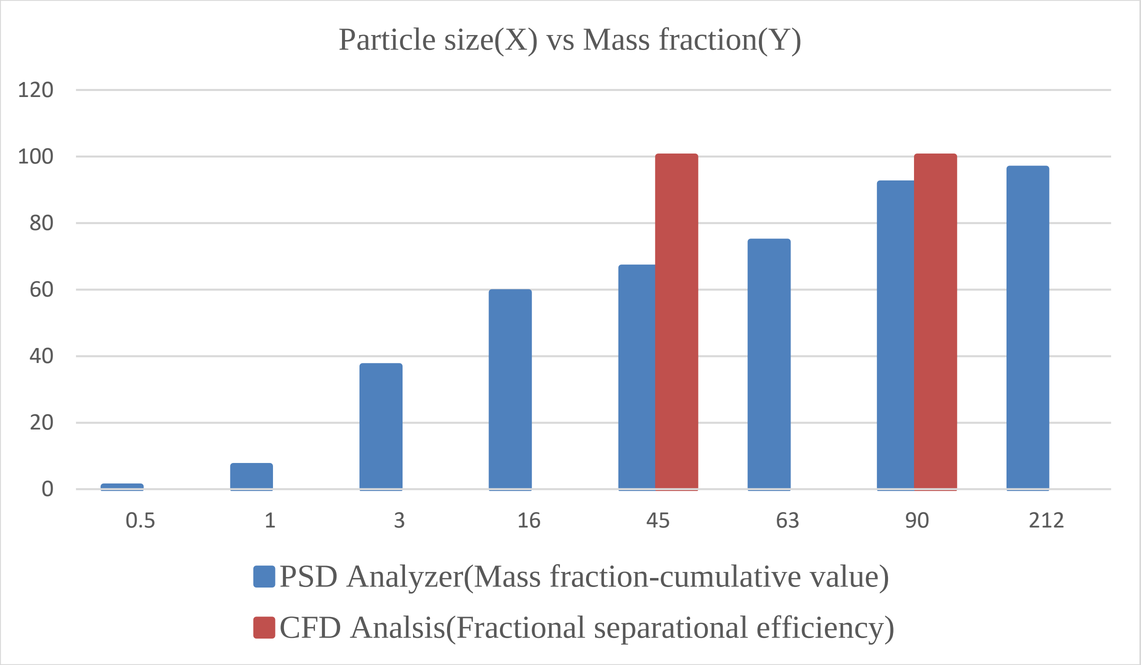


Figure 6.6: Test 2 stream C data analysis between experimental and CFD in a graph

Table 6.7: Theoretical experimental simulation Summary of the efficiency value

Velocity(ms^{-1})		Theoretical		Experimental		Simulation	
		Efficiency Value		PSD Analysis value		CFD Analysis value	
		d-63μm	d-45μm	d-63μm	d-45μm	d-63μm	d-45μm
Test A	**17.22**	**74.9**	67.1	**78.04**	69.43	**100**	0
	16.92	74.6	66.8	75.16	66.19	100	0
	16.55	74.5	66.6	73.78	64.61	0	100
Test B	**13.55**	**72.25**		**78.74**		**0**	0
	12.92	71.68		77.37		100	0
	12.45	71.24		74.42		0	100

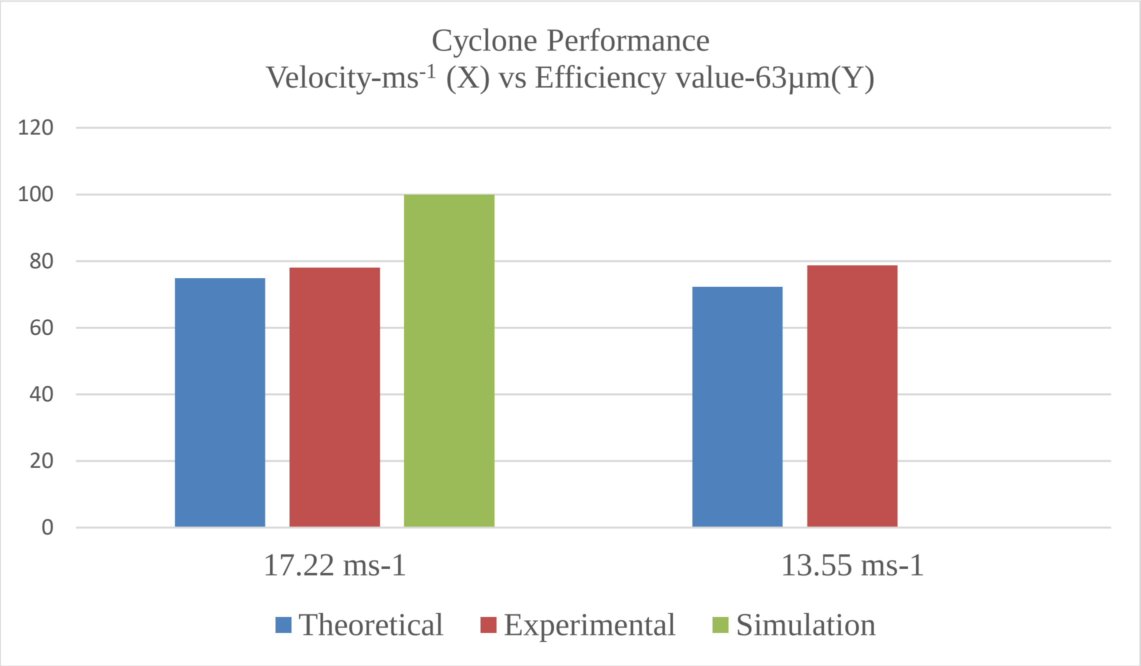


Figure 6.7: Cyclone performance (theoretical, experimental and simulation)

6.1 VALIDATION TWO PHASE GAS AND CALCIUM CARBONATE FLOW

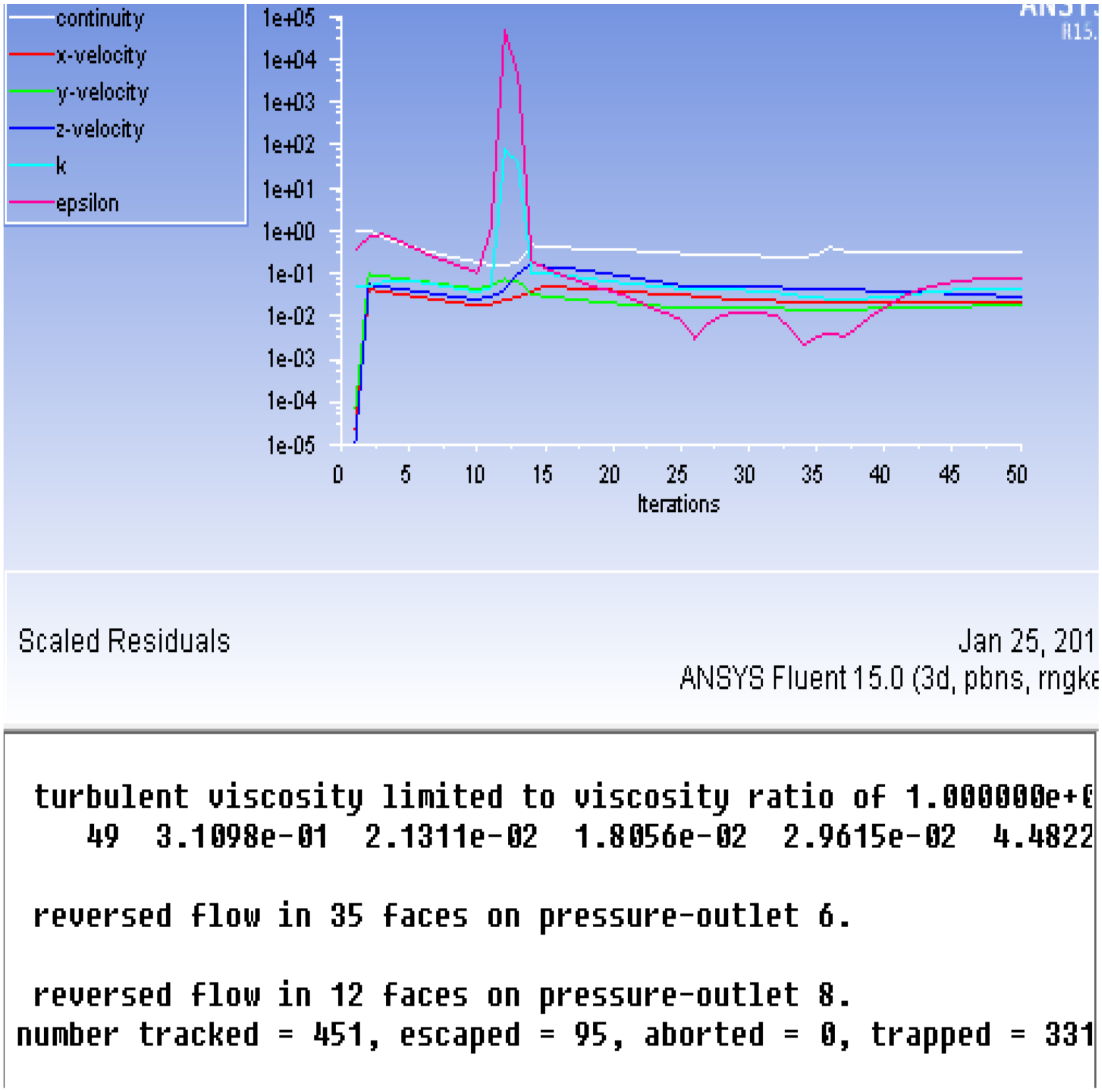
6.1.1 Multiphase model

This model was compared with data obtained under the experimental conditions at INSEE cement plant at Puttalam. Experiments were performed with air under high temperature & pressure conditions, as a function of flow rate & dust load of (11.1 kg/s &10.06 kg/s) Solid particles were calcium carbonate powder density of 800 kg/m^3 in this condition & a mass particle diameters between 0.3μm to 212μm. The cyclone with an inlet velocity & body dimension was studied. In this research, the collection efficiency of the high solids flux inlet in the cement industry cyclone was validated with ANSYS FLUENT WORKBENCH 15.0 software. This difference seemed to be caused by irregularly shaped particles & particle size distribution.

Followings are the volume fraction plots of air & limestone dust respectively. They show the areas of air & limestone dust dominance in the separator. Air shows prominence in the upper region while dust shows prominence in the lower region.

6.1.2 Discrete phase model - Particle tracking

Escaped means out of the upper outlet, ‘trapped’ are the particles collected & ‘incomplete’ are the particle still revolving in the cyclone.



```
 turbulent viscosity limited to viscosity ratio of 1.000000e+
    49  3.1098e-01  2.1311e-02  1.8056e-02  2.9615e-02  4.4822

 reversed flow in 35 faces on pressure-outlet 6.

 reversed flow in 12 faces on pressure-outlet 8.
number tracked = 451, escaped = 95, aborted = 0, trapped = 331
```

Figure 6.8: Particle tracking in ANSYS FLUENT

Figure 6.8 compares the grade collection efficiency of the numerical results obtained with the ANSYS FLUENT WORKBENCH 15.0 with that of the experimental data for different inlet velocities. As expected, the experimental & predicted efficiency of all cases increase as the particle size & inlet velocity increase. Good agreement is observed between the numerical calculation of the two phase model & the experimental data.

According to our plant conditions, feeding material has large mass fraction below 90um in size. But in CFD analysis we tracked same particle numbers (451) for each particle diameters. So we found collection efficiency by using fluent software. we should take cumulative value for all particle diameter.

6.2 RESULTS FOR THE CURRENT STUDY

When the inlet velocity & particle diameter increase, it shows the collection efficiency increases. (Both CFD & experimental analysis).

In this process, most amount of particles are in the range of 63 µm,45 µm, and 16 µm.

According to test 1 experimental and numerical results, it shows that stream A gives the highest collection efficiency. In this operating condition, we can choose optimum particle diameter 63 µm & inlet velocity 17.22 ms^{-1}. According to test 1 optimum operation conditions, we can manage collection efficiency 74.9%. If basic cyclone conical section has no angle, it is easy to increase collection efficiency.

According to test 2 experimental and numerical results, it also shows that stream gives the low collection efficiency compare with test 1.

Furthermore, when analyzing these two results, we have identified that the test 1stream A results got the highest theoretical, experimental and CFD efficiencies.

6.3 THE CONSOLIDATED RESULTS

In here consolidated results of cyclone collection efficiency based on CFD computation presented. The CFD results of collection efficiency are slightly different than the experimental results as shown in figure: 6.7.

1. These differences seemed to be caused by irregularly shaped particles, particle size distribution & particle density. Which were prevented, an accurate modeling of the interaction between particles of difference diameter.
2. Also the model adopted for interaction between the phases, which considered only the gas –solid interaction. The influence of the solid-solid interaction in the performance of cyclone has been studied.
3. In this analysis we considered cyclone performance only. However, we also should consider about kiln operating conditions & burning conditions in the system. Therefor this reason we identified that the experimental results were lower than numerical results.
4. In the normal operation, coating profile buildup in the cyclone. So that actual area in the cyclone is lower than the original value. This reducing area was not included in this CFD analysis.
5. The reason for the efficiency difference between CFD and experimental analysis is, in the CFD analysis 451 particle has been injected per one particle diameter. But in the experimental analysis 100 particle have been injected per every particular particle range.

7 CONCLUSION

In this research, a model that is based on CFD techniques was used to optimize the operational parameters for the cyclone in a cement industry to verify the performance of cyclone collection efficiency. In cement industry cyclones have different shapes & operate at high temperatures with high solid loading flows.

The model was validated with experimental data on inlet velocity & collection efficiency. The results obtained in this research, sensitivity of the model to particle size. Thereby it showed that the ANSYS FLUENT WORKBENCH 15.0 has a considerable potential to predict the collection efficiency. With particle size distribution (PSD) being the key design parameter in the inlet gas stream. However, collection efficiency was much higher, compared to experimental results. The collection efficiency is highly sensitive to both particle density and PSD. There were two possibilities for errors, one was due to agglomeration of the particles & second was due to particles not being spherical.

In the other hand the need of improvement in the cyclone collection efficiency was identified, and redesigned the geometry of the cyclone.

Particle tracking correctly predicted the flow patterns for different size particles. Path lines gave the exact simulations as seen by experimentally.

8 REFERENCES

[1]. Amit Kumar Singh Parihar, Chandrasekhar Joshi and S Sridhar, *“Performance of cyclone for producer gas cleaning: experimental and modeling studies*” Siemens Corporate Research and Technologies, Bangalore, India

[2]. D. Noriler, A.A. Vegan, C. Soars, A.A.C. Barros, H.F. Meier, & M. Mori, “A New role to reduce pressure drop in cyclones using computational fluid dynamic *techniques”*

[3]. Joao Jaime, & Milton Mori, “*Computational fluid dynamics (CFD) Analysis of cyclone separator connected in series.”* Department of chemical engineering, university of Blumenau (FURB), Blumenau Santa Catarina SC 89010-971, Brazil.

[4]. John Dirge, & David Leitch, *“Cyclone collection efficiency; comparison experimental results with theoretical predictions*.” (https://doi.org/10.1080/027868285/08959066)

[5]. Lacier Sing Brar, & Amit Kumara, “*CFD simulation of cyclone separators with different diameters” Department* of mechanical engineering B.T.T Mesra, Ranchi 835215, India.

[6]. Marek Wasilewski, W.S.K.I, Jerzy, “*Application of computational fluid dynamics to optimization of cyclone dust separators, operated in the cement industry*” DUDA- faculty of production engineering and logistics, Opole. University of technology, Poland

[7]. P. A Funk, K. Elayed, K.M Yeater, G.A Holt, D.P White lock, “*Could cyclone performance improve with reduced inlet velocity*?”

[8]. Seyed Ehsan Rafiee, M.M Sadeghiazad, *“Experimental and 3D CFD analysis on optimization of geometrical parameters of parallel vortex tube cyclone separator”*

[9]. Simon LiziaParaschiv, Spiruparaschiv, “*Analysis of cyclone collection efficiency*” Dunarea de jos university, Galati, Romania

[10]. Sujeet Kumar Shukla, Prashant Shukla, Pradyumna Ghosh, *“The effect of modeling of velocity fluctuations on prediction of collection efficiency of cyclone separators”*

[11]. V. Singh, S. Srivastava, R. Chaval, V. Vitankar, B. Basu, M.C. Agrawal, *“Simulation of gas-solid flow and design modification of cement plant cyclones”*

[12]. *“The effect of the particle size and input velocity on cyclone separator process* Bulletin of the Transylvania university of Brasov. Series 11: forestry. Wood industry. Agriculture food engineering. vol.4(53) no.2-2011M.MARINUC F.RUS

[13]. R.H. Perry and D. W Green, “Gas solid separation,” in Perry’s Chemical Engineer’ Handbook ,7th ed. McGrown-Hill,1997, ch.17, pp.17.23-17.31.

[14]. You tube video’s

CFD ANSYS Tutorial – Cyclone separator theory and simulation using DPM | Fluent

CFD Cyclone Simulation

Cyclone Separator Performance Validation with ANSYS – Solid Trust

CFD Tutorial-Cyclone Separator Eulerian model

APPENDIX A

Theoretical efficiency equation on cyclone separator

$$\eta = 1 - \exp\left\{-2\left[\left(\frac{8.\frac{\frac{\pi D^2}{4}.h.\frac{\frac{\pi D^2}{4}(h-s)+\frac{\pi D^2}{4}\left(\frac{ln+s-h}{3}\right).\left(1+\frac{d}{D}+\frac{d^2}{D^2}\right)-\frac{\pi D_e^2\,ln}{4}}{2}}{D^3}}{a-b}\right).\left[\left(\frac{\rho d_p^2 v}{18\,\mu D}\right).1.5\right]\right]^{\frac{1}{3}}\right\}.100$$

cyclone length should be 100% of it's diameter.

η = efficiency of the cyclone

D = diameter cylindrical body

h = upper height of the cyclone

s = depth of penetration of purified gas hose

ln = natural length of cyclone

D_e = outer diameter of central tube exhaust gas purified

ρ = solid particle density

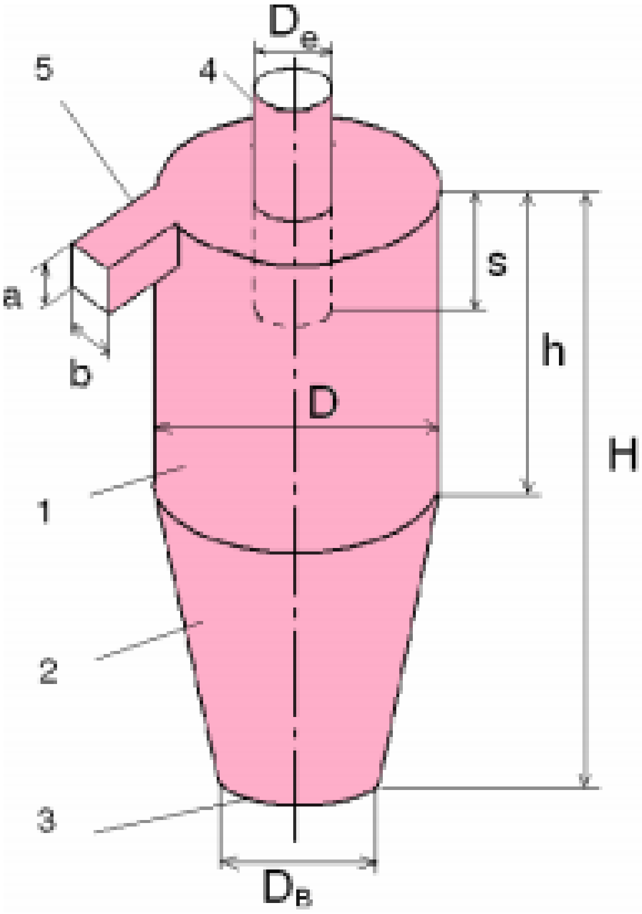
a = height of the cyclone inlet

b = width of the cyclone inlet

μ = gas mixture viscosity

d_p = dimensions of particles

v = the average inlet velocity



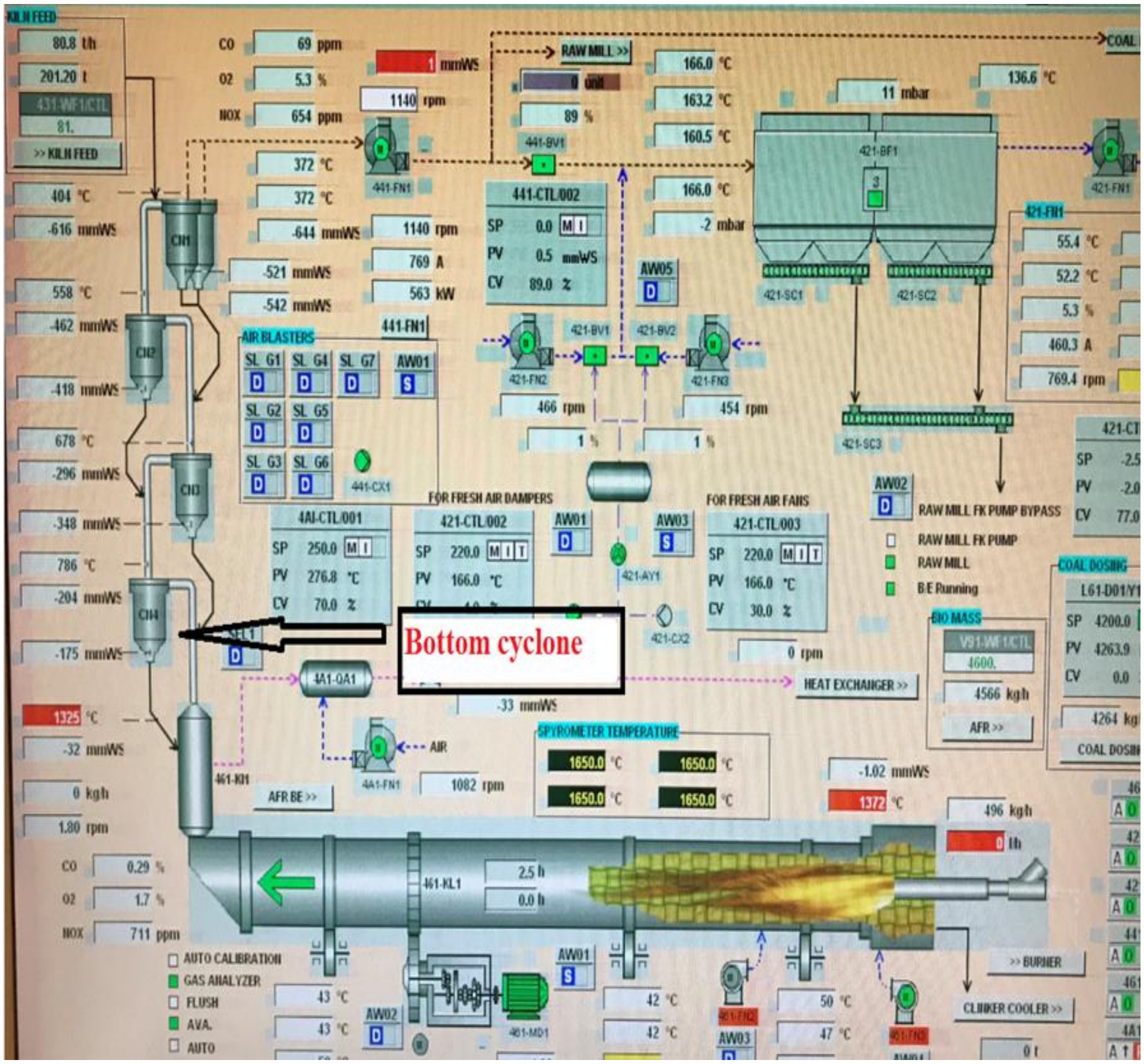
If conical section has an θ angle

$$\eta = 1 - exp\left[-\frac{\rho_p\, Q D_p^2\, \theta_f}{9\mu r_2 w(r_2^2 - r_1^2) ln(r_2/r_1)}\right]$$

D_{P}- -Dimension of particles
r_2-Inner radius of cylindrical body
r_1-Outer diameter of central tube
ρ_p -Particle density
μ-Gas mixture viscosity
Q-Gas flow rate

APPENDIX B

Flow chart of the bottom cyclone and kiln process



APPENDIX C

Ansys operational procedure for CFD Analysis

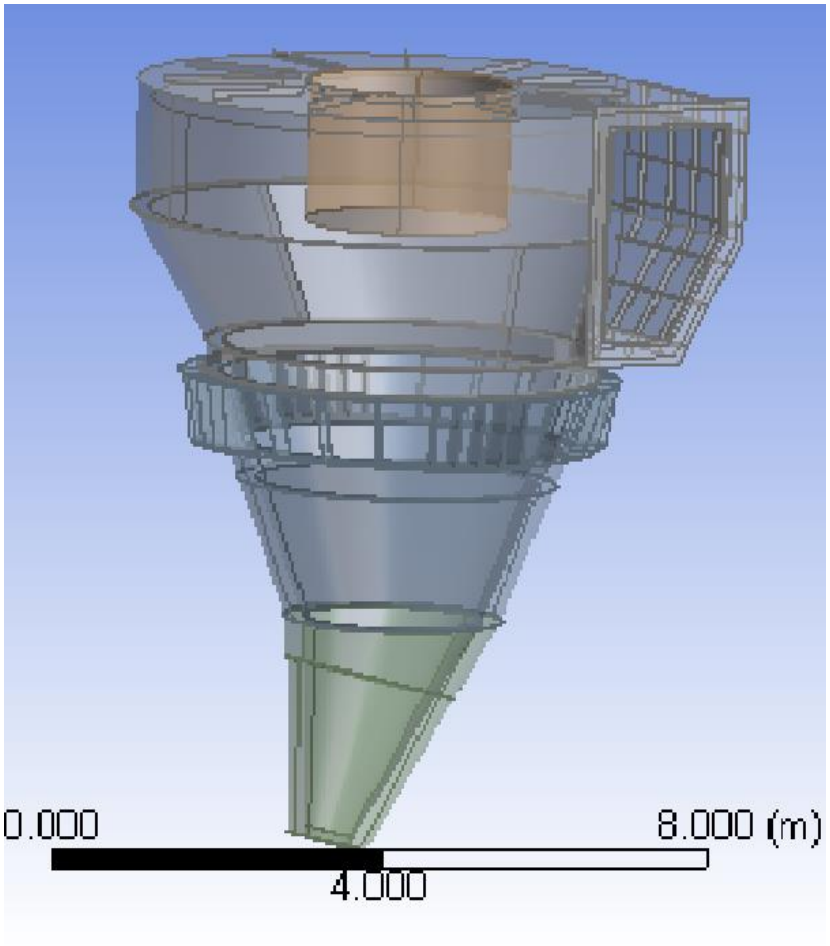
This appendix deals with the computer simulation of the flow, inside a cyclone separator with the help of various turbulence models. Detailed steps show the procedure to run simulation on different turbulences as well as multiphase models.

Given below are the guidelines for making geometry, generating mesh, giving solver parameters and finally post-processing.

Creating Geometry

STEPS

1. Draw the geometry in solid works then save the file
2. Open workbench and select geometry options.
3. Import the solid works file
4. Click the generate button



Cyclone geometry in ANSYS FLUENT

Creating Mesh

Generation of appropriate mesh for cyclone geometry is not a trivial task. The flow inside a cyclone is fully 3 dimensional and complex. Proper simulation of such flow requires careful treatment of the mesh.

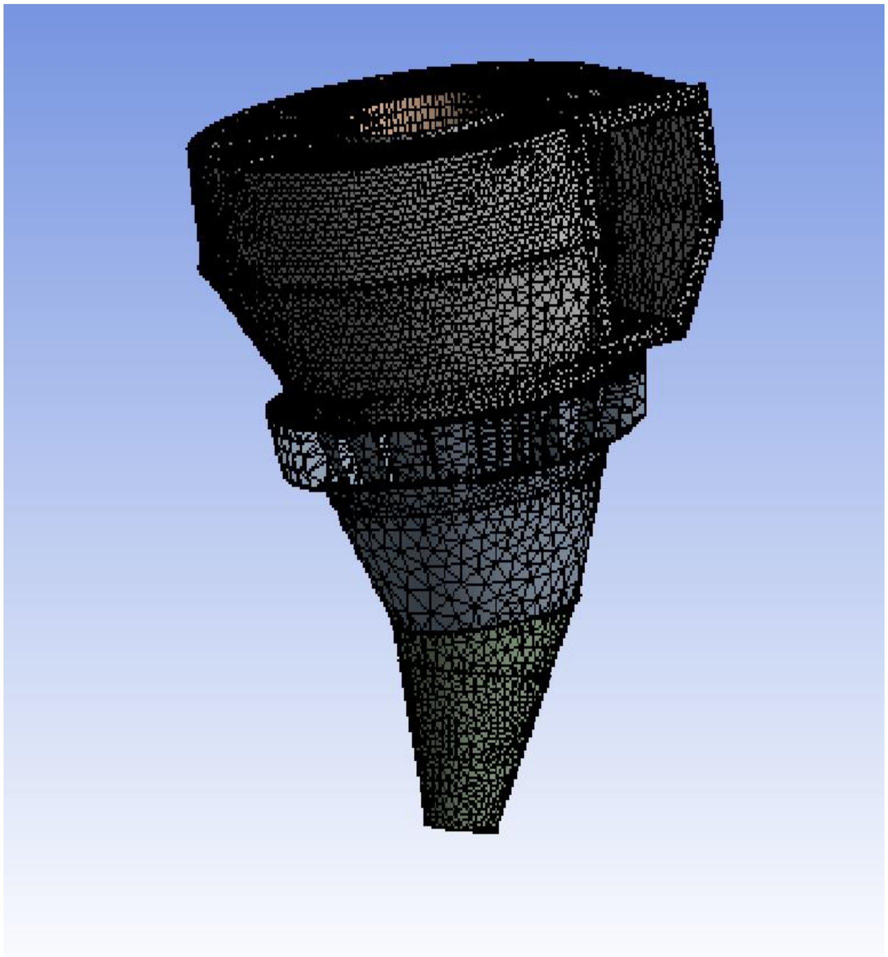
STEPS

1. Click mesh icon, then open the detail of mesh
2. Then change physics preference to the mechanical in the detail of the mesh

3. Click the sizing in the detail of mesh & change relevance center to the coarse
4. Click right click & generate mesh

After generating mesh, we need to find our boundaries

Selected two surfaces as outlet number 1 by creating name selection & entry point selected to be inlet side & other outlet side as outlet number 2



Cyclone mesh in ANSYS FLUENT

Creating Setup

Make sure to green check on the mesh before click on the Setup

Click the setup

1. Click General &choose the steady state simulation &acceleration gravitation -9.81ms^{-2}

2. Go to model & select k-epsilon, RNG model & check the swirl dominated flow, then press ok
3. Click discrete phase on the models & check the interaction with continuous phase & update DPM sources every flow in iteration. keep the maximum number of steps 50000 & step length factor at 5
4. Then Create new injection & choose injection type as surface, then select inlet for the release from surface. Choose calcium carbonate as material & diameter distribution is uniform. Then specify X-velocity of the particle 17.22ms^{-1}& change diameter of the particles up to 3um to 300um.Then put the total flow rate keep as 11.1 kg/s.
5. Press ok
6. Go to material & air as a fluid
7. Go to cell zone condition & go to operating condition, then change operating pressure as 101000Pascal
8. Press ok
9. Select boundary condition & specify velocity inlet, then change velocity magnitude between 17.22 ms^{-1} to 16.92ms^{-1}. Make sure discrete phase boundary condition reflect& press ok. Then go to outlet no 1 as pressure-outlet & similarly pressure outlet no 2 with zero-gauge pressure.

10. Press ok
11. Click the solution method & already select
    a. Pressure velocity coupling scheme-SIMPLE
    b. Spatial Discretization
        - Gradient – Least squares cell based
        - Pressure –second order
        - Momentum-second order upwind
        - Turbulence kinetic energy-second Order upwind
        - Turbulence dissipation rate- second order upwind

12. Select solution controls then change all parameters to 0.1 or 0.2

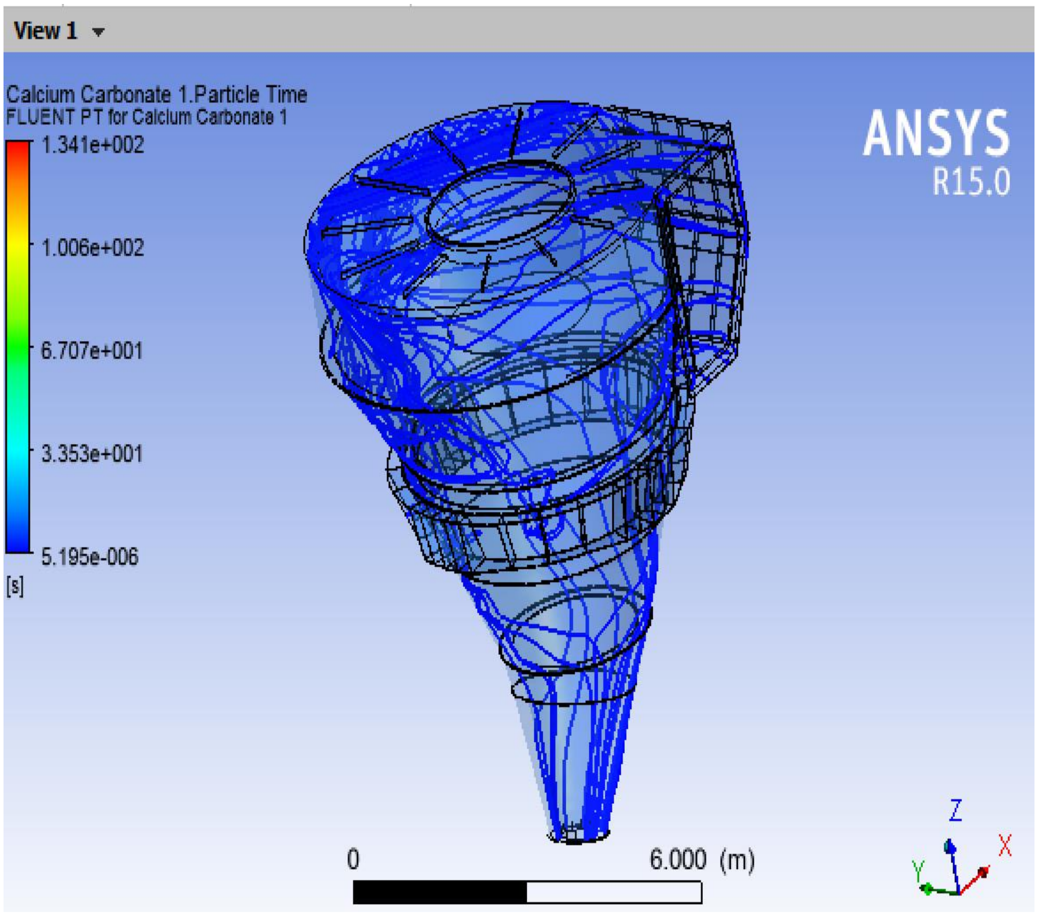
13. Then go to solution initialization & do not use hybrid initialization in this instance because it will run very slowly & it will not converge quickly. So use the standard initialization & computed from the inlet once choose the inlet, crash initialize

14. Go to run calculation & choose approximately 800 number of iteration at beginning once finished the calculation.

Particle tracking

In most of the cases mass load of the inert particle is small comparing to transport gas. If heat transfer between phases is not involved, particle can be, without considerable error, traced within a gas phase in the frame of post processing. It means that first we simulate fluid flow of a gas phase.

Basically tracking of the particle trajectories can go to file & then export particle history data then choose injection 0 & the file type CFD force & write a file somewhere in directory. So in this case call file name & press ok & press write. Once a file was written go to CFD post. Then select the wall document, double click on it & choose the color & transparency in there. Once this is done import fluid particle track file & will show the result. If selected, then we can see the particle tracking or the trajectories of all particles. We can change by double clicking on here, then go to color & then we can change it to variable & it will be based on a residence time. If we have it & its particle time, press Apply & we can see how particles swell inside of the cyclone & get out. We have few particles going outside as well as with air to outlet. Then if we want to see vortex, there the core of the vortex itself, go to location & then go to vortex core region & then defines what level of swirling strength & press Apply.



Calcium carbonate particle time in ANSYS FLUENT