Accepted Manuscript

Development of Novel Hydrocyclone Designs for Improved Fines Classification Using Multiphase CFD Model

Teja Reddy Vakamalla, Veera Bhadra rao Koruprolu, Rakesh Arugonda, Narasimha Mangadoddy

PII: S1383-5866(16)32103-7
DOI: http://dx.doi.org/10.1016/j.seppur.2016.10.026
Reference: SEPPUR 13296

To appear in: Separation and Purification Technology

Received Date: 28 April 2016
Revised Date: 22 August 2016
Accepted Date: 20 October 2016


This is a PDF file of an unedited manuscript that has been accepted for publication. As a service to our customers we are providing this early version of the manuscript. The manuscript will undergo copyediting, typesetting, and review of the resulting proof before it is published in its final form. Please note that during the production process errors may be discovered which could affect the content, and all legal disclaimers that apply to the journal persist.
Development of Novel Hydrocyclone Designs for Improved Fines Classification Using Multiphase CFD Model

Teja Reddy Vakamalla*, Veera Bhadra rao Koruprolu*, Rakesh Arugonda*, Narasimha Mangadoddy1*

*Department of Chemical Engineering, Indian Institute of Technology Hyderabad, India.

Abstract

In this paper, a set of potential hydrocyclone designs are explored for fines classification using Computational Fluid Dynamics (CFD) technique. The CFD model uses Volume of Fluid multiphase model coupled with Reynolds Stress turbulence model for two phase flow predictions. Lagrangian Particle Tracking and Algebraic Slip Mixture model (ASM) modified with shear lift forces and slurry rheology corrected with fines fraction are used to predict the particle classification. Conventional cylindrical-conical design and various novel cyclone designs having a combination of multiple and small cone angles, tapered vortex finder and air core free designs are considered in this study. Simulation results are presented in terms of mean and turbulent flow field along with particle efficiency curve. Predictions show that all the tested novel designs are inherently having the potential for finer cut separation and higher tangential velocity compared to the conventional design. Cyclone design with a small cone angle is further modified by placing various sizes of rods at the center axis of the cyclone. Multiphase simulations with modified ASM model are also carried out for the multiple cone angles design, standard design, and the modified small cone angle design with full length rod at 15 wt% of solids. Performance by multiphase simulations show that multiple cone angles design and modified small cone angle design with the full length rod are the best among the tested designs yielding high separation efficiency, smaller cut-point and minimum coarse particle misplacement due to resultant turbulence minimization.

Keywords: Air core, CFD, Cut diameter, Design, Hydrocyclone, Multiphase
1. Introduction

Hydrocyclones are widely used in mineral processing, food, pharmaceutical and chemical industries due to its simple design, high throughput, and low operational and maintenance costs. Flow behavior inside hydrocyclones is complex and the complete understanding of multiphase behavior of hydrocyclone still remains a research problem. Conventional hydrocyclone design consists of a tangential inlet, cylindrical section, vortex finder, spigot, and conical section with a certain angle responsible for flow reversal thereby flow split. Optimizing the design parameters may have a chance to improve the hydrocyclone performance, indicated by a set of desired performance indices; cut diameter, water split to underflow and sharpness of separation. Exploring novel hydrocyclone designs through the experimental approach is an expensive and time consuming process. Computational Fluid Dynamics (CFD) is proven as an efficient tool to explore new designs virtually in the past [1-5].

Fines generation in various mineral industries is increasing day by day with an increased demand in production and treatment of low grade ores. Bulk produced fines separation requires large hydrocyclones to handle high throughputs but with small cut size separations. Usually the conventional hydrocyclones, having high cut points and coarse and fine size particle entrainment issues, are restricted to classify fines with high efficiency. This requires a change in the design philosophy of hydrocyclones to treat fines with a minimum turbulence on particles. Fines misplacement to underflow occurs due to the insufficient drag force of fine particles to resist the movement along with fluid. Entrainment of fine or slow settling particles between the coarser or faster settling particles discharged as the underflow. Therefore, reducing the water split to underflow may minimize the fines misplacement. Reducing the cone angle may also decrease the water split to underflow [6]. Usually some of the coarse particles with water also move along the wall near the vortex finder and join the overflow as the coarse misplacement in the hydrocyclone. Through CFD studies, Rajamani et al. [7] and Brennan et al. [8] have observed that about 12-18% of the feed slurry is usually short-circuiting to the overflow in conventional hydrocyclones without being subjected to centrifugal action. It is believed that the area between cylindrical wall and vortex finder acts to provide preliminary separation in hydrocyclones. Altering the cylindrical section [9] or vortex finder design [10, 11] may also minimize the coarse
particle misplacement. Alternative approach is to increase the tangential velocity by altering inlet designs [2, 5], thereby enhancing the centrifugal forces to improve the fine particle separation.

It has been observed that the multiphase CFD models have potential to provide quick and reliable route to simulate novel designs for hydrocyclone performance predictions. These CFD models allow us to predict the performance of a hydrocyclone by studying the classification process in terms of air core position, turbulence analysis and particle size segregation [8, 12-19]. Delgadillo and Rajamani [9] have explored six novel 75 mm hydrocyclone designs by modifying the cylindrical chamber and the cone angle. They have utilized the CFD model consist of Large Eddy Simulation (LES) turbulence model coupled with Volume of Fluid (VOF) multiphase model for water-air flow and the discrete phase model (DPM) for particle classification. The predicted separation efficiency and cut point results were compared with the standard hydrocyclone design data. This exploration study yields two novel designs with smaller cut size and better separation efficiency compared with the standard design. Vieira et al. [20] replaces the conical wall of 30 mm hydrocyclone with a filtering wall and observed the improved efficiencies. In their recent CFD studies, Vieira and Barrozo [21] using different vortex finder diameters for the modified hydrocyclone, demonstrated that small vortex finder diameter can be used for the separation of fine particles. However the simulations were performed in a two dimensional geometry lacking realistic turbulent flow conditions of cyclonic flow.

Yang et al. [22] altered the hydrocyclone design from single cone to double cone with multiple combinations and studied the classification performance by both CFD and physical experiments. It was observed that angle between two cones plays a significant role for flow split and separation efficiency, but it has minimal effect on the cut diameter. Inclusion of conical top plate near the vortex finder of 20 mm hydrocyclone was made the circulation flow reduction and enhancement of the tangential velocity [2]. They used DPM model for particle tracking and found an improvement in the cut size of 4-12% compared to the conventional vortex finder design. Hwang et al. [23] modified the inlet and vortex finder designs for fine particle separation. These designs consists number of inlets changed from 1 to 4 with proportionally reduced inlet size, resulted high centrifugal forces along the cylindrical section. They have also used conical vortex finder and guided channel, which were resulted in the increased efficiency levels. But in
the case of guided channel there was no clear information about cut size alteration. Ghodrat et al. [1] used the CFD tool to modify the 75 mm hydrocyclone conical section with the long convex and concave shape bodies. It was observed that the convex conical section is having better separation performance and reduces water split to the underflow than the conventional hydrocyclone. Inner cone hydrocyclone design provides longer residence times and more stable flow field for improved separation than the conventional design [24]. Recently they [25] also studied the effect of coal particle size and density on the separation efficiency. They observed a reduction in the separation efficiency of particle with a reduction in the particle density.

Air core is one of the most important flow characteristics in hydrocyclone [26]. It has strong effect on flow field, performance and cut size. Most of the research works dealt with the measurement of air core shape and size of a given hydrocyclone [27-31]. But far less attention has been paid on the air core influence on separation efficiency and cut size [32, 33], which are the fundamental function of hydrocyclone and potentially have profound consequences for development of improved designs. Xu et al. [34] reported that air core elimination in the cyclones can improve the separation efficiency. Lee and Williams [35] conducted experiments on a 44 mm hydrocyclone with solid rods of various diameters and length to suppress the air core have given a contradictory statement. They reported air core elimination leading to decrement in the efficiency rather an improvement. Chu et al. [36] experimentally studied the effect of solid insertion to eliminate the air core in a 75 mm hydrocyclone using Laser Doppler Anemometer (LDA) in terms of the mean and fluctuating velocity components. It was reported a reduction in the mean and turbulent components of axial, radial velocities, and cut diameter of the separation. An improvement in the sharpness of separation with the air core suppression design was also observed. Efficiency of the hydrocyclone also depends on the solid rod diameter used to suppress the air core. Chu et al. [36] explained the reason behind the efficiency decrement of Lee and Williams work [35] to attribute the disturbed flow due to rod insertion case. Sripriya et al. [31] performed an experimental study in a 100 mm hydrocyclone on air core suppression by placing the solid rod in the center and reported similar results with reduction in the cut-diameter. Evans et al. [37] also studied the effect of solid rod insertion in the center of 45 mm hydrocyclone using Reynolds Stress Model (RSM) as turbulence model. They observed that the 4 mm diameter solid rod improves the efficiency whereas 6 mm diameter solid rod decreases the efficiency. Therefore
suitable selection of the rod dimensions is very important. It was also learnt that insertion of the rod decreases the frictional losses between the air and water, which in turn converted to increase the kinetic vortex motion of the fluid; thereby an improved efficiency is expected.

1.1 Present work

Aim of the present paper is to explore a number of potential novel designs computationally, for high throughput and finer cut size separation with a reduction in the coarse and fine particle misplacement. Predicted performance of the novel designs are compared with the conventional cylindrical-conical standard cyclone design. The fluid flow is assessed in terms of mean and turbulent flow field, air core size and turbulence intensity profiles. Particulate phase is simulated by both DPM and Algebraic Slip Mixture (ASM) multiphase models using ANSYS’s Fluent with help of modified user defined functions (UDF). Particle classification performance of various designs is analyzed fundamentally using size wise distributions and Locus of Zero Vertical Velocities (LZVV) inside the cyclone from the multiphase CFD simulations at moderate solids concentrations.

2 Description of designs

The investigation of various flow base design concepts using a comprehensive CFD model is targeted to reduce the short-circuiting of particles to both overflow and underflow, minimize the turbulence levels inside the hydrocyclone and maximize the throughput and thereby reducing the cut size while separating various size particles through a classifying hydrocyclone.

In order to reduce the coarse particles misplacement around the vortex finder walls a tapered vortex finder (frustum shape) with various thickness levels are replaced in place of regular straight vortex finder tube. Similarly in order to reduce the fines misplacement, in this work, the conventional designs are modified to longer conical bodies having small cone angle sections. Small cone angle sections potentially can reduce the water split to underflow, hence reduction in the fines entrainment. For fine cut separation, high throughput, and increased sharpness of separation, the standard hydrocyclone design is targeted to modify with long conical sections and parabolic bodies. Longer bodies may provide sufficient residence time for fine particles, so that particles can be separated efficiently in the presence of centrifugal forces. In order to reduce the
turbulence intensity levels, thereby fine cut separation is aimed with the hydrocyclone modified by placing rods of different lengths and diameter at the center. All the above mentioned designs are studied in this work and explored for their ability to classify fine particles at improved separation levels. Selection of exact specifications is made from the author’s industrial experience in targeting the minimal turbulence, enhanced residence time of the fine particles and minimal short-circuiting flow along the vortex finder walls. Geometric variables of conventional D2 design are shown in Fig. 1. Complete dimensional details of all the modified designs used in this study are presented in Table 1.

![Fig. 1: Representation of geometrical variables in the conventional design](image)

### 2.1 Multiple cone angle design (D1)

The proposed geometry, D1 design consists of tangential feed port, tapered vortex finder with a short cylinder followed by two conical sections with angles of 20° and 10° as shown Fig. 2 (a). Tapered vortex finder with certain wall thickness may increase the residence time of coarse particles, thus reduce the short circuiting of coarse fraction to the overflow. The small cone angle sections increase the fluid residence time and keeps the vortex flow for long will eventually reduce the amount of water split to the underflow. Therefore a reduction of fine fraction
misplacement can be expected in the underflow. In this design, because of two cones, there is minimal change in the outer conical walls. Hence, the mean tangential velocity field transitions smoothly throughout of the hydrocyclone body.

2.2 Standard design (D2)

It is a standard conventional cylindrical-conical design as shown in Fig. 2 (b). The size of the inner diameter of vortex finder is maintained 0.4 times of the hydrocyclone diameter. Hydrocyclone cone angle is 20°. Spigot size varies 0.2 times of hydrocyclone diameter. Cone force ratio (ratio of spigot to vortex finder diameter) is kept as 0.5, as described by Bradley [38].

2.3 Small cone angle design (D3)

The proposed geometry, D3 is a modification to standard geometry (D2) with a small cone angle. It consists of tangential feed port, tapered vortex finder (smaller thickness compared to D1) with short cylinder followed by long conical section having cone angle of 10°. Small cone angle design also produces a possible sharper separation at high unit capacity because of long conical body [5]. Longer cylindrical bodies may also provide sufficient residence time for the fine particles to get separated to right product compared with short cylindrical body cyclones [39].

2.3.1 Air core free designs (D3_1, D3_2, D3_3)

It is known that turbulence plays an important role for the fines classification. Most of the fines below 10 microns follow the water vortices formed because of turbulence, leading to mixing of the fines rather separating them by size. In order to reduce the turbulence intensity levels and minimize the pressure losses, and thereby enhance the sharpness of separation, the D3 design is further modified to eliminate air core by placing a rod of different lengths, diameters, and shapes at the center. The rod diameter technically should replace only the air core size, but for the sake of flow stability, a smaller size rod than the air core is used. For a given design and operating conditions, the size of the air core can be calculated using the approach described by Narasimha et al. [26]. For the D3 design operating with only water, the estimated air core size is around 15 mm. The selected rod size of 12 mm is used for air core elimination in this design. Design D3_1 is the modified design of D3 with a rod of 12 mm at the center from vortex finder to spigot. Design D3_2 is the modified design of D3 with a rod of 12 mm at the center having a length
from the spigot to the cylindrical-cone intersection. D3_3 is modified design of D3 with a tapered rod of size 25 – 12 mm, varying the size from vortex finder tip to spigot section respectively. Air core elimination may improve the tangential velocity by reducing the pressure losses between the air and water in the hydrocyclone.

2.4 Parabolic body design (D4)

The proposed geometry, D4 is a novel design with the parabolic body ending with a small cone angle. It consists of tangential feed port, tapered vortex finder (thickness equal to D3 design) with very short cylinder followed by long parabolic section ending with an angle of 10° near the spigot section. With this novel design [40], it is expected a further reduction in the turbulence intensity levels of flow due to smooth body change throughout the hydrocyclone length unlike the sudden change from cylindrical to conical shape in the conventional designs.

2.5 Modified standard design (D5)

Suggested geometry, D5 is an altered design of D2 with a modification primarily in the conical section. It consists conventional design having a change in the vortex finder i.e. tapered with an angle of 2°, similar cylindrical section, and conical section with an angle of 10°. It is targeted to
decrease the coarse particle misplacement through tapered vortex finder walls and reduction in the fines misplacement because of lower cone angles.

Table 1: Dimensional details of various hydrocyclone designs (in mm) used in the simulations

<table>
<thead>
<tr>
<th>Design</th>
<th>Geometrical parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>diameter (Dc)</td>
</tr>
<tr>
<td>D1</td>
<td>102</td>
</tr>
<tr>
<td>D2</td>
<td>102</td>
</tr>
<tr>
<td>D3</td>
<td>102</td>
</tr>
<tr>
<td>D3_1</td>
<td>102</td>
</tr>
<tr>
<td>D3_2</td>
<td>102</td>
</tr>
<tr>
<td>D3_3</td>
<td>102</td>
</tr>
<tr>
<td>D4</td>
<td>102</td>
</tr>
<tr>
<td>D5</td>
<td>102</td>
</tr>
</tbody>
</table>

3. Methodology

Schematic of the various hydrocyclone designs and meshes used for the simulation studies are displayed in Fig. 2. All the 3-D unstructured hexahedral meshes are generated using ANSYS ICEM meshing tool. Flow governing equations in terms of Navier-Stokes equations are solved. Air core at low pressure center region is resolved by VOF method. Turbulence is modelled using quadratic pressure strain based RSM. The model equations and strategy for simulating air core and flow field are similar to Narasimha et al., [41] and Brennan [42].

3.1 VOF Model
VOF model is a numerical technique used for tracking the interface between free surfaces [43] by solving the momentum equation (1). Continuity equation (2) is solved for the volume fraction of the air ($\alpha_q$) and this tracks the position of the air core in this problem. In this work, geometric reconstruction scheme with piecewise-linear approach is used to track the interface between air and water. In piecewise-linear approach, the interface between air and water is assumed to have a linear slope within each cell and uses the linear shape for calculation of advection of fluids through the cell phases.

$$\frac{\partial}{\partial t} (\rho u_j) + \frac{\partial}{\partial x_j} (\rho u_j u_j) = -\frac{\partial p}{\partial x_j} + \rho g_j + \frac{\partial}{\partial x_j} \mu \left( \frac{\partial u_j}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

(1)

$$\frac{\partial \alpha_q}{\partial t} + u_j \frac{\partial \alpha_q}{\partial x_j} = 0$$

(2)

$\alpha_q$ is volume fraction of $q^{th}$ phase which varies between 1 and 0. $u_j$ is $j^{th}$ component of the velocity. A constant of 0.078 N/m is used for the surface tension between air and water. The average density, viscosity are calculated by the following manner

$$\rho = \alpha \rho_{water} + (1-\alpha) \rho_{air}$$

$$\mu = \alpha \mu_{water} + (1-\alpha) \mu_{air}$$

(3)

### 3.2 RSM Model

Turbulence is modelled using RSM. In the RSM model, unsteady transport equations given below are solved for individual Reynolds stresses $\overline{u_i' u_j'}$ [44]. Pressure strain is solved using the quadratic model proposed by Speziale et al. [45].

$$\frac{\partial}{\partial t} (\rho \overline{u_i' u_j'}) + \frac{\partial}{\partial x_k} (u_k \rho \overline{u_i' u_j'}) = \phi_{ij} + P_{ij} + D_{T,ij} + D_{L,ij} - \varepsilon_{ij} + F_{ij}$$

(4)

Here $\phi_{ij}$ is pressure strain, $P_{ij}$ is stress production, $D_{T,ij}$ is turbulent diffusion, $D_{L,ij}$ is molecular diffusion, $\varepsilon_{ij}$ is dissipation, $F_{ij}$ is production by system rotation is modeled by the following to close the equations.
\[
\phi_{ij} = -(C_1 \rho \varepsilon + C_1^* p) b_{ij} + C_2 \rho \varepsilon \left( b_{ik} b_{kj} - \frac{1}{3} b_{mn} b_{mn} \delta_{ij} \right) + \left( C_3 - C_3^* \sqrt{b_{ij}} \right) \rho k S_{ij} + C_4 \rho k \left( b_{ik} S_{jk} + b_{jk} S_{ik} - \frac{2}{3} b_{mn} S_{mn} \delta_{ij} \right) + C_5 \rho k \left( b_{ik} \Omega_{jk} + b_{jk} \Omega_{ik} \right)
\] 

\[
p_{ij} = -\rho \left( \frac{u_i \mu_j}{\sigma_k^2} \frac{\partial u_j}{\partial x_k} + \frac{u_j \mu_i}{\sigma_k^2} \frac{\partial u_i}{\partial x_k} \right) \quad D_{r,ij} = \frac{\partial}{\partial x_k} \left( \frac{\mu_i}{\sigma_k} \frac{\partial u_i}{\partial x_k} \right) \quad \sigma_k = 0.82
\]

\[
D_{\text{DPM}} = \frac{\partial}{\partial x_k} \left( \mu \frac{\partial u_j}{\partial x_k} \right) \quad \varepsilon_{ij} = \frac{2}{3} \delta_{ij} \rho \varepsilon
\]

Where \( b_{ij} \) is the Reynolds stress anisotropy tensor, \( \Omega_{ij} \) is mean rate of rotation tensor, \( S_{ij} \) is mean strain rate, \( \mu \) is turbulent viscosity. Turbulent viscosity is computed from the kinetic energy and dissipation rate transport equations as per \( \kappa-\varepsilon \) model and constants used in the quadratic pressure strain are \( C_1 = 3.4, C_1^* = 1.8, C_2 = 4.2, C_3 = 0.8, C_3^* = 1.3, C_4 = 1.25, C_5 = 0.4 \).

Two phase water and air simulations are run using VOF multiphase model coupled with RSM turbulence model for a physical time of 20 seconds for the solution convergence. All the results reported in the paper are time averaged over few thousand iterations equivalent to approximately 2 s of physical time after the solution converged (Net mass flow rate between feed, overflow and underflow is oscillating around zero value). Once the solution is converged, the solid particles of different sizes with a density of 2650 kg/m³ are tracked using DPM.

### 3.3 DPM Model

In this model Lagrangian reference frame is used to solve the properties of discrete/dispersed phase particles along with continuous phase, which is solved in Eulerian reference frame using Navier-Stokes equation as described in aforementioned section. Large number of spherical particles are tracked which include discrete phase inertia, hydrodynamic drag, gravitational force and shear lift forces for both steady and unsteady flows. 5th Runge-Kutta scheme [46] is used to integrate the particle equations of motion. Discrete Random Walk model (DRW) is used to predict the dispersion effect on the particles due to turbulent eddies present in the continuous phase. Basic assumptions behind the DPM is the presence of dispersed phase volume fraction is
very low, even though mass loading is higher, particle-particle interactions and the effects of the
particle volume fraction on the liquid phase are negligible. This model is not suitable if the
volume fraction of secondary phase is high. DPM involves prediction of discrete phase particle
trajectory by integrating force balance in the Lagrangian reference frame by equating particle
acceleration to forces acting on the particle.

\[
\frac{du_p}{dt} = F_D(\bar{u} - \bar{u}_p) + \frac{g(\rho_p - \rho_m)}{\rho_p} \tag{7}
\]

\[
F_D = \frac{18 \mu \cdot C_D \cdot Re_p}{\rho_p d_p^2} \tag{8}
\]

\[
Re_p = \frac{\rho d_p |\bar{u}_p - \bar{u}|}{\mu} \tag{9}
\]

where \( F_D(\bar{u} - \bar{u}_p) \) is drag force acting on the particle, \( C_D \) is drag coefficient, \( \bar{u} \) is continuous
phase velocity, \( \bar{u}_p \) is particle velocity, \( \rho_p \) is particle density, \( \rho \) is fluid density, \( \mu \) is fluid
viscosity, \( Re_p \) is the particulate Reynolds number.

### 3.4 ASM model with additional forces and rheological models

In multiphase simulations, the multiphase model is switched over from VOF to ASM model
before introducing solid particles with full size distribution as dispersive phase. The basic
assumption behind the ASM is that dispersed phases are in equilibrium with continuous phase
and such dispersed phases accelerate rapidly to terminal velocity. ASM is a simplified Eulerian
multiphase CFD methodology where the equations of motion are solved for the slurry mixture
and transport equations for the volume fractions of particulate phases \( p \) dispersed throughout a
continuous water phase \( c \) are also solved

\[
\frac{\partial}{\partial t} \alpha_p + \frac{\partial}{\partial x_i} (\alpha_p u_i) + \frac{\partial}{\partial x_i} (\alpha_p u_{pm,i}) = 0 \tag{10}
\]

\[
u_{pm,i} = u_{pi} - u_i
\]
Phase segregation in equation (10) is accounted for $u_{pm,i}$ which is the drift velocity of the phase $p$ relative to the mixture $m$. This is related to the slip velocity $u_{pc,i}$, which is the velocity of the phase $p$ relative to the continuous water phase $c$ by the formulation

$$u_{pmi} = u_{pci} - \sum_{j=1}^{n} \alpha_k \frac{\rho_k}{\rho_m} u_{sci}$$

$$u_{pci} = u_{pi} - u_{ci}$$

(11)

$u_{pc,i}$ is calculated algebraically in Manninen et al. [47] treatise by an equilibrium force balance and is implemented in Ansys’s Fluent in a simplified form. In this work Fluent has been used with the granular options and the slip velocity calculation has been modified to include (i) a shear dependent lift force based on Saffman’s expression [48]

$$u_{pci} = \frac{d^2(\rho_p - \rho_m)}{18 f_{rep} \mu_c} \left[ g_i - \frac{\partial}{\partial t} u_{mi} - u_{mj} \frac{\partial}{\partial x_j} u_{mi} + 0.75 \frac{\rho_c}{\rho_p - \rho_m} C_{ij} \varepsilon_{ijk} \omega_{mj} u_{pk} \right]$$

(12)

Equation (12) has been implemented in Fluent as a custom slip velocity calculation using a user defined function. $f_{rep}$ has been modelled with the Schiller Naumann drag law [49] but with an additional correction factor for hindered settling based on the Richardson and Zaki correlation [50]

$$f_{rep} = \left(1 + 0.15 \text{Re}^{0.687}_p \right) \alpha_p^{-4.65}$$

(13)

$$\text{Re}_p = \frac{d_p \rho_c |u_{pc}|}{\mu_c}$$

(14)

The lift force is a mechanical force, generated by solid particles as they move through a fluid, directed perpendicular to the flow direction. The inclusion of lift forces for the slip calculation will account the effect shear forces at wall on particles. The lift force expression derived by Saffman for the lift force [48] on a single particle was used here

$$F_{lip} = \frac{\rho_c}{8} \pi d_p^3 C_{lp} \varepsilon_{ijk} \omega_{mj} u_{pk}$$

(15)

The lift coefficient ($C_{lp}$) has been calculated as
The correlation proposed by Mei [51]. This equation is implemented in ANSYS’s Fluent™ by a modified slip velocity \( (u_{pc}) \) user defined function (UDF). The slip velocity of the air phase is disabled and assumed to be zero. Here, \( \alpha_p \) is the volume fraction of particles, \( f_{rep} \) is the drag coefficient, \( d_p \) (m) is the diameter of the particle, \( \rho_c, \rho_p \) (kg/m\(^3\)) are the continuous phase and particle phase density, \( \text{Re}_p \) is the Reynolds number, \( \mu_c \) (kg/ms) is the continuous phase viscosity, \( g_i \) (m/s\(^2\)) is the \( i \)th component of gravity, \( \epsilon_{ijk} \) is the permutation tensor, \( \omega_{mj} \) is the vorticity of the mixture.

Slurry rheology plays a vital role for the particle settling in gravity/centrifugal field at high solids concentration. Therefore the work has been carried out with Newtonian viscosity model corrected with fine fractions below 38 µm [17] to study the particle classification in the hydrocyclone. The equation (17) was obtained by calibrating against the measured viscosity data of various mineral slurries. Here \( \mu_m \), \( \mu_w \) is the mixture and water viscosity.

\[
\mu_m = \left[1 - \frac{\alpha_p}{0.62}\right]^{-1.55} \left(F_{-38\mu m}\right)^{0.39}
\]

### 3.5 Numerics

In all the simulations, the inlet is set to uniform velocity and outlet is set to atmospheric pressure boundary condition with an air back flow volume fraction equal to unity. So that air can be drawn back from both the outlets which enable the simulation to generate air core due to low pressure region at the center. A bounded central differencing scheme is used to discretize momentum equations. Pressure interpolation is calculated by using PRESTO. QUICK scheme is used to solve dispersed phase transport equations. SIMPLE method is used for the pressure velocity coupling. Fixed time step of 1.0×10\(^{-4}\)s is used in all the simulations. Mean flow field data are calculated by running the simulations over a few thousand time steps after the solution converged. In the DPM model, nearly 2000 particles with uniform size and sphericity (0.8) are
injected across feed boundary into the converged flow field. Sphericity is used to account the particle shape effect on the drag force and to simulate realistic behavior of particles. Grid independence check with respect to mean tangential and axial velocity for D3 design with 3 different meshes of 100354, 203399 and 303098 nodes is shown in Fig. 3. It can be observed that, velocity predictions of 203399 and 303098 are close to each other. Therefore, simulations are further continued with 203399 nodes for D3 design.
Fig 3: Grid independence check for D3 geometry in terms of (a) Tangential velocities and (b) axial velocities

4. Results

4.1 Velocity predictions

4.1.1 Tangential velocity

The predicted mean tangential velocity profiles of novel designs D1, D2, D3, D4, and D5 at the mid-section of hydrocyclone are displayed in Fig. 4 (a). It can be seen from the Fig. 4 (a) that at a given feed flow rate, the tangential velocities of all the designs except D4 are higher than the conventional D2 design. Predicted pressure drop (between feed to overflow), as shown in Table 2, is having a direct implication on the tangential velocity profiles in the hydrocyclones. D1 and D3 designs are predicting the higher pressure drops, leading to superior tangential velocity components compared to other designs. This increased tangential velocity is responsible for high centrifugal force, which may lead to an improved performance for fine particle separation through D1 and D3 designs. Design D1 shows peculiar behavior with very high tangential velocity throughout the hydrocyclone body because of the two cones. Design D4’s predicted pressure drop is slightly higher than the design D2. Even though there is slight improvement in the pressure drop data; the predicted tangential velocity component in D4 is lower than the D2. The reason could be attributed to the parabolic body shape, which might have increased flow cross-sectional area compared to D2 design. G-forces (ratio of centrifugal to gravitational forces) are calculated for all the designs and displayed in Table 2.

The predicted mean tangential velocities of the modified D3 designs i.e. D3_1, D3_2, D3_3 are also presented in Fig. 4 (b). One can notice that the modified designs with a rod placing at the center improve the tangential velocities except in the D3_3 design. Eliminating the air core with a rod at the center reduces the frictional losses between the air and water at the interface, which in turn increases the kinetic energy, i.e. improves the tangential velocity component. As explained in the literature [37], the selection of suitable rod diameter and length is important. The design with the tapered rod (D3_3) decreases the tangential velocity. Reason behind the
reduction of tangential velocity component could be ascribed to the rod with higher cross-
sectional end diameter, which might be disturbing the flow field as shown in Fig. 5. It can be
observed that, mean vorticity vectors of D3_3 design shows a disturbance in the flow field near
the vortex finder. A sudden increase in the radial velocities also confirms that a flow field
disturbance is observed near the vortex finder in D3_3 design. Therefore, reduction in the
velocities despite a significant increase with the pressure drops is observed in the simulations.
Fig. 4: Mean tangential velocities of (a) modified designs compared with standard design (b) air core eliminated designs at fixed water mass flow rate of 1.664 kg/s.

Fig. 5: (a) Mean vorticity vectors and (b) Mean radial velocity contours of D3_1, D3_3 designs.

4.1.2 Axial velocity

The CFD predicted mean axial velocity profiles of novel designs D1, D2, D3, D4, and D5 at the mid-section of the hydrocyclone are displayed in Fig. 6 (a). It can be observed that the change in
mean axial velocity component of the tested designs in comparison with the conventional hydrocyclone design is very minimal. This might cause a minor change in the water split to the underflow as shown in Table 2. Similar to tangential velocity profile, the axial velocity component predictions of D1 design also shows a peculiar behavior. A sudden dip in the axial velocity can be observed near the second cone region. All the designs show maximum positive axial velocities at the center and maximum negative velocities near the wall region. It indicates that all the new designs (D1, D3, D4 and D5) may have potential for fine cut size separation. Fig 6 (b) shows the predicted mean axial velocities of the modified D3 designs (D3_1, D3_2, D3_3) in comparison with D3 design’s profile. A small increment in the axial velocities of modified designs observed. This may help the particles to separate with slight improved separation efficiency level than the conventional hydrocyclone design. The reason for finer cut size is explained by using LZVV. In hydrocyclone, always two vortex flows exist. One flows towards upward (forced vortex) and another flows towards downward (free vortex). There exists a point where the vertical/axial velocity becomes zero. LZVV is defined as the locus of the fluid points at which the axial velocity becomes zero. A particle at this position has an equal chance of going either overflow or underflow. At this point, the particle is subjected to an equal outward centrifugal force and inward drag force. The drag force pulls the fine particles in to the inner vortex and centrifugal force pulls the coarse particles in the primary vortex that forms near the hydrocyclone wall. In general finer particles have an orbit less than LZVV and coarser particles have an orbit greater than LZVV. LZVV predictions by various designs are displayed in Fig. 7. A shift in LZVV towards the air core is observed in the modified designs compared with conventional designs. It indicates a reduced flow split in the underflow hence a reduction in the cut size can be expected.
Fig. 6: Mean axial velocities of (a) modified designs compared with standard design (b) air core eliminated designs (D3_1, D3_2, and D3_3) at fixed water mass flow rate of 1.664 kg/s.
Fig. 7: LZVV predictions by modified designs compared with standard design for a fixed water mass flow rate of 1.664 kg/s.

4.2 Air core profiles

CFD predicted contours and the axial variation of air core diameter for the D1, D2, D3, D4, D5 designs is shown in Fig. 8 (a) and (b). The size of the air core significantly varies for most of the designs compared with the conventional design. The shape of the air core appears to be parabolic and similar to the locus of zero vertical velocity (LZVV) shape [38]. In general, from Fig. 8, it can be observed that at the given optimum operating pressure, the air core diameter axially varies from the size of vortex finder to spigot diameter depending on the height of the hydrocyclone and outlet orifice diameters. Precisely, the air core size also depends on axial location for a given inlet pressure [26]. The occupied air core diameter in D1 design is slightly higher than the conventional design at the fixed feed flow rate. The occupied air core area of D3, D4, and D5 designs is smaller than the conventional D2 design. The resulted reduction in the air core diameter is responsible for the increase in water fraction reporting to overflow; hence the lower cut-size of separator is expected during the particle classification process. The air core is completely eliminated in the designs D3_1, D3_2, D3_3 having a rod at the center of hydrocyclone axis.
4.3 Turbulence Intensity

Flow in hydrocyclones is usually turbulent with a Reynolds number ranging from $10^5$ to $10^6$. Reynolds number is calculated based on hydrocyclone diameter as the characteristic length and feed inlet velocity. At this high Reynolds number, turbulent fluctuations inside hydrocyclones
are expected to be significant due to the collision of the inlet stream with the rotating stream, local recirculation’s and flow reversal near the spigot zone. Turbulence is also directly related to large velocity gradients found inside hydrocyclones [8]. Turbulence Intensity \( TI \) is defined as the ratio of root mean square velocity of fluctuations to mean velocity as shown in equation 18.

\[
TI = \frac{u'}{u_{avg}}
\]  

(18)

Fig. 9 shows the turbulence intensity profiles for all the designs calculated along the radius at the intersection of cylindrical and conical sections. From the Fig. 9 (a), it is clearly evident that the turbulence intensity values of D1, D3, D4, and D5 designs are lower than the conventional D2 design. It is known that high turbulence intensity levels are always associated with increased particle dispersion that eventually affects the performance of a given hydrocyclone. With an increase in the turbulence intensity the classification performance of hydrocyclone deteriorates. A significant reduction in the turbulence intensity is observed with the new designs compared with the conventional design (see the Fig. 9 (a)). An improvement in the classification performance of novel designs (D3, D4 and D5) may be expected due to turbulence minimization.
To minimize the turbulence further, the D3 design is modified to eliminate the air core by inserting various sizes of solid rods in the center of the hydrocyclone axis. From the Fig. 9 (b), it can be seen that the turbulence intensities of D3_1, D3_2 are significantly reduced. This indicates that designs D3_1 and D3_2 might have a better fines separation. As shown in the literature [37], the selection of suitable rod size for the air core elimination is very important. Fig. 9 (b) demonstrates that turbulent intensity values of design D3_3 are very high compared with the modified design D3. The expected reason is the flow field disturbances due to the tapered rod in D3_3 design (see the Fig. 5).

### 4.4 Performance predictions by discrete phase model (DPM)

After the solution converged with the two-phase model (VOF coupled with RSM turbulence model), the hydrocyclone design configurations are then subjected to predict the classification performance using DPM model for the particle phase. DPM uses quartz particles having sizes of 3.35, 10.25, 19.37, 28.27, 38 and 63 microns. Solid particles are injected across the feed boundary using instantaneous flow field obtained from the converged two-phase (water-air) flow simulations. The number of particles reported to the underflow and overflow are noted and used
to generate standard classification curve as a function of particle size. The classification efficiency curve is calculated numerically using the fraction of each size feed material that reported to the underflow. The Lagrangian particle tracking for each size is repeated 3 times and averaged to obtain a mean value. The predicted classification curves for various designs are shown in Fig 10. From the Fig 10 (a), it is observed that, for the given fixed flowrate, all the tested designs show a lower cut-size separation than the D2 design. This is because of the shift in LZVV (Fig. 7) location towards air core and enhancement of tangential velocity (Fig. 4). The cut-size for these designs is in the range of 8-13 microns, whereas the conventional design is having around 16 microns. Fig 10 (b) illustrates that rod insertion at the center of D3 design further decreases the cut size to 8-9 microns. Table 2 displays all the tested design cut diameters and their separation sharpness in comparison with the conventional hydrocyclone design. Designs D1 and D3 show an increased efficiency, i.e. design D1 and D3 may have better sharpness of separation and finer cut size compared with the standard design. The cut diameter of air core eliminated design, D3_3 with the tapered rod, is almost equal to design D3 but with the reduced sharpness of separation. Fig. 11 shows the most probable particle trajectories for a typical particle size of 10 microns while classifying them under turbulent and centrifugal forces in all the designs. It is observed that in D1, D2, D3, D4 designs, particle size of 10 microns is always classified to overflow except in D5 design, whereas in the modified D3 designs (D3_1, D3_2), same particle size is getting separated towards underflow stream.
Fig. 10: DPM predicted cut diameter of (a) modified designs compared with standard design (b) air core eliminated designs

Table 2: Predicted two-phase flow parameters by all designs at a mass flow rate of 1.664 kg/s

<table>
<thead>
<tr>
<th>Design</th>
<th>Air core diameter (m)</th>
<th>Water split to underflow (%)</th>
<th>Pressure drop (kPa)</th>
<th>Cut diameter ($d_{50}$)</th>
<th>Imperfection (I)</th>
<th>Sharpness of separation ($\alpha$)</th>
<th>Maximum G-force</th>
</tr>
</thead>
<tbody>
<tr>
<td>D1</td>
<td>23.1</td>
<td>6.58</td>
<td>63.89</td>
<td>13.2</td>
<td>0.23</td>
<td>8.33</td>
<td>443.07</td>
</tr>
<tr>
<td>D2</td>
<td>20.58</td>
<td>7.53</td>
<td>43.89</td>
<td>15.8</td>
<td>0.31</td>
<td>5.1</td>
<td>282.13</td>
</tr>
<tr>
<td>D3</td>
<td>10.48</td>
<td>6.79</td>
<td>77.92</td>
<td>11.6</td>
<td>0.28</td>
<td>7.58</td>
<td>582.13</td>
</tr>
<tr>
<td>D3_1</td>
<td>eliminated</td>
<td>10.17</td>
<td>78.99</td>
<td>8.9</td>
<td>0.28</td>
<td>9.62</td>
<td>686.82</td>
</tr>
<tr>
<td>D3_2</td>
<td>eliminated</td>
<td>13.49</td>
<td>83.27</td>
<td>8.8</td>
<td>0.29</td>
<td>8.62</td>
<td>736.45</td>
</tr>
<tr>
<td>D3_3</td>
<td>eliminated</td>
<td>29.96</td>
<td>111.74</td>
<td>10.6</td>
<td>0.56</td>
<td>4.18</td>
<td>347.55</td>
</tr>
<tr>
<td>D4</td>
<td>13.438</td>
<td>9.22</td>
<td>46.78</td>
<td>12.8</td>
<td>0.33</td>
<td>5.81</td>
<td>517.76</td>
</tr>
</tbody>
</table>
Compared to D1 design, it is believed that the existence of D3 design’s longer conical body is able to provide sufficient residence time for the fines to get classify efficiently under the centrifugal action. Fig. 12 (a) and (b) illustrate the most probable particle trajectories for the typical particle size of 7 and 13 microns in D1 and modified D3 designs respectively. It is observed that the 13 micron size particles are classified to the underflow, whereas the 7 micron size particles are classified to the overflow products. The efficiency of the hydrocyclone is expressed mainly with two parameters namely imperfection levels ($I$), and sharpness of separation ($\alpha$). Generally the slope of the efficiency curve is taken as sharpness of separation. In this paper, authors have used Whiten equation to fit the sharpness of separation. Imperfection levels are calculated by using the following equation (19).

$$I = \left( \frac{d_{75} - d_{25}}{2d_{50}} \right)$$  \hspace{1cm} (19)

Here $d_{25}, d_{50}, d_{75}$ is defined as the point on the efficiency curve at which 25%, 50%, 75% of the feed particles are reported to the underflow. Table 2 also illustrates the typical two-phase flow parameters of hydrocyclone, such as air core diameter, water split to the underflow, imperfection levels, and sharpness of separation for all the tested designs. From Table 2 it is observed that the air core eliminated designs, D3_1 and D3_2, show high pressure drops, low cut size with high sharpness of separation levels.
4.5 Performance predictions by multiphase ASM model

Using the DPM based CFD model, one could mainly simulate the hydrocyclones operating in dilute solids loading (< 5 wt %) conditions. Usually the particle-fluid interactions at high solids loading are not being accounted completely in the DPM model. Industrially, hydrocyclones are capable of operating at very high feed solids concentrations, hence an increase in cut-size is typically observed with high solids loadings. In order to predict the performance characteristics for the improved designs, say D1, D2, D3_1 and D3_2 at high feed solid concentrations, a number of multiphase simulations are carried out at 15 wt% with a quartz (fixed PSD) material using modified Algebraic Slip Mixture (ASM) CFD model.
Fig. 12: DPM predicted path of (a), (b), (c) 7 micron, (d), (e), (f) 13 micron particles in D1, D3_1 and D3_2 designs

In ASM model it is assumed that the dispersed phases are in local equilibrium with the continuous phase and such dispersed phases accelerate rapidly to their terminal velocities. The separation efficiency curve is constructed numerically using the mass fraction of each size feed material that reported to the underflow. Predicted cut diameters, water split to underflow, imperfection levels and sharpness of separation (α) from the multiphase simulations are presented in Table 3. Corresponding multiphase based classification performance curves are also shown in Fig. 13. As expected, the cut size of all the hydrocyclone designs is increased at feed solids concentration of 15%, compared to the DPM model (dilute concentration model) data. D2 design’s predicted cut diameter is 26.8 microns. Conversely, modified designs D1, D3_1, D3_2 show reduced cut diameter of 21.4, 22.2, and 21.4 microns, respectively. It is observed from Fig. 13, that D3_1, D3_2, and D1 hydrocyclones are showing significantly improved performance than the D2 design even at high feed solid loadings. The predicted sharpness of separation for the D2 design is 4.02. Whereas, the modified designs is in the range of 6-8. It is clear from the Fig. 13, that coarse particle entrainment to the overflow is drastically reduced for the modified designs and is believed due to the tapered vortex finder presence in place of conventional straight cylindrical vortex finder. As cited in the literature, one can assume and account the amount of fines misplacement in the underflow is equals to the amount of water recovery in the underflow
Modified designs D1, D3_1, and D3_2 have reduced the water split to the underflow compared to D2 design (see Table 3). This reflects a possible reduction of the fines entrainment to underflow and thereby increases the separation efficiency.

Fig. 13: Multiphase predicted cut diameters of D1, D3_1 designs compared with conventional D2 design

Fig. 14: Mean contours of 3.35 micron volume fraction of (a) D1 (b) D2 (c) D3_1 (d) D3_2 conical sections
Fig. 14 shows the volume fraction contours of 3.35 micron size particle in the conical section of D1, D2, D3_1 and D3_2 designs. The volume fraction level of 3.35 micron particle is minimal at the walls in D3_1 and D3_2 hydrocyclone designs compared to the standard D2 design. It may indicate that the presence of bigger size particles is high towards the wall because of high centrifugal forces. Therefore 3.35 micron particles may have very less chance to escape through the underflow. In other words, the air core eliminated hydrocyclone designs, D3_1 and D3_2, show low levels of fines misplacement than the conventional D2 and the modified D1 designs. Although there is a slight difference in the predicted cut sizes of designs D1, D3_1 and D3_2; Design D3_1 shows high sharpness of separation and minimum fines misplacement. Quantified imperfection values of the designs are also displayed in Table 3. The reasons behind this improvement are believed due to the enhanced high tangential velocities, low turbulence levels, small radial fluctuations as discussed in two-phase flow analysis. High tangential velocities benefit the coarse particles to get separated at maximum efficiency levels. Low turbulence and low radial fluctuations help the fine particles to carry away from the underflow and flow towards the overflow stream.

Table 3: Predicted multiphase flow parameters by novel designs

<table>
<thead>
<tr>
<th>Design</th>
<th>Water split to underflow (%)</th>
<th>Cut diameter (d_{50})</th>
<th>Imperfection (I)</th>
<th>Sharpness of separation (α)</th>
</tr>
</thead>
<tbody>
<tr>
<td>D1</td>
<td>8.15</td>
<td>21.4</td>
<td>0.21</td>
<td>6.13</td>
</tr>
<tr>
<td>D2</td>
<td>9.09</td>
<td>26.8</td>
<td>0.23</td>
<td>4.02</td>
</tr>
<tr>
<td>D3_1</td>
<td>5.88</td>
<td>22.2</td>
<td>0.16</td>
<td>7.82</td>
</tr>
<tr>
<td>D3_2</td>
<td>7.41</td>
<td>21.4</td>
<td>0.2</td>
<td>6.85</td>
</tr>
</tbody>
</table>
Fig. 15: Mean contours of 28.27 micron volume fraction of D1, D2, D3_1, and D3_2 designs in (a) cylindrical and (b) conical sections

Fig. 15 displays the contours of 28.27 micron size particle distribution both in cylindrical and conical sections for all the designs. One can note that the movement of 28.27 micron size particles towards the overflow is higher in the standard D2 design than the all other designs. The coarse particle misplacement in the potential designs D1, D3_1, and D3_2 is low compared with
conventional design and is believed to be due to the tapered vortex finder in place of conventional straight cylindrical tube.

Fig. 16: Particle equilibrium radius of different size particles in (a) standard D2 design and (b) multiple cone angle D1 design
Fig. 16 displays the mean radial position of the presence of maximum volume fraction of the particles in standard D2 and multiple cone angle D1 design. In other words, these graphs display the particle equilibrium radius, which means that, after the entrance of the particle in to the hydrocyclone it reaches an orbit of certain radius based on particle size and tends to rotate in that particle orbit until its exit from the hydrocyclone. In general finer particles have an equilibrium radius lower than LZVV radial position and passes through the overflow. Coarser particles have an equilibrium radius greater than LZVV and pass through the underflow. Near cut size particles usually have an equilibrium radius equal to LZVV and have equal chances to pass through overflow and underflow. From the Fig. 16 (a), (b), it can be observed that fine particles (3.35, 10.25 microns) have an equilibrium radius less than LZVV and coarse particles (38, 63 microns) have an equilibrium radius greater than LZVV in both the designs. In the D1 design the multiphase cut size is near to 19.37 micron, so this particle is showing an equilibrium radius equal to LZVV and whereas the 28.27 micron particle equilibrium radius greater than LZVV. In case of standard D2 design the multiphase cut size is near to 28.27 micron and tends to show equilibrium radius equal to LZVV and the 19.37 micron size particle’s equilibrium radius is lower than LZVV. The main reason for fine cut size in case of new designs (say D1) is the shifting of LZVV towards air core and the same is confirmed through the equilibrium radii.

4.6 Practicality of the novel designs for fabrication and testing

The CFD exploration studies results mainly three potential designs namely D1, D3 and D3_1 suitable for fines classification to further required fabrication and testing steps. The following issues are considered in order to check the practicality of these designs:

- All the three designs have essentially complicated tapered vortex finder (VF) and inlet parts. With the advances in polymer technology that can offer the required material quality in terms of toughness and strength suitable for casting of the tapered vortex finder and complex inlet designs instead of steel parts is possible.

- As required for D3_1, the full length rod insertion and required mechanical supporting arrangement for rod externally is also possible. This requires first the estimation of size of air core that can be formed in the given hydrocyclone design, where CFD inputs could be highly useful.
As recommended for D1 design, two difference size cones can be arranged with the suitable flange designs precisely. The tapered VF, involute inlet sections are again required special high density polyethylene (HDPE) casting routes.

Authors have now ordered the fabrication of these three hydrocyclone designs successfully using polyurethane HDPE grade through third party. Commissioning and performance guarantee tests are in progress and will be available in the near future.

5 Conclusions

The effect of design parameters on the hydrocyclone separation characteristics are investigated computationally.

- All the modified design shows high tangential velocities, fine cut separation and high sharpness of separation compared with the conventional design.

- Tapered vortex finder in place of straight cylindrical tube reduces the coarse particle misplacement to the overflow in the multiple cone angles and small cone angles designs.

- Long cone provides sufficient residence time for all the particles to separate based on the centrifugal/drag forces. This modification obtains finer cut separation than standard design.

- Air core elimination by placing a rod in the center decreases the turbulent levels drastically. This results in the reduction of fines misplacement to the underflow.

- DPM predicts the cut size of 16 microns for conventional design, whereas the modified designs show values in the range of 8-13 microns.

- Multiphase simulations with modified mixture model including lift forces and rheology model also shows modified designs are having high separation efficiency and fine cut size compared with conventional design. The reason for fine cut is also explained by the help of LZVV.
• Cut size ($d_{50}$) of all the designs are increased up to 20 microns with an increase in the feed solids concentration.

6 Acknowledgements

Authors would like to express their sincere thanks to SERB, Department of Science and Technology, Government of India for the financial support and encouragement to undertake these studies under the project (SR/S3/CE/004/2011) during 2011-2014.

References


Graphical abstract
Highlights

1. Novel hydrocyclone designs are proposed using multi-phase CFD model
2. All the modified designs are having finer cut and improved sharpness of separation
3. Tapered vortex finder minimizes coarse particle short circuiting to overflow
4. Finer cut is explained using LZVV thereby particle equilibrium positions are displayed