

Design Modification and CFD Formulation of 1D3D Mechanical Cyclone for Optimal Particle Collection Efficiency

N. D. Azike^{1,*}, S.O. Onyegebu²

^{1,2} Department of Mechanical Engineering, University of Nigeria, Nsukka, Enugu State, NIGERIA.

Abstract

Standard 1D3D mechanical cyclone with standard inlet velocity of 16m/s and cylinder diameter of 0.1524m was modified using Downhill simplex method to develop four modified cyclone designs. Computational Fluid Dynamics (CFD) formulation involving model design, meshing, CFD simulation and postprocessing was carried out on the mechanical cyclones. ANSYS Fluent software was used for the CFD formulation using fine mesh of the default minimum mesh size of the CFD mesher. Cornstarch was used as inlet particulate matter. Reynolds Stress Turbulence Model (RSTM) was used to model the swirling turbulent flow while Discrete Phase Model (DPM) was used to track about 10,000 particles through the simulated cyclones. The DPM result of each mechanical cyclone was used to calculate its CFD, particle collection efficiency result. Standard 1D3D mechanical cyclone obtained particle collection efficiency result of 87.38% using cornstarch and default minimum mesh size for CFD simulation while modified 1D3D mechanical cyclone one (1) recorded the optimal CFD particle collection efficiency result among modified 1D3D mechanical cyclones with 94.00%. Its convergence iteration point also showed improvement in simulation time compared to the standard 1D3D mechanical cyclone. Consequently, it was concluded that, design modification in line with CFD formulation offers an alternative and powerful approach to modeling 1D3D mechanical cyclones performance.

Keywords: Mechanical cyclone; Design modification; Downhill simplex method; Computational Fluid Dynamics (CFD) formulation; Discrete phase modeling (DPM); static pressure; velocity flow field.

1.0 INTRODUCTION

Mechanical cyclones are fluid-solid separation machines for removing particulate matter from air stream or gas. They are mainly used in the field of air pollution control and gas–solid separation. They are also used for aerosol sampling and in industrial applications.

1D3D mechanical cyclones have a barrel length equal to the barrel diameter and cone length equal to three times the barrel diameter. They were reported [1] as the most efficient for fine dust collection. Figure 1 shows 1D3D mechanical cyclone design configurations.

There is a major problem associated with 1D3D mechanical cyclones, even if it has been reported as the most efficient cyclone for fine dust. The problem is that of poor particle collection efficiency. The major reason behind this problem is that the standard design methods used in developing these cyclones such as the Classical Cyclone Design (CCD) method [2,3] by Lapple and/or the Texas A&M Cyclone Design (TCD) method [4] by Parnell

predicts cyclones with poor particle collection efficiency [5].

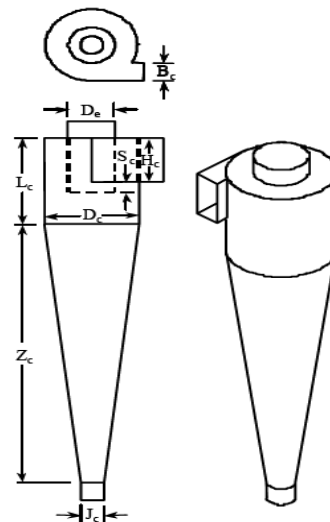


Figure 1: 1D3D mechanical cyclone configurations.

For example, it was reported [6] that the overall particle collection efficiency of 1D3D mechanical cyclone,

*Corresponding author (Tel: +234 (0) 8037889731)

Email addresses: ndukaazike@gmail.com (Azike N.D.), samuel_onyegebu@yahoo.co.uk (Onyegebu S.O)

with cylinder diameter of 0.1524m, using flyash as inlet particulate matter was 85.2%. The result indicates that 1D3D mechanical cyclones have low particle collection efficiency and that there is need for improvement.

Despite the problems associated with 1D3D mechanical cyclones they are still used in all types of power and industrial applications such as; flour mills, cement plants, pharmaceutical industries, steel mills, petroleum coke plants, metallurgical plants, saw mills, vacuum cleaners and other kinds of facilities that process dust.

2.0 METHODOLOGY

The motivation behind this study is to develop 1D3D mechanical cyclones with optimal particle collection efficiency using design modification and Computational Fluid Dynamics (CFD) formulation. Consequently, standard 1D3D mechanical cyclone was designed using standard inlet velocity of 16 m/s and cylinder diameter of 0.1524m (6in.) with cornstarch as inlet particulate matter. The design of the standard 1D3D mechanical cyclone using the standard design method is shown on table 1 below.

Table 1: Design of standard 1D3D mechanical cyclone.

DESIGN PARAMETER	DESIGN FORMULAR	DESIGN VALUE
Inlet velocity	Standard	16m/s
Cylinder diameter	Standard	0.1524m
Inlet width	$B_c = D_c / 4 = 0.1524 / 4$	0.0381 m
Inlet height	$H_c = D_c / 2 = 0.1524 / 2$	0.0762 m
Inlet area	$A_{in} = B_c * H_c$	0.0029 m ²
Gas outlet height below inlet height	$S_c = D_c / 8 = 0.1524 / 8$	0.01905 m
Gas outlet height	$h_g = H_c + S_c$	0.09525 m
Gas outlet diameter	$D_e = D_c / 2 = 0.1524 / 2$	0.0762 m
Dust outlet diameter	$J_c = D_c / 4 = 0.1524 / 4$	0.0381 m
Cylinder section height	$L_c = 1 * D_c = 1 * 0.1524$	0.1524 m
Cone section height	$Z_c = 3 * D_c = 3 * 0.1524$	0.4572 m
Inlet length	$L_i = 1.0 * D_c = 1.0 * 0.1524$	0.1524m
Gas outlet length	$L_g = 0.618 * D_c = 0.618 * 0.1524$	0.09418m

Downhill simplex method was used to carry out design modification on the standard 1D3D mechanical cyclone to develop modified 1D3D mechanical cyclone designs. Six key multivariate process parameters which significantly affect particle collection efficiency of mechanical cyclones was used for the design modification. There are; inlet velocity, inlet area, cylinder height, cone height, vortex finder height and vortex finder diameter. Computational Fluid Dynamics (CFD) formulation which involves model design, meshing, CFD simulation and postprocessing was used to study, analyze, simulate and validate the effect of design modification on the particle collection efficiency of the 1D3D mechanical cyclones. Flow field conditions of the mechanical cyclones as well as CFD particle collection efficiency results were determined. Finally, 1D3D mechanical cyclones with optimal particle collection efficiency was determined.

The softwares that was used for design, meshing, simulation and post-processing include Autodesk Inventor Fusion 2013 R1 and ANSYS workbench using its repository softwares; ANSYS design modeler, ANSYS-ICEM, ANSYS-Fluent and ANSYS-CFD Post.

2.1 Downhill Simplex Method

Downhill simplex method is a nonlinear optimization technique for minimizing an objective function in multi-dimensional space [7]. The target is to obtain global optimum values of the simplex consequently, no linear constrains are applied. The idea of downhill simplex method is to employ a moving simplex in the design space to surround the optimal point and then shrink the simplex until its dimensions reach a specified error tolerance [8]. In n-dimensional space, a simplex is a figure of n+1 vertices connected by straight lines and bounded by polygonal faces. For two variables, the simplex is a triangle and the method is a pattern search that compares functional values at the three vertices of the triangle using reflection, expansion, contraction and shrinkage of the original triangle from a given face. A new triangle is formed in each case and the search is continued with the process generating a sequence of triangles which have different shapes and sizes. Figure 2 illustrates downhill simplex method using a triangle in 2D space.

From figure 2, it is clear that the optimization technique of downhill simplex method involves

modification of an original domain using reflection, expansion, contraction and shrinkage. Therefore, modification of the standard 1D3D mechanical cyclone to develop modified cyclones will involve these techniques.

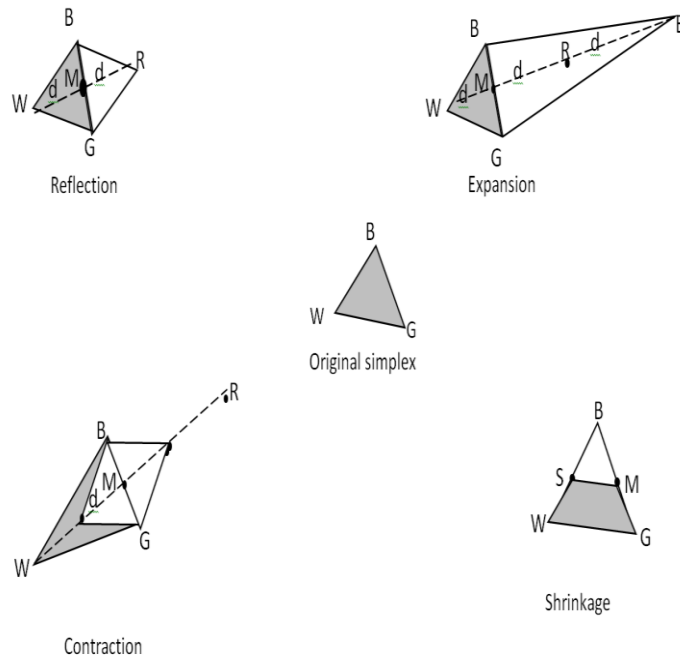


Figure 2: Illustration of downhill simplex method using a triangle in 2D space

2.2 Design Modification

Design modification was carried out using downhill simplex method to develop modified 1D3D mechanical cyclones that will help to improve upon the particle collection efficiency of the standard 1D3D mechanical cyclone. It was carried out by modifying six key multivariate process parameters of the standard 1D3D mechanical cyclone. The process parameters which were used in the design modification and which from research findings significantly affect particle collection efficiency of 1D3D mechanical cyclones are; inlet velocity, inlet area, gas outlet height, dust outlet diameter, cylinder height and cone height. The design modification method involves reflection, expansion, contraction and shrinkage. Reflection involves increasing the multivariate process parameters by their original size, expansion involves increasing the process parameters by twice their original size, and contraction involves increasing the process parameters by half of their original size while shrinkage involves reducing the process parameters by half. Following the design modification, four modified 1D3D mechanical cyclone designs were developed. Table 2 below shows the use of Downhill simplex method for the design modification of the standard 1D3D mechanical cyclone using the six key multivariate process parameters

Table 2: Design modification of standard 1D3D mechanical cyclone using downhill simplex method

Design Parameters	Research cyclone design Values	Modified Mechanical Cyclones Design Values			
		Expansion cyclone values (x3)	Reflection cyclone values (x2)	Contraction cyclone values (x1.5)	Shrinkage cyclone values (x0.5)
(1) Cyclones Inlet velocity(m/s)	16	48	32	24	8
(2) Cyclones inlet area (m ²)	0.0029	0.0087	0.0058	0.0044	0.0015
(3) Cyclones dust outlet diameter (m)	0.0381	0.1143	0.0762	0.05715	0.0191
(4) Cyclones gas outlet height (m)	0.09525	0.2858	0.1905	0.1429	0.0476
(5) Cyclones cylinder section height (m)	0.1524	0.4572	0.3048	0.2286	0.0762
(6) Cyclones cone section height (m)	0.4572	1.3716	0.9144	0.6858	0.2286
Cyclones cylinder diameter (m)	0.1524	0.1524	0.1524	0.1524	0.1524
Cyclones gas outlet diameter (m)	0.0762	0.0762	0.0762	0.0762	0.0762
Cyclones inlet length	0.1524	0.1524	0.1524	0.1524	0.1524

Design Parameters	Research cyclone design Values	Modified Mechanical Cyclones Design Values			
		Expansion cyclone values (x3)	Reflection cyclone values (x2)	Contraction cyclone values (x1.5)	Shrinkage cyclone values (x0.5)
(m)					
Cyclones Gas outlet length (m)	0.09418	0.09418	0.09418	0.09418	0.09418

2.3 Computational Fluid Dynamics (CFD) formulation

Computational Fluid Dynamics (CFD) formulation which involves model design, meshing, CFD simulation and post processing was carried out on the standard and modified 1D3D mechanical cyclone designs using ANSYS Fluent software. This is to determine their particle collection efficiency and flow field characteristics. Flow in mechanical cyclones is considered to be steady-state, incompressible, turbulent flow. For incompressible fluid flow, the equation for continuity and balance of momentum otherwise known as Navier-Stokes equations are;

$$\frac{\partial u_j}{\partial x_j} = 0 \quad (1)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} \quad (2)$$

Where u_i and u_j represents the gas velocity along the coordinate axis in X and Y direction respectively, ρ is the gas density, and ν is the kinematic viscosity of the fluid.

Four main numerical procedures for solving Navier-Stokes equations are [9]; Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), Detached Eddy Simulation (DES) and Reynolds Averaged Navier Stokes (RANS) approach. The most accurate approach is DNS however; DNS can be computationally expensive for high Reynolds number flow mostly associated with mechanical cyclones. Consequently, Reynolds Averaged Navier Stokes (RANS) approach offers a better alternative for high Reynolds number flow since it analyzes flow into two parts; a mean (time-averaged) component and a fluctuating component [9]. Therefore, based on RANS approach, equations (1) and (2) become;

$$\frac{\partial \bar{U}_j}{\partial x_j} = 0 \quad (3)$$

$$\frac{\partial \bar{U}_i}{\partial t} + \bar{U}_j \frac{\partial \bar{U}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_i} + \nu \frac{\partial^2 \bar{U}_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j} (\overline{u'_i u'_j}) \quad (4)$$

$\overline{u'_i u'_j}$ in equation 4 is called the Reynolds stress tensor. All the effects of turbulent fluid motion on the mean flow are lumped into this term by the process of averaging. This will enable great savings in terms of computational requirements. However, due to the process of averaging, the Reynolds stress tensor need to be closed by replacing it with an eddy viscosity multiplied by velocity gradients. Therefore;

$$\overline{u'_i u'_j} = -\nu_t \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) \quad (5)$$

Where ν_t is the turbulent (eddy) kinematic viscosity. In order to make equation 5 valid based on the flow direction, it is rewritten as,

$$\overline{u'_i u'_j} = -\nu_t \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) + \frac{2}{3} \rho \delta_{ijk} \quad (6)$$

δ_{ij} is Kronecker delta. $\delta_{ij}=1$ if $i = j$ and $\delta_{ij}= 0$ if $i \neq j$. K is turbulent kinetic energy given by:

$$k = \frac{1}{2} \overline{u'_i u'_i} \quad (7)$$

The eddy viscosity is treated as a scalar quantity and is determined using the turbulent velocity scale ν and the length scale l .

$$\nu_t \approx \nu l \quad (8)$$

Reynolds stress tensor is closed using two main categories [9]; Eddy Viscosity Models (EVM) and Reynolds Stress Model (RSM). Eddy Viscosity Models (EVM) uses equations which are solved and which may not be precise, time consuming and computationally expensive. As a result, Reynolds Stress Model (RSM) offers an alternative and is regarded as the most appropriate RANS turbulence model for cyclone flows. Under RSM, equation 2 is written as;

$$\frac{\partial \bar{U}_i}{\partial t} + \bar{U}_j \frac{\partial \bar{U}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_i} + \nu \frac{\partial^2 \bar{U}_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j} R_{ij} \quad (9)$$

Where \bar{U}_i is the mean velocity, x_i the position, t the time, \bar{P} the mean pressure, ρ the constant gas density, ν the kinematic viscosity and $R_{ij} = \overline{u'_i u'_j}$ is the Reynolds stress tensor. Here $u'_i = u_i - \bar{u}_i$ is the i th fluid fluctuation velocity component.

In ANSYS Fluent [10], RSM turbulence model provides differential transport equations for evaluation of the turbulence stress components [9], i.e.:

$$\begin{aligned} \frac{\partial}{\partial t} R_{ij} + \bar{U}_k \frac{\partial}{\partial x_k} R_{ij} &= \frac{\partial}{\partial x_k} \left(\frac{\nu_t}{\sigma^k} \frac{\partial}{\partial x_k} R_{ij} \right) - \left[R_{ik} \frac{\partial \bar{U}_j}{\partial x_k} + R_{jk} \frac{\partial \bar{U}_i}{\partial x_k} \right] \\ &- C_1 \frac{\varepsilon}{K} \left[R_{ij} - \frac{2}{3} \delta_{ij} K \right] - C_2 \left[P_{ij} - \frac{2}{3} \delta_{ij} P \right] \\ &- \frac{2}{3} \delta_{ij} \varepsilon \end{aligned} \quad (10)$$

Where the turbulence production term P_{ij} are defined [9] as;

$$P_{ij} = - \left[R_{ik} \frac{\partial \bar{U}_j}{\partial x_k} + R_{jk} \frac{\partial \bar{U}_i}{\partial x_k} \right], \quad P = \frac{1}{2} P_{ij} \quad (11)$$

P is the fluctuating kinetic energy production. The value of the empirical constants are $\sigma^k=1$, $C_1=1.8$ and $C_2=0.6$. The transport equation for the turbulent dissipation rate, ε , is given as:

$$\begin{aligned} \frac{\partial \varepsilon}{\partial t} + \bar{U}_j \frac{\partial \varepsilon}{\partial x_j} &= \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma^\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] - C^{\varepsilon 1} \frac{\varepsilon}{K} R_{ij} \frac{\partial \bar{U}_i}{\partial x_j} \\ &- C^{\varepsilon 2} \frac{\varepsilon^2}{K} \end{aligned} \quad (12)$$

In the above equation the value of the constants are; $\sigma^\varepsilon = 1.3$, $C^{\varepsilon 1} = 1.44$ and $C^{\varepsilon 2} = 1.92$.

2.4 Model design and meshing

Model design was carried out by designing to specification the 1D3D mechanical cyclone to be simulated using Autodesk Inventor Fusion 2013 R1. The design was saved as Initial Graphics Exchange Specification (IGES) file which is a special type of graphical file format that is used for file transfer protocol and allows the digital exchange of information among Computer Aided Design (CAD) systems. Next, ANSYS

workbench was launched and the geometry field used to import the saved design.

The mesh field of ANSYS workbench was refreshed and the edit tool used to enable ANSYS ICEM mesher from the field. Meshing of the designed cyclone in ANSYS ICEM platform was carried out using minimum mesh size which is the length of the smallest mesh that will be used in meshing the mechanical cyclone. It is reflective of friction factor which is a function of roughness factor of material (stainless steel) used in designing mechanical cyclones and viscosity of the fluid. Using the default minimum mesh size, the full mesh of the designed 1D3D mechanical cyclone was developed using cells generated by the ANSYS ICEM code for analyzing flow in mechanical cyclones. Hexahedral fine mesh with nodes at the center is recommended [10] for the meshing and proper simulation of rectangular inlet mechanical cyclones. The nodes are the points where computation of flow field residuals, involving time and energy is carried out during CFD simulation. Multizone setting was selected, with assembly and part based meshing applied, in order to generate the mesh. Create named selection tool was used to create the boundaries of the 1D3D mechanical cyclones that has been meshed. These include; inlet boundary, gas outlet boundary, dust outlet boundary. The wall boundary was created automatically by ANSYS ICEM mesher. Finally, stainless steel was selected as design material during model design of 1D3D mechanical cyclones using Autodesk Inventor Fusion 2013 R1 because of its properties which include; corrosion resistant, high tensile strength, durability, temperature resistant, easy formability and fabrication, low-maintenance (long lasting), attractive appearance, and environmentally friendly (recyclable). Figure 3 show model design and mesh of standard 1D3D mechanical cyclone while figure 4 show that of modified 1D3D mechanical cyclones.

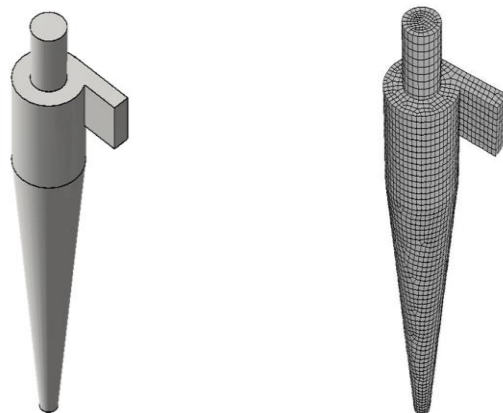


Figure 3: Model design and mesh of standard 1D3D mechanical cyclone.

2.5 CFD numerical settings and simulation

The key to the success of Computational Fluid Dynamics (CFD) simulation lies with the accurate selection of numerical settings. For the CFD simulation of 1D3D mechanical cyclones, Reynolds Stress Turbulence Model (RSTM) was used to model the swirling turbulent flow. Finite volume method was used to discretize the partial differential equations of the model using SIMPLEC (Semi-Implicit Method for Pressure-Linked Equations-Consistent) method for pressure velocity coupling. Least Squares Cell Based was used for the gradient, PRESTO (pressure staggering option) for pressure, and QUICK (quadratic upwind differencing) scheme for momentum and to interpolate the variables on the surface of the control volume. Volume fraction and turbulent kinetic energy was used for turbulent dissipation rate while second order upwind was used for Reynolds stresses.

2.6 Boundary conditions

Inlet boundary condition is numerical value of inlet velocity of each 1D3D mechanical cyclone to be simulated. Other inlet boundary conditions are; air density

of 1.225 kg/m³ and dynamic viscosity of 17.894 x 10⁻⁶ kg/(ms).

Boundary condition at the outlet is outflow where all transport variables have zero normal gradients while wall boundary condition is no-slip. Turbulence intensity of 5% was used following recommendations [10] while hydraulic diameter was calculated from equation 8 below [11] for rectangular inlet mechanical cyclones.

$$D_H = \frac{4 * \text{cross - sectional area}}{\text{wetted perimeter}} = \frac{4ab}{2(a + b)} \quad (13)$$

Where D_H = hydraulic diameter
 a = inlet height
 b = inlet width

Tables 3 and 4 show design parameters, hydraulic diameter and default minimum mesh size of standard and modified 1D3D mechanical cyclones respectively, used for CFD formulation.

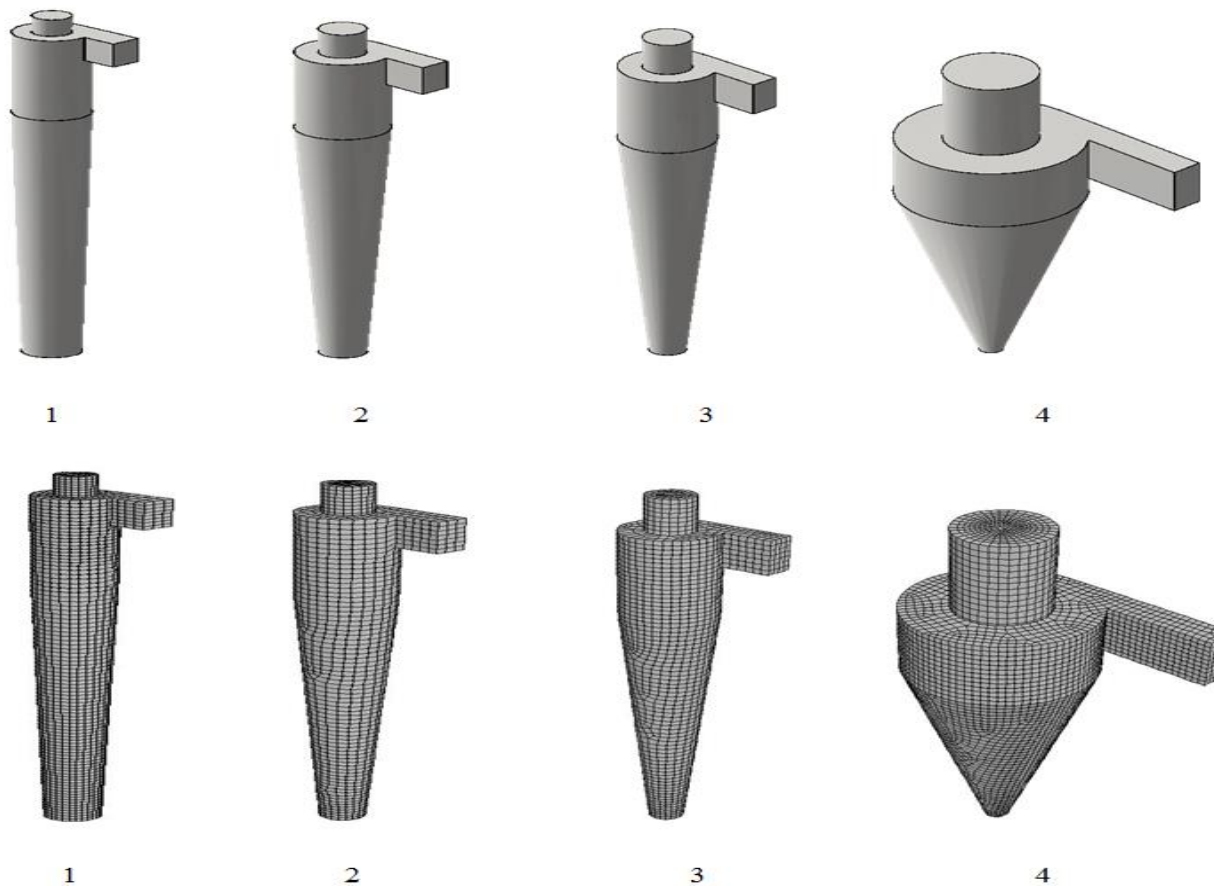


Figure 4: Model design and mesh of modified 1D3D mechanical cyclones.

Table 3: Design parameters, hydraulic diameter and default minimum mesh size of standard 1D3D mechanical cyclone used for CFD formulation

Exp run	Design parameters						Hydraulic Diameter (m)	Default Min. mesh size (m)
	Inlet Vel. (m/s)	Inlet area (m ²)	Dust outlet diam. (m)	Gas outlet height (m)	Cyl. height (m)	Cone height (m)		
1	16	0.0029	0.0381	0.09525	0.1524	0.4572	0.0508	1.1029e-4

Table 4: Design parameters, hydraulic diameter and default minimum mesh size of modified 1D3D mechanical cyclones used for CFD formulation

Exp run	Design parameters						Hydraulic Diameter (m)	Default Min. mesh size (m)
	Inlet Vel. (m/s)	Inlet area (m ²)	Dust outlet diam. (m)	Gas outlet height (m)	Cyl. height (m)	Cone height (m)		
1	48	0.0087	0.1143	0.2858	0.4572	1.3716	0.0879555	2.8358e-4
2	32	0.0058	0.0762	0.1905	0.3048	0.9144	0.0718667	1.9588e-4
3	24	0.0044	0.05715	0.1429	0.2286	0.6858	0.0625333	1.526e-4
4	8	0.0015	0.0191	0.0476	0.0762	0.2286	0.0365333	7.0719e-5

2.7 Discrete Phase Modelling (DPM)

To calculate the trajectories of particles in the flow, the discrete phase model (DPM) was used to track individual particles through the continuum fluid. The equation of particlemotion including the effects of nonlinear drag and gravitational forces is given by;

$$\frac{dx_p}{dt} = u_p \tag{14}$$

$$\frac{du_p}{dt} = \frac{1}{\tau_p} (u - u_p) + g \tag{15}$$

x_p is particle position, g is acceleration of gravity. The slip velocity $(u - u_p)$ in the above equation leads to an unbalanced pressure distribution as well as viscous stresses on the particle surface. This yields a resulting force called drag force, F_d , which is given [12] by7

$$F_d = \frac{1}{\tau_p} \frac{C_d Re_p}{24} \tag{16}$$

Where τ_p is the particle relaxation time given [9] by:

$$\tau_p = \frac{\rho_p d_p^2}{18\mu} \tag{17}$$

The Reynolds number of the particle is defined as;

$$Re_p = \rho_p d_p \frac{|u - u_p|}{\mu} \tag{18}$$

C_d , is drag coefficient which is a function of particle Reynolds number (Re_p). In ANSYS Fluent, it is calculated for spherical particles using developed correlations as a function of the relative Reynolds numbers Rep . The equation of motion for particles was integrated along the trajectory of an individual particle. Collisions between particles and the walls of the cyclone were assumed to be perfectly elastic (coefficient of restitution is equal to 1). Particle–particle collision is negligible with the particle being cornstarch with density of 1520kg/m³. The particle size range is 0.01µm to 0.08µm [13]. The velocity of the particles was assumed to be the same as the inlet velocity of the mechanical cyclone to be simulated. Number of steps was 500,000, while the length scale was

0.07 times the hydraulic diameter. The implicit coupled solution algorithm was selected. Discrete Random Walk Model (DRWM) was used for the stochastic tracking of particles during turbulent dispersion. Collection efficiency statistics were obtained by releasing a specified number (about 10,000) of polydispersed particles (cornstarch) at the inlet surface of the mechanical cyclone that was simulated and by monitoring the number escaping through the outlet. The DPM results obtained from the tracking of particles through the 1D3D mechanical cyclones was used to calculate the CFD, particle collection efficiency results of the mechanical cyclones using equation 19 below [14].

$$\eta(d_p) = \frac{n_{p, \text{trapped}}}{n_{p, \text{injected}}} \quad (19)$$

Where $\eta(d_p)$ = fractional separation efficiency

$n_{p, \text{trapped}}$ = no of particles trapped
 $n_{p, \text{injected}}$ = no of particles injected or tracked

Contrasting results of the simulated 1D3D mechanical cyclones were also obtained. These include pressure and velocity flow field results.

3.0 RESULTS AND DISCUSSION

3.1 Particle Collection Efficiency Results

Particle collection efficiency results of 1D3D mechanical cyclones simulated using cornstarch as inlet particulate matter with default minimum mesh size are shown in the tables below. Table 5 show particle collection efficiency results of standard 1D3D mechanical cyclone while table 6 show results of modified 1D3D mechanical cyclones.

Table 5: DPM and particle collection efficiency results of standard 1D3D mechanical cyclone

Mech. Cycl.	Type of part.	Type of mesh	Conv. iter. Point	Discrete Phase Model (DPM) result				CFD particle Coll. Eff. results (%)
				No. of tracked part.	No. of escaped part.	No. of trapped part.	No. of incomplete particles	
Standard	Corn Starch	Default	1413	10,240	1292	8948	0	87.38

Table 6: DPM and particle collection efficiency results of modified 1D3D mechanical cyclones

Mod. Mech Cycl.	Type of dust part.	Type of mesh	Convergence Iteration Point	Discrete Phase Model (DPM) results				CFD particle Coll. Eff. results (%)
				No. of tracked part.	No. of escaped part.	No. of trapped part.	No. of incomplete part.	
1	Corn starch	Default	1081	10,080	602	9438	0	94.00
2	Corn Starch	Default	819	10,080	1143	8937	0	88.66
3	Corn Starch	Default	1197	10,080	1030	9050	0	89.78
4	Corn starch	Default	Oscillatory iteration	n/a	n/a	n/a	n/a	n/a

3.2 Flow Field Results

Static pressure and tangential velocity contours are used to present flow field results of 1D3D mechanical cyclones simulated in this research work. Static pressure and tangential velocity are used because, research findings [6, 9, 14] indicate that, they are the dominant flow fields in

mechanical cyclones that affect particle collection efficiency. Consequently, they are presented using YZ planes and for 1D3D mechanical cyclones whose CFD simulation converged and results obtained. Figure 3 show static pressure and tangential velocity contour of standard 1D3D while figures 4 and 5 show static pressure and

tangential velocity contour respectively, of modified 1D3D mechanical cyclone.

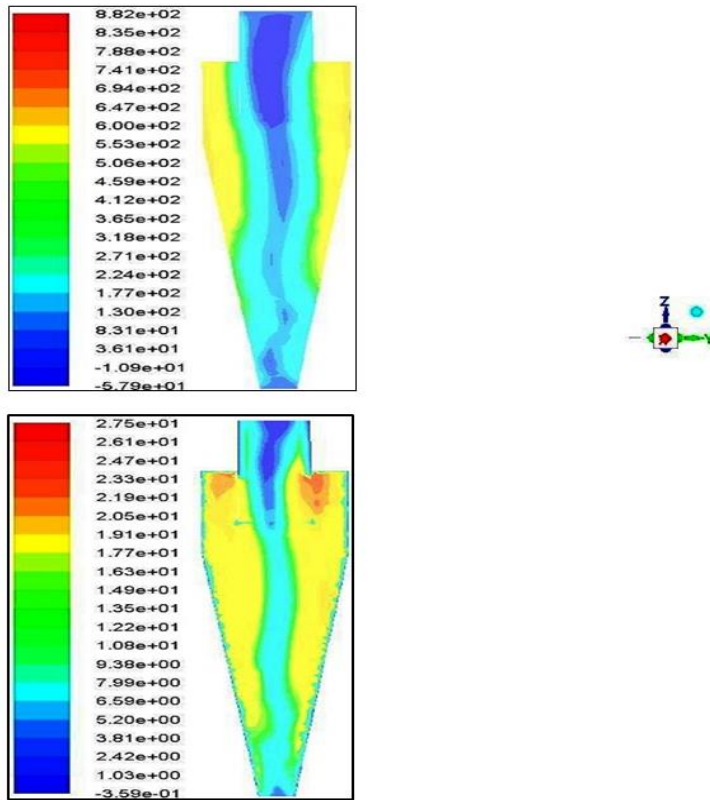


Figure 3: Static pressure and tangential velocity contour of standard 1D3D mechanical cyclone using cornstarch and default minimum mesh size for CFD simulation.

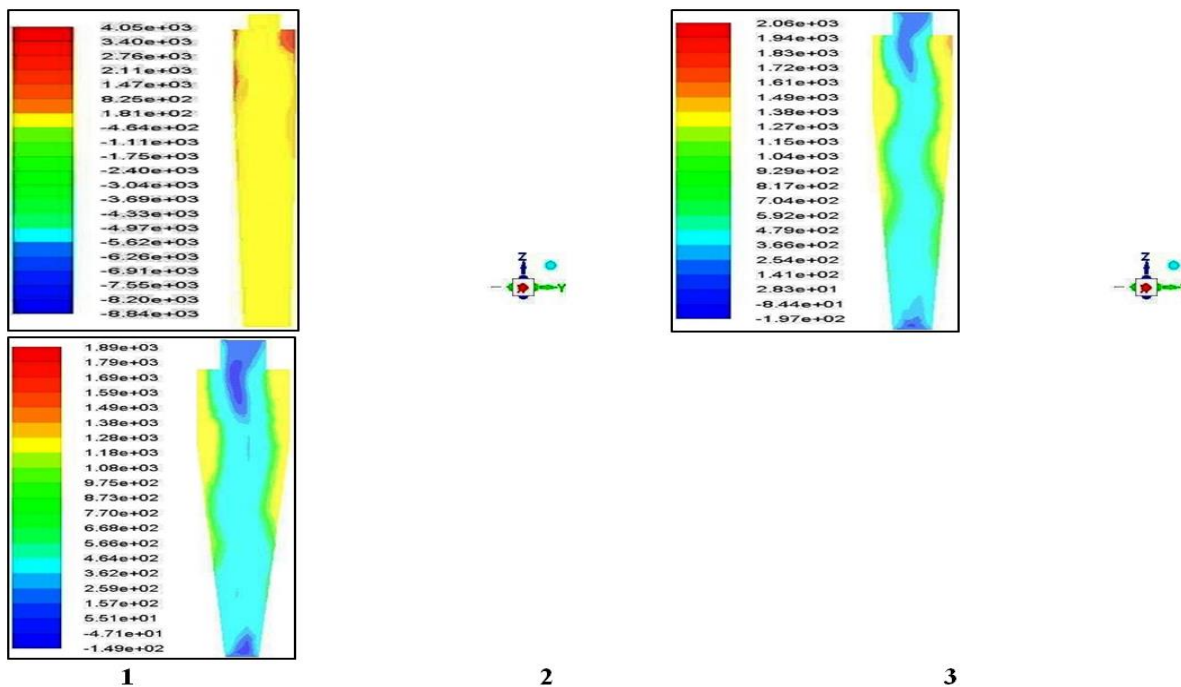


Figure 4: Static pressure contour of modified 1D3D mechanical cyclones using cornstarch and default minimum mesh size for CFD simulation.

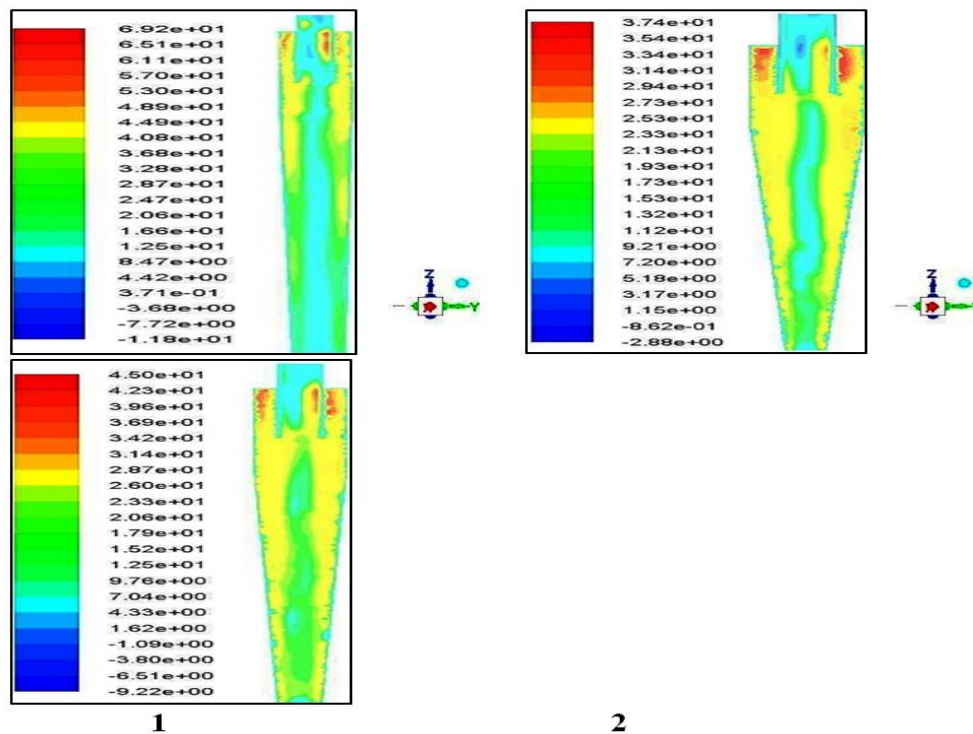


Figure 5: Tangential velocity contour of modified 1D3D mechanical cyclones simulated using cornstarch and default minimum mesh size

3.3 Discussion

Particle collection efficiency results in table 6 show that, standard 1D3D mechanical cyclone recorded 87.38% using cornstarch and default minimum mesh size for CFD simulation.

Design modification of standard 1D3D mechanical cyclone to develop modified 1D3D mechanical cyclones show from table 7 that, modified 1D3D mechanical cyclone 1 recorded optimum CFD particle collection efficiency result among modified 1D3D mechanical cyclones. It recorded an optimum particle collection efficiency result of 94.00% using cornstarch and default minimum mesh size for CFD simulation. This result is higher than that of standard 1D3D mechanical cyclone. From tables 6 and 7, the convergence iteration point of modified 1D3D mechanical cyclone 1 showed improvement in simulation time compared to standard 1D3D mechanical cyclone. Using cornstarch and default minimum mesh size for CFD simulation; standard 1D3D mechanical cyclone converged at 1413 iterations while modified 1D3D mechanical cyclone 1 converged at 1081 iterations. The above results show improvement in CFD iteration and simulation time in modified 1D3D mechanical cyclone 1 compared to standard 1D3D mechanical cyclone. This will bring about energy saving. However, modified 1D3D mechanical cyclone 1 will have

higher energy requirement than standard 1D3D mechanical cyclone. This is because, its static pressure and tangential velocity are higher than that of standard 1D3D mechanical cyclone. From figure 3 using cornstarch and default minimum mesh size for CFD simulation, standard 1D3D mechanical cyclone recorded maximum and minimum static pressure values of 882Pa and -57.9Pa while from figure 4 modified 1D3D mechanical cyclone 1 recorded 4050Pa and -8840Pa respectively. From figure 3, standard 1D3D mechanical cyclone recorded maximum and minimum tangential velocity values of 27.5m/s and -35.9m/s while from figure 5, modified 1D3D mechanical cyclone 1 recorded 69.2m/s and -11.8m/s respectively. Drop in static pressure and change in tangential velocity leads [15] to pressure drop and centrifugal force which are functions [6, 16] of power requirement of the blower needed to introduce the fluid-solid content into mechanical cyclones and have been reported [6, 9] to affecting energy requirements. Thus, modified 1D3D mechanical cyclone 1 will have higher energy requirements than standard 1D3D mechanical cyclone even when its particle collection efficiency is higher. Nevertheless, it was stated [4] that there may be times when it is economically beneficial for a processing industry to incur higher energy costs rather than convert to a filter system, the cost of which may be five to ten times higher than that of a cyclonic abatement system.

Consequently, modified 1D3D mechanical cyclone 1 is considered as the optimal 1D3D mechanical cyclone with optimal particle collection efficiency in this research work.

3.4 Validation of Optimum Particle Collection Efficiency Result

CFD results always need validation by; experimental and CFD results that have analogous parameters. Validation of optimum particle collection efficiency result recorded by modified 1D3D mechanical cyclone 1 in this research work using cornstarch and default minimum mesh size for CFD simulation is hereby carried out using experimental research work [16] on efficiency and pressure drop of cyclones across a range of inlet velocities. In the experiment, particle collection efficiency result of over 99% regardless of inlet velocity was recorded for 1D3D mechanical cyclone with cylinder diameter of 0.1524m using cornstarch as inlet particulate matter. The setting of the above experiment is in line with that of this research work. Here, 1D3D mechanical cyclone with cylinder diameter of 0.1524m was also used with cornstarch as inlet particulate matter. Taking 99% as maximum particle collection efficiency recorded in the experiment, the optimum particle collection efficiency result of 94.00% recorded by modified 1D3D mechanical cyclone 1 in this research is in good matching with the result of the experiment. Percentage error between CFD model result of modified 1D3D mechanical cyclone 1 and experimental result is calculated as -5.0%. Absolute value of above result lies within acceptable limit (5%) [14] of experimental error. Therefore, the particle collection efficiency result of modified 1D3D mechanical cyclone 1 is validated.

Optimum particle collection efficiency result of modified 1D3D mechanical cyclone 1 in this research work is also validated by the result of the experimental research [17] on 1D2D, 1D3D, 2D2D cyclone fractional efficiency curves for fine dust. In the experiment, particle collection efficiency result of 99.3% was recorded by 1D3D mechanical cyclone with cylinder diameter of 0.1524m using cornstarch as inlet particulate matter. The above result is good matching with the optimum particle collection efficiency result of 94.00% recorded in this research work by modified 1D3D mechanical cyclone 1 with cylinder diameter of 0.1524m using cornstarch as inlet particulate matter. Percentage error between CFD result of modified 1D3D mechanical cyclone 1 and experimental result is calculated approximately as -5.0%. Absolute value of above result lies within acceptable limit (5%) [14] of experimental error. Therefore, the particle collection efficiency result of modified 1D3D mechanical cyclone 1 is validated.

4.0 CONCLUSION

Design modification and CFD formulation have been used to understand the effect of process parameters on the performance 1D3D mechanical cyclones and a new optimal cyclone design has been obtained. Consequently, design modification using six key multivariate process parameters in line with CFD formulation offer an alternative and powerful approach to model 1D3D mechanical cyclones performance. The design of 1D3D mechanical cyclone that gave optimal particle collection efficiency and that of standard 1D3D mechanical cyclone can be seen to have been computationally compared to get a clear vision of their differences in performance parameters. The CFD formulation results confirm the superior performance of the design of 1D3D mechanical cyclone that gave optimal particle collection efficiency in comparison with the design of standard 1D3D mechanical cyclone.

REFERENCES

- [1] Wang, L. "A new engineering approach to cyclone design for cotton gins". *Department of Agricultural Engineering, Texas A&M University*. Texas. (2000)
- [2] Cooper, C. C. and Alley, G. C. "Air Pollution Control: A Design Approach". *Prospect Heights. Waveland Press*. Illinois. (2002).
- [3] Davis, W. T. "Air Pollution Engineering Manual". 2nd edition. *Air and Waste Management Association*. John Wiley and Sons. New York. (2000).
- [4] Parnell, C.B. Jr. "Cyclone design for air pollution abatement associated with agricultural operations". *In Proceedings of 1996 Beltwide Cotton Production Conferences*. Nashville, Tennessee. (1996).
- [5] Kasper, P., Mihalski, K.D. and Parnell, C.B. Jr. "Evaluation and development of cyclone theory". *In Proceedings 1993 Beltwide Cotton Production Conferences*. National Cotton Council. New Orleans, Los Angeles. (1993).
- [6] Wang, L." Theoretical study of Cyclone Design". *Department of Biological and Agricultural Engineering, Texas A&M University*. Texas. (2004)
- [7] Salcedo, R.L.R. and Cândido, M.G. "Global optimization of reverse-flow gas cyclones: application to small scale cyclone design". *Separation Sci. Tech* 36(12), (2001), pp 2707–2731.
- [8] Press, H., Flannery, B. P., Teukolsky, S. A. and Vetterling W. T. "Numerical Recipes in FORTRAN 77: The Art of Scientific Computing". *Cambridge University Press*, Cambridge. UK. (1992).
- [9] Elsayed, K. "Analysis and Optimization of Cyclone Separators Geometry Using RANS and LES

- Methodologies”. *Department of Mechanical Engineering, Vrije Universiteit Brussel*. Brussels, Belgium. Uitgeverij University Press. (2011). ISBN 978-94-9069-594-1.
- [10] ANSYS Fluent. “ANSYS FLUENT 17.0 user’s guide”. *ANSYS Fluent Incorporated*. Canonsburg. (2015).
- [11] Christoph, H. “Tutorial for a simple cyclone simulation using ICEM CFD and ANSYS FLUENT”.
<https://www.youtube.com/watch?v=dalhTs4-0-c>. (2014).
- [12] Jayaraju, S. T.” Study of the air flow and aerosol transport in the human upper airway using LES and DES methodology”. *Belgium.Vrije Universiteit*, Brussels. (2009).
- [13] The Engineering Toolbox.
<http://www.engineeringtoolbox.com>
- [14] Hoffmann, A. C. and Stein, L. E. “Gas cyclones and swirl tubes: Principle, Design and Operation”. *Springer*, 2nd edition. (2008).
- [15] Davis, W. T. “Air Pollution Engineering Manual”. *Air and Waste Management Association*. John Wiley and Sons. 2nd edition. New York. (2000).
- [16] Faulkner, W.B. and Shaw, B. W. “Efficiency and pressure drop of cyclones across a range of inlet velocities”. *American Society of Agricultural and Biological Engineers*. 22(1), (2006). 155-161.
<https://www.researchgate.net/publication/228621688>
- [17] Wang, L., Parnell, C. B. and Shaw, B. W. “1D2D, 1D3D, 2D2D Cyclone Fractional Efficiency Curves for Fine Dust”. *Proceedings of the 2000 Beltwide Cotton Conferences*, National Cotton Council, San Antonio, TX. (2000).