Numerical Simulation of Particle Separation in Centrifugal Air Classifiers

by

Mohammad Barimani

B.Sc., The University of Tehran, Iran 2013

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

MASTER OF APPLIED SCIENCE

in

The Faculty of Graduate and Postdoctoral Studies

(Mechanical Engineering)

THE UNIVERSITY OF BRITISH COLUMBIA

(Vancouver)

January 2016

© Mohammad Barimani 2016
Abstract

The demand for fine mineral powder in various industries has stimulated creative methods for separating the fine portion of particles from a mixture. Among the many different types of classifiers invented, centrifugal rotor air classifiers are characterized by their capability in producing ultra-fine products with a cut-size as low as 3um.

Classification occurs due to the size-dependence of aerodynamic and inertial forces acting on particles: coarse particles have a higher ratio of centrifugal force to aerodynamic drag than do fine particles, and therefore are preferentially ejected to the classifier perimeter. Therefore, the high speed rotor located inside the classifier is key to classification. Computational fluid dynamics (CFD) is utilized in this study to investigate the motion of calcium carbonate particles in a rotor classifier. The single phase flow in two- and three-dimensional models of the rotor is computed. Two turbulence models, namely K-Omega and RSM, are applied to close the Reynolds-averaged Navier-Stokes equations.

Once the single phase flow has been computed the motion of solid particles is simulated using the Discrete Phase Model. This model ignores particle-particle interactions and the influence of the particles on the air flow. The motion of the particles is coupled to a statistical model of the turbulent velocity fluctuations. By tracking hundreds of particles, the efficiency for a variety of hypothetical classifiers is estimated. Though the CFD models, in comparison with experiments, cannot accurately predict the absolute cut-size values, they have proved effective in predicting cut-size shifts as a result of rotor geometry modification or alternative operating conditions. Based on these simulations two new rotors were built and the change in cut-size was predicted within 30% accuracy.

Based on the paths of a large number of particles tracked in various operating conditions, regions in the rotor with very high particulate concentrations are identified. We speculate that this elevated concentration makes particle-particle interactions much more important than would be expected based on the feed concentration, which could in turn reduce the acceptance of the smallest particles.
Preface

Experimental measurements used to validate the two-dimensional model, presented in Chapter 3, is the work of George Sterling at Imasco Minerals Inc.

The authors of Chapter 2 are Mohammad Barimani, Dr. Sheldon Green and Dr. Steven Rogak. Dr. Green suggested applying the periodicity principle and the three-dimensional simulations.

The authors of Chapter 3 are Mohammad Barimani, Dr. Sheldon Green and Dr. Steven Rogak. Dr. Rogak and Dr. Green suggested calculating the concentration effects.

The authors of Chapter 4 are Mohammad Barimani, Dr. Sheldon Green and Dr. Steven Rogak.

A version of Chapter 2 and 3 is to be published in a journal paper that has not been determined yet. Mohammad Barimani, Sheldon Green, Steven Rogak, Particle Concentration in Ultrafine Rotor Air Classifiers (2016). I conducted all the simulations and wrote most of the manuscript.
# Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abstract</td>
<td>ii</td>
</tr>
<tr>
<td>Preface</td>
<td>iii</td>
</tr>
<tr>
<td>Table of Contents</td>
<td>iv</td>
</tr>
<tr>
<td>List of Tables</td>
<td>vi</td>
</tr>
<tr>
<td>List of Figures</td>
<td>vii</td>
</tr>
<tr>
<td>Acknowledgments</td>
<td>ix</td>
</tr>
<tr>
<td><strong>1 Introduction</strong></td>
<td>1</td>
</tr>
<tr>
<td>1.1 Fundamental mechanism of classifier operation</td>
<td>1</td>
</tr>
<tr>
<td>1.2 Literature review</td>
<td>4</td>
</tr>
<tr>
<td>1.2.1 Fluid mechanics and CFD</td>
<td>4</td>
</tr>
<tr>
<td>1.2.2 Prior studies of air classification</td>
<td>8</td>
</tr>
<tr>
<td>1.3 Organization of this thesis and key contributions</td>
<td>10</td>
</tr>
<tr>
<td>1.4 Contributions</td>
<td>12</td>
</tr>
<tr>
<td><strong>2 Methods and modeling details</strong></td>
<td>13</td>
</tr>
<tr>
<td>2.1 Introduction</td>
<td>13</td>
</tr>
<tr>
<td>2.2 Domain and appropriate boundary types</td>
<td>13</td>
</tr>
<tr>
<td>2.3 Mesh generation strategy</td>
<td>15</td>
</tr>
<tr>
<td>2.4 Turbulence model</td>
<td>17</td>
</tr>
<tr>
<td>2.5 Convergence</td>
<td>18</td>
</tr>
<tr>
<td>2.6 Particle injection</td>
<td>19</td>
</tr>
<tr>
<td>2.7 Solution schemes</td>
<td>21</td>
</tr>
<tr>
<td>2.7.1 Transient or steady</td>
<td>21</td>
</tr>
<tr>
<td>2.7.2 Compressible or incompressible</td>
<td>22</td>
</tr>
<tr>
<td>2.7.3 Coupling and spatial discretization schemes</td>
<td>22</td>
</tr>
<tr>
<td>2.8 Mesh independence study and discretization error</td>
<td>23</td>
</tr>
</tbody>
</table>
# Table of Contents

2.9 Summary .................................................. 23

3 Results and discussion ...................................... 26
   3.1 Two-dimensional simulation ............................ 26
      3.1.1 Pressure distribution .............................. 26
      3.1.2 Velocity field .................................. 28
      3.1.3 Efficiency curves and cut-size .................. 29
      3.1.4 Comparison with experiments .................... 31
      3.1.5 Particle concentration distribution ............ 32
   3.2 Three-dimensional simulations ......................... 37
      3.2.1 Velocity field .................................. 37
      3.2.2 Efficiency curves and concentration contours ... 37
   3.3 Summary ................................................ 39

4 Optimization procedure ...................................... 42
   4.1 Introduction ........................................... 42
   4.2 Optimization parameters ............................... 44
      4.2.1 Effect of blade angle ............................ 44
      4.2.2 Scale .......................................... 45
      4.2.3 Blade length and number ......................... 46
   4.3 Summary ................................................ 47

5 Thesis conclusions .......................................... 50
   5.1 Future work ........................................... 51

Bibliography .................................................. 53
List of Tables

1.1 Rotor dimensions and working conditions . . . . . . . . . . . . . 10
2.1 Turbulence models used . . . . . . . . . . . . . . . . . . . . . 18
2.2 Spatial discretization schemes for K-Omega . . . . . . . . . . . 23
2.3 Mesh refinement study (3D study) . . . . . . . . . . . . . . . . 24
3.1 Operating Conditions . . . . . . . . . . . . . . . . . . . . . . . 26
3.2 Model Validation . . . . . . . . . . . . . . . . . . . . . . . . . 33
4.1 Rotor dimensions and operating conditions . . . . . . . . . . . . 45
4.2 Rotor dimensions and operating conditions . . . . . . . . . . . . 46
4.3 New conditions . . . . . . . . . . . . . . . . . . . . . . . . . . 46
4.4 Rotor dimensions and operating conditions . . . . . . . . . . . . 47
## List of Figures

1.1 Different sections of the classifier ........................................... 2
1.2 Cut-size and Fishhook .......................................................... 2
1.3 The three major forces acting upon a typical particle ................. 3
1.4 Drag coefficient on spherical objects ..................................... 7
1.5 Different Air Classifier Types [7] ........................................... 8

2.1 Rotor Cross Section ............................................................ 14
2.2 Mesh and Boundary types ..................................................... 15
2.3 3D domain and boundary types ............................................ 16
2.4 3D mesh ............................................................................. 16
2.5 Averaged Tangential Velocity on periodic boundary for different iterations ......................................................... 19
2.6 Particle tracking: (a) using averaged values only and (b) using Discrete Random Walk .............................................. 22
2.7 (a) Velocity magnitude profiles along the periodic line and (b) Average of this velocity versus Number of cells (2D study) ...... 24

3.1 Rotor V_1 ............................................................................ 27
3.2 Contours of constant pressure ($p_a$), (a) with rotation and (b) no rotation .......................................................... 27
3.3 Pressure .............................................................................. 28
3.4 Velocity contours: (a) radial component and (b) tangential component .......................................................... 29
3.5 Radial velocity profile on different arcs ................................. 30
3.6 Tangential velocity profile on different chord lines ......... 30
3.7 Efficiency curves and their dependence on tracking time .......... 31
3.8 $d_{50}$ for two rotors ............................................................... 32
3.9 Radial Location of five particles for three different diameters ........ 34
3.10 Non-dimensional concentration for various particle diameters .... 35
3.11 Concentration contours for different conditions; all variables held constant except: (a) flow rate doubled, (b) rotor rotational velocity decreased by 25%, (c) blade angle is reduced to zero and (d) RSM turbulence model is used .............................................. 36
### List of Figures

3.12 Tangential Velocity Contours on periodic plane .......................... 38
3.13 Radial Velocity profiles ......................................................... 38
3.14 Efficiency curves for different reject percentages of rotor flow rate (28.3 m³/min) ......................................................... 39
3.15 Peak Concentration enhancement as a function of reject flow percentage ......................................................... 40
3.16 Concentration enhancement contours for 3.5% and 5% reject flow ......................................................... 40
3.17 Concentration enhancement (a) without and (b) with swirl inlet ......................................................... 41

4.1 Rotor V_0 with curved blades ......................................................... 43
4.2 Curved-blade rotor efficiency curve: clockwise and counterclockwise rotation ......................................................... 43
4.3 Blade angle possibilities: (a) scooping blade with negative angle, (b) radial blade with zero angle and (c) non-scooping blade with positive angle ......................................................... 44
4.4 non-dimensional cut-size $d^{98}$ versus blade angle ......................................................... 45
4.5 Cut-size as a function of blade length ......................................................... 48
4.6 Contours of absolute tangential velocity for a typical blade shorter than critical length ......................................................... 48
4.7 Cut-size as a function of blade length and number for 53 m/s blade tip speed, 28.3 m³/min flow rate and +30° blade angle ......................................................... 49

5.1 Gama densitometer measures the density of particles outside the rotor ......................................................... 52
Acknowledgments

I would like to pay special tribute to my supervisors Prof. Sheldon Green and Prof. Steven Rogak for supporting this research, their technical guidance and exceptional patience during the past two years. It was a great pleasure to know you and to work with you.

I wish to thank the National Sciences and Engineering Research Council of Canada for their financial support of this research.

Finally, my deepest gratitude goes to my parents for their nonstop support during this time.
Chapter 1

Introduction

Centrifugal air classifiers are used to separate the fine portion from a mixed-size dust [1]. For example, they are used industrially to produce ultra-fine calcium-carbonate with aerodynamic diameter below 5µm. Figure 1.1, the particular centrifugal air classifier studied here, is typical of such classifiers. Air carrying the mixed-size feed is introduced around the outer diameter of the rotating vaned rotor. Nearly all of the air flows through the rotor, radially inward, dragging the finest particles with it. The drag on coarser particles is lower than the centrifugal force acting on them, and consequently they are flung to the outside radius of the classifier, where they may be collected either by gravity or by way of a small airflow. As alluded to above, the underlying principle of centrifugal air classifiers is the balance between the inward drag force due to the air flow and the outward centrifugal force due to the high speed rotation (additional forces due to particle radial acceleration are orders of magnitude less than the other forces). For fine particles, the drag exceeds the centrifugal force and the particles migrate to the “accept” stream.

The performance of a centrifugal air classifier is characterized by the efficiency function $F$, which is the fraction of particles of a given size that end up in the reject stream. The main purpose of classifier modeling is to predict and understand the shape of $F$. The size distribution of the feed, convolved with $F$, determines the size distribution of the reject particles. This efficiency function, $F$, is described by $d^x$, where $x$ percent of all particles with diameter $d$ end up in the coarse fraction (reject stream), as shown in Figure 1.2. The “cut-size” of the classifier output is typically taken as $d^{50}$, $d^{97}$ or $d^{98}$. The fish-hook is a rise in the recovery of fine particles and is shown in a red circle.

1.1 Fundamental mechanism of classifier operation

Each particle experiences forces that are different in nature and their significance. Five forces recognized in the flow are:

- Drag force, $F_d$
- Centrifugal force, outward, $F_c$
1.1. Fundamental mechanism of classifier operation

Figure 1.1: Different sections of the classifier

Figure 1.2: Cut-size and Fishhook
1.1. Fundamental mechanism of classifier operation

The three major forces acting upon a typical particle:

- Gravity force, downward, $F_g$
- Wall collision force, $F_w$
- Particle-particle interaction force, $F_p$

The first two forces are major for separation since they are orders of magnitude larger than others. Drag is the force that pushes the particles inside the rotor to the accept side while the centrifugal force originated from the rotational velocity of air is the force that ejects the particles. The role of gravity is to collect those particles rejected by the centrifugal force. A schematic of these forces are shown in Figure 1.3. The interaction of particles results in unfavorable forces leading to inefficiencies and agglomeration.

For a single particle if drag force is larger than centrifugal drag force, the particle is accepted. Inversely if the centrifugal force dominates the drag force, the particle is coarse and is rejected.

\[ F_d > F_c \implies \text{accepted particle} \]

\[ F_d < F_c \implies \text{rejected particle} \]

If a particle experiences almost equal inward and centrifugal drag forces, the particle shows a probabilistic behavior and depending on the chaotic flow pattern (that
is represented by flow turbulence in this study), a probability could be defined for the fate of the particle:

\[ F_d \simeq F_c \implies \text{accepted/rejected particle} \]

This probabilistic behavior leads to an inefficiency for a range of particles. Figure 1.2 shows an actual versus ideal efficiency curve for a classification process.

The balance of forces could be written for cut-size range as follows:

\[ F_c + F_d = 0 \]  \hspace{1cm} (1.1)

Based on the following assumptions, the cut-size (shown in Equation 1.2) is extracted from Equation 1.3: solid body rotation is assumed; the radial velocity is uniform; Stokes flow is assumed; particles are spheres; and particle-particle interactions are ignored. Equation 1.2 shows that the cut-size is inversely proportional to the rotor angular velocity and also proportional to the square root of air volumetric flow rate.

\[ d_c = \sqrt{\frac{36 \nu \rho_a Q}{\pi \rho_c h D^2 \omega^2}} \]  \hspace{1cm} (1.2)

where \( \omega \) is the angular velocity of the rotor, \( Q \) is volumetric flow rate, \( h \) is rotor height, \( \nu \) is kinematic viscosity, \( \rho_a \) is air density and \( \rho_p \) is particle density.

### 1.2 Literature review

This section is divided into two parts. First we discuss the fundamental governing equations in fluid mechanics in brief and introduce the spherical drag model. The second part explains the similar works starting with a short introduction to various types of classifiers and then the numerical studies focusing particularly on centrifugal air classifiers.

#### 1.2.1 Fluid mechanics and CFD

**Navier-Stokes equations (NSE)**

Based on continuum assumption, fluid flow is generally described by Navier-Stokes equations (NSE) shown below.

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \]  \hspace{1cm} (1.3)
1.2. Literature review

\[ \rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \nabla \cdot \mathbf{\tau} + \rho \mathbf{g} \]  \hspace{1cm} (1.4)

where \( t \) is time, \( \mathbf{u} \) is velocity vector, \( \rho \) is density, \( p \) is pressure, \( \mathbf{\tau} \) is stress tensor and \( \mathbf{g} \) is body acceleration. Equation 1.3 is a form of mass conservation known as continuity. Equation 1.4 is a derivation of Newton’s Second Law of motion applied to an infinitesimal fluid element. A famous form of NSE is extracted based on incompressibility (constant \( \rho \)) and Newtonian fluid (\( \mathbf{\tau} = \mu \nabla \mathbf{u} \)) assumptions:

\[ \nabla \cdot \mathbf{u} = 0 \]  \hspace{1cm} (1.5)

\[ \rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho \mathbf{g} \]  \hspace{1cm} (1.6)

where \( \mu \) is kinematic viscosity. Equation 1.6 is a second order non-linear partial differential equation with no generic form of analytical solution. Due to its non-linearity, NSE are highly sensitive to boundary and initial conditions. This results in chaotic nature of the solution that is commonly known as “turbulence”. The onset of turbulence is usually described by the non-dimensional Reynolds’ number:

\[ Re = \frac{\rho V L}{\mu} = \frac{\text{inertial forces}}{\text{viscous forces}} \]  \hspace{1cm} (1.7)

where \( V \) and \( L \) are the characteristic velocity and length of the flow respectively. A high Reynolds number indicates a high energy flow and hence a turbulent one. A low \( Re \), on the other hand, shows the dominance of viscous to inertia force implying a laminar flow. The \( Re \) number showing the transition from laminar to turbulent flow is dependent on the type of flow and the definition of characteristic parameters. For example, in a simple pipe flow, studies have shown the transition \( Re \) number of 2500.

A key concept in studying turbulence is “energy cascade”: the transfer of turbulent energy in the form of eddies from large scale (the flow length) to the smallest scale (Kolmogorov length). The higher the \( Re \), the smaller the Kolmogorov length. For more information see [2].

Reynolds averaged navier stokes equations (RANS)

Due to the stochastic nature of NSE and its high sensitivity to initial and boundary conditions, a statistical form of NSE is introduced using Reynold Averaging. In this method, the exact solution variables are decomposed into mean and fluctuating components:
1.2. Literature review

\[ \phi = \bar{\phi} + \phi' \]  \hspace{1cm} (1.8)

where \( \phi \) is a scalar such as pressure or velocity components. Substituting this form for each variable into NSE and averaging, the Reynolds Averaged Navier Stokes (RANS) equations are obtained. These equations look like NSE with the addition of one nonlinear term, \( -\rho \bar{u}_i u_j' \), that is commonly known as Reynolds Stresses. In order to close RANS equations, this tensor should be replaced with known terms. Equation 1.9 shows the general form proposed by Boussinesq [3] in Einstein notation.

\[ -\rho \bar{u}_i u_j' = \mu_t (\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij}) - \frac{2}{3} \rho k \delta_{ij} \]  \hspace{1cm} (1.9)

The assumption of a scalar \( \mu_t \) (turbulent viscosity) implies an isotropic turbulence. Different turbulence models are proposed to estimate \( \mu_t \). In the two equation models (such as \( k-\epsilon \) or \( k-\omega \)) two transport equations in addition to momentum equations are solved for turbulent kinetic energy \( k \) and the rate of dissipation of this energy \( \epsilon \) (or the specific rate of dissipation \( \omega \)). An alternative is to skip Boussinesq assumption and solve a transport equation for each of the components of Reynolds Stresses that is called Reynold Stress Model (RSM) that is an an-isotropic turbulence model. For more information see [4, 5].

**Particle drag models**

Solving for the flow structure in the classifier is necessary but not enough. In order to extract important features of a classifier such as the efficiency curve or cut-size, particle trajectories should be generated; this requires recognizing different forces on the particles (as explained in section 1.1). Among these forces, drag force is the one pushing the fine particles to the accept stream and requires to be approximated. To do that, the non-dimensional drag coefficient is defined as follow:

\[ C_d = \frac{F_d}{\frac{1}{2} \rho U^2 A} \]  \hspace{1cm} (1.10)

where \( C_d \) is drag coefficient, \( F_d \) is drag force, \( \rho \) is fluid density, \( U \) is relative velocity magnitude between solid object and fluid medium and \( A \) is the reference area of the solid object. The drag coefficient is a function of the shape of the solid object, the flow regime described by \( Re = \rho U D \mu^{-1} \) (\( D \) is particle diameter), the flow direction and Mach number \( Ma \). At a low Reynolds number (\( Re < 0.5 \), Stokes regime) for spherical objects, the drag coefficient could be approximated as follow:

\[ C_d \approx \frac{24}{Re} \]  \hspace{1cm} (1.11)

\[ C_d \approx \frac{24}{Re} \]  \hspace{1cm} (1.11)

The dependence of drag coefficient to \( Ma \) at small mach numbers is negligible
1.2. Literature review

Figure 1.4: Drag coefficient on spherical objects

\[ C_d = \frac{24}{Re} \]  
(1.11)

This is known as Stokes’ law. Oseen has improved the Stokes calculation for low Re numbers and proposed the following relation:

\[ C_d = \frac{24}{Re} + \frac{9}{2} \quad Re < 1 \]  
(1.12)

At a higher Re number, a noticeable deviation from Equation 1.11 is observed that is shown in Figure 1.4. For the transition zone \((1 < Re < 10^3)\) the following relation is proposed.

\[ C_d = \frac{24}{Re} + \frac{4}{Re^{0.33}} \quad 1 < Re < 10^3 \]  
(1.13)

and for the rest of the transition regime \((10^3 < Re < 2.5 \times 10^5)\) the drag coefficient is almost constant.

\[ C_d = 0.4 \quad 10^3 < Re < 2.5 \times 10^5 \]  
(1.14)

At \(Re \approx 2.5 \times 10^5\), the drag coefficient suddenly drops due to a shift from laminar to turbulent boundary layer.
1.2. Literature review

1.2.2 Prior studies of air classification

Two major types of classifiers are “hydrocyclones” and “air classifiers” that are aimed at wet and dry classification respectively. Literature is filled with empirical and numerical studies of hydrocyclones that are not the focus of this study.

Various types of air classifiers are designed. The choice depends on factors including product material, sharpness of cut, range of fineness, economic considerations etc. Figure 1.5 summarizes the types of industrial air classifiers.

Gravitational and Cascade Air classifiers are usually used to produce the product cut-size of 0.1 to 0.2mm [6, 7]. Centrifugal air classifiers are used to achieve a finer cut-size ranging from 5μm to 1000μm. Many empirical models have been proposed: Wang et al. [8] propose a mathematical model that predicts the cut size of a rotor air classifier as a function of various parameters with an emphasis on rotor blade angle. Validating their theoretical model with dozens of experiments, they conclude that the zero angle of blade (radial blades) results in the lowest cut size. The classifier they study is quite different from ours and in Chapter 4 of this thesis it will be shown that this conclusion is not completely compatible with our simulation results.

Purely empirical or analytical models do not suffice in terms of accuracy as Gimbun et al. [9] compare one CFD and four famous empirical models for hydrocyclones with their own experimental data and conclude that CFD is more accurate.
1.2. Literature review

Within six experiments, CFD resulted in an average deviation of 3.7% while the empirical models predict the cut-off size with average deviations ranging from 12% to 34%.

Dozens of studies employ CFD to quantitatively analyze air classifiers. Karunakumar et al. [10] developed a geometrically complete model of a centrifugal rotor air classifier. Investigating particle trajectories, they recognize three major forces influencing particles motion: air drag, centrifugal force and wall-rebound force. They conclude the high sensitivity of the flow pattern to the internal geometry of the classifier and also the coupling of the classification characteristics to this flow pattern. To discretize the domain and solve the NSE equations numerically, they use a 600000-cell unstructured mesh that seems to be too coarse to capture details of the flow. There is also no discussion on the mesh independence.

In recent years the focus of the CFD simulations has shifted to “fish-hook” effect which is an unexpected increase in the recovery of fine particles in the underflow (reject stream). The occurrence of fish-hook is reported for both hydrocyclones and air classifiers consistently [11].

For air classification, various explanations of fish hook are discovered depending on the type of classifier: Estwariah et. al. [12] simulate a circulating air classifier and emphasize on the importance of bottom vane angles. They conclude that radial vanes (versus angled ones) require less rotational speed of the rotating disk in the center and hence are more efficient. By investigating particle trajectories they assert that the main cause of fish hook is the guide vane rebound effect. However they also conclude that whether angled or radial vanes are used, fish-hook is observed with almost the same characteristics but shifts to a finer or coarser range of particle size. Though this point is discussed vaguely, it seems like that the guide vanes have a strong effect on the cut-size rather than fish-hook.

Guizani et al. [13] study the origins of fish-hook in a rotor air classifier. They use Multiple Reference of Frame (MRF) method and three different turbulence models to solve for the complete geometry of their classifier. They adopt a maximum of one-million-cell hybrid mesh with hexahedral elements in sensitive areas such as blades. Solving in an unsteady mode with a $5 \times 10^{-4}$s time step they discover the unsteady movement and breaking of the vortices in the interior of the rotor. Consequently they conclude that secondary recirculation flows and bubble-type vortex breakdown are the main sources of this inefficiency. They justify the use of RSM turbulence model over standard and Realizable $k-\varepsilon$ by comparing the inlet-to-outlet pressure drop with measurements. Since the overall pressure drop does not represent the details of the flow and is not a parameter of interest for the simulations, validating a complicated model using pressure drop is unreasonable.
1.3 Organization of this thesis and key contributions

The primary goal of this study is to understand the mechanism of separation in a centrifugal rotor air classifier. This includes (1) how the flow looks like and (2) how the particles are affected by this flow. Based on these, we calculate key features of the classifier such as cut-size. Having found the answers to these primary questions, the secondary goal is to look for a more optimal configuration i.e. to explore the possibility to achieve a finer product using the same operating conditions.

The laboratory-scale classifier used to classify calcium-carbonate is shown in Figure 1.1. During experiments the housing was blocked. Hence the rotor is rotating in something that approximates as a simple cylinder. Table 1.1 includes the dimensions of the rotor along with the operating conditions to be modeled.

Since the rotor located inside the classifier is rotating at a high speed, it is impossible to access its interior with the aim of flow measurement. The alternative is to develop a numerical simulation of the flow using CFD that provides the flow information of the inaccessible regions. Simulations are also suitable for modeling hypothetical geometries and conditions that can be useful in our search for a more optimal operating state. However the uncertainties in a CFD solution might be a source of mistrust. Therefore, a CFD solution requires both verification and validation; the latter stands for solving the equations correctly and the former for solving the correct equations. In this study, verification is achieved by using multiple mesh sizes and calculating Grid Convergence Index (GCI) based on averaged and rms velocity magnitudes. Validation is achieved by comparing the CFD results with our rare experimental measurements.

The numerical model is to predict shifts of cut-size \(d_{50}\) or \(d_{98}\) as a result of new working conditions and geometric specifications of the rotor. Also it should be fast enough to meet the optimization requirements. Based on the simulations and particle tracking data, the mechanism of separation and particle motion will be deduced.

According to the requirements of the simulation, we concentrated on the rotor and the simulation of the whole device is avoided leading to the elimination of unnecessary calculations. The periodic nature of the rotor and the symmetry of

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Blade Length</th>
<th>Blade Tip Radius</th>
<th>Blade Angle</th>
<th>Angular speed</th>
<th>Rotor height</th>
<th>Flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>4.445(cm)</td>
<td>14.03(cm)</td>
<td>29.87°</td>
<td>5100(rpm)</td>
<td>14.86(cm)</td>
<td>11.3 (m/min)</td>
</tr>
</tbody>
</table>

Table 1.1: Rotor dimensions and working conditions
1.3. Organization of this thesis and key contributions

the boundary conditions [10] encouraged us to simulate only one blade. A multi-
blok structured mesh with proper boundary layer adaption is generated. The flow
solution and particle tracking is then accomplished using ANSYS Fluent 16.0.

Having defined the domain and interpreted boundary conditions, single-phase
Reynolds Averaged Navier-Stokes equations are numerically solved for two- and
three-dimensional space. In order to capture the turbulent features of the flow, two
turbulence models are implemented and compared in Chapter 3. Finally particles
are injected at appropriate positions. Discrete random walk model is utilized to
track particles of different size in the domain and efficiency curves are then cal-
culated. This will be discussed in details in Chapter 2. Chapter 4 presents the
gometric optimization of the rotor based on the two-dimensional model explained
in Chapter 2.

General simplifications

In this section simplifications for the numerical computations in this study are ex-
plained. They include geometric simplifications and physical assumptions includ-
ing the material properties and the form of differential equations used.

Geometric simplifications

• The focus is shifted to the rotor. The rest of the classifier, including air inlet
and outlet, recirculation zones, particle introduction and the elaborate accept
side and reject side passages, are ignored

• With the aid of periodic boundary conditions the rotor is reduced to the vicinity of only one blade

Physical assumptions

• Air is modeled as an incompressible fluid

• Air properties at room temperature are used

• A homogeneous temperature is assumed: energy equation is avoided

• A steady solution employed

• Spherical particles assumed

• Local influence of particles on the continuum medium is neglected

• Particle-particle interactions are ignored
1.4 Contributions

This study is aimed at solving a current industry problem. The first objective is to have an insight in the flow within a centrifugal rotor air-classifier. CFD has proved useful in providing a detailed understanding of the flow structure. The behavior of the solid particles in the flow, for instance the region within which the particles are being separated and high particle concentration regions, are then deduced.

Aside from the flow characteristics and particle behavior, useful quantitative results e.g. the efficiency curves and cut-size for various operating conditions and geometries are the benefits of this study. Based on these predictions two new rotors were built that resulted in a finer product.
Chapter 2

Methods and modeling details

2.1 Introduction

The simulation of the air classifier includes mesh generation, flow solution and particle tracking. The rotor is the most important part of the classifier in which all the separation related phenomena occur. A complex flow field, significant centrifugal forces and high swirl velocity are observed in this region of every rotor classifier. Hence the rest of the domain including air intake, outlet pipes and fans are ignored and the focus is shifted only to the rotor. Having defined the domain, all the boundary conditions should be identifiable and interpreted mathematically (section 2.2). The next step is to discretize the domain in order to solve the Reynolds-Averaged Navier-Stokes equations numerically. This will be more discussed in Section 2.3. Section 2.4 explains the necessity of incorporating a turbulence model and compares two different Reynolds-Averaged simulations. The injection of the particles in the domain as a post-processing computational procedure is explained in Section 2.6. Discrete Random Walk (DRW) for particle tracking model with its notable advantages will be introduced as well. In Section 2.7 the solution schemes used to solve discretized RANS equations are explained. An example of mesh independence study is presented in Section 2.8 in which uncertainty is reported using the Grid Convergence Index (GCI) method.

2.2 Domain and appropriate boundary types

2D

The development of a 2D model is elaborated in this section. The stratified structure of the rotor (ring structure) as shown in Figure 2.1-a and also the axi-symmetric flow of inlet air justify the 2D approach to the problem as a start. The accuracy of 2D simulations is verified by experimental results and 3D simulations. In addition they feature low costs and high flexibility in modifying geometry and re-meshing.

The 2D domain is obtained by intersecting a horizontal plane with the rotor as shown in Figure 2.1-b.c. The domain consists of 32 straight blades with a circular inlet and a circular outlet. The inherent repeating property of the blades urged us
2.2. Domain and appropriate boundary types

![Figure 2.1: Rotor Cross Section](image)

To incorporate periodic boundaries that have the repeating characteristic (Figure 2.2), this makes a remarkable reduction in the domain area without compromising the validity of the results. The diameter for inlet and outlet boundaries are set according to the housing and outlet pipe dimensions respectively.

The boundary conditions shown in Figure 2.2 are as follows:

- **Velocity-inlet**: Naturally the upstream region is less affected by the diffusive viscous forces caused by blade rotation. Constant radial velocity inlet is applied to this boundary.

- **Pressure-outlet**: A vacuum pressure induced by a fan far away from the rotor creates a controlled flow. A constant pressure in the outlet is chosen since no information is provided regarding the velocity direction or magnitude.

- **Wall**: The blade is the region of solid-fluid contact. A wall boundary condition stands for zero relative velocity and zero pressure gradient normal to the blade.

- **Periodic**: The reduction in geometry due to its repeating characteristic leads to the appearance of two new boundaries on the sides of the blades. Since we are treating all blades equally the information on these two sides must be mapped into another one.

### 3D

The 3D domain is obtained by cutting a slice of the rotor as shown in Figure 2.3 with the boundary types used. A sample mesh is illustrated in Figure 2.3. The new boundary conditions are as follow. The rest of the boundary conditions are the same as in 2D case:
2.3. Mesh generation strategy

Mass-flow-inlet/outlet A constant mass flow rate is applied.

Fully-developed No information for the accept side outlet is provided; a fully-developed flow is assumed that means the rate of change of each physical variable (pressure, velocity and turbulence related variables) perpendicular to the outlet surface is zero.

2.3 Mesh generation strategy

Mesh generation is one of the most crucial stages of developing a CFD model. Both quantity and quality of a mesh is of high importance. A coarse mesh containing few cells leads to inaccurate results while an extremely fine mesh results in a time-consuming computation. Also a mesh without any regard for the physics of the problem (e.g. boundary layer) should be avoided. The sample meshes shown in Figure 2.2 and 2.4 are quadrilateral (hexahedral for 3D) and multi-block structured meshes that possess the periodic property on the sides i.e. each cell on the side has its own counterpart on the other side. Furthermore high aspect-ratio cells are incorporated in the vicinity of the blade to capture the viscous dominant boundary.
2.3. Mesh generation strategy

Figure 2.3: 3D domain and boundary types

Figure 2.4: 3D mesh
layer region.

### 2.4 Turbulence model

An inherent characteristic of a turbulent flow is the significant variations of velocity either in position or time [2], a phenomenon that is commonly observed in turbomachinery and rotating devices. The occurrence of turbulence is mainly judged based on non-dimensional analysis of a certain problem. For stationary geometries, the Reynolds number would characterize the intensity of the turbulence within the domain and it provides the information regarding the necessity of using a turbulence model. Reynolds number is defined according to Equation 2.1 in which $V_r$ is the radial component of velocity, $L$ is the characteristic length scale chosen as the distance between each two neighbor blade tips and $v$ is the kinematic viscosity of air. A high Reynolds number stands for the dominance of inertia to viscous forces that means a turbulent flow is probable. Based on the experimental observations, turbulence in pipe flow occurs at $Re \gtrsim 5000$ [14]. The typical Reynolds number throughout this study varies from 3500 to 9000, proving the necessity of considering the turbulent behavior of the flow.

$$Re = \frac{V_r L}{v} \quad (2.1)$$

For rotating geometries the existence of centrifugal and Coriolis forces leads to new considerations regarding turbulence. Rossby number defined in Equation 2.2 is the ratio of inertial to Coriolis force. A small Rossby number signifies the dominance of rotational forces and implies a turbulent flow. The typical Rossby number encountered in this study ranges from 0.02 to 0.06 that is small enough to consider the flow as turbulent.

$$Ro = \frac{V_r}{\omega L} \quad (2.2)$$

Two different turbulence models are applied. Both models are inherently invented to solve for the Reynolds Averaged Navier-Stokes (RANS) equations and are summarized Table 2.1. The application of Large Eddy Simulations (LES) turbulence models is avoided due to high computational costs.

K-Omega SST (Shear Stress Transport) has low free-stream sensitivity (where information is lacking in our case) and is applicable from the inner part of the flow to the viscous sub-layer without the need to any extra functions [19]. Morgut et al. [20] have successfully simulated the flow around a marine propeller using this turbulence model and have verified the accuracy of the simulations by comparing to their experimental data. Hence this model is used for our 2D simulations.
2.5. Convergence

When the designated numerical scheme was able to produce a solution that meets a pre-defined condition, “convergence” is reached. The convergence of a numerical solution is usually judged by calculating a parameter of interest at each iteration. This parameter could be the residual of an equation or a physical parameter such as pressure drop or velocity at a single point and the condition for convergence could be defined as the variation of this parameter within a small range in a large number of iterations.

In this study the parameter of interest is the average tangential velocity on the periodic boundary. Convergence is reached when the standard deviation of this velocity becomes small enough.
2.6. Particle injection

Figure 2.5: Averaged Tangential Velocity on periodic boundary for different iterations.

The parameter is equal or less than 0.1% of its average magnitude for a range of 10000 iterations. Figure 2.5-a shows a typical convergence study for a 3D simulation from the 50,000th to 60,000th iteration. The average and standard deviation of the averaged tangential velocity in this range is $48.32\, m/s$ and $0.4172\, m/s$ respectively. Figure 2.5-b shows the average tangential velocity on the periodic boundary in a 2D simulation for the range of a 10000 iterations. The average of $46.62\, m/s$ and standard deviation of $2 \times 10^{-5}\, m/s$ shows a perfect convergence in a typical 2D simulation.

2.6 Particle injection

The Discrete Phase Model (DPM) is used to track particles. In this model the fluid flow affects the particles motion but the presence of particles is assumed to have no impact on the fluid. This Euler-Lagrange approach is useful for a multi-phase flow especially when the second phase is small particles or bubbles (dispersed phase).
2.6. Particle injection

The validity of the dilute particle phase assumption depends on three conditions. Firstly, the volume fraction $\phi$ (Equation 2.4) of the dispersed phase must be small ($\phi < 10\%$ according to the Fluent Theory Guide [15]). Secondly, the momentum carried by the dispersed phase should be much smaller than that carried by the continuous phase. Effectively, this means $(\rho_{\text{particle}}/\rho_{\text{air}}) \phi < 1$, or $\phi < 5 \times 10^{-4}$ for calcium carbonate particles in air. Finally, the separation of particles should be sufficient that particle-particle coagulation does not occur over timescales relevant to the classification process. The threshold volume fraction for this effect is more difficult to estimate. In typical industrial classifiers, the feed has a volume fraction of $\phi \sim 10^{-4}$, suggesting that the dilute particle assumption is plausible but not by a large margin. While the one-way coupling assumption is commonly made by researchers to make the simulations tractable, in particular regions in the classifier this ratio might increase and the assumption may not be valid. The ramifications of this assumption will be discussed in the next chapter.

\[
\phi = \frac{\dot{V}_{\text{dispersed phase}}}{\dot{V}_{\text{continuum phase}}} \quad (2.4)
\]

Ignoring particle-particle interactions, Newton’s Second Law applied to a particle in the rotating frame of reference simplifies to:

\[
m \frac{d\vec{u}_p}{dt} = \vec{F}_c + \vec{F}_{co} + \vec{F}_d \quad (2.5)
\]

where $m$ is the particle mass, $\vec{u}_p$ is the particle velocity vector and $\vec{F}_c$, $\vec{F}_{co}$ and $\vec{F}_d$ are centrifugal, Coriolis and drag force respectively. $\vec{F}_c$ and $\vec{F}_{co}$ are frame-dependent fictitious forces and are known for each particle. The drag force $\vec{F}_d$ is modeled by Equation 2.6 with the drag coefficient, $C_d$, given by Equation 2.7 for spherical particles.

\[
\vec{F}_d = \frac{1}{2} \rho |\vec{u}_p - \vec{u}|^2 c_d \pi \frac{d_p^2}{4} \quad (2.6)
\]

\[
c_d = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \quad (2.7)
\]

where $\rho$ is the density of air, $\vec{u}$ is the local velocity of air, and $d_p$ is the particle diameter. $a_1$, $a_2$ and $a_3$ are functions of relative Reynolds number $Re$ defined as follow:

\[
Re = \frac{d_p |\vec{u}_p - \vec{u}|}{\nu} \quad (2.8)
\]

where $\nu$ is the air kinematic viscosity.
2.7. Solution schemes

The boundary conditions set for the DPM are as follow: the blade is an elastic reflect boundary. The inlet and outlet are escape boundaries that means calculations stop if a particles reaches to them. A particle passing through the periodic boundary appears on the other side of the domain.

**Turbulent dispersion of the particles**

The solution to the Reynolds Averaged Navier-Stokes equation does not include the instantaneous values of velocity. Instead it contains an average velocity \( \langle u \rangle \) defined as follow:

\[
\langle u \rangle = u - u'
\]

Where \( u \) is the instantaneous velocity and \( u' \) is its fluctuation. Using only the average velocities while tracking particles is not sensible; hence Discrete Random Walk model (DRW) that also takes into account the fluctuating components is applied. These components are estimated based on turbulent kinetic energy shown in Equation 2.9.

\[
k = \frac{1}{2} \left( (u')^2 + (v')^2 + (w')^2 \right)
\]

Figure 2.6 compares the difference in the paths of a 4-micron particle using averaged velocities and DRW model. For all of the simulations presented in the following chapters, the Discrete Random Walk model is applied due to its accuracy and comprehensiveness in particle tracking.

2.7 Solution schemes

In this part other considerations regarding the simulations are explained. These topics are focused on the form of RANS equations used and the manner they are discretized.

2.7.1 Transient or steady

In a stationary frame (laboratory frame) the rotation of the blades are obviously time-dependent. However using the blade frame of reference, a steady solution can be produced. In all simulations presented in this study a steady solution with frame rotation is incorporated.
2.7. Solution schemes

Figure 2.6: Particle tracking: (a) using averaged values only and (b) using Discrete Random Walk

2.7.2 Compressible or incompressible

Fluid continuum can undergo significant changes in density $\rho$ both in time and space. The assumption of incompressible flow implies treatment of density as a constant scalar independent of either time or space. In most engineering practices, the Mach number, $Ma$, defined in Equation 2.10, explains the validity of this assumption. For $Ma < 0.2$ or 0.3 it is safe to consider the flow as incompressible [17]. In this study the maximum local Mach numbers encountered are between 0.1 and 0.2.

$$Ma = \frac{V}{C} \quad (2.10)$$

where $V$ is the velocity of the fluid particle and $C$ is the speed of sound in the medium.

2.7.3 Coupling and spatial discretization schemes

In order to couple pressure and velocity equations of the discretized RANS equations, various methods have been proposed. For an iterative steady solution in which the difference between iterations is significant, SIMPLE algorithm suits well and hence used in this study. In Table 2.2 a summary of the spatial discretization
2.8 Mesh independence study and discretization error

<table>
<thead>
<tr>
<th>Gradient</th>
<th>Least squares</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>Second Order</td>
</tr>
<tr>
<td>Momentum</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Specific Dissipation Rate</td>
<td>Second Order Upwind</td>
</tr>
</tbody>
</table>

Table 2.2: Spatial discretization schemes for K-Omega

schemes is presented. In the case of using and RSM model, second order upwind scheme is used for all the transport scalars.

2.8 Mesh independence study and discretization error

The sudden rise in the application of CFD in 1980’s led to concerns regarding the verification of the immense volume of codes being developed daily. Roache introduced the Grid Convergence Index (GCI) with the aim of reporting the grid refinement error for every CFD simulation in a uniform fashion. [18]. For every mesh (or grid) a characteristic mesh size $h$ is defined:

$$h = \left[ \frac{1}{N} \sum_{i=1}^{N} (\Delta A_i) \right]^2 \quad (2.11)$$

where $N$ is the number of cells and $A_i$ is the area (or volume for a 3D mesh) of the $i^{th}$ cell. A target value for investigating grid convergence should be set; Figure 2.7-a shows the velocity magnitude along the bisector of two neighbor blades for three different 2-dimensional meshes. Figure 2.7-b shows the average of that velocity for different number of cells. The $GCI_{fine}$ computed according to the Roache method is 0.48% that is small enough to verify a mesh-independent solution.

For a typical 3D simulation, two important values are studied: the average tangential velocity on the periodic surface and the rms of radial velocity on the vertical plane splitting the blade. Table 2.3 summarizes the corresponding values. The grid convergence index for the tangential and radial velocity is 0.08% and 1.4% respectively, which confirms the mesh-independence of the flow solution.

2.9 Summary

In this chapter a CFD solution aiming at tracking particles of different size within a centrifugal rotor air classifier is introduced. The considerations regarding the choice of the domain are explained that dramatically reduced the area (or volume
2.9. Summary

Figure 2.7: (a) Velocity magnitude profiles along the periodic line and (b) Average of this velocity versus Number of cells (2D study)

<table>
<thead>
<tr>
<th>Number of Cells</th>
<th>Average Tangential Velocity on Periodic Face (m/s)</th>
<th>RMS radial velocity on the vertical plane (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse Mesh</td>
<td>72624</td>
<td>50.641</td>
</tr>
<tr>
<td>Medium Mesh</td>
<td>270165</td>
<td>47.241</td>
</tr>
<tr>
<td>Fine Mesh</td>
<td>459198</td>
<td>47.305</td>
</tr>
<tr>
<td>corresponding</td>
<td></td>
<td>0.08%</td>
</tr>
</tbody>
</table>

Table 2.3: Mesh refinement study (3D study)
for 3D). A multi-block structured mesh with high aspect ratio close to the blade aiming to capture the boundary layer effect is then created. Two different turbulence models used in this study are briefly introduced. Discrete phase model is then utilized to track spherical calcium-carbonate particles with various diameters. To better capture the turbulent nature of the flow and its influence on the paths of particles, Discrete Random Walk model is applied that takes into account the effect of velocity fluctuations obtained by solving the Reynolds Average Navier-Stokes equations. Using Grid Convergence Index method (GCI) the mesh-independent solution for both 2D and 3D simulations is verified.
Chapter 3

Results and discussion

In this chapter the details regarding the flow characteristics and particle tracking for a particular rotor is presented. This chapter is divided into two sections that include 2D and 3D simulations separately. The rotor V_1 simulated in this chapter is shown in Fig 4.1 and the operating conditions used are shown in Table 3.1.

3.1 Two-dimensional simulation

3.1.1 Pressure distribution

A rotating rotor imposes centrifugal forces (radially outward) resisting the inward flow of air. Hence extra pressure drop is required to achieve the target flow rate compared to a stationary rotor. Figure 3.2 shows the contours of constant pressure for rotating and non-rotating cases. The 38\(pa\) pressure drop is negligible compared to 4500\(pa\) pressure requirement for the rotor at 5100\(rpm\) implies the strength of the centrifugal forces.

Comparison with rigid body rotation

The pressure drop in a rigid body rotation of a fluid medium can be calculated using the NSE equations in polar coordinate. Neglecting the trivial terms in this equation following relation will be obtained:

\[
\rho \frac{v_\theta^2}{r} = \frac{dp}{dr} \tag{3.1}
\]

where \(r\) is the distance to the axis of rotation and \(v_\theta\) is the tangential velocity. Equation 3.1 is obtained assuming a non-viscous, incompressible fluid rotating

\begin{center}
\begin{tabular}{|c|c|}
\hline
Rotational Velocity & 5100rpm \\
\hline
Flow rate & 28.3 \(m^3/min\) \\
\hline
\end{tabular}
\end{center}

Table 3.1: Operating Conditions
3.1. Two-dimensional simulation

Figure 3.1: Rotor V_1

Figure 3.2: Contours of constant pressure ($p_a$), (a) with rotation and (b) no rotation
3.1. Two-dimensional simulation

rigidly about a vertical axis. Hence the radial velocity \( v_r \) is zero and \( v_\theta = r \omega \). Substituting and solving for pressure:

\[
p - p_0 = \frac{\rho \omega^2 (r^2 - r_0^2)}{2}
\]

where \( p_0 \) is a known pressure at radius \( r_0 \), \( \rho \) is density and \( \omega \) is rotational velocity. Figure 3.3 compares the pressure profile on the bisector of two neighbor blades and the pressure obtained from Equation 3.2 with the initial reference point (\( r_0 \) and \( p_0 \)) taken from the middle of the blades (shown in a red dot). Only in the regions between the blades is observed a quadratic pressure distribution like Equation 3.2. Upstream of the blades, pressure is almost constant due to the non-rotating flow. Downstream of the blades and close to the blades’ inner radius a high deviation is noticed from rigid body rotation since the tangential velocity profile will not follow the rigid body rotation after losing contact with the blade (\( v_\theta \neq r \omega \)).

3.1.2 Velocity field

Figure 3.4-a shows the radial component of the velocity that creates the inward drag force and pushes particles to the accept stream. The existence of a vortex rotating in the reverse direction relative to the rotor is observed. This vortex creates a radial velocity that is much higher than the average velocity on the inlet. The
3.1. Two-dimensional simulation

Figure 3.4: Velocity contours: (a) radial component and (b) tangential component

radial component of velocity is plotted along inlet, outlet and three arcs crossing the blades (Figure 3.5). \( s_b \) is the non-dimensional distance along the arcs.

The outward centrifugal force on particles originates from the air tangential velocity. The absolute tangential velocity contours are depicted in Figure 3.4-b. Figure 3.6 shows this value on three chord lines drawn from the inlet to the outlet: \( s_c = 0 \) is the inlet, \( s_c = 0.4 \) is the outer radius of the rotor and \( s_c = 0.8 \) is the inner radius of the rotor. For \( s_c > 0.8 \), an increase in velocity towards the center of rotation is observed which is resulted from the conservation of angular momentum.

3.1.3 Efficiency curves and cut-size

The efficiency curve shows the percentage of recovered particles in the reject stream for different particle diameters. In our 2D simulations reject stream is not present. Therefore particles are either accepted by passing through the outlet boundary or moving around in the domain without reaching a boundary. For these “stuck” particles, calculation stops when the time given to each of them exceeds a certain value called “tracking time”. If a particle cannot reach the outlet (become accepted) within this time it will be considered as “rejected”. Figure 3.7 shows the efficiency curves as a function of the non-dimensional tracking time defined as below:

\[
t^* = \frac{\text{tracking time}}{\text{residence time}}
\]  

(3.3)

where residence time is the ratio of the volume of the rotor to the volumetric flow rate that equals 0.032s for the conditions stated in Table 3.1. According to this figure cut-size monotonically increases with \( t^* \).
3.1. Two-dimensional simulation

Figure 3.5: Radial velocity profile on different arcs

Figure 3.6: Tangential velocity profile on different chord lines
3.1. Two-dimensional simulation

Figure 3.7: efficiency curves and their dependence on tracking time

A major contribution of 2D simulations is their capability in predicting shifts of cut-size as a result of geometric modifications. This was done with efficiency curves of various cases compared at the same $t^*$. In Figure 3.8, $d^{50}$ of two different cases are shown: rotor V_1 and an imaginary rotor with a short inner radius (blade length of 7.99 cm), operating at the condition shown in Table 3.1. The difference in cut-size in yellow shows that for $t^* \gtrsim 50$, the cut-size shift is converged to a constant value and becomes independent of $t^*$. In order to be far from the threshold of 50 and at the same time avoid doing unnecessary time-consuming calculations, a $t^*$ of 100 is chosen for all 2D calculations.

3.1.4 Comparison with experiments

Based on the cut-size predictions two new rotors were built. Table 3.2 summarizes the new rotors and cut-size shifts calculated by simulations and measured by experiments. The simulations predict that the fineness of the final product of Rotor V_0 with curved blades is dependent on the direction of rotation. This phenomenon is validated by experiments that measured a $1.4 \mu m$ improvement in cut-size. The effect of the blade length is also simulated. According to simulations a shorter blade, Rotor V_2 (with the constant outer radius) has shown $1.035 \mu m$ decrease in the cut-size. This product is too fine compared to our feed particle distribution and
3.1. Two-dimensional simulation

Figure 3.8: $d^{50}$ for two rotors

is not feasible to produce. Therefore Rotor V_2 with a lower angular velocity is tested that produced almost the same product of Rotor V_1 with a high angular velocity.

3.1.5 Particle concentration distribution

For each particle diameter study, 500 particles are injected close to blade tip ($r = 0.149m$) and their radial position is registered and the history of 5 particles is plotted versus time. The very fine particles are accepted immediately without being pushed out by the strong vortex between the blades (Figure 3.9-a). For the particle range close to the cut-size $d^{98}$, the probability of particles being affected by the vortex increases (Figure 3.9-b). The large particles ($d > 1.3d^{98}$) are concentrated outside the rotor with a slight chance of being pulled in for a short time (Figure 3.9-c).

The stochastic trajectories can be used to calculate the concentration probability density. To do so, the radial location was divided into 0.001m intervals. The time that 500 particles of the same size spend at each of these intervals is registered. This study is repeated for particle sizes between 2 and 7 microns (near
### 3.1. Two-dimensional simulation

<table>
<thead>
<tr>
<th>Rotor Name</th>
<th>Blade</th>
<th>Operating Condition</th>
<th>Cut-size Shift (µm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Measured</td>
</tr>
<tr>
<td>V_0</td>
<td>0.01</td>
<td>4.45</td>
<td>14.08</td>
</tr>
<tr>
<td>V_0</td>
<td>0.01</td>
<td>4.45</td>
<td>14.08</td>
</tr>
<tr>
<td>V_1</td>
<td>0</td>
<td>3.92</td>
<td>14.08</td>
</tr>
<tr>
<td>V_2</td>
<td>0</td>
<td>1.91</td>
<td>14.08</td>
</tr>
</tbody>
</table>

Table 3.2: Model Validation

the cut size of the rotor) with 0.05µm step size. To obtain concentration values, the Particle Source in Cell (PSI-C) method is used to convert Lagrangian particle tracking information into Eulerian [21]:

\[ C_j = \frac{\dot{M} \sum_{i=1}^{n} dt_{i,j}}{V_j} \]  

(3.4)

where \( C \) is the particle concentration, \( \dot{M} \) is feed rate, \( V \) is the volume of cell², \( dt \) is particle residence time, \( i \) is the trajectory index, \( n \) is the number of trajectories and \( j \) denotes the cell index. Based on this relation, non-dimensional concentration contours are calculated:

\[ \frac{c_j}{c_{inlet}} = \frac{\sum dt_j}{\sum dt_{inlet}} \frac{r_{inlet}}{r_j} \]  

(3.5)

where \( c_{inlet} \) and \( t_{inlet} \) denote the feed concentration and time spent in the injection radius respectively.

A non-dimensional representation of the outcome for Rotor V_1 with the operating conditions in Table 3.1 is compactly illustrated in Figure 3.10. Several features of this figure warrant explanation. First, for the finest (2µm) particles the color map is nearly uniform. This implies that the fine particles spend almost an equal amount of time in all parts of the rotor, because they are convected with the air. Secondly, Figure 3.10 shows that particles between 3.0 and 3.9 microns have a high concentration (100 to 350 times greater than feed concentration) just inside the inner radius of the rotor. The unwanted presence of these particles inside the rotor could potentially block finer particles from making it to the accept radius. Third, particles between 4 and 7µm have a very high concentration near the outer radius of the rotor. The elevated concentration of these larger particles would also tend to prevent smaller particles from making it to the accept side of the rotor. We

²Here cells are concentric rings with the uniform thickness of 0.001m
3.1. Two-dimensional simulation

Figure 3.9: Radial Location of five particles for three different diameters
3.1. Two-dimensional simulation

Figure 3.10: Non-dimensional concentration for various particle diameters

believe that the increased concentration of particles at these locations might be the reason for the experimentally measured fish-hook effect. It must be noted that the information Figure 3.10 presents is independent of feed size distribution and treats all particle sizes equally.

Particle-particle interactions are commonly neglected based on the assumption of low concentration. Due to the enhanced concentration values in different regions of the rotor, this assumption is not valid when simulating industrial classifiers with a high particle feed rate. Figure 3.10 shows only the concentration when the feed is infinitesimally dilute; though our simulations could predict the cuts-size shifts accurately, they cannot represent the fish-hook effect.

In order to generalize and verify the behavior observed, relevant parameters are manipulated separately. These parameters include angular velocity, blade angle, volumetric flow rate and finally the turbulence model. Figure 3.11-a shows the relative concentration contours after doubling the air volumetric flow rate. Figure 3.11-b depicts the case in which angular velocity is decreased by 25%; though the cut size is shifted in the alternative operating conditions (as was expected according to Equation 1.2), the high concentration of medium size particles inside the rotor and coarse particles outside the rotor is observed in both of these cases. In Figure 3.11-c a radial blade is incorporated and Figure 3.11-d is based on Reynold Stress model. In these two cases the relative high concentration at blade inner and outer radii is observed as well.
3.1. Two-dimensional simulation

Figure 3.11: Concentration contours for different conditions; all variables held constant except: (a) flow rate doubled, (b) rotor rotational velocity decreased by 25%, (c) blade angle is reduced to zero and (d) RSM turbulence model is used.
3.2 Three-dimensional simulations

2D simulations neglect the presence of axial flow for the accept side (inside the rotor) and the reject side (outside rotor). This leads to the introduction of the tracking time concept in 2D. 3D simulations are developed to circumvent this problem. RSM turbulence model with scalable wall function is used in simulating Rotor V_1 at angular velocity of 5100rpm and volumetric air flow rate of 1000CFM. Due to the lack of information for the reject flow rate, different values are used from 2% to 10% of the accept flow rate. The inlet boundary can possess swirl velocity; in reality swirl inlet is realized by mounting stator blades in the periphery of the rotor. In simulations we apply a uniform tangential velocity on inlet boundary. The ratio of the tangential to radial velocity varies from 0 (radial flow) to 3.7 (for a highly tilted stator blade, 85degree).

3.2.1 Velocity field

Figure 3.12 shows the tangential velocity on the periodic face. In this case, the inlet has no swirl inlet and the reject flow rate is 0, a state that is close to the 2D simulation discussed previously. The high tangential velocity occurs starting from the rotor tip down to the rotor outlet; this is in agreement with 2D predictions. The radial velocity profiles in the middle of every deck are shown in the Figure 3.13 and is compared to the 2D simulation result with the same RSM turbulence model.

3.2.2 Efficiency curves and concentration contours

Efficiency curves are functions of the reject flow rate. Since there are no measurements for this flow rate, we examined different values starting from 2% to 11% of the accept flow rate (28.3 m³/min). When the reject flow rate is not zero, there is no “incomplete” particles i.e. all particles go through either the accept or reject boundary. A high velocity on the reject channel will reduce the probability of the finer particles to migrate to the accept side that results in higher recovery ratio as shown in Figure 3.14 for 11% reject. It seems like that the 2% produces the lowest recovery of fine particles which is favorable. However the concentration enhancements pertaining to 2% is much higher that could be unfavorable in reality. This is illustrated in Figure 3.15 that presents the peak concentration enhancement for various reject flow rates. Figure 3.16 illustrates the concentration enhancement as a function of radial position for two different reject flow rates. The accumulation of particles around the outer radius of the rotor is more intense for the 3.5% reject flow rate.

Another mechanism that helps reduce the enhancement level is the inlet with
3.2. Three-dimensional simulations

Figure 3.12: Tangential Velocity Contours on periodic plane

Figure 3.13: Radial Velocity profiles
3.3 Summary

A straight blade rotor is introduced in this chapter. After solving for the single-phase air flow it was shown that in the area confined between blades, the pressure has a quadratic distribution similar to that of rigid body rotation. It is shown that a huge pressure drop is required to compensate for the resistant force of a rotating rotor. Based on the velocity distribution, a big vortex rotating in the opposite direction of the rotor is observed.

The concept of a non-dimensional tracking time, $t^*$, is introduced for 2D simulations and it was shown that efficiency curves depend on it. For a typically large $t^*$, this dependence is eliminated when calculating shifts of cut-size for two different cases. The shift of cut-size from clockwise to anti-clockwise rotation of the curved blade rotor as well as the shift of cut-size from a long to a short straight...
3.3. Summary

Figure 3.15: Peak Concentration enhancement as a function of reject flow percentage

Figure 3.16: Concentration enhancement contours for 3.5% and 5% reject flow
3.3. Summary

Figure 3.17: Concentration enhancement (a) without and (b) with swirl inlet blade have shown agreement with experiments withing 30% accuracy.

Concentration contours as a function of radius in the rotor were produced by tracking hundreds of particles and transforming their paths’ information from Lagrangian to Eulerian using the Particle Source in Cell (PSI-C) method. In our 3D simulations, the presence of high axial velocity inside the rotor and in the reject channel lead to the completion of all particles being tracked and therefore $t^*$ is not defined in 3D simulations. It was shown the concentration enhancement is a function of reject flow rate and the inlet swirl velocity. The elevated concentration is observed for mid-size particles (close to cut-size $d_{98}$) around the blade inner and outer radii. For coarse particles this high concentration is observed only on the outer radius. The degree of concentration enhancement is dependent on the choice of turbulence model or whether a 3D or 2D approach is adopted. However the elevated concentration regions are observed qualitatively in all cases regardless of the aforementioned factors. The high concentration in particular regions makes particle-particle interaction an important factor that can potentially affect the motion of very fine particles. We speculate these high concentration regions might be responsible for the experimentally measured “fish-hook” effect.
Chapter 4

Optimization procedure

4.1 Introduction

The two-dimensional simulations provide a fast means to predict the cut-size $d_{98}$. The absolute values compared to experiments are not equal and almost a $2\mu m$ difference is observed for each simulation. The inaccuracy in our simulations originates from the following possible explanations:

- The devices measuring operating conditions (especially the volumetric flow rate) are not calibrated
- The secondary flows that are not accounted for in 2D simulations
- The rest of simplifying assumptions that are stated in detail in the previous chapters

The simulations, though lack accuracy, are able to predict the behavior or in other words, they are precise. Based on this precision, the behavior of the classifier could be predicted as a result of change in geometry or operating conditions. Accordingly, we tried to predict the fineness of product after reversing the rotor’s direction of rotation. At the time this issue was posed, the rotor with curved blades was installed. Figure 4.1 shows the sketch of this rotor. Simulations were necessary at this stage as it was costly to do experiments without having a clue about the outcome.

The following figure shows the efficiency curves calculated. The $-1.484\mu m$ shift in cut-size $d_{98}$ predicted by simulation compelled the company to do the experiment with the reversed direction of rotation. The outcome was satisfactory as almost the same shift in cut-size ($-1.4\mu m$) was reported. The rare experimental data necessitate relying on this validation point and start looking for a new rotor configuration. Hence different geometrical parameters are altered to look for a more optimized solution. These parameters include blade angle, rotor scale, blade length and the number of blades.
4.1. Introduction

Figure 4.1: Rotor V_0 with curved blades

Figure 4.2: Curved-blade rotor efficiency curve: clockwise and counterclockwise rotation
4.2 Optimization parameters

4.2.1 Effect of blade angle

An important force that drives the particles is the wall rebound force of the blades. The direction of this force is dependent on the attack angle of the blades. This angle is defined as the deviation of the blade cord-line form a radial line that passes through the center of rotation.

Assuming an elastic rebound, Figure 4.3 shows the path of a particle colliding with three blades with different angles. A conventionally negative angle is shown in Figure 4.3-a. A particle colliding with this blade has a higher chance of reaching the accept boundary. A positive angle is assumed as shown in Figure 4.3-c with a lower chance of particles becoming accepted while a radial blade with zero angle (Figure 4.3-b) is the threshold between these two states.

One might imagine that a larger positive angle would be an optimum blade since it acts as a stronger barrier producing a finer product. It must be noted that an angled blade changes the flow structure and affects other physical variables such as turbulence and peak velocity. For instance by applying a very large positive angle (as an extreme case), the blades will block the air flow that obviously is unfavorable. Hence an optimum angle must exist. Figure 4.4 shows the non-dimensional

Figure 4.3: Blade angle possibilities: (a) scooping blade with negative angle, (b) radial blade with zero angle and (c) non-scooping blade with positive angle
4.2. Optimization parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Curvature</th>
<th>Length</th>
<th>Blade Angle</th>
<th>Blade Count</th>
<th>Tip Radius</th>
<th>Angular speed</th>
<th>Flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>0 (straight)</td>
<td>3.92 cm</td>
<td>Target</td>
<td>32</td>
<td>14.08 cm</td>
<td>5100 rpm</td>
<td>28.3 m/min</td>
</tr>
</tbody>
</table>

Table 4.1: Rotor dimensions and operating conditions

Figure 4.4: non-dimensional cut-size $d_{98}$ versus blade angle

The cut-size $d_{98}$ as a function of blade angle. The operating conditions and the type of blade is included in Table 4.1. The optimum angle in this operating condition is between +10 and +20 degrees. The current rotor has an almost +30 degree attack angle and the difference is small. Figure 4.4 also implies the disadvantage of a negative angle that results in a coarse product. This is compatible with the curved blade direction of rotation explained in the introduction: the primary chord-line angle was −30 while both simulation and experiment proved that an angle of +30 leads to a finer product.

4.2.2 Scale

How does scale affect the cut-size? Is a larger rotor better? In order to answer this question thoroughly, other aspects must be taken into account. The major drawback of a larger rotor is that with a higher moment of inertia, it is harder
4.2. Optimization parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Curvature</th>
<th>Length</th>
<th>Blade Angle</th>
<th>Blade Count</th>
<th>Tip Radius</th>
<th>Tip Velocity</th>
<th>Flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>0</td>
<td>3.92 cm</td>
<td>29.87°</td>
<td>variable</td>
<td>Target</td>
<td>75 m/s</td>
<td>28.3 m/s</td>
</tr>
</tbody>
</table>

Table 4.2: Rotor dimensions and operating conditions

<table>
<thead>
<tr>
<th>case number</th>
<th>Blade Count</th>
<th>Tip Radius (cm)</th>
<th>Cut-size d98 (um)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>32</td>
<td>14.08</td>
<td>3.40</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td>18.10</td>
<td>3.51</td>
</tr>
<tr>
<td>3</td>
<td>48</td>
<td>22.11</td>
<td>3.65</td>
</tr>
<tr>
<td>4</td>
<td>56</td>
<td>26.11</td>
<td>3.84</td>
</tr>
<tr>
<td>5</td>
<td>64</td>
<td>30.12</td>
<td>3.95</td>
</tr>
<tr>
<td>6</td>
<td>72</td>
<td>34.12</td>
<td>4.08</td>
</tr>
</tbody>
</table>

Table 4.3: New conditions

to keep the rotational velocity constant: a rigorous balancing, stronger bearings etc. are required that was not feasible in terms of experimentation. Keeping a constant tip-speed solves this issue. Therefore a constant tip-speed of 75 m/s as a restriction is imposed to the problem of scale. As the rotor grows larger, the blades count increases as well so as to keep a constant distance between the tips of each two successive blades. Table 4.2 summarize the parameters involved in this optimization study.

According to Equation 1.2 (shown below) the cut-size must be independent of scale (keeping tip velocity, \( D \omega \), and flow rate, \( Q \), constant). On the other hand, simulations reveal an increase in cut-size when scaling up the rotor. As shown in Table 4.3, by doubling the size of the rotor, \( d_{98} \) increases almost 0.4 \( \mu \)m. As the simulations suggest an unfavorable outcome, a large scale rotor was not experimented. Therefore there is no experimental evidence to corroborate the scale effect optimization.

\[
d_c = \sqrt{\frac{36 \nu \rho s Q}{\pi \rho h D^2 \omega^2}}
\]

4.2.3 Blade length and number

Shorter or longer blade? The length of blade has a dramatic effect on the flow structure. In this part different lengths are simulated while keeping the outer radius
4.3 Summary

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Curvature</th>
<th>Length</th>
<th>Blade Angle</th>
<th>Blade Count</th>
<th>Tip Radius</th>
<th>Tip Velocity</th>
<th>Flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>0</td>
<td>Target</td>
<td>29.87°</td>
<td>32</td>
<td>14.08cm</td>
<td>75m/s</td>
<td>28.3 m³/min</td>
</tr>
</tbody>
</table>

Table 4.4: Rotor dimensions and operating conditions

...of the blade constant. Table 4.4 summarizes the operating conditions.

As is shown in Figure 4.5 there is a critical length for which the finest cut-size is obtained ($L_c \approx 1.9\, cm$). The flow for a blade length smaller that the optimum point ($L < 1.9\, cm$) is similar to an unattached flow as illustrated in Figure 4.6. Due to the short length of the blades, only a fraction of the air flow gains the angular momentum from blades (red color) while the rest of the flow is less influenced by the rotor (yellow and green contours). This phenomenon dramatically weakens the vital centrifugal force responsible for ejecting coarse particles. Hence a much higher $d_{98}$ is predicted in these conditions.

At the critical length the cut-size $d_{98}$ is too small for the feed distribution. Therefore the tip velocity is reduced from 75$m/s$ to 53$m/s$ to increase the cut-size. Using this new condition the blade length study is redone for different blade counts illustrated in Figure 4.6. Rotor $V_2$ is built based on the length optimization. Experiments suggest a close agreement with simulations when rotor $V_1$ was replaced by rotor $V_2$ (see Table 3.2).

As shown in Figure 4.7 The critical length $L_c$ is a function of blade count. Adding blades will increase the contact area and eases the transfer of angular momentum from blade to the flow. Thus increasing the number of blades will reduce the critical length. $L_c$ is also a function of inlet flow rate. A higher flow rate requires longer blades for efficient transfer of momentum but this dependence is not investigated in this study.

4.3 Summary

Based on the two-dimensional model introduced in Chapter 2, a single variable optimization is performed with the aim of producing a finer product. The variables studied include blade angle, rotor scale, blade length and the number of blades. It was shown that there is a critical value associated with blade angle and blade length. At 5100rpm and 28.3$m³/min$, the optimum angle is between +10 and +20 degrees and the optimum blade length is 1.9$cm$. Scaling the rotor up by keeping a constant blade tip-speed, flow rate and blade spacing results in an increase in the cut-size and is unfavorable.
4.3. Summary

Figure 4.5: Cut-size as a function of blade length

Figure 4.6: Contours of absolute tangential velocity for a typical blade shorter than critical length
4.3. Summary

The critical blade length is dependent on the number of blades in a rotor. Producing an even finer product is possible by using a shorter blade and increasing the number of blades at the same time. By adding blades, the distance between them decreases that compensates for the weak angular momentum transfer of a very short blade.

Figure 4.7: Cut-size as a function of blade length and number for 53m/s blade tip speed, 28.3m³/min flow rate and +30° blade angle
Chapter 5

Thesis conclusions

This study was set out to understand and improve the separation of the fine portion of particles from a mixed-sized powder in rotor air classifiers that produce the input material for many industries such as paint and glass-fiber. Computational Fluid Dynamics was employed in this study that provided the detailed quantitative analysis of the flow and the motion of small solid particles. This study is aimed to answer the following issues:

1. The flow structure in a centrifugal rotor classifier
2. The motion of small calcium-carbonate particles
3. Investigating the possibility of a finer product by modifying blade geometry
4. The high particle concentration regions and the effect of reject flow rate

Before using CFD, we conceived the flow to be close to that of a solid body rotation with well-behaved velocity field and that blades are the perfect means of producing the swirl required to induce the centrifugal force on small solid particles. However it turned out, as is shown in detail in Chapter 3, that blades are not ideal for swirl and lead to highly non-smooth radial and tangential velocity profiles. The non-uniform velocity field is a significant source of inefficiency that makes simple-minded mathematical analyses pointless. An instance of such relations is shown in Equation 1.2 that relates cut-size to flow rate, angular velocity of the rotor and rotor diameter. According to the simulations, solid particles ranging from very fine to slightly larger than cut-size penetrate the rotor and oscillate between the inner and outer regions of the blades. Based on this behavior we conclude that “separation” does not occur at the tip of rotor blades, but happens throughout the space contained by the rotor. Hence for a mathematical analysis of a rotor classifier, care must be taken in choosing the effective separation radius; as is shown in Chapter 4, the inner blade radius plays a major role in the fineness of the output product.

The simultaneous introduction of millions of particles in a real classifier, as opposed to our simulations that consider particles as isolated elements, is a considerable discrepancy. It was shown in Chapter 3 that the particle concentration level may dramatically rise, depending on the type of the classifier and the conditions
5.1 Future work

in which it works. The elevated concentration regions occur particularly in the vicinity of the blade tip as large particles struggle to jump out of this region. This renders the dilute assumption (the foundation of all simulations in this study) as a crude guess. The elevated regions not only alter the flow structure but also can potentially inhibit the motion of very fine particles passing through the rotor that may or may not explain the commonly observed fish-hook effect. Two mechanisms are proposed to alleviate this issue: pre-swirl inlet flow and reject flow. The former can be achieved by placing angled static blades about the periphery of the rotor and the latter stands for a forced reject flow close to the blade outer radius.

The shape of the rotor significantly affects the fineness of final product. By keeping the air flow rate and rotor angular velocity constant, a finer cut-size is achieved by shortening the blades to a critical length while keeping the tip diameter constant. The critical blade length makes things more complicated: a blade shorter than the critical length results in flow separation and incomplete angular momentum transfer from blade to fluid medium. In order to circumvent this problem and still use a shorter blade length, the number of blades can be increased that mitigates the momentum transfer and decreases the critical length opening the possibility for even a finer cut-size.

5.1 Future work

The next step is to develop a complete geometry of the classifier. This could be done using a sliding mesh technique to transfer information from the stationary housing flow to the rotating mesh of the rotor. Solving for a transient solution and also getting rid of the problematic boundaries close to the rotor area are among the benefits of a complete geometry. The downside is the high number of elements that must be incorporated in the mesh.

A numerical model that is capable of predicting the interaction of particles is extremely beneficial. It provides information regarding the optimum feedstock distribution and also can predict the quantitative efficiency for fine particles. However this type of simulation requires greater resources as simulations should be performed in transient 3D that includes the whole domain (avoiding periodicity) and also requires an extremely fine mesh to resolve the length scale of the particles depending on the requirement of a particular model applied.

An interesting experiment would be to introduce pre-classified material as feed to the classifier. The efficiency curves of the accept will explain uncertainties we are currently facing; the adverse effect of mid-size particles stuck in the flow will be verified if the efficiency of fine particles are close to 100% i.e. no fine particles are rejected. On the other hand if fine particles are still being rejected like a nor-
5.1. Future work

Figure 5.1: Gama densitometer measures the density of particles outside the rotor

...mal experiment, attributing fish-hook effect to particle-particle interactions will be falsified.

Measurements Using a Gamma densitometer can clarify the intensity of particle concentration on the periphery of the rotor [22]. Figure 5.1 shows a typical gamma-ray passing through the classifier and the vicinity of the rotor. Calcium carbonate particles will absorb this gamma-ray and the intensity of this absorption is proportional to the concentration of particles that is measured by a detector located on the other side of the classifier.
Bibliography


