Performance Recovery of Scale Reduced Cyclone Particle Separators using a Rotating Classifier

Mark J. Parker, The University of Western Ontario

Supervisor: Straatman, Anthony G., The University of Western Ontario
Co-Supervisor: Savory, Eric, The University of Western Ontario

A thesis submitted in partial fulfillment of the requirements for the Doctor of Philosophy degree in Mechanical and Materials Engineering
© Mark J. Parker 2023

Follow this and additional works at: https://ir.lib.uwo.ca/etd

Part of the Other Mechanical Engineering Commons

Recommended Citation
https://ir.lib.uwo.ca/etd/9859

This Dissertation/Thesis is brought to you for free and open access by Scholarship@Western. It has been accepted for inclusion in Electronic Thesis and Dissertation Repository by an authorized administrator of Scholarship@Western. For more information, please contact wlswadmin@uwo.ca.
Abstract

Cyclone particle separators are often used to collect particles that are of value or environmental concern. These devices achieve high Total Collection Efficiency (TCE) with steady pressure drop ($\Delta P$). Increasing the height of the cyclone improves both TCE and $\Delta P$. However, due to the height of the cyclone, these devices are often forced to be placed outside, which can lead to decreased performance over time. The research examines the outcome of scale reducing a cyclone. This is achieved by (i) how the scale reduction changes the flow field, (ii) identifying what flow aspects contribute to the reduction in TCE and (iii) how the TCE can be recovered using a rotating classifier. The scale reduction of the cyclone is performed in two stages: first, the barrel length is reduced in height by a factor of two (Intermediate), then the cone height is reduced by a factor of two (Truncated). Thus, the Truncated cyclone is half the overall height as the initial Full-size cyclone. Experimental and Computational Fluid Dynamics (CFD) with Discrete Phase Modeling (DPM) methods were used, with a newly developed injection method (A-STERP), which has the accuracy of a fully random transient injection and reduces the computational time from over six months to an hour. The experiments served as a validation tool for the flow fields predicted by CFD and for tuning the drag model of a non-spherical particle used in the DPM.

Examination of the flow fields showed that shortcutting in the Truncated cyclone increased from 20% to 36% and, as such, was considered a contributor to the 20% decrease in the TCE. Adding a classifier that rotates at 4000 rpm increased the TCE by 3% in the Truncated cyclone but reduced it by an additional 20% when rotating at 6000 rpm. The drop in TCE at high rotational speeds resulted from particle re-entrainment from the Collection Hopper (CH). Increasing the diameter of the CH or adding a vortex stabilizer will eliminate re-entrainment. Thus, a Truncated cyclone with a vortex stabilizer and a classifier rotating at 6000 rpm can achieve a similar TCE as a Full-size cyclone.
Keywords

Fluid flow experiments, Computational fluid dynamics, Discrete phase modeling, Particle injection, Cyclone particle separators, Dynamic Cyclone particle separators, Cyclone collection hoppers, Shortcutting flow, Particle re-entrainment
Summary for Lay Audience

A cyclone particle separator is a device used to filter particles out of air. The cyclone archives this filtration using a tornado-like flow to throw the particles out of the air. The particles then fall to the bottom of the device and are stored in the collection hopper of the cyclone. For a cyclone to be able to collect most of the particles, it needs to be tall. This project focused on how a cyclone of half the height could achieve a similar ability to collect particles as a cyclone of typical height. This is accomplished by using experiments and simulations to examine what about the flow results in the high collection and changes with a decrease in height.

A new method that allowed for the accurate injection of simulated particles into the computer-modelled flow was developed, which resulted in the time to simulate the particles taking several months to only an hour. This new method was validated against more comprehensive methods on three different particle filtering devices, with the third device being a cyclone particle separator. Using the new method of injecting particles, it was found that reducing the height of a cyclone by half significantly decreased the short cyclone’s ability to remove particles from the airflow. The reduction in the cyclone height increased the amount of flow that prematurely exited the short cyclone, and this was identified as a significant contributor to the decrease in the ability to collect particles. Rotating blades were added inside the cyclone to prevent the flow from prematurely leaving the cyclone. The rotation of the blades forced the air that was prematurely leaving away from the exit and created a strong tornado-like flow in the cyclone. The flow, however, becomes so strong that it pulls particles out of the collection hopper and further decreases the short cyclone’s ability to collect particles. Lastly, the collection hopper was altered by increasing its size or adding a blockage to ensure particles that enter cannot escape, thus allowing the short cyclone with rotating blades to collect most of the particles.
Co-Authorship Statement

Chapter 2: Written by Mark J. Parker, edited by Dr. Eric Savory and Dr. Anthony G. Straatman


Chapter 3: Written by Mark J. Parker, edited by Dr. Eric Savory and Dr. Anthony G. Straatman

Chapter 4: Written by Mark J. Parker, edited by Dr. Eric Savory and Dr. Anthony G. Straatman
Acknowledgments

First, I want to thank my supervising professors, Dr. Anthony G. Straatman and Dr. Eric Savory, for their support, guidance and understanding over the last four years. I also want to thank the contributions of Mike Taylor from TML Tech. Ltd. And Mathew Hengst from Design Built Mechanical Inc. for the help with coordinating the experimental setup and conducting the experiments, as well as the rest of the team at Design Built Mechanical Inc. for fabricating the cyclones and equipment used in the experiments.

I want to express a special thank you to my mom and dad, Sherry and Shawn Parker, who saw and nurtured my potential. I would not have made it this far without their continued support and love. Lastly, I want to thank my loving wife, Rebecca Parker, who is always there to make me smile and laugh even at the most stressful of times and who has proven to be there for me time and time again.
# Table of Contents

Abstract .......................................................................................................................... ii

Summary for Lay Audience .......................................................................................... iv

Co-Authorship Statement ............................................................................................. v

Acknowledgments ......................................................................................................... vi

Table of Contents ......................................................................................................... vii

List of Tables ................................................................................................................ xi

List of Figures ................................................................................................................ xiii

List of Abbreviations, Symbols, and Nomenclature .................................................... xx

Chapter 1 ....................................................................................................................... 1

1 Introduction ............................................................................................................... 1

1.1 Background ........................................................................................................... 1

1.2 Literature Review ................................................................................................. 4

1.2.1 Cyclone ............................................................................................................ 5

1.2.2 The Flow Structure of a Cyclone ..................................................................... 11

1.2.3 Predicting Particle Motion in Numerical Domains .......................................... 31

1.2.4 Literature Review Summary ............................................................................ 33

1.3 Objectives of the Present Work ........................................................................... 34

1.4 Methods and Scope of Research ......................................................................... 35

1.5 Thesis Outline ....................................................................................................... 36

1.6 References ............................................................................................................. 37

Chapter 2 ....................................................................................................................... 47

2 Statistically accurate discrete phase modelling of particle cloud generation using Aggregate Steady Random Particle injection ......................................................... 47

2.1 Introduction ......................................................................................................... 47

2.2 Discrete Phase Modeling ...................................................................................... 50
4.6 Conclusion .................................................................................................................. 155

4.7 References ................................................................................................................ 157

Chapter 5 ......................................................................................................................... 160

5 Summary, Contributions and Suggestions for Future Research ......................... 160

5.1 Summary and Contributions .................................................................................. 160

5.2 Suggestions for Future Research ........................................................................... 164

  5.2.1 Classifier Blade Design ..................................................................................... 164

  5.2.2 The Particle Location Distribution .................................................................... 164

  5.2.3 Development of Dynamic Cyclone Design Guidelines .................................. 165

5.3 Summary .................................................................................................................. 165

5.4 References ............................................................................................................... 165

Curriculum Vitae ............................................................................................................. 167
List of Tables

Table 1.1: Geometric parameter values for Stairmand Cyclone design. ............................... 6

Table 1.2: Non-dimensional flow parameters for experimental datasets. ............................... 20

Table 1.3: Non-dimensional flow parameters for experimental datasets. ............................... 22

Table 1.4: Non-dimensional flow parameters for CFD results. ............................................. 26

Table 2.1: Grid independence results for the Generic Chamber, the Baffled Pre-Separator and the Cyclone shown in Fig. 2.5. ......................................................................................................................... 65

Table 2.2: Summary of the approximate times required for conducting DPM simulations using randomized transient injection, surface injection and A-STERP. Included in the table are the summary results of TCE percent difference between and the percent compared to randomized transient injection. ........................................................................................................... 82

Table 3.1: Summary of dimensions for the three cyclones constructed and tested in the experimental study. ................................................................................................................................................................. 92

Table 3.2: Summary of instrumentation used and their reported accuracies. ......................... 97

Table 3.3: Summary of experimentally measured total collection efficiency, pressure drop and pressure coefficient in the Full size, Intermediate and Truncated cyclones. ................. 100

Table 3.4: Summary of numerical predicted pressure drop and pressure coefficient........... 108

Table 3.5: Summary of numerical predicted Total Collection Efficiency (TCE). ................. 114

Table 3.6: Summary of percent difference of the numerical results compared to the experimentally measured values. ...................................................................................................................... 117

Table 4.1: Summary of dimensions for the three cyclones constructed and tested in the experimental study. ................................................................................................................................. 127

Table 4.2: Summary of collection hoppers tested in the CFD study. .................................... 130
Table 4.3: Summary of instrumentation used and their reported accuracies. .................. 131

Table 4.4: Summary of experimentally measured cyclone performance in the Truncated cyclone at various classifier rotational speeds. ................................................................. 134

Table 4.5: Summary of experimentally measured cyclone performance in the Full size, Intermediate and Truncated cyclones with a classifier rotating at 4000 rpm. .................... 135

Table 4.6: Summary of numerical predicted pressure drop and pressure coefficient and classifier fluid power for the different cyclone and collection hopper (CH) configurations at two rotational speeds. ........................................................................................................... 139

Table 4.7: Summary of the numerically predicted peak tangential velocity. .................. 142

Table 4.8: Summary of the numerically predicted shortcutting flow. ............................ 146

Table 4.9: Summary of the numerically predicted peak tangential velocity along a radial line inside the collection hoppers. .............................................................................. 146

Table 4.10: Summary of the numerically predicted total collection efficiency. ............ 148

Table 4.11: Summary of the percent difference between the numerically predicted and experimentally measured total collection efficiency, pressure drop, pressure coefficient. .. 149

Table 4.12: Summary of numerical modeling and experimental (shown in parentheses) results obtained in Chapter 3 for standard cyclones with a collection hopper. ............... 151
List of Figures

Figure 1.1: Schematic of a porous media filter ......................................................... 2

Figure 1.2: Schematic of an electrostatic precipitator device ..................................... 3

Figure 1.3: Schematic of an inertial separating device (baffle type) ............................ 4

Figure 1.4: Schematic of a cyclone describing the various aspects of the device and showing key dimensions ................................................................. 5

Figure 1.5: Multiple cyclone systems, parallel cyclones shown(right) and series cyclones (left) ................................................................. 9

Figure 1.6: Schematic of dynamic cyclone ................................................................. 11

Figure 1.7: Sensitivity analysis of the Burgers-Rott tangential velocity, $U\theta$ (Eq. 1.5). Sensitivity to the empirical constant ‘K’ shown in figures (A), and the sensitivity to the location of $R\theta max$ shown in figures (B). With figures (i) showing the predicted $U\theta$ across $r$ normalized by $R = D/2$, and figures (ii) showing the $U\theta$ normalized by $U\theta max$ and $r$ by $R\theta max$ ................................................................................. 15

Figure 1.8: Sensitivity analysis of the Burgers-Rott axial velocity, $Uz$ (Eq. 1.5). Sensitivity to the empirical constants ‘$W1$’, ‘$W2$’, and the location of $R\theta max$ shown in figures (A), (B), and (C) respectively .......................................................................................................................... 16

Figure 1.9: Comparison of simple analytical models for $U\theta$. Rankine vortex model based on Eq. 1.4, and Burgers-Rott model based on Eq. 1.5 ................................................................. 18

Figure 1.10: Comparison of experimentally measured mean velocity profiles (normalized by $U\delta in$ and $R$) for different cyclone geometries and operating parameters. Velocity profiles are for an axial position in the cylinder section of the cyclone and below the inlet. Tangential velocity (A), Axial velocity (B) ..................................................................................................... 20
Figure 1.11: Comparison of experimentally measured mean tangential velocity profile (normalized by $U\theta_{max}$ and $R\theta_{max}$) for different cyclone geometries and operating parameters and analytical solutions for $U\theta$. ................................................................. 23

Figure 1.12: Comparison of velocity profiles (normalized by $U_{in}$ and $R$) for various turbulence closure models used. Tangential velocity $U\theta$ profiles are shown in figures (i) and axial velocity $Uz$ profiles shown in figures (ii). Hu et al. [82] data shown in (A), Hoekstra et al. [77] data shown in (B), and Liu et al. [79] data shown in (C). .................................................. 25

Figure 1.13: Comparison of numerically predicted mean velocity profile using an RSM turbulence closure model (normalized by $U_{in}$ and $R$) for different cyclone geometries and operating parameters. Velocity profiles are for an axial position in the cylinder section of the cyclone and below the inlet. Tangential velocity (A), axial velocity (B) ......................... 26

Figure 1.14: Comparison of numerically predicted mean tangential velocity profile (normalized by $U\theta_{max}$ and $R\theta_{max}$) for different cyclone geometries and operating parameters and analytical solutions for $U\theta$. ................................................................. 27

Figure 1.15: Time history of the fluctuating tangential velocity normalized by the nominal inlet velocity with time normalized by a macro time scale, showing how the radial position affects the magnitude of the fluctuations. Hoekstra et al. [77] data shown in (A), and Liu et al. [79] data shown in (B). ................................................................. 28

Figure 1.16: Histograms of the fluctuating tangential velocity normalized by the inlet velocity at the geometric axis of rotation. Solero and Coghe [80] data shown in (A), Hoekstra et al. [77] data shown in (B), and Liu et al. [79] data shown in (C). .................................................. 29

Figure 1.17: Schematic of a cyclone depicting shortcutting, blue line shows a typical bulk flow path and red line shows a shortcutting flow path. ................................................................. 31

Figure 1.18. A Rosin-Rammler particle size distribution ................................................................. 32

Figure 2.1: Illustration of a typical rectangular inlet surface with an edge-refined computational mesh. The points shown are for the injection locations using the surface injection approach, which collocates the injection locations with the mesh cell centres. ...... 53
Figure 2.2: Cumulative size distribution of the powder considered in this study. The cumulative fractions are given based on particle number $Y_d$ and on particle mass $M_d$. 55

Figure 2.3: Plot and table showing the distributed and cumulative size distribution computed using a random number generator. 55

Figure 2.4: Three distribution patterns generated using a MATLAB script that prescribes a random surface location and a random particle size within a specified range and preserving the specified size distribution. The left and middle patterns each have 2000 randomly assigned injection points of different random injection positions and particle size, while the right pattern has 5000 random injection points. 57

Figure 2.5: Flow chart detailing the A-STERP injection process. 59

Figure 2.6: Geometric models of the generic rectangular chamber (a), the baffled pre-separator (b), and cyclone (c). All dimensions are in mm. Red areas represent area/position of interest for grid independence criteria. In all cases, the Cartesian coordinate system has $x$ pointing into the inlet, with the $y$-$z$ plane oriented on the inlet face. 61

Figure 2.7: Illustration of the complex airflow in the generic chamber using flow path lines with color indicating velocity magnitude. 68

Figure 2.8: Total collection efficiency (TCE) for the generic chamber shown as a function of the number of randomly located injection points, which also include random-sized particles within the specified range and distribution. 69

Figure 2.9: Plot showing the specific total collection efficiency (TCE) and the cumulative average (aggregate) TCE as a function of the number of distribution patterns as predicted using the A-STERP approach to injection modelling. Each distribution pattern has 5000 randomly assigned injection locations with random-sized particles within the specified range and distribution of particles. 70

Figure 2.10: Predicted fractional collection efficiency (FCE) obtained for different methods of particle injection into the chamber. A-STERP results are shown for 9 frames with 3500 injection points. 71
Figure 2.11: Illustration of the complex flow in the baffled pre-separator using flow path lines with color indicating velocity magnitude. ................................................................. 72

Figure 2.12: Total collection efficiency (TCE) for the baffled pre-separator shown as a function of the number of randomly located injection points, which also include random-sized particles within the specified range and distribution. ................................................................. 73

Figure 2.13: Predicted fractional collection efficiency (FCE) obtained for different methods of injecting particles into the baffled pre-separator. A-STERP results are shown for 15 frames each with 3500 injection points. ................................................................. 74

Figure 2.14: Illustration of the airflow in the cyclone using flow path lines with color indicating velocity magnitude. .................................................................................. 75

Figure 2.15: Total collection efficiency (TCE) for the cyclone shown for A-STERP as a function of the number of randomly located injection points and random particle size distribution, which also include random-sized particles within the specified range and distribution. .................................................................................. 76

Figure 2.16: Predicted fractional collection efficiency (FCE) obtained for different methods of injecting particles into the cyclone. A-STERP results for 25 frames with 4000 injection points. .................................................................................. 77

Figure 2.17: Inlet and monitoring point velocity with time for the time-periodic flow in the baffled pre-separator. .................................................................................. 78

Figure 3.1: Cyclone schematic showing components ................................................. 88

Figure 3.2: Three cyclone geometries, Full-size (left), Intermediate (centre), Truncated (right), with the horizontal line showing the radial line used for the velocity profiles and the shortcutting ribbon. .................................................................................. 93

Figure 3.3: Photos of Full-size (right) and Truncated (left) cyclones used in the physical experiments. .................................................................................. 95

Figure 3.4: Physical experimental setup schematic with bounding box of CFD domain...... 97
Figure 3.5: Particle size distribution measured from the experiments and used in the numerical simulations. ................................................................. 98

Figure 3.6: Fractional collection efficiency (FCE) for Portland cement. ......................... 99

Figure 3.7: Effect of shape factor $\Phi$ on the predicted total collection efficiency (TCE) for the Truncated cyclone with a collection hopper. ................................................................. 107

Figure 3.8: Normalized tangential velocity profile along the radial line shown in Fig. 3.2 for the three cyclone geometries with and without a collection hopper. ......................... 109

Figure 3.9: Rankine vortex tangential velocity profiles along the radial line shown in Fig. 3.2 for the three cyclone geometries with and without a collection hopper. ......................... 110

Figure 3.10: Normalized axial velocity profiles (positive indicates flow towards the outlet and negative indicates flow towards the collection hopper) along the radial line shown in Fig. 3.2 for the three cyclone geometries with and without a collection hopper. ......................... 110

Figure 3.11: Tangential projection of velocity vectors for the three cyclone geometries with and without a collection hopper. Full-size (left), Intermediate (middle) and Truncated (right), with a collection hopper (top), and without a collection hopper (bottom). ......................... 112

Figure 3.12: Predicted fractional collection efficiencies (FCE) of the three cyclone geometries with and without a collection hopper. ................................................................. 114

Figure 4.1: Dynamic cyclone schematic showing components. ........................................ 125

Figure 4.2: Three cyclone geometries, Full-size (left), Intermediate (centre) and Truncated (right), with the horizontal line showing the radial line used for the velocity profiles and the shortcutting ribbon. Geometry of classifier shown in top right. ................................. 128

Figure 4.3: Five collection hopper geometries, Collection Hopper 1 (top left), Collection Hopper 2 (top centre), Collection Hopper 3 (top right), Collection Hopper 4 (bottom left), Collection Hopper 5 (bottom right), with the horizontal line showing the radial line used for the velocity profiles inside collection hopper. ................................................................. 129
Figure 4.4: Physical experimental setup schematic with a bounding box representing the CFD domain.

Figure 4.5: Particle size distribution measured from the experiments and used in the numerical simulations.

Figure 4.6: Portland cement Total Collection Efficiency (TCE) for the Truncated dynamic cyclone at various rotational rates (\( \omega \)), where \( \omega = 0 \) represents a standard cyclone.

Figure 4.7: Portland cement fractional collection efficiency (FCE) for the three dynamic cyclone geometries with the classifier rotating at 4000 rpm.

Figure 4.8: Normalized tangential velocity profile along the radial line shown in Fig. 4.2 for the three cyclone geometries in the standard and dynamic configurations (\( \omega = 4000 \) rpm).

Figure 4.9: Rankine vortex tangential velocity profiles along the radial line shown in Fig. 4.2 for the three cyclone geometries in standard and dynamic configurations (\( \omega = 4000 \) rpm).

Figure 4.10: Normalized axial velocity profiles (positive indicates flow towards the outlet, and negative indicates flow towards the collection hopper) along the radial line shown in Fig. 4.2 for the three cyclone geometries in the standard and dynamic configurations (\( \omega = 4000 \) rpm).

Figure 4.11: Normalized tangential velocity profile along the radial line shown in Fig. 4.2 for the five collection hopper geometries on the dynamic Truncated cyclone, for (A) \( \omega = 4000 \) rpm and (B) \( \omega = 6000 \) rpm.

Figure 4.12: Rankine vortex tangential velocity profiles along the radial line shown in Fig. 4.2 for the five collection hopper geometries on the dynamic Truncated cyclone, for (A) \( \omega = 4000 \) rpm and (B) \( \omega = 6000 \) rpm.

Figure 4.13: Normalized axial velocity profiles (positive indicates flow towards the outlet, and negative indicates flow towards the collection hopper) along the radial line shown in Fig.
4.2 for the five collection hopper geometries on the dynamic Truncated cyclone, for (A) \( \omega = 4000 \) rpm and (B) \( \omega = 6000 \) rpm. .............................. 145

Figure 4.14: Normalized tangential velocity profile along the radial line shown in Fig. 4.3 for the five collection hopper geometries on the dynamic Truncated cyclone, for (A) \( \omega = 4000 \) rpm and (B) \( \omega = 6000 \) rpm. ................................................................. 147

Figure 4.15: Predicted fractional collection efficiencies of the three cyclone geometries in standard and dynamic configuration (\( \omega = 4000 \) rpm). ................................................................. 149

Figure 4.16: Predicted fractional collection efficiencies for the five collection hopper geometries on the dynamic Truncated cyclone, for (A) \( \omega = 4000 \) rpm and (B) \( \omega = 6000 \) rpm. ................................................................. 150

Figure 5.1: Summary of numerically predicted total collection efficiencies for the various cyclones and collection hoppers studied in Chapters 4 and 5. ................................................................. 162
### List of Abbreviations, Symbols, and Nomenclature

#### Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A-STERP</td>
<td>Aggregate steady random particle</td>
</tr>
<tr>
<td>CH1</td>
<td>Benchmark collection hopper</td>
</tr>
<tr>
<td>CH5</td>
<td>Benchmark collection hopper with cone vortex stabilizer</td>
</tr>
<tr>
<td>CH4</td>
<td>Benchmark collection hopper with flat vortex stabilizer</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational fluid dynamics</td>
</tr>
<tr>
<td>DBM</td>
<td>Design build Mechanical Inc.</td>
</tr>
<tr>
<td>DPM</td>
<td>Discrete phase modeling</td>
</tr>
<tr>
<td>CH2</td>
<td>Double diameter of benchmark collection hopper</td>
</tr>
<tr>
<td>LES</td>
<td>Large eddy simulation</td>
</tr>
<tr>
<td>LDA</td>
<td>Laser doppler anemometry</td>
</tr>
<tr>
<td>LDV</td>
<td>Laser doppler velocimetry</td>
</tr>
<tr>
<td>NRMS</td>
<td>Normalized root mean square</td>
</tr>
<tr>
<td>PIV</td>
<td>Particle image velocimetry</td>
</tr>
<tr>
<td>PVC</td>
<td>Precession vortex core</td>
</tr>
<tr>
<td>CH3</td>
<td>Quadruple diameter of benchmark collection hopper</td>
</tr>
<tr>
<td>RNG</td>
<td>Renormalization group</td>
</tr>
<tr>
<td>RSM</td>
<td>Reynolds stress model</td>
</tr>
<tr>
<td>URANS</td>
<td>Unsteady Reynolds-averaged Navier-Stokes</td>
</tr>
<tr>
<td>VS</td>
<td>Vortex stabilizer</td>
</tr>
</tbody>
</table>
### Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S$</td>
<td>Actual surface area of particle</td>
<td>m$^2$</td>
</tr>
<tr>
<td>$Re_{c2}$</td>
<td>Alternative cyclone Reynolds number</td>
<td>-</td>
</tr>
<tr>
<td>$\theta_c$</td>
<td>Angle of cone</td>
<td>m</td>
</tr>
<tr>
<td>$G_{\theta}$</td>
<td>Angular momentum</td>
<td>kgm$^2$s$^{-1}$</td>
</tr>
<tr>
<td>$z$</td>
<td>Axial coordinate</td>
<td>m</td>
</tr>
<tr>
<td>$G_z$</td>
<td>Axial momentum</td>
<td>kgms$^{-1}$</td>
</tr>
<tr>
<td>$U_z$</td>
<td>Axial velocity</td>
<td>ms$^{-1}$</td>
</tr>
<tr>
<td>$D$</td>
<td>Barrel diameter</td>
<td>m</td>
</tr>
<tr>
<td>$H_b$</td>
<td>Barrel height</td>
<td>m</td>
</tr>
<tr>
<td>$R$</td>
<td>Barrel radius</td>
<td>m</td>
</tr>
<tr>
<td>$K$</td>
<td>Burgers-Rott Vortex empirical constant</td>
<td>m$^2$s$^{-1}$</td>
</tr>
<tr>
<td>$W_1$</td>
<td>Burgers-Rott Vortex empirical constant</td>
<td>ms$^{-1}$</td>
</tr>
<tr>
<td>$W_2$</td>
<td>Burgers-Rott Vortex empirical constant</td>
<td>ms$^{-1}$</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Classifier rotational speed</td>
<td>rads$^{-1}$</td>
</tr>
<tr>
<td>$b_1$</td>
<td>Coefficient for particle drag coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$b_2$</td>
<td>Coefficient for particle drag coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$b_3$</td>
<td>Coefficient for particle drag coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$b_4$</td>
<td>Coefficient for particle drag coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$m_c$</td>
<td>Collected mass of particles</td>
<td>kg</td>
</tr>
<tr>
<td>$D_h$</td>
<td>Collection hopper major diameter</td>
<td>m</td>
</tr>
<tr>
<td>$H_H$</td>
<td>Collection hopper major height</td>
<td>m</td>
</tr>
<tr>
<td>$D_h$</td>
<td>Collection hopper minor diameter</td>
<td>m</td>
</tr>
<tr>
<td>$H_h$</td>
<td>Collection hopper minor height</td>
<td>m</td>
</tr>
<tr>
<td>$H_c$</td>
<td>Cone height</td>
<td>m</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>------------------------------------------------------------------------------</td>
<td>--------</td>
</tr>
<tr>
<td>$D_d$</td>
<td>Cone minimum diameter</td>
<td>m</td>
</tr>
<tr>
<td>$C_{grid1}$</td>
<td>Criterion of interest for coarse mesh level</td>
<td>-</td>
</tr>
<tr>
<td>$C_{grid2}$</td>
<td>Criterion of interest for fine mesh level</td>
<td>-</td>
</tr>
<tr>
<td>$b$</td>
<td>Cyclone inlet duct width</td>
<td>m</td>
</tr>
<tr>
<td>$a$</td>
<td>Cyclone inlet duct height</td>
<td>m</td>
</tr>
<tr>
<td>$L_i$</td>
<td>Cyclone inlet duct length</td>
<td>m</td>
</tr>
<tr>
<td>$Re_{c1}$</td>
<td>cyclone Reynolds number</td>
<td>-</td>
</tr>
<tr>
<td>$d$</td>
<td>Diameter of particle</td>
<td>m</td>
</tr>
<tr>
<td>$C_{exp}$</td>
<td>Experimentally measured value for a given parameter</td>
<td>-</td>
</tr>
<tr>
<td>$\Delta t_{fp}$</td>
<td>Flow timestep</td>
<td>s</td>
</tr>
<tr>
<td>$u_\theta$</td>
<td>Fluctuating tangential velocity</td>
<td>ms$^{-1}$</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Fluid density</td>
<td>kgm$^{-3}$</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Fluid viscosity</td>
<td>kgm$^{-1}$s$^{-1}$</td>
</tr>
<tr>
<td>$FCE_{act}$</td>
<td>Fractional collection efficiency at given particle diameter</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>yielded by from the randomized transient simulations</td>
<td></td>
</tr>
<tr>
<td>$FCE_{per}$</td>
<td>Fractional collection efficiency at given particle diameter</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>yielded by from the surface or A-STERP injection simulations</td>
<td></td>
</tr>
<tr>
<td>$FCE$</td>
<td>Fractional Collection Efficiency</td>
<td>-</td>
</tr>
<tr>
<td>$\vec{g}$</td>
<td>Gravitational acceleration</td>
<td>ms$^{-2}$</td>
</tr>
<tr>
<td>$\vec{F}_g$</td>
<td>Gravitational force</td>
<td>N</td>
</tr>
<tr>
<td>$GI$</td>
<td>Grid index</td>
<td>-</td>
</tr>
<tr>
<td>$H_{cl}$</td>
<td>Height of classifier</td>
<td>m</td>
</tr>
<tr>
<td>$\theta_b$</td>
<td>Inclination angle</td>
<td>rad</td>
</tr>
<tr>
<td>$U_{in}$</td>
<td>Inlet velocity</td>
<td>ms$^{-1}$</td>
</tr>
<tr>
<td>$\vec{u}_\theta$</td>
<td>Instantaneous tangential velocity</td>
<td>ms$^{-1}$</td>
</tr>
<tr>
<td>$T$</td>
<td>Macro timescale</td>
<td>s</td>
</tr>
</tbody>
</table>

xxii
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D_{CL}$</td>
<td>Major diameter of classifier</td>
<td>m</td>
</tr>
<tr>
<td>$m_{total}$</td>
<td>Mass flow of injected particles (CH2)</td>
<td>kgs$^{-1}$</td>
</tr>
<tr>
<td>$m_p$</td>
<td>Mass flow rate of given particle size</td>
<td>kgs$^{-1}$</td>
</tr>
<tr>
<td>$M_{d_1}$</td>
<td>Mass fraction of particles</td>
<td>-</td>
</tr>
<tr>
<td>$U_{\theta_{max}}$</td>
<td>Max tangential velocity</td>
<td>ms$^{-1}$</td>
</tr>
<tr>
<td>$U_{z_{max}}$</td>
<td>Maximum axial velocity</td>
<td>ms$^{-1}$</td>
</tr>
<tr>
<td>$\bar{d}$</td>
<td>Mean particle diameter</td>
<td>m</td>
</tr>
<tr>
<td>$U_{z_{min}}$</td>
<td>Minimum axial velocity</td>
<td>ms$^{-1}$</td>
</tr>
<tr>
<td>$D_{cl}$</td>
<td>Minor diameter of classifier</td>
<td>m</td>
</tr>
<tr>
<td>$FCE_{N_{RMS}}$</td>
<td>Normalized root mean square of fractional collection efficiency curves</td>
<td>-</td>
</tr>
<tr>
<td>$n_p$</td>
<td>Number of particles in a parcel</td>
<td>-</td>
</tr>
<tr>
<td>$Y_{d_1}$</td>
<td>Number fraction of particle</td>
<td>-</td>
</tr>
<tr>
<td>$N$</td>
<td>Number of blades in classifier</td>
<td>-</td>
</tr>
<tr>
<td>$N_s$</td>
<td>Number of samples</td>
<td>-</td>
</tr>
<tr>
<td>$C_{num}$</td>
<td>Numerically predicted value for the given parameter</td>
<td>-</td>
</tr>
<tr>
<td>$U_{out}$</td>
<td>Outlet Velocity</td>
<td>ms$^{-1}$</td>
</tr>
<tr>
<td>$m_{pc}$</td>
<td>Parcel mass</td>
<td>kg</td>
</tr>
<tr>
<td>$\bar{\omega}_p$</td>
<td>Particle angular velocity</td>
<td>rads$^{-1}$</td>
</tr>
<tr>
<td>$x_p$</td>
<td>Particle coordinate</td>
<td>m</td>
</tr>
<tr>
<td>$y_p$</td>
<td>Particle coordinate</td>
<td>m</td>
</tr>
<tr>
<td>$z_p$</td>
<td>Particle coordinate</td>
<td>m</td>
</tr>
<tr>
<td>$\rho_p$</td>
<td>Particle density</td>
<td>kgm$^{-3}$</td>
</tr>
<tr>
<td>$d_l$</td>
<td>Particle diameter bin size</td>
<td>m</td>
</tr>
<tr>
<td>$C_d$</td>
<td>Particle drag coefficient</td>
<td>-</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>------------------------------------</td>
<td>-------</td>
</tr>
<tr>
<td>$\vec{F}_d$</td>
<td>Particle drag force</td>
<td>N</td>
</tr>
<tr>
<td>$m_p$</td>
<td>Particle mass</td>
<td>kg</td>
</tr>
<tr>
<td>$I_p$</td>
<td>Particle momentum of inertia</td>
<td>kgm²</td>
</tr>
<tr>
<td>$\tau_r$</td>
<td>Particle relaxation time</td>
<td>s</td>
</tr>
<tr>
<td>$C_\omega$</td>
<td>Particle rotational drag coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$T_p$</td>
<td>Particle temperature</td>
<td>K</td>
</tr>
<tr>
<td>$\zeta$</td>
<td>Particle spin</td>
<td>-</td>
</tr>
<tr>
<td>$\Delta t_p$</td>
<td>Particle timestep</td>
<td>s</td>
</tr>
<tr>
<td>$V_p$</td>
<td>Particle velocity</td>
<td>ms⁻¹</td>
</tr>
<tr>
<td>$u_p$</td>
<td>Particle velocity component</td>
<td>ms⁻¹</td>
</tr>
<tr>
<td>$v_p$</td>
<td>Particle velocity component</td>
<td>ms⁻¹</td>
</tr>
<tr>
<td>$w_p$</td>
<td>Particle velocity component</td>
<td>ms⁻¹</td>
</tr>
<tr>
<td>$PD$</td>
<td>Percent difference</td>
<td>-</td>
</tr>
<tr>
<td>$\Delta c_p$</td>
<td>Pressure coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$\Delta P$</td>
<td>Pressure drop</td>
<td>Pa</td>
</tr>
<tr>
<td>$A_p$</td>
<td>Projected particle surface area</td>
<td>m²</td>
</tr>
<tr>
<td>$r$</td>
<td>Radial coordinate</td>
<td>m</td>
</tr>
<tr>
<td>$R_{\theta max}$</td>
<td>Radial coordinate of U_\theta max</td>
<td>m</td>
</tr>
<tr>
<td>$U_r$</td>
<td>Radial velocity</td>
<td>ms⁻¹</td>
</tr>
<tr>
<td>$Re_\omega$</td>
<td>Relative angular particle Reynolds number</td>
<td>-</td>
</tr>
<tr>
<td>$\vec{\nu}$</td>
<td>Relative fluid-particle velocity</td>
<td>ms⁻¹</td>
</tr>
<tr>
<td>$\vec{\Omega}$</td>
<td>Relative particle-fluid angular velocity</td>
<td>rads⁻¹</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
<td>-</td>
</tr>
<tr>
<td>$RMSD$</td>
<td>Root Mean Square Difference</td>
<td>-</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>------------------------------------------------------------------</td>
<td>------------------</td>
</tr>
<tr>
<td>$C_{RL}$</td>
<td>Rotational lift coefficient</td>
<td></td>
</tr>
<tr>
<td>$\vec{F}_{RL}$</td>
<td>Rotational lift force</td>
<td>N</td>
</tr>
<tr>
<td>$\Phi$</td>
<td>Shape factor</td>
<td></td>
</tr>
<tr>
<td>$n$</td>
<td>Spread parameter</td>
<td></td>
</tr>
<tr>
<td>$s$</td>
<td>Surface area of a sphere having the same volume as the particle</td>
<td>m²</td>
</tr>
<tr>
<td>$S$</td>
<td>Swirl number</td>
<td></td>
</tr>
<tr>
<td>$\theta$</td>
<td>Tangential coordinate</td>
<td>rad</td>
</tr>
<tr>
<td>$U_\theta$</td>
<td>Tangential velocity</td>
<td>ms⁻¹</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
<td>s</td>
</tr>
<tr>
<td>$\vec{T}$</td>
<td>Torque applied to particle</td>
<td>Nm⁻¹</td>
</tr>
<tr>
<td>$M$</td>
<td>Total collected mass</td>
<td>kg</td>
</tr>
<tr>
<td>$TCE_{act}$</td>
<td>Total collection efficiency yielded by from the randomized transient simulations</td>
<td></td>
</tr>
<tr>
<td>$TCE$</td>
<td>Total Collection Efficiency</td>
<td></td>
</tr>
<tr>
<td>$y$</td>
<td>Value for the domain with a collection hopper</td>
<td></td>
</tr>
<tr>
<td>$Y$</td>
<td>Value for the domain without a collection hopper</td>
<td></td>
</tr>
<tr>
<td>$Q$</td>
<td>Volume flow rate of continuous phase</td>
<td>m³s⁻¹</td>
</tr>
<tr>
<td>$V_{INJ}$</td>
<td>Volume of injected particles of given size</td>
<td>m³</td>
</tr>
<tr>
<td>$S$</td>
<td>Vortex finder height</td>
<td>m</td>
</tr>
<tr>
<td>$D_x$</td>
<td>Vortex finder inside diameter</td>
<td>m</td>
</tr>
<tr>
<td>$D_v$</td>
<td>Vortex finder outside diameter</td>
<td>m</td>
</tr>
</tbody>
</table>
Chapter 1

1 Introduction

1.1 Background

Transportation of a discrete phase in a continuous phase is a common occurrence in industry. The discrete phase can be a solid particular matter or a liquid droplet when the continuous phase is gas, or a solid particular matter when the continuous phase is liquid. Applications of a discrete phase being transported by a continuous phase include, but are not limited to: cyclones (pneumatic [1-11] and hydraulic [12-14]), settling/storage chambers [15, 16], waste water treatment equipment [17, 18], water spray from fire safety nozzles [19], and respiratory systems [20]. To limit the scope, a solid discrete phase carried in a gas continuous stream such as pneumatically transported particulate matter will be addressed in this work. Pneumatic transport is a widely utilized industrial process for transferring products or playing a role within processes like fluidized bed reactors. The material undergoing pneumatic transport can either be of value, as is the case for petroleum refining, pharmaceutical production, powder coating, mineral processing, and cement [21-26], or a potential pollutant, as in cases where the particles are filtered from the airstream to avoid their release into the atmosphere [26-29]. Irrespective of the intended purpose of the material, pneumatically transported powder requires separation from the carrying gas stream. Common mechanisms to achieve the separation include porous media filtration, electromagnetic separation, and inertia separation [30].

A primary approach for removing particulate matter transported pneumatically involves the utilization of porous media (schematic shown in Fig. 1.1), such as fabric (bag house filter), fiber, or granular (granular bed filter) materials [30-33]. Porous media filters operate by creating a physical barrier that prevents the pneumatically transported powder from passing through, while allowing the gas stream to flow through the pores [30, 33, 34]. The particulate matter that cannot pass through the porous media becomes trapped on the upstream surface of the material forming a cake-like deposit [30, 34]. The effectiveness of
a porous media filter depends on the size of the pores; smaller pores improve particle collection performance but also increase the pressure drop across the filter [30, 32, 34]. By adjusting the size of the pores, it is possible to design porous media filters that capture specific particle sizes without causing unnecessary additional pressure drop [27, 34, 35]. In porous media particle separators, particles become lodged within the pores, gradually reducing the passage size over time [31, 34, 35]. This reduction in passage size leads to a growing pressure drop with continued use of the device [30, 31, 34, 35]. To dislodge the particulate matter from the surface of the porous media and restore its functionality, the operation needs to be paused, and a strong reverse flow is applied [27, 30].

Another approach for removing pneumatically transported particulate matter involves the application of electrostatic forces, as is the case for the electrostatic precipitator device (shown in Fig. 1.2) [30, 36, 37]. In this method, the particulate matter in the pneumatically transported stream is exposed to an electric field, causing the discrete phase particles to acquire an electric charge [30, 37-39]. These charged particles are subsequently carried by the continuous phase and directed towards electrode plates, which exert an attractive force on the particulate matter [30, 37-39]. This leads to the accumulation and aggregation of the particles onto the plates, forming a cake-like deposit [30, 37, 39]. As the cake builds up it weakens the attraction between the particulate matter and the plates [30, 39]. At regular intervals, this deposit is dislodged from the plates using a process termed "rapping." The dislodged cake descends and collects in a hopper [30, 37-40]. The frequency and intensity of the rapping process are critical factors, as improper execution can lead to cake breakage.

Figure 1.1: Schematic of a porous media filter.
and the reintroduction of particulate matter into the air stream. An advantage of utilizing electrostatic forces for particulate removal from gas flows is the incredible collection efficiency, which can reach as high as 99.9% for submicron particles [30, 41, 42]. Additionally, this method is characterized by a low pressure drop. However, a drawback is the potential risk of particulate re-entrainment during the rapping process [30, 37, 40].

The final method under consideration for removing particulate matter from a gas stream relies on the utilization of the particles' inertia. In this approach, pneumatically transported particulate matter is directed into an inertia separation apparatus, which could take the form of a cyclone particle separator or a baffle-type separator (shown in Fig. 1.3) [11, 43-49]. Within the apparatus, the gaseous stream is forced by the geometry of the device to make a sharp change in direction [43, 44, 46, 50]. Particles having significant inertia struggle to alter their direction and consequently deviate from the bulk flow [44, 46-48]. Cyclones have an additional advantage over baffle-type separators, in that the enclosed vortex created by the cyclone imparts centrifugal force on the particles to assist in removing the particulate matter from the bulk flow [11, 24, 27, 28, 30, 45, 46, 48, 49, 51-57]. Once the particulate matter is separated from the bulk flow, it is allowed to settle into a designated collection area [23, 28, 56, 57]. Inertia separators have the benefit of maintaining a consistent pressure drop and collection performance [27, 31, 34, 35]. This advantage stems
from the fact that they do not trap the particulate matter onto a separating/filtering surface, thereby preventing the formation of a cake-like deposit. An added advantage of not trapping the particulate matter within the device is the reduction of material wastage, which is beneficial when the particulate matter holds value.

The cyclone particle separator was selected for research for its advantages over the other filtering devices and its ability to be integrated into pneumatic transport systems. The remainder of this chapter will focus on synthesizing the current literature on cyclone particle separators, followed by the objectives and outline of the thesis.

1.2 Literature Review

The synthesis of the current literature starts with a detailed look at how a cyclone operates and achieves particle separation both as a stand-alone unit and in a system. The literature review will then turn to methods of analyzing the vortex flow responsible for particle separation inside a cyclone. The flow field is directly responsible for separating the inertial separation present in a cyclone. Thus, understanding the flow structure and how to analyze the flow is vital to the research presented in this thesis. Lastly, the literature synthesis will address how particles are modelled in a numerical simulation.
1.2.1 Cyclone

Gas cyclone particle separators (a schematic is depicted in Fig. 1.4) have a simple design where, aside from the tangential inlet, the entire geometry maintains axisymmetry. The primary components of the cyclone include the inlet, a cylinder on top of a conical body, a connected collection hopper at the cone’s base, and a vortex finder inside the upper portion of the cylinder, serving as the outlet for the flow [24, 28, 56]. Contaminated air enters the cyclone, with a velocity $U_{in}$ and mass flow rate $\dot{m}$, through the tangential inlet, and due to the axisymmetric configuration, the airflow acquires a prominent tangential velocity ($U_{\theta}$), thus generating a strong vortex [1-3]. As the airflow swirls, it simultaneously descends along the cyclone walls towards the collection bin and ascends towards the vortex finder or outlet near the central axis [23, 28, 56, 57]. This results in an axial velocity component.

![Figure 1.4: Schematic of a cyclone describing the various aspects of the device and showing key dimensions.](image)

Figure 1.4: Schematic of a cyclone describing the various aspects of the device and showing key dimensions.
Additionally, the flow must progress radially to transition from the outer downward trajectory to the central upward trajectory, leading to a radial velocity component \((U_r)\) [54, 58, 59]. The prominent tangential velocity of the vortex flow results in a centrifugal force on the particles comprising the contaminating powder [11, 24, 27, 28, 30, 45, 46, 48, 49, 51-57]. The centrifugal forces move the particles radially away from the neutral axis and towards the cyclone walls. The particles near the wall travel with the downward flow towards the collection hopper and are finally removed from the bulk flow, with the aid of gravity, and contained in a collection hopper [24, 27, 28, 30].

Stairmand [60] established a high efficiency cyclone design in 1951, and the design is commonly used in both industry and as a benchmark in research literature [28, 56, 57, 61-63]. A Stairmand cyclone geometry is established by selecting the value of the diameter \((D)\) of the cyclone and the various ratios of the other dimensions are summarized in Table 1.1. The \(D\) of a Stairmand cyclone is determined by the required volumetric flow rate \((Q)\) of air through the device and by setting an inlet velocity \((U_{in})\). By equating \(Q\) to \(U_{in}\) multiplied by the inlet area \((A = ab)\), and substituting in the ratios for the inlet height \((a = 0.5D)\) and width \((b = 0.2D)\) and rearranging for \(D\) yields Eq. 1.1.

\[
D = \sqrt{\frac{10Q}{U_{in}}}
\]  
(1.1)

### 1.2.1.1 Cyclone Device Performance

The performance of a cyclone can be determined based on the pressure drop \((\Delta P)\) across the device, the Total Collection Efficiency (TCE) (defined by Eq. 1.2) and the Fractional Collection Efficiency (FCE) (defined by Eq. 1.3) [1, 2, 49, 57, 63, 64].

\[
TCE = \frac{M_c}{M}
\]  
(1.2)

<table>
<thead>
<tr>
<th>(H_b/D)</th>
<th>(H_c/D)</th>
<th>(D_d/D)</th>
<th>(D_e/D)</th>
<th>(S/D)</th>
<th>(a/D)</th>
<th>(b/D)</th>
<th>(L_i/D)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5</td>
<td>2.5</td>
<td>0.375</td>
<td>0.50</td>
<td>0.50</td>
<td>0.50</td>
<td>0.20</td>
<td>1.00</td>
</tr>
</tbody>
</table>
\[ FCE(d) = \frac{m_c(d)}{m(d)} \]  

(1.3)

Where, for a given time period, the total collected mass is \( M_c \) and the collected mass of particles having a diameter \( d \) are \( m_c(d) \). Similarly, the total mass injected is \( M \) and the mass of injected particles with a diameter of \( d \) are \( m(d) \). The \( \Delta P \) across the device is representative of the fluid power \( (= Q\Delta P) \) required to operate the device, whereas the TCE and FCE are related to the collection of particles.

The performance analysis of cyclone particle separators has three main categories: geometry parameters operating conditions, and the material properties of the particulate matter. Geometric parameters, commonly investigated for their impact on cyclone performance, include cyclone length and inlet dimensions. Extending the cyclone length has been shown to positively influence both collection efficiency and \( \Delta P \) across the cyclone [23]. Brar et al. [57] investigated the effect of both conical (\( H_c \)) and barrel (\( H_b \)) section lengths, revealing that increasing \( H_b \) can increase TCE by up to 9.72% and reduce \( \Delta P \) by as much as 37.5% compared to a Stairmand cyclone design. Increasing \( H_c \) also raised TCE by up to 10.8% and reduced \( \Delta P \) by as much as 31.6% [57]. The study demonstrated that increasing \( H_b \) has a more prominent effect on TCE, whereas increasing \( H_c \) has a greater impact on \( \Delta P \). However, excessively lengthy cyclones are not ideal due to increased cost and potential reductions in collection efficiency [56, 65]. Also, there may be space limitations to consider. Research by Hoffman et al. [61] and Hoffman et al. [65] indicated that a decrease in collection efficiency exists for extended cyclones and is the result of the vortex ending on the conical section wall rather than above the collection hopper. The inlet pipe/duct height \( (a) \), width \( (b) \), and angle also influence cyclone performance [55, 63, 64]. Reducing \( a \) or \( b \) generates a vortex with higher \( U_\theta \) (compared to the Stairmand cyclone design), yielding increased collection efficiency [63]. However, this modification also raises \( \Delta P \) across the cyclone [63]. Angling the inlet pipe/duct downward has demonstrated increased collection efficiency across a range of inlet velocities while simultaneously reducing \( \Delta P \) [55, 64].

Operating conditions that influence cyclone performance include the inlet velocity \( (U_{in}) \) of
the continuous phase and the rate of particulate matter entering the cyclone. Increasing $U_{in}$ leads to a stronger vortex, subsequently enhancing the centrifugal force and collection efficiency [2, 11, 46, 56, 57, 64]. However, a powerful vortex can result in particle collisions of greater intensity, potentially causing particle breakage and leading to a revised particle size distribution with a greater number of small particles [46, 66]. Consequently, excessively high $U_{in}$ values can negatively impact TCE [67]. Increased $U_{in}$ values also contribute to increased $\Delta P$ across the cyclone particle separator [2, 11, 46, 56, 57, 64]. The rate of particulate matter entering the cyclone is often expressed as a solid loading rate, representing the mass flow rate of solid particulate matter ($\dot{m}_{total}$) relative to the volume flow rate of the continuous phase ($Q$) entering the cyclone (measured in kg/m³) [56]. Increasing the solid loading rate has been shown to enhance collection efficiency [9, 11, 56, 68]. Achieving this involves increasing $\dot{m}_p$ or decreasing $Q$. Reducing $Q$ without altering cyclone geometry would necessitate a decrease in $U_{in}$, which is not preferred as described above. Thus, increasing $\dot{m}_p$ leads to the improved collection efficiency linked to higher solid loading rates [9, 11, 56, 68]. This improvement can be attributed to an increased frequency of particle-particle collisions [11, 56, 68]. Such collisions can result in particle agglomeration, altering the particle size distribution in favour of larger particles [11, 56, 68].

Given that the cyclone particle separator operates on the basis of inertia separation, its ability to collect particulate matter is influenced by particle mass ($m_p$) and size, which directly affect the inertia and drag forces on the particles [1, 2, 4, 5, 7, 9-11, 13, 15-18, 69]. Larger or denser particles possess increased inertia, increasing the likelihood of separation from the bulk flow [26, 46, 48, 56]. Fractional collection efficiency (FCE) serves as a valuable metric for evaluating the cyclone's ability to collect particles of varying sizes. However, since particulate matter typically comprises particles spanning a range of sizes, the size distribution of the particulate matter also influences device performance [26, 46, 48, 56, 66]. The particulate matter's particle size distribution can influence the TCE, whereas the FCE is decoupled from the particle size distribution.
1.2.1.2 Cyclone System Performance

An alternative approach to enhancing the collection performance of a cyclone particle separator device is to optimize the collection system as a whole. This can involve utilizing multiple cyclones instead of a single cyclone (shown in Fig. 1.5) [10, 24, 46, 70, 71], or integrating alternative separating devices downstream [27, 30, 34-37, 41, 42]. Additionally, a newer strategy involves incorporating a rotary classifier to transform a standard cyclone into a dynamic cyclone [1, 2, 26].

Although using a single cyclone is often more economically feasible due to space constraints and other factors, employing multiple cyclones can enhance the overall collection efficiency of the cyclone system [10, 24, 46, 70-71]. Multiple cyclones can be arranged in various configurations: serial [10, 70], parallel [71, 72], or a combination of both [46]. In serial configurations, the outlet of one cyclone feeds directly into the next,

Figure 1.5: Multiple cyclone systems, parallel cyclones shown(right) and series cyclones (left).
resulting in cleaner air entering sequential cyclones. However, sequential cyclones experience reduced solid loading rates, leading to lower probabilities of particulate matter agglomeration and consequently reduced collection efficiency [70, 72]. Moreover, if the first cyclone lacks sufficient centrifugal forces and rotations to remove fine particulate matter, expecting significant capture of escaped particulate matter in sequential cyclones may not be realistic [10]. Therefore, achieving substantial improvement in collection efficiency using a serial multi-cyclone system compared to a single cyclone would require individual cyclones with different geometric designs [10].

On the other hand, a parallel multi-cyclone configuration splits the contaminated air stream before it enters the cyclones, ensuring that each cyclone receives a similar solid loading rate and mass flow rate [71, 72]. Running multiple cyclones in parallel has demonstrated improved overall collection efficiency compared to a single cyclone, provided that the mass flow rate and pressure drop of each individual cyclone remain identical to those of the other cyclones in the system [71, 72]. However, any operational discrepancies among cyclones, such as a blockage in the collection bin of a single cyclone, can lead to an uneven distribution and compromise the collection efficiency of a parallel multi-cyclone system [72].

In cases where a cyclone particle separator setup fails to achieve the required or desired collection performance, an additional particle filtering device, such as a porous media filter, can be introduced downstream of the cyclone device [27, 30, 34-37, 41, 42]. These secondary devices are designed to capture very fine particulate matter and can quickly form cake-like deposits, leading to increased pressure drop [30, 32, 34, 37, 39]. Integrating the cyclone upstream of the secondary filtering device, instead of using the secondary filter independently, is beneficial because the cyclone can remove the bulk of the material, thus reducing the rate at which cake-like deposits form.

An innovative method to enhance cyclone collection efficiency involves incorporating a rotary bladed classifier, effectively converting a standard cyclone into a dynamic cyclone [1, 2, 26]. This externally powered classifier is positioned concentrically with the cyclone body and is placed inside or below the vortex finder (as depicted in Fig. 1.6) [1, 2]. Aspects
of the classifier design, such as rotational speed ($\omega$), number of blades ($N$), and blade inclination angle ($\theta_b$) have been studied [1, 2, 26]. Experiments conducted with a small-scale lab setup of a dynamic cyclone reveal that running the classifier at a frequency faster than the natural swirl frequency significantly improves collection efficiency compared to a standard cyclone with matching geometric parameters [1, 2, 26]. Negative inclination angles and the use of fewer blades have also been shown to enhance collection efficiency [1]. The improvement in collection efficiency is attributed to the rotary classifier increasing the tangential velocity throughout the cyclone, thereby elevating the centrifugal forces on the particulate matter [23, 26]. The classifier enhances the likelihood of particle collisions and agglomeration due to increased turbulence created near the classifier, ultimately raising particle inertia [1, 68].

1.2.2 The Flow Structure of a Cyclone

The swirling flow enclosed within the cyclone can be separated into two distinct regions: the outer and the inner. It's important to note that the precise location of this division is not fixed, and a phenomenon known as the precession vortex core (PVC) is evident [53, 73]. The swirling motion in the outer region is represented as a free vortex, while the inner region a forced vortex [56, 73]. These two forms of vortex represent the extreme cases, being zero and infinite viscosity for the free and forced vortexes, respectively [56].

Figure 1.6: Schematic of dynamic cyclone.
However, when dealing with a gas medium, particularly air, the viscosity falls between neither zero nor infinity. Consequently, the cyclone demonstrates a quasi-free vortex in the outer region and a quasi-forced vortex in the inner region [58, 74].

The flow structure created by the tangential inlet and contained by cyclone body has been studied analytically, experimentally, and numerically. Analytical models suggested for the vortex flow structure present in a cyclone particle separator are the Rankine vortex [57, 63, 75] and the Burgers-Rott vortex [76, 77]. Experimental measuring techniques employed for cyclone particle separator studies include a five-channel pressure probe [78], Particle Image Velocimetry (PIV) [79], and Laser Doppler Anemometry (LDA), also known as Laser Doppler Velocimetry (LDV) [77, 80–82]. Numerical investigations of the cyclone particle separator allow for detailed analysis of the internal flow but requires selection of proper turbulence closure models such as the k-ε model, Reynolds Stress Model (RSM) or the use of Large Eddy Simulation (LES) [48, 51, 67, 75, 83, 84]. The intricate and challenging nature of the swirling flow within cyclones emphasizes the difficulty of experimental measurement, numerical prediction, and analytical models.

1.2.2.1 Analytical Flow

When a swirling flow displays characteristics of both forced and free vortices, it can be termed a Rankine Vortex. A Rankine vortex possesses distinct attributes: it only has a tangential ($U_\theta$) velocity component, the velocity only relies on the radial coordinate ($r$), and a radial position ($R_{\theta \text{max}}$) exists where $U_\theta$ reaches its maximum value ($U_{\theta \text{max}}$) [85]. Additionally, the velocity has a linear increase with $r$ for smaller radii and an inverse decrease with $r$ at larger radii [85]. The velocity equation set for the Rankine vortex, as delineated by Giaiotti and Stel [85], and resembling the formulation by Batterson et al. [86], is as follows:

$$
\vec{U} = \begin{cases} 
U_r = 0 \\
U_\theta = \begin{cases} 
U_{\theta \text{max}} \frac{r}{R_{\theta \text{max}}} & \text{if } 0 \leq r < R_{\theta \text{max}} \\
U_{\theta \text{max}} \frac{R_{\theta \text{max}}}{r} & \text{if } R_{\theta \text{max}} \leq r \leq R 
\end{cases} \\
U_z = 0
\end{cases}
$$

(1.4)
The Rankine vortex is essentially a fusion of the free and forced vortex phenomena, thus adhering to the assumption of either zero or infinite viscosity. While alternative models (such as the Burgers-Rott vortex) strive to replicate real-world physics more accurately, they inherently involve a greater level of complexity [86, 87].

The Burgers-Rott vortex stands as an exact solution within the Navier-Stokes framework, encompassing the axial velocity component and formulated for a flow driven by suction [86, 87]. The velocity equation set characterizing the Burgers-Rott vortex is expressed as follows:

$$\vec{U} = \begin{cases} 
U_r &= 0 \\
U_\theta &= \frac{K}{2\pi r} \left[ 1 - \exp \left( -\frac{r^2}{R_{\theta max}^2} \right) \right] \\
U_z &= W_1 + W_2 \exp \left( -\frac{r^2}{R_{\theta max}^2} \right) 
\end{cases} $$

(1.5)

with $K$, $W_1$, and $W_2$ being empirical constants.

The Rankine vortex stands as a straightforward analytical model tailored for flows that exhibit radial symmetry around an axis and possess a single significant velocity component, denoted as $U_\theta$ [85]. This singular velocity component requirement is distinctly emphasized by the analytical equation of the Rankine vortex model previously presented (Eq. 1.4). Another implication of the Rankine vortex model is its assumption of an infinitely distant far-field, leading to $U_\theta$ diminishing to zero only at an infinite distance from the axis of rotation [86]. Consequently, it overlooks the no-slip boundary condition relevant to situations where the vortex is confined, as is the case in cyclone particle separators.

An inherent limitation of the Rankine vortex model (apart from its reliance on infinite or zero viscosity assumptions) lies in its dependency on the knowledge of the flow itself [86], particularly the flow variables $U_{\theta max}$ and $R_{\theta max}$. This reliance renders the analytical model inadequate for standalone prediction of the flow field, preventing its use in flow prediction.
As previously explained, the single velocity component \( U_\theta \) in the Rankine vortex model is not the case for cyclones, where all three velocity components are present. However, the dominant velocity component remains \( U_\theta \), which is approximately three times greater than \( U_{in} \) for cyclones [73, 74]. In contrast, the radial velocity component \( U_r \) typically is an order of magnitude less than \( U_{in} \) [73], rendering it negligible in comparison to \( U_\theta \). Moreover, the maximum axial velocity component \( U_z \) tends to be of similar scale to \( U_{in} \) [73, 74], thus exerting minimal influence when compared with \( U_\theta \).

In comparison to the Rankine vortex model, the Burgers-Rott analytical model introduces greater complexity by incorporating \( U_z \) and serving as a solution to the Navier-Stokes equations [86]. This increased complexity requires the inclusion of empirical constants to solve for \( U_\theta \) and \( U_z \), as evident from Eq. 1.5. Beyond the empirical constants, the Burgers-Rott model also demands knowledge of the position \( R_{\theta max} \) (a requirement similar to the Rankine vortex model). The empirical constant \( K \), utilized in the computation of \( U_\theta \), was estimated by Long [88] for two distinct scenarios: a vortex within a highly viscous liquid (one of the cases the Burgers-Rott model was devised for, representing a draining sink or bathtub [87]) and flows constrained by a rigid boundary at the base (a second case addressed by the Burgers-Rott model, resembling tornado-like flow [87]). Long [88] determined the empirical constant \( K \) for the tornado-like flow to be a 750 \( m^2s^{-1} \), while values of \( W_1 \) or \( W_2 \) (used in \( U_z \) calculation) were not obtained.

A sensitivity analysis was conducted for this thesis to assess the impact of the empirical constant, \( k \), used in the Burgers-Rott model, along with the positioning of \( R_{\theta max} \), on both \( U_\theta \) and \( U_z \). The outcomes of this analysis are illustrated in Fig. 1.7 for \( U_\theta \) and Fig. 1.8 for \( U_z \).

In Fig. 1.7 A-i, the calculated \( U_\theta \) exhibits an ascending trend with increasing \( K \), which is expected given \( K \)'s role as a scaling variable in Eq. 1.5. However, when \( U_\theta \) is normalized by \( U_{\theta max} \), the curves converge, as depicted in Fig. 1.7 A-ii. Fig. 1.7 B-i highlights the influence of \( R_{\theta max} \) on the magnitude of \( U_{\theta max} \). As \( R_{\theta max} \) approaches the rotation center, \( U_{\theta max} \) rises. Normalizing the curves in Fig. 1.7 B-ii by \( U_{\theta max} \) and \( R_{\theta max} \) again leads to the curves converging onto each other.
Figure 1.8 A and B illustrate the sensitivity of $U_z$ to $W_1$ and $W_2$, respectively. As anticipated, in Fig. 1.8 A, the second term in Eq. 1.5 approaches zero at positions far from the rotation axis, resulting in $U_z$ approximating $W_1$. Similarly, at the axis of rotation, the second term equates to $W_2$, and $U_z$’s magnitude becomes the sum of $W_1$ and $W_2$, as shown in Fig. 1.8 B. Figure 1.8 C-i demonstrates $U_z$’s sensitivity to the positioning of $R_{\theta_{\text{max}}}$. It was observed that $R_{\theta_{\text{max}}}$ dictates the spatial range where the decrease in $U_z$ occurs. For $R_{\theta_{\text{max}}}$ values near the rotation axis, $U_z$ drops from its maximum to its minimum at around a $r/R$ value of 0.2. Conversely, when $R_{\theta_{\text{max}}}$ is located farther from the rotation axis, the drop to $U_z$’s minimum occurs at greater $r/R$ values. Normalizing these curves by their corresponding $R_{\theta_{\text{max}}}$ leads to their convergence, as depicted in Fig. 1.8 C-ii. However, no
variable was identified that could yield convergence of $U_z$ for different $W_1$ and $W_2$ values.

The observed convergence of these curves, irrespective of $K$ or $R_{\theta_{\text{max}}}$ values, facilitates the comparison of experimental and numerical data with the Burgers-Rott model. This compatibility holds without necessitating precise knowledge of either $K$ or $R_{\theta_{\text{max}}}$. However, in cases where $U_{\theta_{\text{max}}}$ is known, it becomes feasible to estimate a specific $K$ value for individual experimental or numerical datasets related to cyclones or confined vortices. This estimation can be achieved through the following rearrangement of Eq. 1.5:

$$K \cong \frac{2\pi R_{\theta_{\text{max}}} U_{\theta_{\text{max}}}}{1 - \exp(-1)}$$  \hspace{1cm} (1.6)

Additionally, when $U_{z_{\text{max}}}$ and $U_{z_{\text{min}}}$—the maximum and minimum values of $U_z$,

Figure 1.8: Sensitivity analysis of the Burgers-Rott axial velocity, $U_z$ (Eq. 1.5). Sensitivity to the empirical constants ‘$W_1$’, ‘$W_2$’, and the location of $R_{\theta_{\text{max}}}$ shown in figures (A), (B), and (C) respectively.
respectively—are determined through numerical or experimental analysis of cyclones or confined vortices, it becomes possible to estimate the values of $W_1$ and $W_2$ individually for each case. This estimation can be achieved by rearranging Eq. 1.5 as follows:

\[
W_1 \approx U_{z_{\min}} \\
W_2 \approx U_{z_{\max}} - U_{z_{\min}}
\]

(1.7)

The Rankine vortex model and the Burgers-Rott model describe swirling flows characterized by an outer region of free vortex and an inner region of forced vortex [85-87]. Both models yield a normalized velocity profile for $U_\theta$. When additional data are available from experiments or numerical simulations, both models can provide actual $U_\theta$ values (employing estimated empirical constants in the case of Burgers-Rott) [86]. Moreover, the Burgers-Rott model can provide a velocity profile for $U_z$ given relevant experimental or numerical data, whereas the Rankine model necessitates $U_z$ to remain zero.

For comparison of these two analytical models, Eqs. 1.4 and 1.5 were assessed for $U_\theta$ using $R_{\theta_{\max}}$ set to 1 and $R$ set to 8 ($U_{\theta_{\max}}$ in Eq. 1.4 and K in Eq. 1.5 were both set to 1 as well). The resulting curves were normalized by their respective $U_{\theta_{\max}}$ and $R_{\theta_{\max}}$ values, and these outcomes are displayed in Fig. 1.9. Interestingly, the normalized Burgers-Rott curve does not align with $U_{\theta_{\max}}$ at the specified $R_{\theta_{\max}}$; instead, it shifts slightly further from the axis of rotation. Conversely, the Rankine model corresponds to $U_{\theta_{\max}}$ at the specified $R_{\theta_{\max}}$. Consequently, even with numerical or experimental data, the Burgers-Rott model does not achieve precise alignment and introduces a shift in the curve.

Another distinction between the Rankine and Burgers-Rott models is the transition from the forced vortex inner flow to the free vortex outer flow. The Rankine model displays a sharp peak attributed to the change in functions at $R_{\theta_{\max}}$, while the Burgers-Rott model features a smoother transition. Both models tend toward zero as they approach the axis of rotation. Additionally, as the distance from $R_{\theta_{\max}}$ increases, both models tend toward zero,
yet the Burgers-Rott model trails behind the Rankine model in this regard.

1.2.2.2 Experimental Flow

The swirling flow observed in cyclones and similar devices generates intense turbulence characterized by high anisotropy [58, 82, 89], leading to intricate flow patterns that pose challenges for both experimental measurements [77-82] and numerical predictions [77, 79, 82]. In the literature, three distinct methods have been utilized to measure the internal turbulent flow field within cyclones: five-channel pressure probe [78], Particle Image Velocimetry (PIV) [79], and Laser Doppler Anemometry (LDA), also known as Laser Doppler Velocimetry (LDV) [77, 80-82].

The five-channel probe is an intrusive device that measures the pressure difference between the five holes using a pressure transducer. These pressure measurements are then used to calculate the velocity at the point the probe was measuring. Using this device to map the internal flow velocity also requires knowledge of the internal flow direction [78].

Both Particle Image Velocimetry (PIV) and Laser Doppler Anemometry (LDA) methods are less invasive compared to the probe. These approaches employ small seed particles that ideally track the flow without influencing it. Given that cyclones are employed to separate particles from the flow, using such seed particles becomes more challenging. In both methods, a laser illuminates these seed particles, enabling their tracking by comparing their
positions across successive camera frames. The distinction between PIV and LDA lies in their illumination strategies. PIV employs a laser sheet to illuminate a broad flow area simultaneously, whereas LDA employs a laser to illuminate a single point within the flow [90]. Notably, since both PIV and LDA need a view of the internal flow, the light's refraction through the curved sidewall needs correction, or the cyclone's geometry must be adjusted.

Before conducting a comparative analysis of experimentally measured flow fields from various literature sources for cyclone particle separators with different geometries and operating conditions, it's essential to calculate the Reynolds number (Re) [56, 73] and the Swirl (S) number [53, 77, 79]:

\[
Re_{c1} = \frac{U_{in} D \rho}{\mu} \tag{1.8}
\]
\[
S_g = \frac{\pi D x D}{4ab} \tag{1.9}
\]

where \( \rho \) is the density of the fluid and \( \mu \) is the viscosity of the fluid.

The Reynolds number utilizes the cyclone's diameter as a characteristic length scale, and the swirl number quantifies the ratio of angular momentum \( (G_\theta) \) to axial momentum \( (G_z) \), defined in Equations 10 and 11 respectively [77, 91].

\[
G_\theta = 2\pi \int_0^\infty \rho r^2 (\overline{U_\theta} \overline{U_z} + \overline{u_\theta u_z}) dr \tag{1.10}
\]
\[
G_z = 2\pi \int_0^\infty r \left( (\overline{p} - p_\infty) + \rho \left( \overline{U_z^2} + \overline{u_z^2} \right) \right) dr \tag{1.11}
\]

The Re and S values for the experimental datasets analyzed in this review are summarized in Table 1.2, with the assumption of air's viscosity being \( 1.81 \times 10^{-5} \text{ kgm}^{-1}\text{s}^{-1} \). For industrial cyclone particle separators, Re typically falls in the range of \( 10^5 \) and S is usually between 1.5 and 4.0 [77]. The datasets listed in Table 1.2 investigated devices within the range of industrial cyclones.
The velocity profiles of $U_\theta$ and $U_z$ for the datasets listed in Table 1.2 are shown in Figs. 1.10 A and B, respectively. These velocity profiles are taken from experimental data for cyclone particle separators of different geometries. Although, the axial position ($z$) is not consistent between the datasets examined, each curve pertains to a $z$ position inside the cylindrical section of the cyclone, with the exception of Liu et al. [79], which was in the conical section of the cyclone. The velocity components being examined are normalized by the corresponding $U_{in}$ of that dataset. Similarly, the radial positions ($r$) at which the measurements were taken are normalized by the radius of the cyclone ($R = D/2$). Normalizing by $U_{in}$ and $R$ is common within the literature [74, 77, 78], and allows for the

<table>
<thead>
<tr>
<th>Dataset</th>
<th>$Re$</th>
<th>$S$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reydon and Gauvin [78]</td>
<td>$7.08 \times 10^5$</td>
<td>1.5</td>
</tr>
<tr>
<td>Hoekstra et al. [77]</td>
<td>$1.21 \times 10^5$</td>
<td>1.8</td>
</tr>
<tr>
<td>Liu et al. [79]</td>
<td>$1.24 \times 10^5$</td>
<td>2.4</td>
</tr>
<tr>
<td>Solero and Coghe [80]</td>
<td>$1.20 \times 10^5$</td>
<td>2.4</td>
</tr>
<tr>
<td>Hu et al. [82]</td>
<td>$1.89 \times 10^5$</td>
<td>2.8</td>
</tr>
<tr>
<td>Liu et al. [81]</td>
<td>$1.51 \times 10^5$</td>
<td>Na</td>
</tr>
</tbody>
</table>

Figure 1.10: Comparison of experimentally measured mean velocity profiles (normalized by $U_{in}$ and $R$) for different cyclone geometries and operating parameters. Velocity profiles are for an axial position in the cylinder section of the cyclone and below the inlet. Tangential velocity (A), Axial velocity (B).
datasets to be compared despite the difference in the size of the cyclone or the magnitude of $U_{in}$.

Liu et al. [81], Reydon and Gauvin [78], and Hoekstra et al. [77] presented their $U_{\theta}$ velocity profiles for a $r/R$ having a range of zero to one, thus the results shown from Hu et al. [82], Solero and Coghe [80], and Liu et al. [79] were only examined over this range as well. By only showing this range, it assumes that the $U_{\theta}$ velocity profiles are axi-symmetrical, although this is not necessarily the case if a precessing vortex is present [53, 80].

The position of $R_{\theta max}$ is independent of $z$ within the same cyclone geometry [82]. Thus, the trends of the five curves shown in Fig. 1.10 A can be compared despite the different $z$ values at which the measurements were taken. The $U_{\theta}$ velocity profiles shown in Fig. 1.10 A clearly indicate the existence of a quasi-free vortex away from the geometric axis of rotation and a quasi-forced vortex near the axis of rotation, thus displaying the typical Rankin vortex structure [74, 79, 81, 82]. The position of $R_{\theta max}/R$ also does not seem to be linearly dependent on the value of $Re$ or $S$, as Hu et al. [82] had the second largest value for $Re$ and largest value for $S$ while having the second smallest $R_{\theta max}/R$ position. While other experimental data sets with lower $Re$ and $S$ had larger and smaller $R_{\theta max}/R$ positions. However, Solero and Coghe [80] and Liu et al. [79] had the same $S$ and very similar $Re$, and not only do these data sets have similar $R_{\theta max}/R$ positions, they also have similar $U_{\theta max}/U_{in}$ values. The positions of $R_{\theta max}$ and the magnitude of $U_{\theta max}$ normalized by $U_{in}$ for each dataset are summarized in Table 1.3. Also of interest in Fig. 1.10 A, was how the $U_{\theta}$ velocity profile in the quasi-free region collapsed for Hu et al. [82], Liu et al. [81] and Liu et al. [79]; with the velocity profile from Solero and Coghe [80] almost collapsing onto the others.

The $U_z$ velocity profiles are shown in Fig. 1.10 B collapse onto each other in the quasi-free vortex region of the flow; however, they behave quite differently in the quasi-forced vortex region. As is typical for a cyclone particle separator $U_z$ is negative (traveling towards the dust hopper) near the cyclone wall and is positive (travelling towards the vortex finder) near the geometric axis of rotation [24, 53, 58, 80]. There is also a depression in the $U_z$ profiles at the positions very close to the axis of rotation for all but Hu et al. [82]. This
depression of axial velocity can be significant enough to cause a reversal of flow, resulting in negative $U_z$ [53], and this happens to be the case for the dataset of Solero and Coghe [80]. The drop in $U_z$ very close to the geometric axis of rotation is the result of a reverse flow zone that is often present when the $Re$ and $S$ are greater than $10^4$ and 0.6, respectively [80]. For each of the experimental datasets shown in Fig. 1.10 B, $Re$ was higher than $1.2 \times 10^5$ and $S$ was larger than 1.5 (for the datasets where this information was available) which indicates that the reversed flow zone was to be expected. Interestingly, the only dataset not exhibiting the reversed flow is Hu et al. [82] which had the second largest $Re$ and largest $S$ values.

The experimental data were once again plotted together to compare the $U_\theta$ velocity profile, this time normalized by $U_{\theta max}$ and $R_{\theta max}$, shown in Fig. 1.11. Both analytical models (which are now independent of empirical values: $R_{\theta max}$, $U_{\theta max}$, and $K$) were also plotted in Fig. 1.11 for comparison to the experimental velocity profiles. Figure 1.11 shows that, with the exception of Hoekstra et al. [77], the tangential velocity profiles closely resemble the Burgers-Rott model. This suggests that the Burgers-Rott model, which is a solution to the Navier-Stokes equations [8], is better suited as an analytical model than the Rankine model for cyclone particle separators. This is due to the Rankine model following a truly free-vortex outer flow and forced vortex inner flow [7], while the actual flow in a cyclone is a quasi-free/quasi-forced vortex flow [2, 6].

<table>
<thead>
<tr>
<th>Dataset</th>
<th>$R_{\theta max}/R$</th>
<th>$U_{\theta max}/U_{in}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hoekstra et al. [77]</td>
<td>0.12</td>
<td>2.36</td>
</tr>
<tr>
<td>Liu et al. [79]</td>
<td>0.33</td>
<td>2.46</td>
</tr>
<tr>
<td>Solero and Coghe [80]</td>
<td>0.29</td>
<td>2.49</td>
</tr>
<tr>
<td>Hu et al. [82]</td>
<td>0.21</td>
<td>2.58</td>
</tr>
<tr>
<td>Liu et al. [81]</td>
<td>0.32</td>
<td>1.59</td>
</tr>
</tbody>
</table>
Employing Computational Fluid Dynamics (CFD) techniques enables the numerical prediction of the internal flow within a cyclone. CFD offers the advantage of providing insights into the entire flow field without the need for intrusive instruments, a benefit not present in experiments. However, CFD methods come with their own set of challenges and limitations.

One primary challenge lies in configuring the solver to utilize an appropriate turbulence closure model. Another limitation of CFD pertains to the necessity for validation through physical experiments to establish trustworthiness in the predicted results.

One primary challenge lies in configuring the solver to utilize an appropriate turbulence closure model. Another limitation of CFD pertains to the necessity for validation through physical experiments to establish trustworthiness in the predicted results.

Hu et al. [82], Hoekstra et al. [77], and Liu et al. [79] each performed CFD simulations of the cyclones used in their respective experimental work. Furthermore, Hoekstra et al. [77] and Liu et al. [79] tested several turbulence closure models including: the Reynolds Stress Model (RSM), renormalization group (RNG) k-ε, and k-ε). Additionally, Hu et al. [82] presented CFD results using the RSM turbulence closure model. The performance of CFD simulations using the various turbulence closure models is compared to corresponding experimental data in Fig. 1.12. The $U_\theta$ velocity profiles are shown in plots labelled with ‘i’ and the $U_z$ velocity profiles are labelled with ‘ii’.

Figure 1.11: Comparison of experimentally measured mean tangential velocity profile (normalized by $U_{\theta_{\text{max}}}$ and $R_{\theta_{\text{max}}}$) for different cyclone geometries and operating parameters and analytical solutions for $U_\theta$. 

1.2.2.3 Numerical Flow

Employing Computational Fluid Dynamics (CFD) techniques enables the numerical prediction of the internal flow within a cyclone. CFD offers the advantage of providing insights into the entire flow field without the need for intrusive instruments, a benefit not present in experiments. However, CFD methods come with their own set of challenges and limitations.

Hu et al. [82], Hoekstra et al. [77], and Liu et al. [79] each performed CFD simulations of the cyclones used in their respective experimental work. Furthermore, Hoekstra et al. [77] and Liu et al. [79] tested several turbulence closure models including: the Reynolds Stress Model (RSM), renormalization group (RNG) k-ε, and k-ε). Additionally, Hu et al. [82] presented CFD results using the RSM turbulence closure model. The performance of CFD simulations using the various turbulence closure models is compared to corresponding experimental data in Fig. 1.12. The $U_\theta$ velocity profiles are shown in plots labelled with ‘i’ and the $U_z$ velocity profiles are labelled with ‘ii’.
The comparison of Hu et al. [82] CFD to the experimental results (shown in Figs. 1.12 A-i and A-ii) revealed that $U_\theta$ was consistently overpredicted by the CFD model, although the trends between the two were similar for both $U_\theta$ and $U_z$ [82]. The comparison of the $U_\theta$ velocity profiles obtained through CFD and experiments for both Hoekstra et al. [77] (Fig. 1.12 B-i), and Liu et al. [79] (Fig. 1.12 C-i), showed that the RSM turbulence closure model has the best agreement to the experimental data. The RSM turbulence closure model was the only closure model to show the typical Rankine/Burgers-Rott $U_\theta$ velocity profile [77], while the k–$\varepsilon$ turbulence closure model solves the flow as if it were entirely a quasi-forced vortex [77]. The RSM proving to be the superior turbulence closure model can be explained by the fact that the other closure models assume isotropic turbulence [79], while the cyclone in reality has anisotropic turbulence [58, 82, 89]. The evaluation of the $U_z$ velocity profiles for the CFD turbulence models and experimental data for Hoekstra et al. [77] (Fig. 1.12 B-ii), and Liu et al. [79] (Fig. 1.12 C-ii), showed no turbulence closure model outperforming the other consistently. Since all the turbulence closure models examined had issues with the flow depression at the geometric axis of rotation, the RSM was deemed best suited for predicting the $U_\theta$ velocity profile and was comparatively adequate at predicting the $U_z$ velocity profile.

As already indicated, the mean velocity profiles for $U_\theta$ obtained through the RSM turbulence closure model (shown in Fig. 1.13 A) shows the quasi-forced vortex towards the axis of rotation and the quasi-free vortex towards the cyclone walls [77, 79, 82]. Unlike the experimental data, the CFD simulations can provide the velocity profile right to the cyclone wall (which has a no-slip boundary condition), which explains the sudden drop in the $U_\theta$ at high $r/R$ values for Hu et al. [82], Hoekstra et al. [77]. The normalized (by $R$) radial position values where the flow transitions from the quasi-free vortex to the quasi-forced vortex ($R_{\theta_{max}}$) for the three CFD simulations are summarized in Table 1.4. Also tabulated in Table 1.4 are the $U_{\theta_{max}}$ values found at $R_{\theta_{max}}$ normalized by their corresponding $U_{in}$. 
Figure 1.12: Comparison of velocity profiles (normalized by $U_{in}$ and $R$) for various turbulence closure models used. Tangential velocity ($U_\theta$) profiles are shown in figures (i) and axial velocity ($U_z$) profiles shown in figures (ii). Hu et al. [82] data shown in (A), Hoekstra et al. [77] data shown in (B), and Liu et al. [79] data shown in (C).
The $U_z$ velocity profiles for the three CFD simulations were also compared (shown in Fig. 1.13 B). The velocity profiles show the expected flow in the negative $z$ direction near the wall of the cyclone and the positive $z$ direction towards the geometric axis of rotation, as is typical for flows in a cyclone particle separator [24, 53, 58]. However, only Hoekstra et al.’s [77] simulations predicted a depression in the $U_z$ very close to the axis of rotation that typically exists for vortex flows with similar $Re$ and $S$ values [82].

Table 1.4: Non-dimensional flow parameters for CFD results.

<table>
<thead>
<tr>
<th>Dataset</th>
<th>$R_{\theta\text{max}}/R$</th>
<th>$U_{\theta\text{max}}/U_{in}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hu et al. [82]</td>
<td>0.18</td>
<td>3.00</td>
</tr>
<tr>
<td>Hoekstra et al. [77]</td>
<td>0.13</td>
<td>2.63</td>
</tr>
<tr>
<td>Liu et al. [79]</td>
<td>0.37</td>
<td>2.42</td>
</tr>
</tbody>
</table>

The $U_z$ velocity profiles for the three CFD simulations were also compared (shown in Fig. 1.13 B). The velocity profiles show the expected flow in the negative $z$ direction near the wall of the cyclone and the positive $z$ direction towards the geometric axis of rotation, as is typical for flows in a cyclone particle separator [24, 53, 58]. However, only Hoekstra et al.’s [77] simulations predicted a depression in the $U_z$ very close to the axis of rotation that typically exists for vortex flows with similar $Re$ and $S$ values [82].

Figure 1.14 shows the numerical predicted tangential velocity profiles normalized by $U_{\theta\text{max}}$ and $R_{\theta\text{max}}$ and compared to both analytical models. The tangential velocity profiles collapse quite well (qualitatively) onto each other when normalized by $U_{\theta\text{max}}$ and $R_{\theta\text{max}}$. Furthermore, the Burgers-Rott model is again shown to be a better representation of the tangential velocity distribution when compared to the Rankine model. The reason for this...
The presence of the PVC in a cyclone particle separator needs an assessment of the fluctuating tangential velocity \(u_\theta\) [53, 80]. By definition, the PVC cannot be seen in mean velocity profiles that assume axisymmetric flow. To observe the PVC in the cyclone, \(u_\theta\) at the geometric axis of rotation is examined [53, 77]. At this specific position, \(U_\theta\) is null for both analytical models discussed (Rankine vortex and Burgers-Rott), and experimental and numerical data also indicate a trend towards zero. In the absence of a PVC, the geometric axis of rotation transforms into a stagnation point within the flow (in the \(\theta\) direction), suggesting minimal \(u_\theta\) values. However, if a PVC is present, then higher levels of turbulence would be expected as the PVC is the result of instability of the flow [80].
indicating that $u_\theta$ should be significant at the axis of rotation [77].

Figure 1.15 illustrates $u_\theta$ for two distinct radial locations: the first resides precisely at the axis of rotation, while the second is situated a slight distance from the axis (such that it is located within the free vortex zone of the flow). The fluctuating velocity is normalized by $U_{in}$, and the measurement time is normalized by a macro timescale ($T$) as defined by Hoekstra et al. [77], expressed in Eq. 1.12.

$$T \doteq \frac{D}{U_{in}}$$  \hspace{2cm} (1.12)

The timeseries data of $u_\theta$ from Hoekstra et al. [77] is presented in Fig. 1.15 A (with $T$ at 47 ms), while that from Liu et al. [79] is depicted in Fig. 1.15 B (with $T$ at 18 ms). In Fig. 1.15 A, the timeseries of $u_\theta$ reveals substantial values (twice the magnitude of $U_{in}$) at the axis of rotation, in contrast to the fluctuations within the free vortex region (half the magnitude of $U_{in}$). The pronounced $u_\theta$ values at the axis of rotation strongly indicate the presence of a PVC within the cyclone. Additionally, the timeseries data of $u_\theta$ at the axis of rotation exhibit low-frequency oscillations, attributed to the PVC’s influence [77].

Analyzing the data in Fig. 1.15 B, the magnitude of $u_\theta$ is notably smaller than that observed

![Figure 1.15: Time history of the fluctuating tangential velocity normalized by the nominal inlet velocity with time normalized by a macro time scale, showing how the radial position affects the magnitude of the fluctuations. Hoekstra et al. [77] data shown in (A), and Liu et al. [79] data shown in (B).](image)
in Fig. 1.15 A. Nonetheless, $u_\theta$ remains greater at the axis of rotation compared to the free vortex region [79], reinforcing the notion of a contained PVC within the cyclone. Furthermore, in the timeseries $u_\theta$ data shown in Fig. 1.15 B, a low-frequency oscillation is visible, similar to the effect of the PVC as noted in the case of Hoekstra et al. [77].

In order to provide a clearer illustration of the presence of a PVC in cyclones, the timeseries $u_\theta$ data shown in Fig. 1.15 has been transformed into histograms representing the count of measurements falling within specified $u_\theta$ bins (shown in Fig. 1.16). The histogram for the $u_\theta$ data from Solero and Coghe [80] has been incorporated and is depicted in Fig. 1.16 A (the data from Hoekstra et al. [77] is shown in Fig. 1.16 B, and the data from Liu et al. [79] is shown in Fig. 1.16 C). In Fig. 1.16 A, it is evident that the cyclone utilized by Solero and Coghe [80] yielded substantial $u_\theta$ values, attaining magnitudes nearly ten times that of $U_{in}$.

![Histograms of the fluctuating tangential velocity normalized by the inlet velocity at the geometric axis of rotation. Solero and Coghe [80] data shown in (A), Hoekstra et al. [77] data shown in (B), and Liu et al. [79] data shown in (C).](image-url)
Beyond the magnitude, the bimodal distribution of the histogram is of particular interest. This distribution has been attributed to the coexistence of two distinct velocity regimes [80], a phenomenon that can also be attributed to the presence of a PVC. Given the three-dimensional nature of the cyclone flow, the PVC can oscillate around the axis of rotation without passing through it directly. Consequently, this scenario would lead to more frequent measurement of higher fluctuations as compared to smaller fluctuations, since the stagnation point would seldom align with the axis of rotation. A similar bimodal distribution is observed in Fig. 1.16 B, while it is absent in Fig. 1.16 C. Notably, the latter two histograms are based on a limited time sample (with a maximum value of $t/T$ at 20 for Fig. 1.16 B and 15 for Fig. 1.16 C). Therefore, it is speculated that if a more extended time sample of $u_\theta$ were employed to construct the histogram in Fig. 1.16 C, a bimodal distribution might emerge. This assumption is founded on the observation that the distribution in Fig. 1.16 C is not centred around zero, implying the potential presence of a PVC.

### 1.2.2.5 Flow Shortcutting

An additional flow feature that is present in cyclone particle separators is that it is possible for the flow to leave the cyclone prematurely by escaping under the vortex finder (shown in Fig. 1.17) and making far fewer rotations around the cyclone as it travels axially downwards, an effect referred to as shortcutting [50, 55, 57, 62, 64]. The shortcutting flow has an adverse effect on the vortex strength below the vortex finder due to the decreased volume of air [50]. The Muschelknautz and Trefz model, presented in [56] incorporates shortcutting flow and estimates the shortcutting to be 10% of the mass flow rate of the continuous phase. However, numerical simulations have predicted the degree of shortcutting to be more severe than this [55, 62, 64], and it can be as high as 32.5% [55]. Cyclone designs that result in a high degree of shortcutting flow tend to have reduced separation efficiencies [64]. Therefore, reducing shortcutting flow can improve the collection efficiency of a cyclone particle separator.
1.2.3 Predicting Particle Motion in Numerical Domains

The application of Discrete Phase Modeling (DPM) enables the injection and tracking of particles within a numerical domain, offering the ability to predict particle motion within devices like cyclones [1-11] and settling chambers [15, 16]. The process of using DPM for particle motion prediction involves three steps: injecting particles into the domain, tracking their movement through the domain, and concluding the tracking at their final position. Injecting particles into the numerical domain requires the specification of initial particle properties such as position, velocity and temperature, in addition to the particle diameter and density [92]. Particle density is dependent on the material composition, while particle diameter is influenced by the particle size distribution. The initial position, velocity, and temperature depend on the upstream process being numerically simulated. When considering the inlet distribution, factors such as injection locations and particle size distribution deserve attention. Stovin & Saul [15] investigated a storage chamber in combined sewer overflow structures, focusing on injection location and the number of simulations. Particle location was tested by injecting particles with a Rosin-Rammler [69] distribution (an example distribution is shown in Fig. 1.18) at discrete points. The predicted efficiency showed a spread of 37%, with maximum efficiency achieved from positions
toward the right edge of the inlet. The impact of injection location on performance has been explored by Egarr et al. [14] and Patel et al. [18]. Both studies injected particles along different lines on the inlet surface, observing varied performance for each line. Patel et al. [18] also experimented with point injections and concluded that varying point locations impacted predicted performance. Stovin & Saul [15] suggested repeating DPM simulations with varied parameters to reduce the efficiency deviation to ±2.5% at 99% confidence, necessitating changing the initial particle location in repeated runs.

The common approach for particle introduction in separation devices employs a surface injection method [5, 8, 12-14]. The surface injection method fixes injection locations on the chosen surface to the cell centres of the surface mesh [14, 92]. Thus, this method cannot account for changes in predicted particle motion due to injection location without altering the mesh as well.

Particle tracking in DPM can be accomplished through one-way coupling or two-way coupling. One-way coupling entails the continuous phase affecting discrete phase flow (particles), while the discrete phase has no impact on the continuous phase [2, 8, 11, 69]. In contrast, two-way coupling considers the discrete phase's impact on the continuous phase flow [9-11, 69], requiring conservation equations for each inter-dispersed phase and resulting in greater computational demands. One-way coupling simplifies computational

Figure 1.18. A Rosin-Rammler particle size distribution.
requirements but is suitable only when the dense (particle) phase minimally influences continuous phase flow structure, as in dilute flows (where the dense phase occupies less than 10% of volume) [2, 7, 9, 11]. Notable applications of dilute flows include pneumatic particle transport systems and filtration devices like separation chambers and cyclones.

Particle tracking concludes when one of three conditions is met. The first two conditions are particle collision with a boundary with a trapping or escaping condition [1, 15, 92]. The third condition is reaching the maximum number of particle steps [5, 14, 15, 92]. While the trap boundary condition halts particle tracking due to trapping, the particle's energy at the trapping instant remains in the domain [15, 92]. In contrast, the escape boundary condition allows both the particle and its energy to exit the domain [15, 92]. The maximum particle steps represent the iterations a particle will be tracked for [15, 92], and addresses situations where particles remain within the domain without encountering a termination boundary [15, 92]. Ensuring simulation termination in cases where particles are caught in recirculation zones prevents indefinite particle tracking [15].

1.2.4 Literature Review Summary
The literature synthesis highlighted how the cyclone geometry, in particular the height of the cyclone, affects both the collection performance and the pressure drop. The literature also showed that increasing the inlet velocity yields a stronger vortex and, as a result, increases the collection performance but at the cost of increased pressure drop. Despite the geometric or operating parameters, the Burgers-Rott analytical model was able to represent an accurate tangential velocity profile. However, the Burgers-Rott model, as well as the Rankine model, require pre-existing knowledge of the tangential velocity profile and thus cannot be used as a prediction tool of the flow in a way that CFD modelling can.

The conducted literature review on cyclone particle separators highlighted gaps in the literature. The literature showed how the performance of a cyclone is affected by an increase in the cyclone height; however, how shortening the height of a cyclone affects the flow field and, consequently, the collection performance still needs to be addressed. When it comes to numerically modelling the cyclone particle separator, there are also inconsistencies in including the collection hopper in the numerical domain and what the
effect of neglecting the collection hopper has on the predicted flow field. Furthermore, numerically injecting particles into the cyclone particle separator is commonly performed using a static injection that is grid-dependent. At the same time, the literature for non-cyclone devices has shown that the location of injections affects the predicted particle motion.

1.3 Objectives of the Present Work

The present work aims to achieve a reduced scale cyclone particle separator without sacrificing the collection performance. The literature review reveals that lengthening a cyclone particle separator increases the collection performance and reduces the pressure drop, with the implication that reducing the length of the cyclone would have a reversed effect and the performance of a cyclone would be reduced. Analyzing the effect of reducing the scale of a cyclone using an analytical model such as Rankine Vortex or Burgers-Rott would not be effective, as the models required pre-existing knowledge of the internal flow and, thus, experimental and numerical methods are needed. CFD is a powerful tool for evaluating the flow features inside cyclone particle separators. When coupled with DPM, the particle motion through the device can be predicted, yielding collection performance predictions. However, the surface injection method (prominently used in literature for injecting particles into cyclones) is ineffective, as the predicted particle performance of devices has shown to be dependent on the injected particle position. The specific objectives of the present work to address the gaps in the literature and achieve the goal of this thesis are outlined as follows:

1. Develop a reliable numerical method for injecting and tracking particles into a CFD domain. The technique will allow for fast and highly accurate predictions of bulk particle motion through a CFD prediction of the flow inside a device. The method will account for the chaotic motion of particles entering devices that have a dilute pneumatic transport of particles. The model will be numerically verified using a randomized transient injection with transient DPM tracking for three unique devices.

2. Identify flow features that result in collection performance degradation when a cyclone particle separator is scaled down. Internal flow features can be examined
where particles are predicted to travel in undesirable directions, such as shortcutting or re-entrainment from the collection hopper, using the results gained from numerical modelling of the cyclone devices. The flow features in these areas will be compared to the corresponding flow field of a highly efficient cyclone particle separator to determine the extent of the differences between the two flow fields.

3. Alter the identified flow features from the previous objective to improve the collection performance of the scaled down cyclone particle separator with the addition of a rotating classifier. The method of altering the flow field to increase the collection performance of the cyclone needs to maintain the scale reduced cyclone height, as well as the advantages cyclones have over other separating and filtering devices, steady pressure drop and not permanently trapping the particles in the device.

1.4 Methods and Scope of Research

The research was conducted using physical experiments and numerical simulations (CFD and DPM) to achieve the abovementioned objectives. The industrial partner Design Build Mechanical Inc. (DBM), New Brunswick, Canada, fabricated the experimental setup, including the cyclones and rotating classifier. The experiments were also conducted in DBM’s facilities. The physical experiments served the purpose of tuning the DPM for the selected powder used in the experiments and validating the numerically predicted flow fields and collection performance. The numerical simulations allow for analyzing the internal flow of the cyclone throughout the entire domain and without the use of intrusive measurement devices. However, the numerical simulations result in simplifications of real-world physics. Thus, even with tuning and validation, the simulations yield information on trends but may not represent actual predicted values of collection performance.

The initial cyclone designed and fabricated by DBM was 1670 mm tall and had a diameter of 496 mm. The scaled reduced cyclone maintained the same diameter but was only 835 mm tall. For both the numerical simulations and the physical experiments, the inlet velocity of all cyclones was set to 12.7 ms$^{-1}$. The solid particle mass flow rate in the physical experiments was 8.5 gs$^{-1}$. However, in the numerical simulations, the mass flow rate of the powder did not affect the predicted results and was arbitrarily set to 10 gs$^{-1}$. The powder
used in the experiments was Portland cement with a 1-15 μm particle range. The classifier added to the cyclone to achieve a dynamic cyclone was rotated at 3000-6000 rpm in the experiments and simulations.

Additional details pertaining to the physical experiments and numerical simulations are presented in the following chapters.

### 1.5 Thesis Outline

The outline below details how the remaining chapters of this thesis address the objectives described in Section 1.3. The chapters each of which represents a journal manuscript, will appear as follows

- Chapter 2: A computationally efficient method of injecting and tracking dilute pneumatically transported particles into a numerical domain using DPM is presented. The Aggregate STEady Random Particle (A-STERP) takes advantage of the short computational time of steady DPM simulations. It introduces temporal randomization by considering the aggregate, cumulative average of results obtained from sequential steady simulations using files of randomized injection points and particle sizes. The A-STERP method is computationally validated for its ability to predict the total and fractional collection efficiency of a defined distribution of particles inside three devices: a generic collection chamber, a baffled pre-separator, and a cyclone. The results yielded by A-STERP are compared to those obtained from a randomized transient injection method and show in all cases, to be just as accurate while requiring only a small fraction of the computational time.

- Chapter 3: This chapter describes an experimental and numerical investigation into the performance degradation of cyclone particle separators when they are scaled down. Three conventional (non-dynamic, tangential inlet) cyclones with varying heights are investigated: Full-size, Intermediate and Truncated, with the Full-size being the closest to a Stairmand cyclone design and the Truncated having half the height of the Full-size. The numerical investigation utilizes the A-STERP injection and tracking model described in Chapter 2. The experimental study validates the numerically predicted results and tunes the drag model imposed on the simulated
particles. The numerical investigation allows identification of particle motion and quantification of the degree of particle shortcutting and re-entrainment, which leads to lower collection performance.

- Chapter 4: Recovering the collection performance in scaled down cyclone particle separators is the focus of this chapter. Focusing on the locations responsible for the reduction of the collection performance from scaling down the cyclone, identified in Chapter 3, methods of altering the flow in these areas are investigated numerically. The investigation is, again, conducted numerically and experimentally. Improving the flow under the vortex finder causing shortcutting utilizes a rotating classifier yielding a dynamic cyclone design. The rotating classifier's introduction increases the intensity of the vortex, which introduces increased particle re-entrainment. The particle re-entrainment gained from the rotating classifier is then addressed by examining different collection hopper designs.

- Chapter 5: Key Chapters 2-4 findings are summarized and discussed as an ensemble, with conclusions drawn, along with recommendations for future research.

1.6 References


that determine the efficiency of a hydrodynamic vortex separator,” 2004.


2012.


Chapter 2

Statistically accurate discrete phase modelling of particle cloud generation using Aggregate Steady Random Particle injection

2.1 Introduction

Particle motion tracking through Discrete Phase Modeling (DPM) is available in Computational Fluid Dynamics (CFD) software and has been used for numerous applications including, but not limited to: cyclones (pneumatic [1-11] and hydraulic [12-14]), settling/storage chambers [15, 16], waste water treatment equipment [17,18], water spray from fire safety nozzles [19], and respiratory systems [20].

DPM particle tracking can be performed via one-way coupling or two-way coupling. One-way coupling results in the continuous phase affecting the flow of the discrete phase (particles), while the discrete phase has no impact on the flow of the continuous phase [2, 3, 9, 21]. In contrast, two-way coupling includes the impact of the discrete phase on the flow of the continuous phase [3, 10, 11, 21], but requires conservation equations to be solved for each inter-dispersed phase, resulting in significantly higher computational times. One-way coupling is thus a simplification that reduces the computational resources required, but is only suitable when the dense (particle) phase does not have a significant effect on the flow structure of the continuous phase. Such is the case in in dilute flows, which are defined as those where the dense phase occupies less than 10% of the space by volume [2, 3, 8, 10]. Numerous important applications of dilute flows exist including, for example, pneumatic particle transport systems, and filtration devices such as separation chambers and cyclones. In this Chapter, the focus is restricted to very dilute flows (<< 10%) that are typically seen in particle separation devices.

There are numerous parameters and settings that require consideration in DPM simulations. These include, but are not limited to: the particle size, particle density, wall boundary condition, step length factor, maximum number of wall reflections, maximum number of time steps, inlet distribution, and number of simulations. Several other parameters, such as reactivity and thermophysical properties related to influences by electrical, magnetic or
high-temperature fields, can also be considered but are omitted in the present study. The mass and the size of the particles are dependent upon material properties of the solid phase being represented by the DPM [1-3, 7, 11-13, 15-18] and thus, will change from application to application. However, both will affect the predicted particle motion as it will alter the particle’s inertia and drag force [1-3, 5, 6, 8, 10, 11, 13, 15-18, 21], thereby affecting how quickly the discrete phase will be affected by changes in the surrounding continuous phase [3, 4, 6, 9, 12, 15]. The boundary conditions used are also mostly dependent on the application being simulated. The two options for boundaries through which the dense phase can pass, resulting in termination of particle tracking, are trap and escape [1, 15, 22]. There is an additional option (reflect) for boundaries through which the dense phase cannot pass resulting in the continuation of particle tracking [6, 15, 22]. The reflect boundary condition is used to deflect a particle off of a boundary to keep the particle inside the domain [1, 15, 18, 22]. Though the trap boundary condition terminates the tracking of the particle, since it is trapped in place, the energy the particle contained at the instant it was trapped remains in the domain [15, 22]. The escape boundary condition allows the energy of the particle to leave the domain along with the particle [15, 22].

The step length factor and maximum number of time steps are solution controls [6, 14, 15, 22]. The step length controls the integration time step [14, 15, 22]. Stovin & Saul [15] found that doubling or halving the step length had an insignificant effect on the predicted performance. In contrast, Egarr et al. [14] found that changing the step length factor by three orders of magnitude could affect the predicted performance and concluded that a smaller length scale is more accurate. The maximum number of time steps is the maximum number of time steps a given particle will be tracked for [15, 22] and is used for particles remaining in the domain and never reaching a boundary that allows for termination of motion tracking [15, 22]. It is necessary to ensure that the simulation will end if a particle gets caught in a recirculation zone, which would otherwise result in particle tracking going on indefinitely [15].

In terms of inlet distribution, consideration is required in terms of injection locations and particle size distribution. The most common approach for introducing particles into a particle separation device is to use a surface injection approach [6, 9, 12-14], which in its
simplest form, fixes injection locations on the selected inlet surface to the cell centres of the surface mesh [14, 22]. Particles of a given size distribution can be assigned by this approach, but the number of injection locations is limited by the density of the surface mesh. Finally, in terms of the number of simulations, Stovin & Saul [15] recommend repeating the DPM simulations up to 50 times with varying parameters to reduce the deviation in the predicted efficiency to ± 2.5% at a confidence level of 99%. In this case, the particle size distribution must be changed between repeated runs to change the outcome of the DPM predictions. Stovin & Saul [15] examined a storage chamber that is frequently used in combined sewer overflow structures wherein the parameters of particular interest were the injection location and the number of simulations conducted. The particle location was tested by injecting particles containing a Rosin-Rammler [21] particle size distribution at 9 discrete locations. The predicted efficiency of particles released from each location had a maximum spread of 37% (the maximum efficiency was achieved from a position towards the right edge of the inlet, while the lower efficiencies corresponded to positions near the top left edges). The impact of injection location on predicted performance has also been investigated by Egarr et al. [14] and Patel et al. [18]. Both authors injected particles along different lines located on the inlet surface and found that the predicted performance changed for each line. Patel et al. [18] also tested point injections on the inlet surface and concluded that the predicted performance was affected by the different point locations.

From the literature reviewed, DPM is shown to be suitable for predicting particle motion inside particle separation devices. Thus, using an appropriate CFD solver to simulate the continuous phase coupled with a DPM solver for the discrete phase can yield useful predictions of the separation performance of a device. It is also clear from prior literature that the injection of particles, particularly the location, can significantly impact the path the particle will travel, irrespective of the DPM used, although previous studies generally only consider bulk changes in the injection location. Furthermore, the injection methods used in past studies usually simplify how the particle is introduced to the flow/device by using fixed injection locations; be it a specific point, a uniform line, or at cell centers of a surface. In physical devices, the particles enter in random locations across the inlet boundary that would also be constantly changing in time.
This Chapter presents an approach for accurate characterization of a cloud of particles injected into devices used for filtration via particle separation. The Chapter considers steady and transient one-way coupled DPM methods and compares different injection approaches ranging from simple surface injection to complete replication of how the particle cloud would be introduced in a physical device; i.e. transient injection randomized in surface location and time. A new approach that is both statistically accurate and computationally efficient is proposed for injecting particles into a domain. The proposed approach adopts features already available in commercial CFD software, but uses a novel strategy for particle injection that takes into account the randomness of surface location, particle size and time. This approach has the advantage of running steady sequential simulations on a known flow field that – with a sufficient number of randomly generated input files – are shown to aggregate to the quality of a fully random, transient approach, but with greatly reduced run-times. To this end, the proposed approach is called the Aggregate STEady Random Particle (A-STERP) approach for particle injection in DPM.

2.2 Discrete Phase Modeling

To achieve accurate physical characterization of a cloud of particles injected into a device – and subsequent tracking and capture/separation – both the particle tracking method and the particle injection approach must be taken into consideration. Since the focus of this Chapter is on dilute particle flows, only steady and transient variants of the Euler-Lagrange DPM approach are considered. These are used herein without modification, but are combined with various particle injection strategies to achieve accurate cloud characterization and capture/separation. The commercial software ANSYS FLUENT is used for all the simulations and is run on the same workstation such that comparisons of wall-clock times are meaningful. A brief description of Euler-Lagrange particle tracking methods is given below followed by descriptions of particle injection approaches and A-STERP.

2.2.1 Particle Tracking

One-way coupled DPM can be conducted for both steady and transient (unsteady) particle tracking. Steady particle tracking can be performed when the continuous phase is represented by a single time frame. The steady particle tracking method computes particle
paths in a given flow field that any particle of the same size released from the same location will take. Steady particle tracking only requires running the discrete phase solver and is well-suited for stationary flows. Transient particle tracking can be performed when the continuous phase is to be solved along with the particle phase and thus, not represented by a single time frame. The transient tracking method tracks a parcel of particles that are of the same size and uses a single particle in the parcel to calculate the trajectory of the parcel. In transient particle tracking, the motion of particles is affected by the unsteadiness in the flow and thus, this method more accurately replicates the physics of particle motion in flows that are inherently non-stationary. Another advantage of transient tracking is that it provides a framework for introducing temporal randomness in particle simulations, which is necessary for the most accurate possible characterization of particle cloud injection in stationary, time-periodic and non-stationary flows. These benefits, however, come at the cost of substantially larger computational resources, as transient tracking requires solving the continuous phase and the discrete phase simultaneously until a steady-state of particles exists inside the domain.

While the DPM method affects particle motions within the domain, the particle injection approach affects the characterization of a cloud of particles inserted into a flow. Ultimately, it is of interest to model the randomness of an injected cloud in both space and time and this can be accomplished by combining the appropriate DPM method with an appropriate particle injection approach.

### 2.2.2 Particle Injection

For steady injection of a particle, the following parameters are required: the location/coordinates \((x_p, y_p, z_p)\), the mass flow rate of the injection \((\dot{m}_p)\), the initial velocity \(V_p = (u_p, v_p, w_p)\), the temperature \((T_p)\), and the particle diameter \((d)\). For transient injections, the mass flow rate parameter is replaced with parcel mass \((m_{pc})\), the number of particles in a parcel \((n_p)\) and two additional parameters; the particle timestep \(\Delta t_p\) and flow-timestep \(\Delta t_{fp}\), one of which must be selected for particle treatment. There are also several methods to inject particles into a domain, including (but not limited to): Single, Group, Cone, Surface, and File (as defined in ANSYS FLUENT).
As the name implies, the *Single* injection method only injects particles at a single location (specified anywhere inside the domain). In contrast, the *Group* and *Cone* injection methods allow for specifying the number of particles injected. For *Group* injection, the injections are spaced uniformly between two specified points, resulting in the specified number of particles being injected along a single straight line. *Cone* injection injects all the particles at a single location, but varies the direction of the initial velocity to disperse the particles downstream. The *Surface* injection method permits selection of a surface through which the particles are injected into the domain. By default, the injection locations are set to the centre of each cell that lies on the selected surface. Lastly, the *File* injection method allows the user to specify all the injection parameters directly and is, therefore, the most versatile. Using the *File* method, any number of particles can be injected at any position with any initial velocity, temperature, diameter, and mass flow rate.

For particle separation devices that take in a dilute, dispersed “cloud” of particles, both the *Surface* and *File* injection approaches are appealing to consider. The surface injection method is commonly used in modelling suspended particles flowing in a continuous fluid phase as it distributes injected particles across a plane as opposed to a single point or straight line. Furthermore, the surface injection method can inject particles of a uniform size or of a specified size distribution – Rosin-Rammler, for example. However, in either case, particles are injected at the cell centres of the specified surface and thus, the number and position of injection locations is fixed by the surface mesh. This introduces the obvious issue that the grid distribution on the surface also defines the spatial distribution of the particle cloud. Also, noting that it is common practice to refine the mesh near the bounding surfaces (edges) of a flow field, wall refinement has the effect of concentrating particle injections near the walls of the injection surface, which is not generally desired nor physically correct. Figure 2.1 shows a typical rectangular inlet surface with edge grid refinement and resulting in the particle distribution shown, illustrated for the case of a generic particle size distribution.
This connection between surface mesh and particle injection location can be overcome using File injection, which, as noted above, permits complete control over the injection number and location, particle size distribution of the injections and all other particle injection parameters.

2.2.3 The Injection File

In File injection, a distribution pattern, defined using the location/coordinates \((x_p, y_p, z_p)\) is generated using a specified number of injections, which need not have any correspondence to the number or location of cells on the injection surface. Each injection is given a uniformly distributed, random coordinate that falls on a surface plane parallel to the continuous phase inlet, offset by a small amount (say 0.001m) to ensure that all the particles are injected into the domain.

Each injection is also given a random particle diameter \(d\), within a desired particle size range, while at the same time ensuring that the distribution pattern matches the size distribution of the powder being modelled. Though other options are equally applicable,
the particle size distribution used throughout this Chapter is for a fine artificial powder described by a Roslin-Rammler distribution [21] with a particle density of 1550 kg m⁻³. The fraction by number of particles \( Y(d_l) \) in the powder with diameters less than a specified particle diameter \( d_l \) is given by:

\[
Y(d_l) = 1 - \exp\left(-\left(\frac{d_l}{\bar{d}}\right)^n\right)
\]  

(2.1)

The mean diameter \( \bar{d} \) and spread parameter \( n \) are set to 2.5μm and 3.5, respectively. The mass fraction of particles \( M(d_l) \) in the powder with a diameter less than \( d_l \) can then be obtained from:

\[
M(d_l) = \sum_{d=d_{\text{min}}}^{d_l} \frac{(Y(d) - Y(d-1))d^3}{\sum_{d=d_{\text{min}}}^{\infty} (Y(d) - Y(d-1))d^3}
\]  

(2.2)

The resulting cumulative size distribution of the fine artificial powder used in this study is shown in Fig 2.2.

By discretizing the cumulative particle size curve, a particle size can be coupled to a range of the cumulative fraction undersize, shown by Fig. 2.3. Then, a uniformly distributed random-generated number between zero and unity can represent the cumulative fraction undersize. The particle size in which the random number falls is selected for the injection particle diameter. When desired, a uniform particle size can also be achieved by setting all the injections to the desired particle diameter.

With the diameter of each injection in the distribution pattern known, the total volume of injections can be calculated (summing the volume of each spherical injection, \( V_{\text{INJ}} \)).
Multiplying the desired total mass flow rate \( \dot{m}_{\text{total}} \) of powder by the ratio of the specific injection volume to the total injected volume yields the mass flow rate for the specific injection:

\[
\dot{m}_{p} = \dot{m}_{\text{total}} \frac{V_{INJ}}{\Sigma V_{INJ}} 
\]  

(2.3)
The last required parameters ($u_p$, $v_p$, $w_p$, and $T_p$) can also be set, but for the purpose of this study are held constant for all injections in the distribution pattern. For all simulations presented herein, $T_p$ is set to 300 K (the air flow temperature) and the particle velocity is set to the mean inlet flow velocity and is assumed normal to the injection surface.

A MATLAB™ script was written to automate the process of creating injection files for the distribution patterns simulated. Figure 2.4 shows examples of three different distribution patterns of particles characterized by the particle size range and distribution given in Fig. 2.3 and applied to the rectangular surface mesh given in Fig. 2.1. The left and middle distribution patterns in Fig. 2.4 each have 2000 randomly located injection points, while the right pattern has 5000. Comparing the images confirms (qualitatively) that both location and particle size are randomly prescribed on the surface within the defined surface and particle size range and distribution.

If a single injection file is provided for a steady DPM simulation, the discrete phase is characterized by continuous injection of the same randomized particle distribution, and the approach is referred to as Steady Injection. Running this process is computationally inexpensive because all the particles flow through a single set of particle paths that need only be computed once. Though this is an improvement over simple surface injection, this approach is still not physically realistic since a truly random cloud has random particles released from (different) random locations at different instants in time.

### 2.2.4 Randomized Transient (Unsteady) Injection

The general approach for introducing a spatially and temporally randomized particle cloud into a stationary, time-periodic or non-stationary flow is to conduct transient injection in which a new randomized distribution of injection locations and particle sizes is applied at each time-step of the solution of the continuous phase, thereby also generating a new set of particle paths at every time step of the solution. By this approach, temporal randomness is introduced at the resolution of the continuous phase simulation, as dictated by the time-step size of the continuous phase. Though this imposes a limit on the number of different particle injection “frames”, the time-step size of the Eulerian solver is usually much smaller
than it needs to be for characterizing temporal randomness of the particle cloud. To conduct transient simulations, the injection file must be set up to contain an injection frame for each time-step of the Eulerian solver, which for fine temporal resolution, can require tens of thousands of frames. In practice, however, the injection file can be set to repeat after a specified flow time has passed such that the injection simulation can continue to some time scale of the device using a much smaller number of repeating frames, thereby avoiding the need for a massive input file. While this last point addresses the issue of injection file size for a given transient simulation, the fact remains that the Eulerian solver is required to run with a new injection frame at each time-step, thereby also requiring the calculation of new particle paths at each time-step. To this end, while the transient approach may yield the most physical characterization of the particle flow, the time required for simulations is extraordinarily high compared to that described in section 2.2.3; literally a comparison of hours/days for randomized transient simulations compared to seconds/minutes for steady (random) injection.

Figure 2.4: Three distribution patterns generated using a MATLAB script that prescribes a random surface location and a random particle size within a specified range and preserving the specified size distribution. The left and middle patterns each have 2000 randomly assigned injection points of different random injection positions and particle size, while the right pattern has 5000 random injection points.
2.2.5 Aggregate Steady Random Particle Injection (A-STERP)

An approach is proposed in this Chapter that achieves the random temporal resolution of the particle cloud using the file injection method, whilst achieving the accuracy of the randomized-transient injection approach described in section 2.2.4. In essence, by the proposed approach, we take advantage of the benefits of the fast processing time of steady particle tracking, while introducing temporal randomness by conducting a number of sequential steady simulations with different injection files defined with randomized distribution points and particle sizes. The results for the temporal cloud resolution are achieved by taking the aggregate of the steady DPM simulations, which preserve the particle distribution, the mass flow rate and, as will be shown in the validation section, the quality of results of the randomized transient approach. The so-called Aggregate STEady Random Particle (A-STERP) injection approach simulates several distribution patterns sequentially (without running the continuous phase solver) to ensure that the DPM results are independent of the injection position of the particle. Distribution patterns are generated using the approach described in section 2.2.3 and simulated until the cumulative average of a given quantity like collection efficiency achieves a steady value. A flow chart detailing the A-STERP injection process is shown in Fig. 2.5.

The A-STERP approach is not mathematically different from steady injection or transient injection and utilizes the tools available in ANSYS FLUENT (and other commercially available CFD software) without modification. Application of A-STERP is extremely straightforward for stationary flows wherein the continuous phase can be described by a single frame in time. A-STERP is also shown in section 2.4 to be applicable to time-periodic and non-stationary flows, provided the continuous phase is described by an approach that accurately captures the flow. A-STERP is described herein for three different applications of particle separation, each with increasing complexity. The
geometric models and the numerical approach are described briefly such that results of this work can be easily replicated in future studies.

2.3 Numerical Simulations

2.3.1 Computing Hardware

All simulations conducted in the this Chapter were performed on the same workstation to ensure that any comparison of computational time is the result of the methods used and not the limitations of the workstation. In addition, while the simulations were being conducted, no other programs (except for core processes that are required to run the workstation) were running. The workstation used had an Intel® Core™ i9-10900 CPU that can operate at 2.80GHz with 20 logical processors (10 dual cores). However only 19 of the logical
processors were dedicated to performing the simulations. The workstation also had 64GB of RAM available for the simulations.

2.3.2 Geometric Models and Mesh Generation Approach

Three collection devices of various complexity (shown in Fig. 2.6) were selected to investigate the effect of the injection method used in DPM and to validate the A-STERP approach for particle injection. The first device is a generic rectangular chamber with offset rectangular inlet and outlet ducts. The second device is the baffled preseparator from the work of Wang et al. [23] and the third is the cyclone considered in the work of Zhou et al. [1]. All three devices were simulated as particle separation devices; thus, particles (discrete phase) are injected at the inlet with the inflow of air (continuous phase). Furthermore, the separation process is less than 100% effective for all three devices and small particles escape with the “cleaned” continuous phase. The two performance parameters that are used to quantify the particle separation effectiveness of the devices are the Total Collection Efficiency (TCE) and the Fractional Collection Efficiency (FCE), which are defined as:

\[
TCE = \frac{M_c}{M} \tag{2.4}
\]

\[
FCE(d) = \frac{m_c(d)}{m(d)} \tag{2.5}
\]

where \(M_c\) is the total collected mass and \(m_c(d)\) are the collected mass for particles having a diameter \(d\). Similarly, \(M\) is the total mass injected and \(m(d)\) is the mass injected of particles with a diameter of \(d\).

The generic rectangular chamber (Fig. 2.6a) was created to test various injection and tracking methods on a simple domain that requires little computational time. The contaminated continuous phase enters this chamber in the lower duct. The simple chamber is then assumed to collect any particles (discrete phase) that make contact with any wall. The “cleaned” continuous phase with any un-separated discrete phase exits through the upper duct.
Figure 2.6: Geometric models of the generic rectangular chamber (a), the baffled pre-separator (b), and cyclone (c). All dimensions are in mm. Red areas represent area/position of interest for grid independence criteria. In all cases, the Cartesian coordinate system has $x$ pointing into the inlet, with the $y$-$z$ plane oriented on the inlet face.
The baffled pre-separator from the work of Wang et al. [23] (Fig. 2.6b) was selected to validate the injection and tracking methods on an actual particle separation device. The contaminated continuous phase enters the pre-separator through the circular pipe denoted as “inlet” in the figure. The inlet flow comprising the contaminated continuous phase impinges on a baffle, which redirects the flow downwards into the chamber. While the bulk of the discrete phase falls to the bottom of the chamber, the “cleaned” continuous phase proceeds upwards and exits through the rectangular, off-centred outlet duct.

The cyclone particle separator from the work of Zhou et al. [1] (Fig. 2.6c) was selected as a final case to demonstrate the efficacy of the proposed particle injection and tracking approach in a large-scale simulation. While Zhou et al. [1] added a rotating classifier to the cyclone (making it a dynamic cyclone), the classifier is neglected in the present study, resulting in the modelling of a conventional cyclone. In practice, the bottom of the conical section is connected to a collection bin (not shown in Fig. 2.6c) where particles removed by the cyclone accumulate. As it is common to use a vortex stabilizer at the top of the collection bin to prevent the core vortex from entering the bin, the entrance to the bin can be considered as a wall that terminates particle paths, making it unnecessary to model the bin itself. Thus, to simplify the geometry, the collection bin is not modelled.

The computational meshes for the three devices were generated using Pointwise V18.4R4. The geometric models were divided into topological blocks, which were then subdivided into hexahedral cells. This meshing approach results in structured meshes that are computationally efficient. The generic rectangular chamber was divided into 12 topologically rectangular blocks, while the pre-separator required 40 rectangular blocks, with an additional 15 blocks constructed as a “butterfly” grid for the inlet pipe. The baffle in the pre-separator is treated as a thin feature and thus has no thickness and requires no blocking. The cyclone is the most complex geometric model and required a total of 89 blocks, wherein because of the circular shape, all but one (the inlet block), were constructed as butterfly grids. Unlike the baffle in the pre-separator, the vortex finder in the cyclone was not modeled as a thin feature, thus requiring a block structure. For the chamber and the cyclone, cell refinement was applied to all blocks, with a surface representing a physical wall. The thickness (perpendicular to a wall) of the cells on a wall was set to 0.001m. The
pre-separator was a smaller device, compared to the chamber or cyclone, thus using a uniform cell distribution resulted in the cells adjacent to walls being less than 0.001m thick. While further details on the grid development are omitted for brevity, this short description of the blocking approach is included to assure the reader that the computational meshes were of similar high-quality for all devices considered.

2.3.3 Continuous Phase

ANSYS FLUENT 20-R1 was used to simulate the continuous phase flow through the three devices. The numerical setup was identical for the chamber and the pre-separator devices. Both were treated as transient simulations with a time step of 0.001s. The standard $k$-$\epsilon$ turbulence closure model [21, 24] was used, and the near-wall treatment was set to enhanced wall treatment. For the cyclone device, the Reynolds Stress Model (RSM) was used to model turbulence, consistent with literature on simulating cyclone particle separators [21, 25-27]. One issue with using the RSM (7-equation model) is the additional computational resources it requires, compared to a two-equation model like $k$-$\epsilon$. However, the additional computational resources were reduced by initially using the $k$-$\epsilon$ model to develop the bulk flow and then restarting the simulation with the RSM model to achieve higher accuracy. In this case, the $k$-$\epsilon$ model is treated as steady, but the simulation approach is switched to transient when using RSM. A time step of 0.001s for the RSM simulation was employed as it ensured stable convergence.

Second-order upwinding was used to model advection in all transport equations, and second-order discretization was used for pressure. An exception was the advection in momentum for the cyclone device, which was set to QUICK as this ensured the numerical setup was similar to Zhou et al. [1]. Pressure-velocity coupling was maintained using the SIMPLEC scheme and all of the equations solved for the continuous phase were forced to converge on each time step achieving globally scaled residuals of less than $10^{-4}$.

2.3.3.1 Boundary Conditions

The boundary conditions for the continuous phase of all three devices are similar, but with differences at the inlets and outlets. Near-wall treatment was set to the enhanced wall treatment for all simulations regardless of turbulence closure model, as recommended in
previous literature [22]. A uniform (plug flow) velocity boundary condition was applied to the inlet surfaces of all three devices (1m$^{-1}$ for the chamber, 17m$^{-1}$ for the pre-separator, and 8m$^{-1}$ for the cyclone). The default FLUENT values for turbulence intensity and turbulent viscosity ratios were used for the generic chamber and the pre-separator. For the cyclone, the turbulent kinetic energy and dissipation rate were specified as 0.18m$^{2}$s$^{-2}$ and 3.75m$^{2}$s$^{-3}$, respectively, using the method presented by Zhou et al. [1]. The outlet boundary condition for both the chamber and the pre-separator was set as an outflow, while for the cyclone, a pressure outlet boundary condition was specified with a gauge pressure of 0Pa. As the continuous phase is considered incompressible, the actual value of the specified pressure is irrelevant and does not affect the continuous phase or the discrete phase motion.

2.3.3.2 Grid Independence

The model settings used for solving the continuous phase were tested on several meshes for each device until the monitored data no longer changed (within 1%) with an increase in the mesh density. The percent change, or Grid Index ($GI$), is defined as:

$$GI = \left| \frac{C_{grid1} - C_{grid2}}{C_{grid2}} \right| \times 100\%$$

where $C$ is a criterion of interest, and the subscripts $grid1$ and $grid2$ represent subsequent coarse and fine mesh levels, respectively. Two criteria were evaluated to ensure the continuous phase results were grid-independent. The first criterion was the pressure drop across the device. The second criterion for the chamber and pre-separator were the area-average velocity magnitude at the outlet and the velocity magnitude at a central point inside the domain, respectively (shown in Fig. 2.6b). The second criterion for the cyclone was the area-average tangential velocity that occurs on a plane that is normal to the cyclone axis of symmetry located where the cylindrical section meets the conical section of the cyclone (shown in Fig. 2.6c). The grid-independence test revealed that the continuous phase model requires a mesh of 78,792 cells for the chamber, 56,224 cells for the pre-separator, and 3,012,570 cells for the cyclone. Table 2.1 shows the summary of the grid-independence tests for all devices.
Table 2.1: Grid independence results for the Generic Chamber, the Baffled Pre-Separator and the Cyclone shown in Fig. 2.5.

<table>
<thead>
<tr>
<th>Device</th>
<th>Grid</th>
<th>Cells</th>
<th>Pressure Drop Condition [Pa]</th>
<th>GI for the Pressure Drop Condition [%]</th>
<th>Velocity Condition [ms⁻¹]</th>
<th>GI for the Velocity Condition [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Generic Chamber</td>
<td>1</td>
<td>78,792</td>
<td>1.23</td>
<td>-</td>
<td>1.00</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>160986</td>
<td>1.22</td>
<td>0.82%</td>
<td>1.00</td>
<td>0.05%</td>
</tr>
<tr>
<td>Baffled Pre-Separator</td>
<td>1</td>
<td>26,116</td>
<td>327</td>
<td>-</td>
<td>12.21</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>56,224</td>
<td>364</td>
<td>10.16%</td>
<td>13.48</td>
<td>9.42%</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>114,629</td>
<td>367</td>
<td>0.82%</td>
<td>13.61</td>
<td>0.96%</td>
</tr>
<tr>
<td>Cyclone</td>
<td>1</td>
<td>549,760</td>
<td>216</td>
<td>-</td>
<td>12.59</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>920,112</td>
<td>227</td>
<td>4.85%</td>
<td>12.57</td>
<td>0.16%</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>1,506,974</td>
<td>230</td>
<td>1.30%</td>
<td>12.78</td>
<td>1.64%</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>3,051,329</td>
<td>246</td>
<td>6.50%</td>
<td>12.80</td>
<td>0.16%</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>4,648,140</td>
<td>248</td>
<td>0.81%</td>
<td>12.86</td>
<td>0.47%</td>
</tr>
</tbody>
</table>

2.3.4 Discrete Phase Modeling

As this Chapter focuses primarily on how the injection of particles into a domain affects the simulation of particle clouds, the FLUENT DPM settings for tracking the particles are not critical, provided the same settings are used for all the injection approaches being compared. In other words, while the DPM settings can change the outcome, they do not influence the approach adopted to reach the outcome. To this end, the DPM Physical Models options such as Saffman Lift and Particle Collisions are all turned off, as these settings also do not influence the injection method. Additionally, the discrete phase does not affect the continuous phase for the applications being considered, since the particles are small and the discrete phase makes up a very small volume fraction of the domain. For all the simulations, the spherical drag model was used and rotation of the particles is not considered.

2.3.4.1 DPM Boundary Conditions

Although the inlet boundary condition for the discrete phase is not critical, to ensure that no particles are lost through the inlet, the DPM boundary condition at the inlet was set in all cases to “reflect” (elastic). The baffle in the pre-separator and the cyclone walls also use
a “reflect” DPM condition with the coefficient of restitution set to 1.0. The “escape” condition was used at the outlet of each device, as any particles that reach these boundaries would escape with the continuous phase. The walls of the generic chamber and the baffled pre-separator, as well as the base of the cyclone, had “trap” conditions imposed, because particles that encounter a wall in the generic chamber and baffled pre-separator or enter the cyclone bin are considered collected and thus, tracking of the particle can be terminated. Using the “trap” condition at the cyclone bin instead of the “escape” condition allows for differentiation of the particles terminated at the cyclone outlet from those terminated at collection locations. All simulations use the same particle size distribution with the same discrete particle sizes, as described in section 2.2.3. This resulted in particles being injected with a diameter of 1-6.5μm with 0.5μm increments, yielding 12 different particle sizes. The total mass flow rate of the powder was also consistent at 0.001kgs\(^{-1}\) for all injection methods on all models.

### 2.4 Results and Discussion

For each device given in Fig. 2.6, simulations are first presented to qualitatively illustrate the flow field of the continuous phase under the conditions described in section 2.3, followed by results of DPM tracking obtained using the randomized transient approach, the simple surface injection approach and the A-STERP approach. In every case, the converged continuous phase flow field shown is the starting point for DPM simulations, regardless of the injection method, and in each case this continuous flow field is predicted as stationary when the inlet condition is steady. In section 2.4.4, periodicity is introduced in the baffled pre-separator by imposition of a time-varying boundary condition at the inlet. This case is used to demonstrate application of A-STERP for time-periodic flows and, by extension, to non-stationary flows.

As noted in section 2.2, for randomized transient injection, particles are released from the inlet surface using injection files generated for each time step of the DPM simulation and run simultaneously to the point of steady-state convergence of the particle model, wherein steady-state is achieved when the “loading” of the domain is unchanging. The steady-state loading refers to the point where the mass flux of particles injected at the inlet surface of the domain is balanced by the sum of the mass flux of particles at the outlet (escaped) and
the mass flux of particles separated by the device (trapped). Once steady-state loading is achieved, two additional seconds of flow simulation are run and the TCE is calculated and used as a criterion for assessing the influence of the number of random injection locations. For the randomized transient DPM simulations, the time-step size for the Euler solver is assumed to be the same as that for the Lagrangian solver, and an injection frame is provided for each time-step.

For A-STERP calculations, both the number of random injection locations and the number of randomized injection files are tested for their sensitivity on the results of TCE. Note that the mass flow rate of particles injected does not change as a function of the number of injection points or the number of injection files, since the mass flowrate of particles in each trajectory path is scaled to give the same mass flow rate for each case simulated while also preserving the prescribed distribution of particle sizes. Ultimately, it is the DPM results for TCE and FCE that are independent of the number of injection locations and injection frames that define the accuracy of the approach. These results are compared to similar results obtained using simple surface injection in terms of accuracy and time required to achieve the simulated DPM result.

2.4.1 The Generic Chamber
The three-dimensional flow produced by the uniform inlet condition is illustrated using fluid path lines in Fig. 2.7. The figure shows that the flow takes a straight path through the inlet duct (lower bottom duct), but disperses quickly inside the chamber due to the sudden expansion and impingement on the back wall. Inside the chamber, the flow forms a set of swirling motions that fill the rectangular chamber, driving the bulk flow upward towards the outlet duct. The flow gathers at the outlet duct and leaves the domain with higher velocity airflow concentrated towards the top of the exit duct. Though seemingly chaotic, the swirling structure of the air flow throughout the chamber is predicted by ANSYS FLUENT as stationary and thus, can be represented by a single frame (in time).

For the randomized transient injection simulations of the generic chamber, 10,000 distribution patterns (frames) were generated and written to an injection file to achieve up to 10s of injections using a time step of 0.001s. Though fewer frames could have been
used in a repeating cycle (as noted in section 2.2.4), a different frame for each time step was used here to create the most randomized injection that could be achieved with the specified continuous phase time step. The DPM simulation showed that steady-state loading of the domain to within 0.02% was achieved after approximately 5 seconds of simulation time (i.e. 5000 time steps). From this point, the particles injected and terminated by escaping and trapping were sampled for 2 additional seconds to calculate the TCE and FCE.

To ensure that the particle cloud was not dependent on the specified frame structure in the injection file, a systematic study was conducted whereby the number of random locations and random particle sizes (within the range and distribution specified) was increased until the predicted TCE was unchanging. The number of random injection locations tested was increased in increments of 1000 from 1000 to 5000 and are shown by a solid line with solid triangular markers in Fig. 2.8. The figure shows that the TCE is under-predicted for 1000 injection points, but then quickly rises to near the converged level after doubling, and then
converges to approximately 6.1% after 3000-4000 injection points. In terms of wall-clock time, the transient DPM simulations took approximately 4-6 hours each to reach steady-state loading for each test, depending on the number of injection points.

For simple surface injection, particles are released from the cell centres of the injection surface, but still preserving the mass flux of particles and the specified particle distribution. For the generic chamber, the inlet surface is comprised of 300 non-uniform rectangular cells from which 3600 total particles are released to achieve the mass flux using the prescribed particle distribution. This results in a TCE of 2.2%, which is considerably lower than that predicted by the more realistic, randomized transient simulation described above. However, the simple surface injection result required only a few seconds to compute.

Using the A-STERP approach for particle injection, a converged solution is achieved when the predicted TCE is independent of the number of randomized injection locations (i.e. injection files or frames) and the number of sequential simulations. To this end, a study was first conducted using 30 different random injection files, wherein the number of injection points in each file was changed from 500 to 5000 in increments of 500. The results of this study are given in Fig. 2.8 using a solid line and open square markers to facilitate comparison with the randomized transient results. The figure shows that the DPM
solution for TCE converges to approximately 5.9% within 3500 injection points, which is similar to the number required for the randomized transient simulations. Figure 2.9 shows the results of a study that assesses the number of randomized injection files required. For this study, the number of random injection points per frame is fixed at 5000 and only the location of the points and the particle sizes are random. This plot shows that the cumulative average TCE converges to approximately 5.9% after only 9 frames, which suggests that the huge number of time levels used in the fully-transient approach (1000 per second of simulated time) is far more than necessary to capture the temporal randomness of the injected particle cloud. The individual points given as solid circles in Fig. 2.9 show the TCE results of individual injection patterns and illustrate clearly why the aggregate approach based on several injection patterns is important in order to achieve a constant, steady-state result.

Finally, Fig. 2.10 compares the results of FCE for the three injection methods considered. A-STERP results are presented for a case run with 20 injection files each with 3500 randomized injection locations. This figure shows agreement to within a maximum of 9% between the fully-transient and A-STERP approaches, but under-prediction of up to 70%.

![Figure 2.9](image-url)

Figure 2.9: Plot showing the specific total collection efficiency (TCE) and the cumulative average (aggregate) TCE as a function of the number of distribution patterns as predicted using the A-STERP approach to injection modelling. Each distribution pattern has 5000 randomly assigned injection locations with random-sized particles within the specified range and distribution of particles.
using the surface injection method.

### 2.4.2 The Baffled Pre-separator

The three-dimensional flow produced by the steady, uniform inlet condition is illustrated in Fig. 2.11 using flow path lines. The figure shows that the air entering through the circular inlet duct impinges on the baffle where it is forced to disperse, resulting in a seemingly chaotic swirling pattern that fills the rectangular chamber, driving the flow upward towards the offset rectangular outlet duct. The flow velocity in the lower part of the chamber is seen to be slow relative to that observed on the baffle and at the inlet and outlet ducts, to facilitate separation and collection of injected particles and to avoid re-entrainment of collected particles. At the exit of the chamber, the flow is highly non-uniform, but with no backflow predicted. Once again, the flow using a steady uniform inlet condition is predicted by ANSYS FLUENT to be stationary and can be represented by a single frame in time.

Following the same approach described in section 2.4.1 for the generic chamber, 10,000
distribution patterns were generated and written to an injection file to achieve up to 10s of injections using a time step of 0.001s for fully transient injection simulations of the baffled pre-separator. In this case, the DPM simulation required only 2 seconds of simulation time (i.e. 2000 time steps) to achieve steady-state loading of the domain to within 0.03%. From this point, the particles injected and terminated by escaping and trapping were sampled for 2 additional seconds to calculate the TCE and FCE. The results of this study are given in Fig. 2.12, which shows that the converged TCE for the chamber is 82.3% and is achieved with as few as 1000 injection points on each pattern. In terms of wall-clock time, the fully-transient DPM simulations took approximately 4-6 hours to reach steady mass loading for each test depending on the specified number of injection points.

For simple surface injection, the inlet surface of the baffled pre-separator is comprised of 576 non-uniform rectangular cells and 6912 total particles are released to achieve the mass flux using the prescribed particle distribution. This results in a TCE of 81.1%, which is
only 1.2% lower than that predicted by the fully-transient simulation described above. However, the simple surface injection result required only a few seconds to compute on the stationary continuous phase flow field.

Using the A-STERP approach for surface injection, the required number of injection patterns was determined to be 15 following the approach described for the generic chamber in section 2.4.1. To determine the appropriate number of injection points, 15 injection files were used, and the number of injection points in each frame was changed from 500 to 5000 in increments of 500. The results of this study are given in Fig. 2.12 using a solid line and open square markers. The figure shows that the DPM solution for TCE converges to approximately 82.3% for more than 3500 injection points, which, again, is similar to the number required for the randomized transient simulations. The figure also shows that the TCE prediction using the A-STERP method is essentially identical to that predicted by the fully-transient approach, except requiring only minutes to run, since the aggregate is comprised of the cumulative average of only 15 sequential steady-state DPM simulations using different random injection files.

Figure 2.13 compares the results of FCE for the three injection methods considered. This

![Figure 2.12: Total collection efficiency (TCE) for the baffled pre-separator shown as a function of the number of randomly located injection points, which also include random-sized particles within the specified range and distribution.](image)
The figure shows that while surface injection is only accurate for a medium particle size (3µm), the A-STERP results effectively mirror the randomized transient results to within a fraction of a percent across the full range of particle sizes.

### 2.4.3 The Cyclone

The three-dimensional flow produced by the uniform inflow of air is illustrated using flow path lines in Fig. 2.14. The figure shows that the flow entering the cyclone through the tangential inlet develops a well-organized vortex between the wall and the vortex finder. The swirling flow travels downwards in a helical motion while being driven towards the cyclone centreline by the conical barrel, which causes the tangential flow to speed up. At the bottom, the vertical flow direction reverses, while maintaining a high-speed swirling motion, and progresses upwards through the core of the cyclone towards the vortex finder and the exit duct. In contrast to the generic chamber and the baffled pre-separator, the cyclone flow is organized throughout and has no secondary swirling structures. The flow is predicted by ANSYS FLUENT to be stationary for the steady inlet condition imposed.

Once again, following the approach described in section 2.4.1 for the generic chamber, 10,000 distribution patterns were generated and stored in an injection file to achieve up to

![Graph](image_url)

Figure 2.13: Predicted fractional collection efficiency (FCE) obtained for different methods of injecting particles into the baffled pre-separator. A-STERP results are shown for 15 frames each with 3500 injection points.
10s of injections using a time step of 0.001s for fully transient injection simulations of the cyclone (the file was set to repeat if more than 10,000 time step are required). In this case, due to the lengthy simulation time required, simulations were only conducted using frames with 1000 injection points. To this end, the cyclone case was included here to prove the point that randomized transient DPM simulations are not always practical, despite their accuracy, and that a less computationally intensive approach is necessary. The DPM simulation for the cyclone required 15 seconds of simulation time (i.e. 15,000 time steps) to achieve steady-state loading of the domain to within 0.02%, which required nearly 6 weeks (1000 hours) of dedicated computational time. As with the other cases, the particles injected and terminated by escaping and trapping were then sampled for 2 additional seconds to calculate the TCE and FCE for this single DPM simulation. Though not necessarily independent of the number of injection points, the cyclone TCE was 71.2%.

For simple surface injection, the inlet surface of the cyclone is comprised of 738 non-uniform rectangular cells and 8856 total particles are released to achieve the mass flux using the prescribed particle distribution. This results in a TCE of 60.3%, which is 10.9% lower than that predicted by the fully-transient simulation described above. However, the

Figure 2.14: Illustration of the airflow in the cyclone using flow path lines with color indicating velocity magnitude.
simple surface injection result required only a few minutes to compute.

Applying the A-STERP approach for surface injection resulted in the requirement of 25 injection files. An independence study was then done to determine the suitable number of randomized injection locations. Once again, the number of injection points in each frame was changed from 500 to 5000 in increments of 500. The results of this study are given in Fig. 2.15 using a solid line and open square markers. The figure shows that the DPM solution for TCE converges to approximately 67.8% for more than 4000 injection points. This compares reasonably well with the single randomized transient case that was produced using 1000 injection locations. Based on results presented in sections 2.4.1 and 2.4.2 for the generic chamber and the baffled pre-separator, respectively, it is safe to conclude that the converged TCE produced using the A-STERP approach is sufficiently accurate. This result was obtained from the cumulative average of 25 sequential steady-state DPM simulations and required only 1-2 hours to produce.

Finally, Fig. 2.16 compares the results of FCE for the three injection methods considered for the cyclone DPM simulations. Though the A-STERP results are close to the randomized transient results, the transient result has not been shown to be injection-location-

![Figure 2.15: Total collection efficiency (TCE) for the cyclone shown for A-STERP as a function of the number of randomly located injection points and random particle size distribution, which also include random-sized particles within the specified range and distribution.](image)
independent. To this end, the most important result from this figure is the difference between the simple surface injection approach and A-STERP. Though the trend is similar across the range of particle sizes, the surface injection predicts FCE values that are as much as 15% lower than A-STERP.

2.4.4 A-STERP applied to a time-varying flow

Though A-STERP is performed using the steady particle tracking method, an additional test will show that it can also be used when the flow field varies periodically with time and, by extension, when the flow is non-stationary. To demonstrate this, a time-periodic flow field is introduced into the baffled pre-separator by imposing a sinusoidal inlet condition whereby the uniform inlet velocity has a frequency of 10Hz, an amplitude of $2.5\text{ms}^{-1}$, and an offset of $17.5\text{ms}^{-1}$. The inlet velocity and the velocity at the monitoring point indicated by a red dot in Fig. 2.6b are plotted as a function of time in Fig. 2.17. Though not shown, the flow field produced by the sinusoidal inlet condition produces a similar complex flow field to that shown in Fig. 2.11, except pulsing at the imposed frequency and producing a time-periodic flow throughout the domain. As shown in Fig. 2.17, the velocity at the

![Figure 2.16: Predicted fractional collection efficiency (FCE) obtained for different methods of injecting particles into the cyclone. A-STERP results for 25 frames with 4000 injection points.](image)
monitoring point has a slightly lower amplitude (2.0ms\(^{-1}\)), an offset of 13.2 ms\(^{-1}\), and lags the inlet velocity by 0.006s. Note that further details of the time-varying flow field are not important as it is simply produced to demonstrate the application of A-STERP.

As before, the fully transient DPM method is applied first to determine the best estimate of the TCE with which to compare A-STERP. The number of injection points used for the fully transient injection method was, again, tested to ensure that the predicted TCE is independent of injection file parameters. The time-varying flow required 4000 randomized injection points in each distribution pattern, which is an increase of 500 from the stationary flow. The fully transient injection method was run using 0.001s time steps until steady loading was achieved (approx. 5000 time steps) and then for an additional flow period to calculate a TCE of 82.5%. Because the fully transient DPM requires solving the continuous phase regardless of whether the flow is stationary, time-periodic or non-stationary, the time required for the time-periodic flow was the same that for the stationary flow; i.e. 4-6 hours, depending on the number of randomized injection points per frame of the injection file.

Figure 2.17: Inlet and monitoring point velocity with time for the time-periodic flow in the baffled pre-separator.
Since A-STERP takes advantage of not running the continuous phase solver, the time-periodic flow field needs to be accounted for in a manner that, when subjected to particle injection, produces accurate estimates of TCE. Two approaches were tested; one where the periodic flow was described by a single frame representing the time-average, and a second approach where one period of the flow was represented by a number of frozen frames within a period. Using the first approach, the time-averaged flow field was calculated using successive time steps that span one or more even numbers of periods (1-5 periods was seen to produce a nearly identical time-averaged flow field). A-STERP was then applied to the time-averaged flow field. Independence testing in terms of the number of injection patterns and locations showed that 18 patterns each having 3500 injection points resulted in a TCE of 82.8%, which is nearly identical to the fully transient DPM prediction. Including the calculation to produce the time-averaged flow field, this simulation took only a few minutes.

Using the second approach for characterizing the time-periodic flow field, a single period was discretized into 4 time frames, then 8 time frames, then 16, and so on. A-STERP was then carried out on the individual time frames using 18 patterns each having 3500 injection points. The TCE was then calculated from the average of the TCEs predicted from the individual time frames, and was seen to converge to 82.3% when 8 or more time frames were used to characterize one period of the flow. Application of this approach required only a few minutes per time frame used.

As both of the approaches using A-STERP on a time-varying flow are seen to produce equally accurate estimates of TCE, the time-averaging approach is favoured due to its simplicity. The accuracy of the time-averaging approach to represent the time-periodic flow field also implies that A-STERP can be used for unsteady flows that are more generally non-stationary, which makes the method even more appealing. The more important outcome is that A-STERP can be used to produce accurate simulations of particle cloud injection and tracking/capture, which means that simulations can be produced in a small fraction of the time required for fully-transient injection, which is the seemingly obvious choice for time-periodic and non-stationary flows.
2.4.5 Discussion of Results

The preceding sections show very similar results for the randomized transient and A-STERP approaches, while the simple surface injection approach was shown to be the least accurate. In terms of running the cases, the surface injection method requires very little setup work because ANSYS FLUENT (and other commercial CFD software) is set up to make defining a particle size distribution straightforward. This simple distribution is run for injection locations that are collocated with cell centres on the injection surface and a single DPM simulation then computes the particle paths and particle fluxes in each path.

The A-STERP injection method requires additional time for both the setup and the simulation of the particles, since the injection locations are no longer defined by the grid. In this case, injection files are required for a specified number of random locations on the inlet surface and for a number of different injection patterns. The time required to generate the random distribution patterns is on the order of minutes and depends only on the number of points and patterns required. A-STERP is then run as a sequence of steady-state DPM simulations, each of which take only minutes, and the resulting data are processed by considering the cumulative average of the sequential DPM results for a desired quantity like TCE.

The randomized transient injection method was, by far, the most computationally intensive. Without knowing \textit{a priori} how many randomized injection files are required, the generation of thousands of injection files alone can take hours/days for each device considered, since, by default, one might assume that the number of random injection patterns needs to correspond to the number of time-steps of the transient simulation. However, since this process can be drastically reduced with knowledge of the required number of injection frames, it can be assumed that the time taken to produce input files may not be greater than that for A-STERP. Additional calculations of the baffled pre-separator using 50 repeating frames in the injection file produced results that were nearly identical to those achieved by using a different frame for every time step. Regardless of the number of injection frames required, transient DPM simulations still require hours to weeks to run, depending upon the time required to achieve steady-state loading of the device.
The approximate time requirements for setting up and running the three injection methods on each of the devices is summarized in Table 2.2. Included in the table are summary results of TCE and errors of the A-STERP and Surface injection methods compared to the randomized transient method. Error calculations are not included for the cyclone because the randomized transient result was not established as being independent of the number of injection locations.

The table highlights the enormous differences in computational time for the randomized transient approach compared to both steady-state approaches. The table also illustrates the accuracy of the A-STERP approach compared to simple surface injection for only a modest amount of additional computational time. Further comparisons are made by considering the FCE results given in Figs. 2.10, 2.13 and 2.16. In this case, the Normalized Root Mean Square (NRMS) \( (FCE_{NRMS}) \) defined as:

\[
FCE_{NRMS} = \frac{1}{n_d} \sum_{d=d_{min}}^{d_{max}} \frac{(FCE_{per}(d) - FCE_{act}(d))^2}{TCE_{act}}
\]

is computed, where \( n_d \) is the number of discrete particle sizes injected and the subscript “act” refers to results from the randomized transient simulations. A-STERP gives an almost identical curve to that from the fully transient method for the generic chamber (see Fig. 2.10) resulting in a NRMS of 0.053 (5.3%). For the pre-separator, the FCE curves (see Fig. 2.13) were also very close yielding an NRMS of nearly zero. In contrast, the NRMS for the generic chamber using surface injection was 0.641 (64.1%) and for the baffled pre-separator was 0.013 (1.3%).
Thus, considering the summary given in Table 2.2, combined with the additional comparisons of the FCE, some closing remarks can be made. First, while simple surface injection is fast and can accommodate a random particle distribution, the lack of control on particle injection location and the fact that a single distribution is used to characterize the random particle cloud render the approach unreliable for calculations of TCE and FCE. To capture randomness in both space and time, a more sophisticated approach is required. The A-STERP approach is shown to give results that approach the accuracy of randomized transient DPM simulations, but in a small fraction of the computational time. A-STERP is easy to implement as it takes advantage of tools available in existing commercial CFD software. In this manner, the use of A-STERP makes the characterization of clouds of distributed particles randomized in both space and time and simulated using DPM practical even in large-scale calculations.

Table 2.2: Summary of the approximate times required for conducting DPM simulations using randomized transient injection, surface injection and A-STERP. Included in the table are the summary results of TCE percent difference between and the percent compared to randomized transient injection.

<table>
<thead>
<tr>
<th>Device</th>
<th>Parameter</th>
<th>Randomized Transient Injection</th>
<th>A-STERP</th>
<th>Surface Injection</th>
</tr>
</thead>
<tbody>
<tr>
<td>Generic Chamber</td>
<td>Time Required to Simulate Injection(s)</td>
<td>Days</td>
<td>Minutes</td>
<td>Seconds</td>
</tr>
<tr>
<td></td>
<td>TCE [%]</td>
<td>6.1</td>
<td>5.9</td>
<td>2.2</td>
</tr>
<tr>
<td></td>
<td>PD in TCE [%]</td>
<td>-</td>
<td>3.3</td>
<td>93.9</td>
</tr>
<tr>
<td>Baffled Pre-Separator</td>
<td>Time Required to Simulate Injection(s)</td>
<td>Days</td>
<td>Minutes</td>
<td>Seconds</td>
</tr>
<tr>
<td></td>
<td>TCE [%]</td>
<td>82.3</td>
<td>82.3</td>
<td>81.1</td>
</tr>
<tr>
<td></td>
<td>PD in TCE [%]</td>
<td>-</td>
<td>0.0</td>
<td>1.5</td>
</tr>
<tr>
<td>Cyclone</td>
<td>Time Required to Simulate Injection(s)</td>
<td>Months</td>
<td>Hours</td>
<td>Minutes</td>
</tr>
</tbody>
</table>
2.5 Conclusion
A novel approach called Aggregate STEady Random Particle (A-STERP) injection has been introduced, which takes advantage of the short computing times required for steady DPM simulations, while introducing temporal randomness by conducting sequential steady DPM simulations on different injection files containing random injection locations and particle sizes. Results for the TCE are derived by this approach from the cumulative (aggregate) average of the sequential DPM simulations and are shown to converge to an accurate result within less than 30 different injection frames (files). Validation of the A-STERP approach was done by considering the collection efficiencies of a generic chamber, a baffled pre-separator and a cyclone. In all cases A-STERP produced results that were within a few percent of a fully randomized transient approach, but in a small fraction of the computational time. Although A-STERP has been tested using the CFD software ANSYS FLUENT, the method applies to any CFD software that can model a discrete phase from an injection of particles. Finally, while application of A-STERP is naturally suitable to stationary flows, it was also shown to produce accurate results for a time-periodic flow provided an estimate of the flow field is represented by a single time-averaged frame, or by a series of 8 or more instantaneous frames that capture one period of the flow.

2.6 References


5th Int. Conf. on Sustainable Techniques and Strategies in Urban Water Management, Lyon, France, 2004.


Chapter 3

3 Scale Reduction of Cyclone Particle Separators: Part I – Performance Degradation

3.1 Introduction

Cyclone particle separators are widely used in the pharmaceutical, oil and gas, wood mill, sand cement and plastic industries, among others, as a means to remove pneumatically transported powders [1-9]. The powder being collected can either be of value, as is the case for pharmaceutical production processes using fluidized reactors [6-8, 10], or a potential pollutant, as in cases where the particles are filtered from the airstream to avoid their release into the atmosphere [4, 9]. Cyclone separators are typically required to be tall to achieve the required separation performance, but there is considerable interest in reducing the overall scale of these devices such that they can be located inside production/processing facilities to prevent issues with temperature changes and moisture. While simple scale reduction is known to negatively impact performance, such reductions can potentially be recovered by introduction of dynamic classifiers or carefully designed collection hoppers. The present study explores the impact of vertical scale reduction of a conventional cyclone to provide an understanding of the flow-structure features that need to be modified in the scaled-down cyclone to recover the separation performance of the full-scale cyclone.

A cyclone particle separator (shown in Fig. 3.1) utilizes a tangential inlet to create a vortex contained inside the barrel and cone body of the cyclone device [1, 9, 10]. The prominent tangential velocity of the vortex flow results in a centrifugal force on the particles comprising the contaminating powder [1, 2, 4, 5, 9, 10]. The centrifugal forces move the particles radially away from the neutral axis and towards the cyclone walls. The flow near the cyclone walls has an axial velocity component that is away from the inlet and towards the collection hopper [1, 2, 8, 9]. The particles near the wall travel with the downward flow towards the collection hopper and are finally removed from the bulk flow, with the aid of gravity, and contained in a collection hopper [4, 5, 9, 10].

The performance of a cyclone can be determined based on the pressure drop across the device, the Total Collection Efficiency (TCE) (defined by Eq. 3.1) and the Fractional
Collection Efficiency (FCE) (defined by Eq. 3.2) [2, 3, 11-14].

\[
TCE = \frac{M_c}{M} \quad (3.1)
\]

\[
FCE(d) = \frac{m_c(d)}{m(d)} \quad (3.2)
\]

where the total collected mass is \( M_c \) and the collected mass for particles having a diameter \( d \) is \( m_c(d) \). Similarly, the total mass injected is \( M \) and the mass of injected particles with a diameter of \( d \) is \( m(d) \). The pressure drop across the device is representative of the energy required to operate the device, whereas the TCE and FCE are related to the collection of particles.
Some advantages of using a cyclone particle separator over porous media particle separators (filters) is that the particles are not trapped inside the device and the pressure drop across the device is constant [4, 15-17]. Porous media particle separators result in particles becoming trapped inside the pores, resulting in the particles reducing the passage size over time [15-17]. The reduced passage size results in an increasing pressure drop with continued use of the device [5, 15-17]. The drawback of the cyclone particle separator compared to a porous media separator is that the cyclone does not use a physical barrier, resulting in fine particles, that have low inertia, being allowed to travel with the bulk flow and escape with the cleaned exhaust air flow [3-5, 13]. The porous media separator is a physical barrier that can be designed to collect desired particle sizes and its performance is not dependent on the particle’s inertia [3, 4, 15, 16]. Furthermore, as particles are trapped in the pores and the passageways are reduced, smaller particles are able to be trapped [15, 16].

The vortex present in a cyclone particle separator resembles a Rankine vortex with a free vortex (inviscid fluid rotation) near the walls and a forced vortex (solid body rotation) near the core [1, 2, 12, 18, 19]. The peak tangential velocity component ($U_\theta$) can be several multiples of the inlet velocity ($U_{in}$) [2, 11, 12, 18, 19], while the axial velocity ($U_z$) is of a similar magnitude to the inlet velocity [2, 11, 12, 19], with the radial velocity component ($U_r$) being only a small fraction of the inlet velocity. The axial velocity profile for a given radial line that passes through the neutral axis shows that the axial velocity is negative (towards the collection hopper) near the wall and positive near the neutral axis. However, at the neutral axis the axial velocity reduces in magnitude and, in some cases, reverses and becomes negative again [2, 9, 11, 19]. The peak positive axial velocity occurs near the radial position that corresponds to the vortex finder [2, 9, 11, 19].

It is possible for the flow to leave the cyclone prematurely by escaping under the vortex finder and making far fewer rotations around the cyclone as it travels axially downwards, an effect referred to as shortcutting [2, 11, 18-20]. The shortcutting flow has an adverse effect on the vortex strength below the vortex finder due to the decreased flow rate [20]. The Muschelknautz and Trefz model, presented in [1] incorporates shortcutting flow and estimates the shortcutting to be 10% of the mass flow rate of the continuous phase.
However, numerical simulations have predicted the degree of shortcutting to be more severe than the 10% proposed by the Muschelknautz and Trefz model [11, 18, 19], and it can be as high as 32.5% [18]. Cyclone designs that result in a high degree of shortcutting flow tend to have reduced separation efficiencies [11]. Therefore, reducing shortcutting flow can improve the collection efficiency of a cyclone particle separator.

The literature [1, 2, 9, 12, 19, 21] shows that a high-performance conventional cyclone needs to follow, or be similar to, a Stairmand design [22] where the ratio of the barrel length ($H_b$) to the cyclone diameter ($D$) is 3:2 and the overall length ($H$) of the cyclone to the cyclone diameter is 4:1. The collection performance of cyclone particle separators has been linked to the number of rotations made by the continuous phase as it travels through the device [8]. Increasing the length of the barrel section of the cyclone has been shown to both increase the collection efficiency (by as much as 9.7%) and reduce the pressure drop across the device (by as much as 37.5%) [2]. Increasing the conical section length also increases the collection efficiency (by as much as 10.8%) and decreases the pressure drop (by as much as 31.6%) [2]. Thus, increasing the overall length of the cyclone particle separator has a beneficial effect on the collection efficiency and pressure drop [8]. However, using an extremely long cyclone will result in a decrease in the collection efficiency [1, 23], this reduction being attributed to the vortex contained in the cyclone terminating on the wall of the conical section and not above the collection hopper [21, 23].

The collection efficiency of a cyclone can also be affected by the inlet pipe/duct height ($a$), width ($b$), and angle ($\theta_i$) [11, 12, 18]. Decreasing the height or width of the inlet creates a vortex with higher tangential velocities, resulting in improved collection efficiency [12]. However, the decreased inlet height or width also results in an increased pressure drop across the cyclone [12]. Angling the inlet pipe/duct downwards has also been shown to increase the collection efficiency over a range of inlet velocities while reducing the pressure drop across the cyclone [11, 18].

When numerically simulating a cyclone particle separator using computational fluid dynamics (CFD) methods, the literature recommends use of a Reynolds Stress Model (RSM) turbulence closure over a k-ε model, as it is more accurate for swirling flows [2, 3,
The numerical domain simulated can include a collection hopper [14, 19, 25], or the collection hopper can be neglected wherein the surface that would be open to the collection hopper is treated as a wall [2, 12, 13, 18]. Wan et al. [25] showed that by including the collection hopper in the simulation, particle re-entrainment from the collection hopper can be modelled. To model the motion and collection of particles numerically, the drag, centrifugal, gravity, and lift forces on the particles must be accurately modelled. The drag force is commonly modeled as if the particles are spherical [3, 12, 13, 16, 25], although a non-spherical drag model can also be used [26]. The lift force can be modeled using Saffman lift [3, 13], although this is only recommended for sub-micron particles when using the commercial software FLUENT [26]. Centrifugal and gravity forces are inherently included in any formulation including particles.

This brief literature survey illustrates many of the important considerations necessary in the design and development of cyclone separators. Combinations of barrel and cone height and diameter, inlet size and airflow velocity all lead to different performance characteristics across a spectrum of particle size. In addition, the avoidance of shortcutting is noted to be critical for maintaining high collection efficiencies in cyclone separators. Of central interest in this Chapter is the scale reduction of cyclone separators such that they occupy a smaller overall volume while preserving the required performance characteristics.

The first step in this process is to understand how scale reductions to a cyclone impact the internal flow field and identify which flow feature(s) are impacting the particle separation performance. The current Chapter presents a combined experimental and numerical study that explores the impact on particle separation efficiency of reducing the vertical scale of a cyclone for a single operating condition established from a conventional cyclone. The experimental study provides data for total and fractional collection efficiency for three different cyclones operating under the same throughflow conditions. The numerical study presents detailed results for the flow structure for the three cyclone models and, following calibration, it also provides results for the total and fractional collection efficiencies that are compared to the experiments. An important element of the numerical study is consideration of whether it is necessary to include the collection hopper in simulations since the previous studies noted above have reported mixed results. The remainder of this
Chapter presents the cyclone geometries under consideration, details of the experimental study, details of the numerical study with comparison to experimental results, and discussion of the key results of performance reduction and geometric models.

3.2 Cyclone Geometry

The cyclone particle separator used in the present experimental study was fabricated by Design Build Mechanical Inc. (DBM), New Brunswick, Canada, and inspired by an existing cyclone designed for a specific application. The barrel diameter was $D = 496$ mm, with the remaining dimensions influenced by the standard textbook *Gas Cyclones and Swirl Tubes* [1]. This “conventional” DBM (Full-size) cyclone was then modified twice to consider how substantial changes to the original geometry would influence the collection performance. The first modification was a reduction of the height of the barrel section by a factor of two while leaving all other dimensions the same (Intermediate cyclone). The second modification was to reduce the height of the conical section by a factor of two. Combining both modifications resulted in a cyclone half the height of the original cyclone and is referred to herein as the Truncated cyclone. The three cyclones are shown for comparison in Fig. 3.2 and their dimensions are summarized in Table 3.1. By maintaining the dimensions of the inlet, outlet and collection hopper, the Intermediate and Truncated cyclones could be installed in the same setup as the original Full-size cyclone (with the collection hopper being positioned higher above the ground).

Table 3.1: Summary of dimensions for the three cyclones constructed and tested in the experimental study.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Full-size Cyclone</th>
<th>Intermediate Cyclone</th>
<th>Truncated Cyclone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Barrel Diameter ($D$)</td>
<td>496 mm</td>
<td>496 mm</td>
<td>496 mm</td>
</tr>
<tr>
<td>Barrel Length ($H_b$)</td>
<td>670 mm</td>
<td>335 mm</td>
<td>335 mm</td>
</tr>
<tr>
<td>Cone Minimum Diameter ($D_d$)</td>
<td>121 mm</td>
<td>121 mm</td>
<td>121 mm</td>
</tr>
<tr>
<td>Cone Length ($H_c$)</td>
<td>1000 mm</td>
<td>1000 mm</td>
<td>500 mm</td>
</tr>
<tr>
<td>Angle of Cone ($\theta_c$)</td>
<td>79.4°</td>
<td>79.4°</td>
<td>69.4°</td>
</tr>
<tr>
<td>Vortex Finder Inside Diameter ($D_x$)</td>
<td>252 mm</td>
<td>252 mm</td>
<td>252 mm</td>
</tr>
<tr>
<td>Vortex Finder Outside Diameter ($D_y$)</td>
<td>256 mm</td>
<td>256 mm</td>
<td>256 mm</td>
</tr>
<tr>
<td>Vortex Finder Length ($S$)</td>
<td>247 mm</td>
<td>247 mm</td>
<td>247 mm</td>
</tr>
<tr>
<td>Cyclone Inlet Duct Length ($L_i$)</td>
<td>329 mm</td>
<td>329 mm</td>
<td>329 mm</td>
</tr>
<tr>
<td>Cyclone Inlet Duct Height ($a$)</td>
<td>247 mm</td>
<td>247 mm</td>
<td>247 mm</td>
</tr>
<tr>
<td>Cyclone Inlet Duct Width ($b$)</td>
<td>75 mm</td>
<td>75 mm</td>
<td>75 mm</td>
</tr>
</tbody>
</table>
The common collection hopper mounted to the bottom of the cyclone consisted of a cylinder placed on top of a Truncated cone. The collection hopper major diameter \((D_H)\) and minor diameter \((D_h)\) were 250 mm and 117 mm, respectively. The length of the collection hopper cylindrical \((H_H)\) section was 252 mm, and its conical \((H_h)\) section had a length of 254 mm. The total volume of the collection hopper was \(77.5 \times 10^6\) mm\(^3\).

The geometric swirl parameter \((S_g)\):

\[
S_g = \frac{\pi D_x D}{4ab}
\]  
(3.3)
is a measure of the ratio of the angular \( \theta \) momentum \( G_{\theta} \) and the axial \( z \) momentum \( G_z \) defined in Equations 3.4 and 3.5, respectively [27, 28].

\[
G_{\theta} = 2\pi \int_0^\infty \rho r^2 (\overline{U_\theta} \overline{U_z} + \overline{u_\theta u_z}) \, dr \\
G_z = 2\pi \int_0^\infty r \left( (\overline{\rho} - p_\infty) + \rho \left( \overline{U_z^2} + \overline{u_z^2} \right) \right) \, dr
\]

(3.4)

(3.5)

where \( \rho \) is the air density, \( p \) is the static pressure, \( p_\infty \) is the static pressure at an infinite radial position, \( U \) is the mean velocity component, and \( u \) is the fluctuating velocity component. As this parameter is dependent on geometric parameters that are common between the three cyclone designs, \( S_g = 5.3 \) for all of the cyclones in this Chapter. For a given cyclone inlet velocity, air density and dynamic viscosity \((U_{in} = 12.7 \text{ms}^{-1}, \rho = 1.225 \text{kgm}^{-3}, \text{and } \mu = 1.7894 \times 10^{-5} \text{kgm}^{-1}s^{-1})\) the cyclone Reynolds number:

\[
Re_{c1} = \frac{U_{in} D \rho}{\mu}
\]

(3.6)

is \( 4.31 \times 10^5 \) for all three cyclone geometries as, once again, the working fluid and key geometric variables are common between the three cyclone designs. This magnitude of \( Re_{c1} \) is typical of industrial cyclones [2, 27]. An alternative cyclone Reynolds number [1] incorporates the total height of the cyclone \((H = H_b + H_c)\) and is described by:

\[
Re_{c2} = \frac{U_x D_x \rho}{\mu} \frac{4 H}{D_x \left( \frac{D}{D_x} - 1 \right)}
\]

(3.7)

where \( U_x \) is the average axial velocity inside the vortex finder. Equation 3.7 yields \( Re_c = 3.52 \times 10^5, 4.40 \times 10^5, \text{and } 7.04 \times 10^5 \) for the Full-size, Intermediate and Truncated cyclones, respectively. Although the value of \( Re_{c2} \) scales with the total height of the cyclone and, thus, is different for each cyclone geometry considered in this Chapter, they are all of similar magnitude.
3.3 Experimental Details

3.3.1 The Cyclones
The cyclone used in the physical experiments (shown in Fig. 3.3) was modular, allowing for the conical section to be swapped between the Full-size and the Truncated cone, or a barrel extension to be added or removed. To achieve the Full-size cyclone, the longer cone and the barrel extension were used, whereas the Truncated cyclone configuration had the barrel extension removed and utilized the shorter conical section. This modular setup allowed for the inlet and outlet of the cyclone to be common thereby ensuring that all of the measured differences in collection efficiency and pressure drop were related to changes in cyclone geometry. The continuous phase through the cyclone was driven by suction created by a vacuum truck connected to the end of the outlet duct.

3.3.2 Instrumentation and Measurements
A schematic of the physical experimental setup, showing the location of the instrumentation, is presented in Fig. 3.4. A Venturi tube was placed upstream of the cyclone to determine the mass flow rate of the continuous phase. The pressure drop between the Venturi entrance and the Venturi throat was measured using a differential

Figure 3.3: Photos of Full-size (right) and Truncated (left) cyclones used in the physical experiments.
pressure transducer. The temperature and absolute static pressure of the continuous phase were measured downstream of the Venturi tube to ensure that they did not disturb the flow entering the Venturi tube. A differential pressure transducer was connected to the cyclone inlet, with the other end being connected 2.9 m downstream of the cyclone outlet. This configuration facilitated measurement of the static pressure drop across the cyclone and reduced the effect of the swirling flow that exited the cyclone [1]. The instrumentation used in the experimental setup is summarized in Table 3.2.

The discrete particle phase was introduced downstream of the temperature and absolute pressure transducer instrumentation (see Fig. 3.4). The powder is fed using a hopper feeder that utilizes an auger system that can be adjusted to transfer a desired mass flow rate of powder. A load cell was mounted under the hopper feeder to provide measurements of mass that are used to calculate the total mass of powder injected into the cyclone and the solid loading (ratio of the mass flow rate of the discrete phase to the volume flow rate of the continuous phase). Powder that was not swept into the continuous phase flow settled on the bottom of the pipe and was collected between each experimental test. The mass collected at the bottom of the pipe was not considered as part of the mass injected into the cyclone. The same load cell was used to measure the collected discrete phase after each experimental run.

The TCE of each run is determined using the load cell. However, to determine the FCE, a sample of each of the injected powder and the collected powder was subject to particle size analysis. Comparing the mass-weighted particle size distribution between the injected and collected powder allows calculation of the FCE curve.
Figure 3.4: Physical experimental setup schematic with bounding box of CFD domain.

Table 3.2: Summary of instrumentation used and their reported accuracies.

<table>
<thead>
<tr>
<th>Measurement</th>
<th>Instrument</th>
<th>Measurement Range</th>
<th>Manufactured Reported Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Venturi Pressure Drop</td>
<td>OMEGA PX142-030D5V</td>
<td>0–30psi (0–207 kPa)</td>
<td>±0.75% FS</td>
</tr>
<tr>
<td>Temperature</td>
<td>OMEGA ON-920TA-44005</td>
<td>-80–100°C</td>
<td>±0.2°C</td>
</tr>
<tr>
<td>Absolute Static Pressure</td>
<td>OMEGA PX142-030A5V</td>
<td>0–30psi (0–207 kPa)</td>
<td>±0.75% FS</td>
</tr>
<tr>
<td>Load Cell</td>
<td>OMEGA LCEB-50</td>
<td>0–50 lbf (0–23 kg)</td>
<td>±0.03% FS</td>
</tr>
<tr>
<td>Cyclone Pressure Drop</td>
<td>OMEGA PX142-001D5V</td>
<td>-1–1psi (-7–7 kPa)</td>
<td>±0.75% FS</td>
</tr>
</tbody>
</table>
3.3.3 Experimental Operation

The powder used in the experiments was Portland cement as it has a particle size distribution (shown in Fig. 3.5) with a significant number of particles which are difficult for cyclones to collect. Of the powder that was released by the feeding system 5% would fall to the bottom of the inlet pipe and be collected in the trap, resulting in 95% of the powder that went through the feeder entering the cyclone. The particle size distribution of the powder that was collected in the trap was similar to the Portland cement that was in the feeding system and is shown in Fig. 3.5 for comparison. Using the initial and trap powder distribution and the ratio of the mass through the feeder to that collected in the trap, the particle size distribution of the powder entering the cyclone was found (shown in Fig. 3.5).

Due to the low ratio of powder in the trap to the powder through the feeder, and the similar particle size distribution, the powder entering the cyclone had an almost identical size distribution to the initial Portland cement.

The experiments were carried out with a volume flow rate through the cyclone of $0.319 \pm 0.007 \, \text{m}^3\text{s}^{-1}$, yielding an average cyclone inlet velocity of $12.7 \pm 0.3 \, \text{ms}^{-1}$. This inlet velocity is within the range found in the literature [2, 11, 12, 19, 21]. The hopper feeder released powder at a rate of $8.5 \pm 0.01 \, \text{gs}^{-1}$, yielding a solids loading of $26.6 \pm 0.6 \, \text{gm}^3$.

![Figure 3.5: Particle size distribution measured form the experiments and used in the numerical simulations.](image-url)
3.3.4 Experimental Results

The cyclones yielded TCE values that followed the trend in the literature, with the best TCE (88%±0.3) being achieved by the Full-size cyclone, and the worst TCE (82%±0.3) being achieved by the Truncated cyclone, while the Intermediate cyclone had a TCE of 85%±0.3. The higher TCE of the Full-size cyclone was the result of the Full-size having improved FCE for particles under the size of 7 µm. The FCE curves for the three cyclone geometries are shown in Fig. 3.6. The pressure drop measured across the cyclone was similar between the three geometries. The pressure drop and the pressure coefficient $\Delta c_p$, defined as:

$$\Delta c_p = \frac{\Delta p}{\frac{1}{2} \rho U_{in}^2}$$

(3.8)

for each cyclone geometry is summarized in Table 3.3.

Figure 3.6: Fractional collection efficiency (FCE) for Portland cement.
3.4 Numerical Study

3.4.1 Computational Domain

The computational domain contained all of the components inside the red bounding box shown in the experimental setup schematic (Fig. 3.4). The numerical domain started at the inlet pipe, where the absolute pressure transducer was located, and included the inlet piping, the cyclone and the outlet piping up to the downstream pressure measurement location. The three cyclone geometries were meshed in Pointwise, maintaining consistent mesh densities. The cyclones were created using a structured mesh that required the cyclone to be blocked in a butterfly mesh style.

3.4.2 Continuous Phase Formulations

ANSYS FLUENT 20-R1 was used to simulate the flow through the cyclones, solving for continuity:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0
\]  

(3.9)

and momentum:

\[
\frac{\partial}{\partial t} (\rho \vec{u}) + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla p + \nabla \cdot \mu \left( (\nabla \vec{u} + \nabla \vec{u}^T) - \frac{2}{3} \nabla \cdot \vec{u} I \right) + \rho \vec{g}
\]  

(3.10)

The cyclones were initially simulated using the \( k-\varepsilon \) turbulence closure model (2-equation model) to develop the bulk flow structure and then restarted with the Reynolds Stress Model (RSM) turbulence closure model (7-equation model) to achieve higher accuracy [24, 29-31]. For these Unsteady Reynolds-Averaged Navier-Stokes (URANS) simulations

Table 3.3: Summary of experimentally measured total collection efficiency, pressure drop and pressure coefficient in the Full size, Intermediate and Truncated cyclones.

<table>
<thead>
<tr>
<th>Cyclone</th>
<th>TCE [%]</th>
<th>Pressure Drop [kPa]</th>
<th>( \Delta c_p )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full-Size</td>
<td>88±0.3</td>
<td>0.9±0.05</td>
<td>9.1±0.7</td>
</tr>
<tr>
<td>Intermediate</td>
<td>85±0.3</td>
<td>1.1±0.05</td>
<td>11.1±0.7</td>
</tr>
<tr>
<td>Truncated</td>
<td>82±0.3</td>
<td>1.2±0.05</td>
<td>12.1±0.8</td>
</tr>
</tbody>
</table>
a time step of 0.001 s was employed for all of the cyclone configurations as it ensured stable convergence.

Second-order upwinding was used to model advection in all of the transport equations apart from advection in momentum, which was modeled using QUICK [13]. Second-order discretization was used for pressure. The pressure-velocity coupling was maintained using the SIMPLEC scheme, and all of the equations solved for the continuous phase were forced to converge on each time step, achieving globally scaled residuals of less than $10^{-3}$. Further reductions in the residuals were time-consuming and resulted in imperceptible changes to the flow field and overall pressure drop.

### 3.4.2.1 Boundary Conditions

The boundary conditions for the continuous phase were the same for all of the cyclones considered in this Chapter. Near-wall treatment was set to the enhanced wall treatment for all of the simulations, as recommended in the literature [26, 32, 33]. A fully developed velocity profile was created and applied as the inlet boundary condition. The developed velocity profile was obtained by solving a simple three-dimensional pipe flow which had a uniform (plug flow) velocity inlet, an outflow outlet boundary condition, and no-slip wall boundary conditions. The pipe had the same diameter as the inlet pipe of the cyclones. The uniform velocity was set to achieve a mass flow rate equivalent to the experiments (0.318 kgs$^{-1}$). A pressure outlet boundary condition, specified with a gauge pressure of 0 Pa, was used. As the continuous phase is considered incompressible, the actual value of the specified pressure is irrelevant and does not affect the continuous phase or the discrete phase motion. A no-slip boundary condition was applied to all of the cyclone and collection hopper walls.

### 3.4.3 Particle Phase Formulation

Similar to the continuous phase, Discrete Phase Modeling (DPM) used in FLUENT requires assumptions, initial conditions, and boundary conditions. The particle motion was predicted, using FLUENT’s Discrete Phase Modeling (DPM) [24]. The particles tracked in FLUENT using DPM assumes that the particles are perfectly spherical and rely on the applied particle drag law to account for any irregularity in the particle shape. Other
assumptions included in the DPM were no particle collisions, agglomeration or attrition.

The particle motion was predicted, using FLUENT’s DPM [24], by balancing the inertia of the particle and the forces acting on it (Eq. 3.11) and using an ordinary differential equation for the particle angular momentum (Eq. 3.12):

\[ m_p \frac{d\vec{u}_p}{dt} = \vec{F}_d + \vec{F}_g + \vec{F}_{RL} \]  
\[ I_p \frac{d\vec{\omega}_p}{dt} = \frac{\rho}{2} \left( \frac{d_p}{2} \right)^5 C_\omega |\vec{\Omega}| \cdot \vec{\Omega} = \vec{T} \]

where \( m_p \) is the particle mass, \( \vec{u}_p \) is the particle velocity, \( t \) is time, \( \vec{F}_d \) is the drag force term, \( \vec{F}_g \) is the gravitational force term, and \( \vec{F}_{RL} \) is the rotational lift force term. In Eq. 3.12, \( I_p \) is the particle’s moment of inertia, \( \vec{\omega}_p \) is the particle angular velocity, \( \rho \) is the fluid density, \( d_p \) is the particle diameter, \( C_\omega \) is the particle’s rotational drag coefficient, \( \vec{\Omega} \) is the relative particle-fluid angular velocity, and \( \vec{T} \) is the torque applied to the particle.

The drag force \( \vec{F}_d \) in Eq. 3.11 is expressed by:

\[ \vec{F}_d = m_p \frac{\vec{u} - \vec{u}_p}{\tau_r} = m_p \frac{\vec{V}}{\tau_r} \]  

the gravity force \( \vec{F}_g \) by:

\[ \vec{F}_g = m_p \frac{\vec{g}(\rho_p - \rho)}{\rho_p} \]  

and the rotational lift force \( \vec{F}_{RL} \) by:

\[ \vec{F}_{RL} = \frac{1}{2} A_p C_{RL} \rho \left| \frac{\vec{V}}{\vec{\Omega}} \right| (\vec{V} \times \vec{\Omega}) \]

where \( \vec{u} \) is the fluid phase velocity, \( \rho \) is the fluid density, \( \rho_p \) is the density of the particle, \( \vec{g} \) is the gravitational acceleration, \( \tau_r \) is the particle relaxation time, \( A_p \) is the projected particle surface area, \( C_{RL} \) is the rotational lift coefficient, and \( \vec{V} \) is the relative fluid-particle
velocity.

In Eq. 3.12, the particle inertia is expressed as:

\[ I_p = \frac{\pi}{60} \rho_p d_p^5 \]  

(3.16)

the rotational drag coefficient \( C_\omega \) by:

\[ C_\omega = \frac{6.45}{\sqrt{Re_\omega}} + \frac{32.1}{Re_\omega} \]  

(3.17)

and the relative fluid-particle angular velocity \( \vec{\Omega} \) by:

\[ \vec{\Omega} = \frac{1}{2} \nabla \times \vec{u} - \vec{\omega}_p \]  

(3.18)

where \( d_p \) is the particle diameter and \( Re_\omega \) is the relative angular particle Reynolds number. The particle relaxation time \( \tau_r \) in Eq. 3.13 is expressed as:

\[ \tau_r = \frac{\rho_p d_p^2}{18\mu} \frac{24}{C_d Re} \]  

(3.19)

where \( Re \) is the relative particle Reynolds number:

\[ Re = \frac{\rho d_p |\vec{u}_p - \vec{u}|}{\mu} \]  

(3.20)

and the rotational lift \( C_{RL} \) in Eq. 3.13 as:

\[ C_{RL} = 2\varsigma \]  

(3.21)

with spin \( \varsigma \) expressed as:

\[ \varsigma = \frac{|\vec{\omega}_p| d_p}{2|\vec{u}_f - \vec{u}_p|} \]  

(3.22)

And the relative angular particle Reynolds number \( Re_\omega \) is defined as:
\[ Re_\omega = \frac{\rho d_p^2 |\Omega|}{4\mu} \]  

(3.23)

In Eq. 3.19, \( C_d \) is the particle drag coefficient expressed as:

\[ C_d = \frac{24}{Re}(1 + b_1 Re^{b_2}) + \frac{b_3 Re}{b_4 + Re} \]  

(3.24)

where \( b_1 \rightarrow b_4 \) are coefficients expressed as:

\[ b_1 = \exp(2.3288 - 6.4581\Phi + 2.4486\Phi^2) \]  

(3.25)

\[ b_2 = 0.0964 + 0.5565\Phi \]  

(3.26)

\[ b_3 = \exp(4.905 - 13.8944\Phi + 18.4222\Phi^2 - 10.2599\Phi^3) \]  

(3.27)

\[ b_4 = \exp(1.4681 + 12.2584\Phi - 20.7322\Phi\Phi^2 + 15.8855\Phi^3) \]  

(3.28)

and \( \Phi \) is a shape factor defined as:

\[ \Phi = \frac{s}{S} \]  

(3.29)

where \( s \) is the surface area of a sphere having the same volume as the particle, and \( S \) is the actual surface area of the particle. For the purposes of calculating particle mass, drag force, \( Re \), and \( Re_\omega \), the particle size \( d_p \) should be the diameter of a sphere having the same volume.

### 3.4.3.1 A-STERP Injection

Particles were introduced into the computational domain using the Aggregate STEady Random Particle (A-STERP) injection method described in Chapter 2 [34]. In this method, the TCE is used to establish steady values with an increasing number of distribution patterns or increasing number of particles injected in each pattern. It was found that 50 patterns containing 10000 particle injections in each pattern was sufficient for each device to achieve a steady TCE value.

The powder size distribution used in the numerical simulation was based on the size
distribution calculated (in section 3.3.3) to enter the cyclone. The numerically simulated Portland cement is shown in Fig. 3.5 for direct comparison to the physical powder. Figure 3.5 shows that the simulated and physical Portland cement distributions entering the cyclone were practically identical, allowing for direct comparison between the experiment and the numerical simulation results.

The A-STERP injection method uses either a snapshot of the flow or a time-averaged flow field to account for a quasi-steady flow. The simulated flow fields in the cyclones were time-averaged for a time span of 0.2 seconds of simulated flow time, which resulted in 10 rotations of the forced vortex portion of the Rankine vortex.

The location for A-STERP particle injection was at the inlet to the cyclone (shown in Fig. 3.4) rather than at the pipe inlet. This reduced the computational resources required for solving the particle motion since the particles did not need to be tracked through the pipe leading to the cyclone inlet.

3.4.3.2 Discreet Phase Modeling (DPM) Boundary Conditions

A “reflect” DPM condition with a coefficient of restitution set to 1.0 was assigned to the walls of the cyclone. This is an ideal collision where the particle retains all of its kinetic energy and was consistent for all of the simulations, allowing for comparison of the numerically predicted results. The “escape” condition was used at the outlet of the cyclone, such that any particles that reached this boundary were considered as having escaped with the continuous phase. For the particle simulations of the cyclones without a collection hopper, the surface at the bottom of the cyclone was specified as a ‘trap’ boundary condition, thus terminating the tracking of any particle that contacted the bottom of the cyclone. For the particle simulations of the cyclones with a collection hopper, the walls of the collection hopper also had a ‘reflect’ boundary condition, allowing for particles to enter and leave the collection hopper with the continuous phase.

3.4.4 Calibration of the Numerical Drag Model using Experimental Data

The simulated Portland cement was defined as having the same density as that of the physical Portland cement, as well as having a similar particle size distribution (shown in
Fig. 3.5 for comparison). Although in the simulations the Portland cement was assumed to be comprised of spherical particles, which was not physically correct, the numerical drag model could be calibrated (tuned) to account for irregularities in the particle shapes, thus relegating the assumption of spherical particles as immaterial. The numerical drag model needs to be tuned to ensure that the drag being applied to the simulated (spherical) Portland cement closely matches the actual drag experienced by the irregularly shaped Portland cement particles. The numerical drag model is defined in Eq. 3.24 and is a function of the particle Reynolds number \( (Re) \) and the shape factor \( (\Phi) \). The particle Reynolds number (defined by Eq. 3.20) and, hence, the drag, are functions of both the relative velocity of the flow around the particle and the particle size, as the density and the viscosity of the continuous phase is constant for all the particles. Since \( \Phi \) is a ratio of surface areas, and thus, a geometric parameter, it was the only parameter that affected the calculated particle drag and so it was the chosen tuning variable for calculating the physically accurate particle drag.

To tune the drag model the TCE values obtained from the experiments were compared to their numerical counterparts. The drag model was modified and the TCE examined to determine if the change improved the agreement between the numerical and experimentally obtained TCE. The simulation of the Truncated cyclone geometry with the collection hopper was the best candidate for tuning the numerical drag model used in predicting the motion of the simulated Portland cement particles because it was the experiments that produced the largest difference between the mass of particles that were collected and those that escaped. The variation of the predicted TCE for the Truncated cyclone with collection hopper with increasing \( \Phi \) is shown in Fig. 3.7. Setting \( \Phi \) to be 0.325 resulted in the numerically predicted TCE matching the experimentally measured TCE for the Truncated cyclone with the collection hopper. This shape factor takes into account the drag induced by the irregular shape of the physical Portland cement, but also includes any effect not specifically modeled such as particle collisions, agglomeration and attrition. The 0.325 value for \( \Phi \) was then used for all subsequent calculations.
3.4.5 Numerical Results

All comparisons made here between the predicted flow fields of the cyclones, with and without a collection hopper were carried out using the time-averaged flow fields utilized for the DPM simulations, as described in section 3.4.3.1.

3.4.5.1 Flow Field

The pressure drop calculated from the numerical simulations was taken from a similar position as the cyclone pressure drop measurement from the experiments. The predicted pressure drop from the Full-size and Intermediate cyclone simulations were minimally impacted by the inclusion of a collection hopper. For both geometries, the addition of the collection hopper in the simulation yielded a pressure drop increase of less than 20 Pa. The inclusion of the collection hopper in the Truncated cyclone simulation yielded a pressure drop increase of 60 Pa. The Full-size cyclone with a collection hopper, being the largest cyclone simulated, yielded the smallest pressure drop across the cyclone, being 600 Pa. Reducing the height of the barrel section increased the pressure drop to 730 Pa for the Intermediate cyclone, and reducing the height of the conical section increased the pressure drop to 830 Pa for the Truncated cyclone. Reducing the cone section, resulting in the smallest cyclone with collection hopper geometry (Truncated cyclone with collection

![Figure 3.7: Effect of shape factor $\Phi$ on the predicted total collection efficiency (TCE) for the Truncated cyclone with a collection hopper.](image)
hopper) gave the largest pressure drop, being 830 Pa. The pressure drop and pressure
coefficient values for the six simulated cyclones are summarized in Table 3.4.

Velocity profiles are examined on a radial line formed by the intersection of the central
(vertical) plane perpendicular to the cyclone inlet and the horizontal circular plane at the
seam between the barrel/cone (see Fig. 3.2 for location). The velocity profiles are
normalized by the inlet bulk velocity ($U_{in} = 12.7\text{ms}^{-1}$) and the radius of the barrel
($D/2 = 0.248\text{ m}$).

The tangential velocity profiles at the radial line for all six cyclones are shown in Fig. 3.8.
The tangential velocity profile is minimally affected by the inclusion/exclusion of the
collection hopper for all three cyclone geometries. Comparing the tangential velocity
between the three cyclones reveals that shortening the overall height of the cyclone
increases the magnitude of the tangential velocity component. This increase in the
tangential velocity occurred with both a reduction in the height of the barrel and with the
reduction in the height of the cone. The profiles shown in Fig. 3.8 resemble a Rankine
vortex where the transition between the free and forced vortex regions experiences the
highest tangential velocity. This transition occurred at a normalized radial position of 0.22,
0.23, and 0.21 for the Full-size, Intermediate, and Truncated cyclones, respectively. The
peak normalized tangential velocity was 1.9, 2.2, and 2.5, for the Full-size, Intermediate,
and Truncated cyclones, respectively. Using the peak tangential velocity its location to

Table 3.4: Summary of numerical predicted pressure drop and pressure coefficient.

<table>
<thead>
<tr>
<th>Cyclone</th>
<th>$\Delta P$ [Pa]</th>
<th>$\Delta c_p$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full-Size with collection hopper</td>
<td>600</td>
<td>6.1</td>
</tr>
<tr>
<td>Full-Size without collection hopper</td>
<td>620</td>
<td>6.3</td>
</tr>
<tr>
<td>Intermediate with collection hopper</td>
<td>730</td>
<td>7.4</td>
</tr>
<tr>
<td>Intermediate without collection hopper</td>
<td>750</td>
<td>7.6</td>
</tr>
<tr>
<td>Truncated with collection hopper</td>
<td>830</td>
<td>8.4</td>
</tr>
<tr>
<td>Truncated without collection hopper</td>
<td>860</td>
<td>9.0</td>
</tr>
</tbody>
</table>
normalize $\theta$ and $r$ respectively, new tangential velocity profiles (shown in Fig. 3.9) are created that can be compared to a single Rankine vortex profile defined by Giaiotti and Stel [35], and resembling the formulation by Batterson et al. [36]:

$$U_\theta = \begin{cases} \frac{U_{\theta \text{max}} \cdot r}{R_{\theta \text{max}}} & \text{if } 0 \leq r < R_{\theta \text{max}} \\ \frac{U_{\theta \text{max}} \cdot R_{\theta \text{max}}}{r} & \text{if } R_{\theta \text{max}} \leq r \end{cases}$$

(3.30)

where $U_{\theta \text{max}}$ is the peak tangential velocity and $R_{\theta \text{max}}$ is the radial position were $U_{\theta \text{max}}$ occurs. When the cyclone tangential velocity profiles were compared directly to a Rankine vortex, it was found that all of the cyclones had a higher tangential velocity in the free vortex region, and do not exhibit an exact solid body rotation in the forced vortex region. However, the overall trend of all the cyclone tangential velocity profiles were similar to the analytical Rankine vortex model.

The axial velocity profiles at the radial line are shown in Fig. 3.10. The presence of the collection hopper again had minimal impact on the velocity profile for the Full-size and Intermediate cyclone geometries, while the Truncated cyclone’s axial velocity profile was
Figure 3.9: Rankine vortex tangential velocity profiles along the radial line shown in Fig. 3.2 for the three cyclone geometries with and without a collection hopper.

Figure 3.10: Normalized axial velocity profiles (positive indicates flow towards the outlet and negative indicates flow towards the collection hopper) along the radial line shown in Fig. 3.2 for the three cyclone geometries with and without a collection hopper.
affected by the presence of the collection hopper in the simulation. A common trend for all of the simulated cyclones was the negative axial velocity near the cyclone wall and in the core, while between these two parts the axial velocity was positive (negative axial velocity was towards the collection hopper and positive axial velocity was towards the outlet). The Full-size cyclone had the lowest maximum axial velocity (0.37), whereas the Intermediate cyclone maximum magnitude was 0.49. The Truncated cyclone had a maximum magnitude of 0.42 when the collection hopper was included but increased to 0.64 when it was not.

To compare the flow fields inside the entire cyclone, the tangential projected velocity vectors are shown in Fig. 3.11. By using the tangential projection, the tangential (out of/into the page) velocity component is removed. The flow vector fields in Fig. 3.11 show that the flow away from the collection hopper exhibited similar vortex structures regardless of the presence of the collection hopper. A noted exception was for the Truncated cyclone in the core region (below the vortex finder), which is consistent with the comparison of the axial velocity in Fig. 3.10. The velocity vectors show several recirculation zones. It is important to note that the recirculation zones shown by the projected velocity vectors were small compared to the dominant vortex in the cyclone. A common recirculation zone between the three cyclone geometries was found on the opposite side to the inlet and at the top of the cyclone (A). A second common recirculation zone was found in the barrel section of the cyclones below the inlet (B). This recirculation zone spans the length of the barrel and, thus, is circular in shape for the Truncated and Intermediate cyclones and appears skewed for the Full-size cyclone. A third recirculation zone was present in the Full-size and Intermediate cyclones, but not in the Truncated cyclone. This third recirculation zone was opposite to the second recirculation zone but was not elongated with the length of the barrel (C).

The projected velocity vectors show flow shortcutting, which was identified in the introduction as an important flow feature. The bulk of the continuous phase flow comprises the vortex with an axial component directed towards the collection bin near the walls of the cyclone and towards the vortex finder/outlet near the neutral axis. However,
Figure 3.11: Tangential projection of velocity vectors for the three cyclone geometries with and without a collection hopper. Full-size (left), Intermediate (middle) and Truncated (right), with a collection hopper (top), and without a collection hopper (bottom).
some of the flow will shortcut the cyclone and escape under the vortex finder without traveling further down the cyclone. The projected velocity vectors in Fig. 3.11 show shortcutting in each cyclone and on either side of the vortex finder. By creating a rotated surface (ribbon shown in Fig. 3.2) directly under the vortex finder (with the same radius as the vortex finder) that extends towards the conical section by 2 mm (encompassing the location of shortcutting found in Fig. 3.11) the mass flow rate of the shortcutting fluid can be calculated. This method of calculating the degree of shortcutting flow is similar to that used by Xiang and Lee [18]. The Full-size and Intermediate cyclones experience 26% and 29% of the inlet mass flow rate shortcutting, respectively. This shows that reducing the height of the barrel section has a minimal impact on shortcutting. However, reducing the height of the cone (Truncated cyclone) resulted in the mass flow of fluid shortcutting increasing to 36% of the inlet mass flow rate. The 26% shortcutting in the Full-size cyclone was higher than the 10.6% reported by Xiang and Lee [18], but was similar to the 22.2% found in Qian and Zhang [19].

### 3.4.5.2 Predicted Collection Efficiencies

The Full-size cyclone was able to achieve a high TCE, as expected from its design. The Full-size cyclone was able to collect 98% and 97% of the simulated Portland cement, with and without the collection hopper, respectively. Reducing the height of the barrel section (Intermediate standard cyclone) resulted in the TCE for the simulated Portland cement reducing to 94% and 93%, respectively. The value of the TCE for either of the Intermediate cyclone configurations still classifies it as a high-performance cyclone. However, when the height of the cone was decreased the TCE significantly reduced, with the Truncated cyclone only achieving a TCE of 82% when the collection hopper was included and 62% when it was not. The predicted TCE for each simulation is summarized in Table 3.5.

The predicted FCE curves from the numerical simulations of the three cyclone geometries with and without the inclusion of a collection hopper are shown in Fig. 3.12 for a particle range of 1-15 μm (the full range of the particle size distribution). Similar to the TCE, the FCE curves for the Full-size and Intermediate cyclone geometries were minimally affected
by the inclusion of a collection hopper. Furthermore, these two geometries had similar FCE curves. The Full-size and Intermediate cyclones yielded FCE curves with 100% collection efficiency for the largest particles examined and a gradual decline in collection performance with decreasing particle size. The FCE curve for the Truncated cyclone with a collection hopper shows a moderate reduction in collection efficiency with reduction in particle size. Without the collection hopper the Truncated cyclone yielded an FCE curve with a sudden drop in collection efficiency from 7 μm to 5 μm, with the collection efficiency being less than 25% for any particle smaller than 5 μm.

Table 3.5: Summary of numerical predicted Total Collection Efficiency (TCE).

<table>
<thead>
<tr>
<th>Cyclone</th>
<th>TCE [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full-Size with collection hopper</td>
<td>98</td>
</tr>
<tr>
<td>Full-Size without collection hopper</td>
<td>97</td>
</tr>
<tr>
<td>Intermediate with collection hopper</td>
<td>94</td>
</tr>
<tr>
<td>Intermediate without collection hopper</td>
<td>93</td>
</tr>
<tr>
<td>Truncated with collection hopper</td>
<td>82</td>
</tr>
<tr>
<td>Truncated without collection hopper</td>
<td>62</td>
</tr>
</tbody>
</table>

Figure 3.12: Predicted fractional collection efficiencies (FCE) of the three cyclone geometries with and without a collection hopper.
3.5 Discussion

The Full-size cyclone, being the closest to a Stairmand cyclone design, yielded the highest TCE and the lowest pressure drop of the cyclones evaluated in both the numerical and experimental setups. The Full-size cyclone having the lowest pressure drop equates to it requiring the lowest energy input to drive the contained vortex. The numerically predicted TCE values for the Full-size cyclones was minimally impacted by not including the collection hopper, which implies that very little re-entrainment was occurring. The TCE decreased by 1% when eliminating the collection hopper from the numerical domain. Comparing the experimentally measured TCE for the Full-size cyclone to the numerically predicted TCE for simulations with and without the collection hopper resulted in a percent difference (PD) of 11% and 10%, respectively, where percent difference is defined as:

\[
PD = \frac{|C_{exp} - C_{num}|}{C_{exp}}
\]  

with \(C_{exp}\) being an experimentally measured value for a given parameter, and \(C_{num}\) being the numerically predicted value for the same parameter. The percent difference in the TCE for the Full-size cyclone is of a similar magnitude to the 9% percent difference found in Bogodage and Leung [14].

Similar to the TCE, the numerically predicted pressure drops across the Full-size cyclone with and without collection hopper were similar; the pressure drop across the Full-size cyclone without a collection hopper was reduced by 3% from the pressure drop value obtained with a collection hopper. The numerically predicted pressure drop was greater than the experimentally measured value and comparison of the values yields differences of 31% and 33% for simulations with and without the collection hopper. This magnitude of deviation in the pressure drop is consistent with values found in literature, having a range of 15% to 36%, while Qian and Zhang [11, 19] managed to keep the deviation less than 15%.

The Intermediate cyclone design yielded the second highest TCE and second lowest pressure drop both experimentally and numerically. Similar to the Full-size cyclone, the numerical results for the Intermediate cyclone were minimally affected by inclusion of the
collection hopper in the numerical domain. The TCE and pressure drop were once again reduced by 1% and 3%, respectively, with the removal of the collection hopper. The percent difference for the numerically predicted TCE of the Intermediate cyclone geometry with and without a collection hopper were 11% and 9%, respectively, while the corresponding errors in pressure drop were 32% and 34%.

The Truncated cyclone yielded the lowest TCE and highest pressure drop of the cyclone geometries examined. Thus reducing the height of the cyclone not only results in collection performance degradation but also requires additional energy input. The effect of neglecting the collection hopper in the numerical domain was noticeable in this cyclone geometry. The removal of the collection hopper resulted in the predicted TCE being reduced by 20% when compared to the predicted TCE for the truncated cyclone with a collection hopper. The reduction in the predicted TCE yielded an error in the TCE of 24% when the collection hopper was not included in the domain. The error in the TCE for the truncated cyclone with a collection hopper was 0% as this was the case used for tuning the drag model to the experimentally measured TCE.

Neglecting the collection hopper for the truncated cyclone resulted in a reduction in the pressure drop of 7% of the value obtained with a collection hopper. The Truncated cyclone numerical simulations yielded errors in the pressure drop of 26% and 31% with and without a collection hopper, respectively. The error in pressure drop was of similar magnitude to the other cyclone geometries in this study as well as Bogodage and Leung [14]. The percent difference in the pressure drop and the TCE for each of the simulations conducted are summarized in Table 3.6.

The trend in the TCE decreasing and the pressure drop increasing with reduction in the height of the cyclone, found in both the numerical and experimental investigation, is consistent with Brar et al. [2] who found that making the cyclone taller by lengthening either the cylinder or conical section yielded lower pressure drops and higher TCEs. Reducing the length of the barrel section of the Full-size cyclone by a factor of 2 (Intermediate cyclone) reduced the TCE by 3% in the experiments and 4% in the numerical simulations. This change in TCE is of similar magnitude as the 5% decrease in in the TCE
found in Brar et al. [2] when going from their A3 to A1 cyclones (where the barrel length was reduced by a factor of 2.33). This barrel reduction resulted in a 22% increase in the measured pressure drop and 21% increase in the numerically predicted results when the collection hopper is included (22% without collection hoppers). Again this increase in the pressure drop is of similar magnitude as the 21% pressure drop increase Brar et al. [2] found when going from cyclones A3 to A1. Reducing the total height of the Full-size cyclone by a factor of 2 (Truncated cyclone) reduced the TCE by 6% in the experiments and 15% in the numerical simulations with collection hoppers (34% without collection hoppers). The reduction of the total height of the cyclone yielded a 33% increase in the experimentally measured pressure drop and a 44% increase in the numerically predicted pressure drop for the collection hopper domains (38% without collection hoppers).

The presence of the collection hopper in the numerical simulations had minimal impact on both the continuous and discrete phases (based on the pressure drop and TCE) for the Full-size and Intermediate cyclone geometries. This was further confirmed by examination of the internal flow parameters that included normalized velocity (tangential and axial) profiles (shown in Fig. 3.8 and Fig. 3.9) and the velocity vectors. The Root Mean Square Difference (RMSD):

<table>
<thead>
<tr>
<th>Cyclone</th>
<th>PD of ΔP [%]</th>
<th>PD of TCE [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full-Size with collection hopper</td>
<td>31</td>
<td>11</td>
</tr>
<tr>
<td>Full-Size without collection hopper</td>
<td>33</td>
<td>10</td>
</tr>
<tr>
<td>Intermediate with collection hopper</td>
<td>32</td>
<td>11</td>
</tr>
<tr>
<td>Intermediate without collection hopper</td>
<td>34</td>
<td>9</td>
</tr>
<tr>
<td>Truncated with collection hopper</td>
<td>26</td>
<td>0</td>
</tr>
<tr>
<td>Truncated without collection hopper</td>
<td>31</td>
<td>24</td>
</tr>
</tbody>
</table>

Table 3.6: Summary of percent difference of the numerical results compared to the experimentally measured values.
\[ RMSD = \sqrt{\frac{\sum_{i=1}^{N}|y(i) - Y(i)|^2}{N}} \]  

(3.32)

with \( y \) being a value for the domain with a collection hopper, and \( Y \) being a value for the domain without a collection hopper, was calculated for both the tangential and axial velocity profiles. The RMSD was 0.02 for both of the velocity profiles for the Full-size cyclone, while the Intermediate cyclone yielded RMSD values of 0.06 and 0.03 for the tangential and axial velocity profiles, respectively. These four values of RMSD suggest very close agreement between the velocity fields from the domains with and without collection hoppers, where a RMSD value of zero represents an exact match.

The collection hopper did have a meaningful impact on the numerical simulation of the Truncated cyclone, with both the TCE and pressure drop being affected. The impact of the collection hopper on the Truncated cyclone was also found in the internal flow, with the tangential and axial velocity profiles yielding RMSD values of 0.19 and 0.10, respectively.

From the numerical particle tracking of the cyclones with collection hoppers, the number of times a particle track entered/exited the collection hopper was recorded to determine particle re-entrainment. The particle re-entrainment analysis revealed that a negligible number of particles (less than 0.01% of the injected particles) were re-entrained. Furthermore, these re-entrained particles were often re-captured, resulting in re-entrainment not being a factor in the change of collection performance with a change in cyclone height. Additionally, the impact of the collection hopper on the TCE for the truncated cyclone resulted from the change in the internal flow and not from re-entrainment.

The numerical particle tracking also allowed examination of the degree of particles shortcutting. Using the same ribbon to calculate the amount of shortcutting air flow, the particles that passed through the ribbon and proceeded to the cyclone outlet represented the mass of shortcutting particles. For the cyclones with collection hoppers 1%, 2% and 3% of the injected particle massshortcutted under the vortex finder for the Full-size, Intermediate and Truncated cyclones, respectively. Reducing the length of the barrel section resulted in
a significant increase in the mass of shortcutting particles, while reducing the length of the conical section only marginally increased the mass of shortcutting particles. This trend is the opposite of that seen for shortcutting in the continuous phase, where reducing the conical section had a more significant impact. Compared to the Full-size cyclone, the intermediate cyclone yielded an additional 3% and 2% of the fluid and particle shortcutting, respectively. When compared to the Intermediate cyclone, the Truncated cyclone yielded an additional 7% and 1% of fluid and particle shortcutting, respectively. This shows that although the shortcutting particles require shortcutting of the continuous phase, the degree of shortcutting particles does not trend with increasing fluid shortcutting. Neglecting the collection hopper in the numerical domain resulted in a 0.1% decrease in the predicted mass of shortcutting particles for all three cyclone geometries.

3.6 Conclusion
The performance of three cyclone particle separators was investigated both experimentally and numerically. The first cyclone (Full-size) was of a typical scale, closely resembling a Stairmand cyclone design. The second cyclone (Intermediate) had the same plan dimensions as the Full-size with the exception that the barrel height was halved. The third cyclone (Truncated) also had the same plan dimensions as the Full-size with the exception that both the barrel and conical heights were halved. The experiments were conducted on a modular cyclone setup using a vacuum truck to draw air through the cyclone. The powder used as the solid contaminant to be separated out was Portland cement, which had a median particle size of 7μm. The numerical simulations were carried out using Computational Fluid Dynamics (CFD) and Discrete Phase Modeling (DPM) tools in the FLUENT software. The numerical particle injections were conducted using the Aggregate STEady Random Particle (A-STERP) injection method, and the numerically applied drag on the particles used a non-spherical drag model with the shape factor being tuned based on the Total Collection Efficiency (TCE).

The experimental and numerical investigation allowed for evaluation of the effect that reducing the size of a cyclone has on its performance. The experimental results showed that each reduction in height reduced the TCE by approximately 3% resulting in the Truncated cyclone failing to capture approximately 6% of the injected solids. The pressure
drop, when compared to the Full-size cyclone, increased by 22% and 33% for the Intermediate and Truncated cyclones. The numerical results showed a similar trend to the experiments; both showed the expected result that reducing the cyclone height results in a cyclone that collects less of the injected solids while requiring additional energy to operate. The key flow features leading to this result are the increase in shortcutting between the inlet and classifier and the higher tangential velocity across the cyclone leading to increased entrainment with decreasing height.

The numerical investigation into the three cyclone geometries also allowed for examination of the affect of excluding the collection hopper in the numerical domain. For the Full-size and Intermediate cyclone geometries, the structure of the internal flow was not significantly impacted by the exclusion of the collection hopper. However, the numerical results for the truncated cyclone were dependent on the collection hopper being included in the numerical domain. The predicted TCE of the Truncated cyclone varied by 20% and the difference in the pressure drop was almost twice that of the Full-size and Truncated cyclones. The impact of including the collection hopper was also seen in the internal flow analysis, through both the velocity profiles and the flow structure visualization using velocity vectors. This demonstrates that including the collection hopper in the numerical domain is not always a necessity, but certain circumstances can lead its absence resulting in incorrect results. The height of the cyclone particle separator has been identified as being a determinant of when the collection hopper should be included.

3.7 References


“ANSYS FLUENT, release 20 R1, help system, FLUENT user guide.” ANSYS Inc., 2023.


B. E. Launder, “Second-moment closure: present... and future,” *Int. J. Heat Fluid...


Chapter 4

4 Scale Reduction of Cyclone Particle Separators: Part II – Methods of Performance Recovery

4.1 Introduction

The present Chapter continues the investigation of scale reduction of cyclone separators, aiming to reduce their overall volume while preserving the required performance characteristics. The initial step, conducted in Chapter 3 [1], involves understanding how scale reductions impact the particle separation performance and identifying changes in the flow field in scaled-down cyclones that cause these changes in performance. In Chapter 3, scale reduction of the Full-size cyclone yielded a drop in the Total Collection Efficiency (TCE) (defined by Eq. 4.1) and the Fractional Collection Efficiency (FCE) (defined by Eq. 4.2) [2]:

\[ TCE = \frac{M_c}{M} \]  \hspace{1cm} (4.1)

\[ FCE(d) = \frac{m_c(d)}{m(d)} \]  \hspace{1cm} (4.2)

where the collected mass is \( M_c \) and the collected mass for particles with a diameter \( d \) is \( m_c(d) \). Similarly, the mass injected is \( M \), and the mass of injected particles with a diameter of \( d \) is \( m(d) \). The numerical simulation results from Chapter 3 showed that the scale reduction in the cyclone increased the amount of shortcutting flow. This increase in shortcutting flow was deemed a key flow feature that led to the decreases in the TCE and FCE.

Jiao et al. [3] were able to eliminate flow shortcutting in the cyclone they studied by adding a rotating classifier, thereby transforming the conventional cyclone into a dynamic cyclone [3-7]. The rotating classifier is placed concentrically within the cyclone, creating a common neutral axis (see Fig. 4.1). It can be positioned inside the vortex finder [4], directly below the vortex finder [5], or even replace the vortex finder altogether [3]. Experimental and numerical studies have demonstrated that the rotating classifier enhances the collection.
efficiency, particularly for particles smaller than 10 μm, compared to cyclones of similar geometry and operating conditions without a rotating classifier [3-5]. Increasing the rotational speed of the classifier further improves the collection efficiency [3-5]. The presence of the rotating classifier directly impacts the tangential velocity of the flow field within the cyclone [3-5]. The tangential velocity beneath the blades increases with the blade speed, while the tangential velocity outside of the blades remains similar, provided the inlet speed remains consistent [3-5]. Moreover, the rotating classifier affects the pressure drop across the cyclone. As the rotational speed increases, the pressure drop increases [5], indicating that the classifier works against the device's flow-driving mechanism through the cyclone.

Figure 4.1: Dynamic cyclone schematic showing components.
The classifier used in the dynamic cyclones had flat rectangular blades [3-7]. Jiao et al. [3] suggested that angling the blades away from the inlet (negative inclination angle $\theta_b$) should result in improved collection performance over radial straight blades ($\theta_b = 0^\circ$) or positive inclination angles. Zhou et al. [5] determined that the FCE and TCE were negligibly impacted by a negative inclination angle over radial straight blades ($\theta_b = 0^\circ$). However, a positive inclination angle does decrease the FCE and TCE. Zhou et al. [5] tested classifiers with 20, 30 and 40 blades, and Jiao et al. [3] tested classifiers with 30 and 50 blades. Jiao et al. [3] concluded that the number of blades in the classifier had a minimal effect on the tangential velocity distribution and that 30 blades were preferential. Zhou et al. [5] found that 30 blades on their classifier managed to slightly increase collection performance over 20 blades (both collected all particles larger than 8 μm), but having 50 blades on the classifier reduced the collection performance (less than 70% of 8 μm particles are collected). Zhou et al. [5] attributed the decreases in the collection performance when the classifier had 50 blades to the increases in the blocked cross-sectional area created by the classifier.

The dynamic cyclone in Safikhani et al. [6] included a flat disk vortex stabilized (also called vortex limiter) in the collection hopper. The velocity contours showed that the vortex stabilizer reduced the axial velocity below it. Vortex stabilizers are also found in conventional cyclones [8-10]. The vortex stabilizer found in Karagoz et al. [10] was a flat disk similar to the one in Safikhani et al. [6], whereas the vortex stabilizers in Hoekstra et al. [8] and Celis et al. [9] had a cone shape that pointed up towards the vortex finder. Hoekstra et al. [8] stated that the vortex stabilizer was added to reduce the amount of particles being re-entrained from the collection hopper.

This Chapter explores, experimentally and numerically, how such performance reductions can be recovered by introducing rotating classifiers and different types of collection hoppers. The experimental study provides data on total and fractional collection efficiency for three cyclones operating under the same flow conditions. The numerical study offers detailed flow structure results and predicted collection efficiencies for the three cyclone geometries and five collection hopper designs. The objectives of the presented study are:

(i) to determine the impact of classifier rotational speed on the flow field and collection
efficiency for the scaled reduced cyclone; (ii) to deter particle re-entrainment from the collection hopper by altering the collection hopper design; and (iii) to determine a rotational speed and collection hopper combination that recovers the collection performance of the conventional, Full-size cyclone from Chapter 3 while minimalizing any additional energy input.

4.2 Cyclone Geometry

The three cyclone particle separators in a standard configuration used in the experimental study were fabricated by Design Build Mechanical Inc. (DBM), New Brunswick, Canada, and experimentally studied in Chapter 3. The common barrel diameter was $D = 496$ mm, and the total cyclone height ($H$) was 1670 mm, 1335 mm and 835 mm for the Full-size, Intermediate and Truncated cyclones, respectively. The three cyclones are shown for comparison in Fig. 4.2, and their dimensions are summarized in Table 4.1.

The rotating classifier also fabricated by DBM was based on the classifier from Zhou et al. [5]. The classifier (shown in Fig. 4.2) has 20 radial straight blades and was added to the conventional cyclones to create a dynamic cyclone. The classifier major diameter ($D_{CL}$) and minor diameter ($D_{CL}$) are 241 mm and 70 mm, respectively, and the blade height ($H_{CL}$) is 75 mm. The classifier was co-axially located directly under the vortex finder of the

Table 4.1: Summary of dimensions for the three cyclones constructed and tested in the experimental study.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Full-size Cyclone</th>
<th>Intermediate Cyclone</th>
<th>Truncated Cyclone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Barrel Diameter ($D$)</td>
<td>496 mm</td>
<td>496 mm</td>
<td>496 mm</td>
</tr>
<tr>
<td>Barrel Height ($H_b$)</td>
<td>670 mm</td>
<td>335 mm</td>
<td>335 mm</td>
</tr>
<tr>
<td>Cone Minimum Diameter ($D_d$)</td>
<td>121 mm</td>
<td>121 mm</td>
<td>121 mm</td>
</tr>
<tr>
<td>Cone Height ($H_c$)</td>
<td>1000 mm</td>
<td>1000 mm</td>
<td>500 mm</td>
</tr>
<tr>
<td>Angle of Cone ($\theta_c$)</td>
<td>79.4°</td>
<td>79.4°</td>
<td>69.4°</td>
</tr>
<tr>
<td>Vortex Finder Inside Diameter ($D_x$)</td>
<td>252 mm</td>
<td>252 mm</td>
<td>252 mm</td>
</tr>
<tr>
<td>Vortex Finder Outside Diameter ($D_x$)</td>
<td>256 mm</td>
<td>256 mm</td>
<td>256 mm</td>
</tr>
<tr>
<td>Vortex Finder Height ($S$)</td>
<td>247 mm</td>
<td>247 mm</td>
<td>247 mm</td>
</tr>
<tr>
<td>Cyclone Inlet Duct Length ($L_i$)</td>
<td>329 mm</td>
<td>329 mm</td>
<td>329 mm</td>
</tr>
<tr>
<td>Cyclone Inlet Duct Height ($a$)</td>
<td>247 mm</td>
<td>247 mm</td>
<td>247 mm</td>
</tr>
<tr>
<td>Cyclone Inlet Duct Width ($b$)</td>
<td>75 mm</td>
<td>75 mm</td>
<td>75 mm</td>
</tr>
</tbody>
</table>
cyclone such that the top of the classifier was co-planer to the bottom of the vortex finder. The blade height of 75 mm ensured that the classifier was located inside the barrel section and not protruding into the conical section for the Truncated and Intermediate cyclone.

The impact of five different collection hoppers (see Fig. 4.3) on the Truncated dynamic cyclone were numerically investigated. The benchmark collection hopper (CH1) was the same as the common collection hopper used in Parker et al. [1]. The major diameter \( D_H \) and minor diameter \( D_h \) of the benchmark collection hopper are 250 mm and 117 mm, respectively. The height of the collection hopper cylindrical \( H_H \) section is 252 mm, and its conical \( H_h \) section has a height of 254 mm. The total volume of the collection hopper is \( 19.4 \times 10^6 \text{ mm}^3 (0.0194 \text{ m}^3) \). The modified collection hoppers (CH2 and CH3) have the same dimensions as the benchmark collection hopper, but with vortex stabilizers added.

Figure 4.2: Three cyclone geometries, Full-size (left), Intermediate (centre) and Truncated (right), with the horizontal line showing the radial line used for the velocity profiles and the shortcutting ribbon. Geometry of classifier shown in top right.
inside the hopper. CH2 had a vortex stabilizer that was a flat plate located 100 mm below the top of the collection hopper and had a diameter of 150 mm. CH3 had a cone-shaped vortex stabilizer with the point toward the cyclone. The conical Vortex Stabilizer (VS) has a diameter of 150 mm, and the rim is located 100 mm below the top of the collection hopper. The angle of the cone is 120°. The other two new collection hoppers had the same height as the benchmark hopper, but with their overall diameters increased by a factor of 2 (CH4) and 4 (CH5). The major diameter of CH3 is similar to the cyclone barrel diameter. The five collection hoppers investigated using CFD are shown in Fig 4.3 and are summarized in Table 4.2.

Figure 4.3: Five collection hopper geometries, Collection Hopper 1 (top left), Collection Hopper 2 (top centre), Collection Hopper 3 (top right), Collection Hopper 4 (bottom left), Collection Hopper 5 (bottom right), with the horizontal line showing the radial line used for the velocity profiles inside collection hopper.
4.3 Experimental Study

4.3.1 The Cyclones

The physical experiments utilized the DBM fabricated cyclones and setup from Chapter 3, with the addition of a rotating classifier under the vortex finder, which required a motor to power it. The motor was mounted above the cyclone and used a Variable Frequency Drive (VDF) to control the operating frequency, allowing for controlling the motor rotational speed. The classifier and its shaft were held by a bearing support inside the vortex finder. This mounting configuration allowed for the modular barrel sections to be interchanged without removing the classifier, vortex finder, outlet, or inlet. In this manner, the Full-size, Intermediate, and Truncated cyclone geometries could all be tested as dynamic cyclones.

4.3.2 Instrumentation and Measurements

Figure 4.4 shows a schematic of the physical experimental setup with the location of the instrumentation. A Venturi tube was used to measure the mass flow rate of the continuous phase and was placed upstream of the cyclone. A differential pressure transducer was connected to the Venturi entrance and throat to measure the pressure differential. The absolute static pressure and temperature of the continuous phase were measured at a position downstream of the Venturi tube, but upstream of the cyclone inlet to ensure that these two sensors did not disturb the flow entering the Venturi tube. A second differential pressure transducer was connected to the cyclone inlet, with the other end being connected 2.9 m downstream of the cyclone outlet. This configuration facilitated the measurement of

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Major Diameter [mm]</th>
<th>Minor Diameter [mm]</th>
<th>Volume [mm$^3$]</th>
<th>Vortex Stabilizer</th>
</tr>
</thead>
<tbody>
<tr>
<td>CH1 (Benchmark CH)</td>
<td>250</td>
<td>117</td>
<td>$19.4 \times 10^6$</td>
<td>N/A</td>
</tr>
<tr>
<td>CH2 (Flat VS)</td>
<td>250</td>
<td>117</td>
<td>$19.4 \times 10^6$</td>
<td>Flat</td>
</tr>
<tr>
<td>CH3 (Cone VS)</td>
<td>250</td>
<td>117</td>
<td>$19.4 \times 10^6$</td>
<td>Cone</td>
</tr>
<tr>
<td>CH4 (Double CH)</td>
<td>500</td>
<td>234</td>
<td>$77.5 \times 10^6$</td>
<td>N/A</td>
</tr>
<tr>
<td>CH5 (Quad CH)</td>
<td>1000</td>
<td>468</td>
<td>$310.1 \times 10^6$</td>
<td>N/A</td>
</tr>
</tbody>
</table>

Table 4.2: Summary of collection hoppers tested in the CFD study.
static pressure drop across the cyclone and reduced the impact that the swirling flow at the cyclone exit would have on measurement [11]. The rotational speed of the classifier was measured using a digital photo tachometer. The rotational speed was measured at the exposed shaft connected to the classifier. The instrumentation used in the experimental setup is summarized in Table 4.3.

The discrete particle phase was introduced downstream of the absolute pressure transducer and temperature instrumentation (see Fig. 4.4). The powder is fed using a feeding system comprised of a hopper feeder and a motorized auger. A load cell was used to provide measurements of mass that are used to calculate the total amount of powder injected into the cyclone and collected in the collection hopper. Powder not swept into the continuous phase flow settled on the bottom of the pipe and was not considered part of the mass injected into the cyclone. The solid loading (ratio of the mass flow rate of the discrete phase to the volume flow rate of the continuous phase) is calculated by considering the injected mass flow of powder and the simultaneous mass flow rate of air measured by the Venturi.

Table 4.3: Summary of instrumentation used and their reported accuracies.

<table>
<thead>
<tr>
<th>Measurement</th>
<th>Instrument</th>
<th>Measurement Range</th>
<th>Manufactured Reported Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Venturi Pressure Drop</td>
<td>OMEGA PX142-030D5V</td>
<td>0–30 psi (0–207 kPa)</td>
<td>±0.75% FS</td>
</tr>
<tr>
<td>Temperature</td>
<td>OMEGA ON-920TA-44005</td>
<td>-80–100°C</td>
<td>±0.2°C</td>
</tr>
<tr>
<td>Absolute Static Pressure</td>
<td>OMEGA PX142-030A5V</td>
<td>0–30 psi (0–207 kPa)</td>
<td>±0.75% FS</td>
</tr>
<tr>
<td>Load Cell</td>
<td>OMEGA LCEB-50</td>
<td>0–50 lbf (0–23 kg)</td>
<td>±0.03% FS</td>
</tr>
<tr>
<td>Cyclone Pressure Drop</td>
<td>OMEGA PX142-001D5V</td>
<td>-1–1 psi (-7–7 kPa)</td>
<td>±0.75% FS</td>
</tr>
<tr>
<td>Classifier Rotational Speed</td>
<td>DT-2234C+ Digital Photo Tachometer</td>
<td>2.5–9999 rpm</td>
<td>±0.05%+1 rpm</td>
</tr>
</tbody>
</table>
The load cell allowed for determining the TCE of each run, while particle size analysis was required to determine the FCE. A sample of each injected powder and the collected powder was analyzed for its mass-weighted particle size distribution, and for comparison to determine the FCE curve.

4.3.3 Experimental Operation

The powder used in the experiments was Portland cement, with the particle size distribution shown in Fig. 4.5. Of the powder that was released by the feeding system, 95% entered the cyclone, with the remaining 5% falling to the bottom of the inlet pipe and collected in the trap. The particle size distribution of the stock Portland cement powder, the powder collected in the trap and the powder entering the cyclone are shown in Fig. 4.5 for comparison. Because the powder in the trap is a small fraction of the particle mass injected, and because it has a similar particle size distribution compared to the stock Portland cement, the powder entering the cyclone is also considered to have a nearly identical size distribution as the stock Portland cement.

Figure 4.4: Physical experimental setup schematic with a bounding box representing the CFD domain.
The volume flow rate through the cyclone during the experiments was $0.319 \pm 0.007 \text{ m}^3\text{s}^{-1}$, yielding an average velocity of $12.7 \pm 0.3 \text{ ms}^{-1}$ at the cyclone inlet. The powder entered the cyclone at a rate of $8.5 \pm 0.003 \text{ gs}^{-1}$, yielding a solid loading of $26.6 \pm 0.5 \text{ gm}^{-3}$.

The dynamic cyclone experiments were conducted with classifier rotational speeds ($\omega$) ranging from $3000 \pm 2.5$ to $6000 \pm 4 \text{ rpm}$. A pulley ratio of 1.79 between the motor and the classifier was required to achieve the desired rotational speed of 6000 rpm. The classifier rotational speed was set by varying the frequency output of the VFD until the digital photo tachometer measured the desired rotational speed.

4.3.4 Experimental Results

In the Truncated dynamic cyclone, the rotating classifier was run at $3000 \pm 2.5$, $4000 \pm 3$, $4500 \pm 3.3$ and $6000 \pm 4 \text{ rpm}$. The TCE from the experimental measurements for the Truncated dynamic cyclone with the dynamic classifier is shown in Fig. 4.6, wherein it is observed that the highest TCE ($85\% \pm 0.3$) was achieved when the classifier was rotating at 4000 rpm. The pressure drop across the cyclone ($\Delta P$) and pressure coefficient $\Delta c_p$:

![Figure 4.5: Particle size distribution measured form the experiments and used in the numerical simulations.](image)
\[ \Delta c_p = \frac{\Delta p}{\frac{1}{2} \rho u_{in}^2} \quad (4.3) \]

at the various classifier rotational speeds in the truncated cyclone geometry are shown in Table 4.4.

The Full-size and Intermediate cyclone geometries were also tested in a dynamic cyclone configuration with the classifier rotating at 4000 rpm. At this rotational speed, a TCE value of 85%±0.3 was achieved for the Truncated and Intermediate cyclone and 86%±0.3 for the Full-size cyclone. The similarity in the TCE values between the three dynamic cyclone geometries extended to the FCE curves as shown in Fig. 4.7. The TCE, \( \Delta P \) and \( \Delta c_p \) for the three cyclone geometries are summarized in Table 4.5.

![Graph showing TCE vs. classifier rotational speed](image)

Figure 4.6: Portland cement Total Collection Efficiency (TCE) for the Truncated dynamic cyclone at various rotational rates (\( \omega \)), where \( \omega = 0 \) represents a standard cyclone.

Table 4.4: Summary of experimentally measured cyclone performance in the Truncated cyclone at various classifier rotational speeds.

<table>
<thead>
<tr>
<th>( \omega ) [rpm]</th>
<th>TCE [%]</th>
<th>( \Delta P ) [kPa]</th>
<th>( \Delta c_p )</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>82±0.3</td>
<td>1.4±0.05</td>
<td>14.0±1.0</td>
</tr>
<tr>
<td>4000</td>
<td>85±0.3</td>
<td>1.6±0.05</td>
<td>16.0±1.0</td>
</tr>
<tr>
<td>4500</td>
<td>80±0.3</td>
<td>1.9±0.05</td>
<td>19.0±1.0</td>
</tr>
<tr>
<td>6000</td>
<td>70±0.3</td>
<td>3.0±0.05</td>
<td>30.0±2.0</td>
</tr>
</tbody>
</table>
4.4 Numerical Study

4.4.1 Computational Domain

The computational domain is defined by the contents within the bounding box illustrated in the experimental setup schematic (Fig. 4.2). The numerical domain starts at the inlet pipe where the absolute pressure transducer is located and includes the inlet piping, the cyclone, the collection hopper and the outlet piping up to the downstream pressure measurement location. The cyclones, collection hoppers and classifier were initially meshed separately and were later combined into the final mesh, maintaining individual mesh zones. This modular meshing strategy allowed for changing the collection hopper or the cyclone geometry (the classifier mesh was a common element) without creating each new configuration from scratch.

![Figure 4.7](image.png)

Figure 4.7: Portland cement fractional collection efficiency (FCE) for the three dynamic cyclone geometries with the classifier rotating at 4000 rpm.

Table 4.5: Summary of experimentally measured cyclone performance in the Full size, Intermediate and Truncated cyclones with a classifier rotating at 4000 rpm.

<table>
<thead>
<tr>
<th>Cyclone</th>
<th>TCE [%]</th>
<th>ΔP [kPa]</th>
<th>Δc_p</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full-Size</td>
<td>86±0.3</td>
<td>1.5±0.05</td>
<td>15.2±0.9</td>
</tr>
<tr>
<td>Intermediate</td>
<td>85±0.3</td>
<td>1.5±0.05</td>
<td>15.2±0.9</td>
</tr>
<tr>
<td>Truncated</td>
<td>85±0.3</td>
<td>1.6±0.05</td>
<td>16.0±1.0</td>
</tr>
</tbody>
</table>
4.4.2 Continuous Phase Formulations

The flow through the cyclones was simulated using ANSYS FLUENT 20-R1. Initially, the cyclones were simulated using the k-ε turbulence closure model, a 2-equation model, to establish the bulk flow structure. Subsequently, the simulations were restarted with the Reynolds Stress Model (RSM) turbulence closure model, a 7-equation model, to attain higher accuracy [12-15]. This two-step approach allowed for a more timely completion of the flow simulations and improved the precision in capturing the turbulence characteristics within the cyclone. The Mesh Motion settings in FLUENT were used to simulate the rotation of the classifier on the simulated continuous phase. The classifier mesh was simulated with a rotational speed of 4000 rpm for each geometric configuration based on the improved TCE results observed in the experiments. Additional simulations were performed with the classifier rotating at 6000 rpm for the Truncated cyclone geometry with each of the five collection hoppers. A time step of 0.0002 s was employed for all cyclone configurations as it ensured several time steps between each blade passage and stable and timely convergence.

In the numerical simulations, second-order upwinding was employed to model advection in all transport equations, except for advection in momentum, which was set to use the QUICK scheme [5]. For pressure, a second-order discretization method was utilized. The pressure-velocity coupling was accomplished using the SIMPLEC scheme. At each time step, all the equations related to the continuous phase were iteratively solved until they converged, achieving globally scaled residuals of less than 10⁻³.

4.4.2.1 Boundary Conditions

The boundary conditions for the continuous phase were the same for all simulations considered in this Chapter. Near-wall treatment was set to the Enhanced Wall Treatment, as recommended in the literature [16-18]. A fully developed velocity profile for a duct flow was applied as the inlet boundary condition. A pressure outlet boundary condition was specified with a gauge pressure of 0 Pa. As the continuous phase is considered incompressible, the actual value of the specified pressure is irrelevant and does not affect the continuous phase or the discrete phase motion. A no-slip boundary condition was applied to all walls.
4.4.3 Particle Phase Formulation
A representative flow snapshot is required for the time-saving A-STERP injection method described in Chapter 2 [19]. Use of a snapshot of the flow on the dynamic cyclone mesh was problematic for particle tracking, as particles encountered the classifier as if it were stationary, which caused them to collide on the back side of the blades. When the blades are rotating, the particles are moving more slowly and are unable to collide on the backside of the blades. To rectify this problem, the time-averaged continuous phase flow was loaded onto a mesh with the blades being treated as part of the fluid domain (thus appearing as a non-dynamic cyclone mesh). This approach results in the particles not being able to collide with the classifier at all, and any effect that the classifier has on the predicted particle motion is due entirely to how the classifier affected the continuous phase.

4.4.3.1 A-STERP Injection
The A-STERP injection method required 50 patterns containing 10000 injections in each pattern to achieve a steady TCE prediction for each device. The particle size distribution injected in each frame matched the distribution of the Portland cement that entered the cyclone in the experiments and is shown in Fig. 4.5. As the particle motion of interest is in the cyclone, the particles are injected near the cyclone inlet as opposed to the pipe inlet, thus reducing the computational resources required for particle tracking.

A drag model for non-spherical particles was applied to the particles injected into the numerical domains. This drag model is described in Chapter 3 and uses a shape factor ($\Phi$):

$$\Phi = \frac{s}{S}$$

(4.4)

where $s$ is the surface area of a sphere having the same volume as the particle, and $S$ is the actual surface area of the particle. In Chapter 3, the shape factor was tuned to achieve a numerically predicted TCE that matched the experimentally measured TCE for the truncated cyclone in a non-dynamic configuration and for Portland cement. The result of tuning $\varphi$ using the TCE value was $\Phi = 0.325$, and this value was used in the present Chapter.
4.4.3.2 Discreet Phase Modeling (DPM) Boundary Conditions

Two different types of boundary conditions were used for the discrete phase. The first was the "escape" condition, which was used at the outlet of the cyclone, as any particles that reach this boundary are considered to escape with the continuous phase. The second boundary condition was the "reflect" DPM condition and was assigned to the walls of the cyclone and collection hopper. The “reflect” condition had a coefficient of restitution set to 1.0, yielding an ideal collision where the particle retains all its kinetic energy. Setting the walls of the collection hopper to "reflect" allowed particles to enter and leave the collection hopper with the continuous phase, thus capturing the effects of particle re-entrainment.

4.4.4 Numerical Results

All comparisons made here between the predicted flow fields of the cyclones were carried out using the time-averaged flow fields utilized for the DPM simulations, as described in section 4.4.3.

4.4.4.1 Flow Field

The data for the pressure drop values calculated from the numerical simulations was taken from a similar position as the cyclone pressure drop measurements during the experiments. The predicted pressure drop from the Full-size, Intermediate and Truncated cyclone simulations with the benchmark collection hopper (CH1) and for classifier rotation of 4000 rpm are all 1.1 kPa. For the Truncated cyclone geometry, the pressure drop increased to 1.2 kPa when a vortex stabilizer was added to the collection hopper (CH2 and CH3) or the collection hopper diameter was increased (CH4 and CH5). The pressure drop across the truncated cyclone increased to 2.4 kPa for all collection hoppers when the classifier rotation was increased to 6000 rpm. The pressure drop and pressure coefficient values for each of the simulated cyclones are summarized in Table 4.6.
Velocity profiles are examined on a radial line formed by the intersection of the central (vertical) plane perpendicular to the cyclone inlet and the horizontal circular plane at the seam between the barrel/cone (see Fig. 4.2 for location). The velocity profiles are normalized by the inlet bulk velocity \(U_{in} = 12.7 \text{ ms}^{-1}\) and the radius of the barrel \((D = 0.496 \text{ m})\).

The tangential velocity profiles at the radial line for the Full-size, Intermediate and Truncated cyclone simulations with the benchmark collection hopper (CH1) and for classifier rotation of 4000 rpm are shown in Fig. 4.8. The Truncated cyclone geometry yielded the most prominent peak normalized tangential velocity \(U_{θmax}\) while the Full-size yielded the lowest. The peak normalized tangential velocities were 2.4, 2.8, and 3.0 for the Full-size, Intermediate, and Truncated cyclones, respectively, and occurred at normalized radial positions \(R_{θmax} = 0.23, 0.18, 0.19\), respectively. Using the values of \(U_{θmax}\) and \(R_{θmax}\) to normalize \(U_θ\) and \(r\), respectively, allows for comparison to a single Rankine vortex profile defined by Giaiotti and Stel [20] and resembling the formulation by Batterson et al. [21]:

\[
U_θ = \begin{cases} 
U_{θmax} \frac{r}{R_{θmax}} & \text{if } 0 \leq r < R_{θmax} \\
U_{θmax} \frac{R_{θmax}}{r} & \text{if } R_{θmax} \leq r
\end{cases}
\]
The comparison to the Rankine vortex described by Eq. 4.5 is shown in Fig. 4.9. The Truncated and Intermediate cyclone geometry with a classifier rotating at 4000 rpm yield tangential velocity profiles that resemble a Rankine vortex with a forced vortex core and a free vortex annulus. However, the Full-size cyclone geometry deviates from a Rankine vortex structure in the forced vortex core. Note that the Radial line used to display the tangential velocity profile is 13 mm below the classifier for the Intermediate and Truncated cyclone geometries (due to the reduced barrel height), while for the Full-size cyclone, the radial line is 348 mm from the classifier. The deviation from a Rankine vortex in the Full-size cyclone is attributed to the effect that the classifier has on the flow field away from the classifier.

The axial velocity profiles at the radial line are shown in Fig. 4.10. The three simulated cyclone geometries with a classifier rotating at 4000 rpm maintain the typical cyclone axial velocity profile with negative (towards the collection hopper) axial velocity near the cyclone wall and positive (towards the outlet) axial velocity away from the wall. However, the three cyclone geometries also show a chaotic axial velocity region at the core, resulting from the classifier trying to pull air in from below.
Figure 4.9: Rankine vortex tangential velocity profiles along the radial line shown in Fig. 4.2 for the three cyclone geometries in standard and dynamic configurations ($\omega = 4000 \text{ rpm}$).

Figure 4.10: Normalized axial velocity profiles (positive indicates flow towards the outlet, and negative indicates flow towards the collection hopper) along the radial line shown in Fig. 4.2 for the three cyclone geometries in the standard and dynamic configurations ($\omega = 4000 \text{ rpm}$).
A similar comparison of velocity profiles was conducted for the Truncated cyclone geometry with each collection hopper and with the classifier rotating at 4000 rpm and 6000 rpm. The normalized tangential velocity profiles for simulations at 4000 rpm and 6000 rpm are shown in Fig. 4.11 A and B, respectively. The values of $U_{\theta max}$ and $R_{\theta max}$ for the ten simulations are summarized in Table 4.7. For the simulations with the classifier rotating at 4000 rpm, the values of $U_{\theta max}$ and $R_{\theta max}$ are within 0.1 and 0.2, respectively, for each collection hopper except for CH5, which yielded a larger $U_{\theta max}$ value and a lower $R_{\theta max}$ value. This trend is also present in the simulations at 6000 rpm. Using the values of $U_{\theta max}$ and $R_{\theta max}$ in Table 4.7, the velocity profiles in Fig. 4.12 A and B were generated for comparison to a Rankine vortex for 4000 rpm and 6000 rpm, respectively. The curves shown in Fig. 4.12 A are similar to each other, as is the case in Fig. 4.12 B, with the predicted free vortex being stronger than the Rankine vortex and the predicted forced vortex being stronger near the transition point and weaker near the centre when compared to the analytical model.

The normalized axial velocity profiles for the Truncated cyclone geometry with each collection hopper are shown in Fig. 4.13 A and B for the classifier rotating at 4000 rpm and 6000 rpm, respectively. The curves shown in Fig. 4.13 display a similar trend to the normalized axial velocity profiles shown in Fig. 4.10. Figure 4.13 shows that the collection hoppers with vortex stabilizers (CH2 and CH3) yield more substantial negative axial flow at the centre of the cyclone compared to those with no vortex stabilizer present. Additionally, increasing the diameter of the collection hopper reduced the negative axial flow at the centre, with CH5 maintaining a positive axial velocity at the centre.

Table 4.7: Summary of the numerically predicted peak tangential velocity.

<table>
<thead>
<tr>
<th>Cyclone</th>
<th>$U_{\theta max} / U_{in}$</th>
<th>$R_{\theta max} / D$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4000 rpm</td>
<td>6000 rpm</td>
</tr>
<tr>
<td>Truncated CH1</td>
<td>3.0</td>
<td>4.6</td>
</tr>
<tr>
<td>Truncated CH2</td>
<td>3.1</td>
<td>4.7</td>
</tr>
<tr>
<td>Truncated CH3</td>
<td>3.1</td>
<td>4.7</td>
</tr>
<tr>
<td>Truncated CH4</td>
<td>3.0</td>
<td>4.5</td>
</tr>
<tr>
<td>Truncated CH5</td>
<td>3.3</td>
<td>4.4</td>
</tr>
</tbody>
</table>
Figure 4.11: Normalized tangential velocity profile along the radial line shown in Fig. 4.2 for the five collection hopper geometries on the dynamic Truncated cyclone, for (A) $\omega = 4000$ rpm and (B) $\omega = 6000$ rpm.
Figure 4.12: Rankine vortex tangential velocity profiles along the radial line shown in Fig. 4.2 for the five collection hopper geometries on the dynamic Truncated cyclone, for (A) $\omega = 4000$ rpm and (B) $\omega = 6000$ rpm.
Figure 4.13: Normalized axial velocity profiles (positive indicates flow towards the outlet, and negative indicates flow towards the collection hopper) along the radial line shown in Fig. 4.2 for the five collection hopper geometries on the dynamic Truncated cyclone, for (A) $\omega = 4000$ rpm and (B) $\omega = 6000$ rpm.
The mass flow rate through the 2 mm ribbon under the vortex finder shown in Fig. 4.2, was calculated for each simulated case and normalized by the mass flow rate through the device and is summarized in Table 4.8. This ribbon is around the rotating classifier, which results in the mass flow rates being radially outward. Thus, the rotating classifier eliminates any flow from shortcutting under the vortex finder. The negative values in Table 4.8 indicate that flow heading towards the cyclone's outlet is diverted back towards the cyclone walls, resulting in a fluid parcel travelling through the cyclone multiple times.

The tangential velocity profiles at the radial line inside the collection hopper shown in Fig. 4.3, for all of the collection hoppers and for classifier rotational speeds of 4000 and 6000 rpm are shown in Figs. 4.14 A and B, respectively. All tangential velocity profiles shown in Fig. 4.14 are normalized by the inlet bulk velocity ($U_{in} = 12.7 \text{ms}^{-1}$) and the radius of the barrel ($D = 0.496 \text{m}$). The normalized values of $U_{\theta \text{max}}$ and $R_{\theta \text{max}}$ for the velocity profiles shown in Fig. 4.14 are summarized in Table 4.9.

<table>
<thead>
<tr>
<th>Cyclone</th>
<th>Shortcutting Flow [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4000 rpm</td>
</tr>
<tr>
<td>Full-Size</td>
<td>CH1</td>
</tr>
<tr>
<td>Intermediate</td>
<td>CH1</td>
</tr>
<tr>
<td>Truncated</td>
<td>CH1</td>
</tr>
<tr>
<td>Truncated</td>
<td>CH2</td>
</tr>
<tr>
<td>Truncated</td>
<td>CH3</td>
</tr>
<tr>
<td>Truncated</td>
<td>CH4</td>
</tr>
<tr>
<td>Truncated</td>
<td>CH5</td>
</tr>
</tbody>
</table>

Table 4.8: Summary of the numerically predicted shortcutting flow.

<table>
<thead>
<tr>
<th>Cyclone</th>
<th>$U_{\theta \text{max}} / U_{in}$</th>
<th>$R_{\theta \text{max}} / D$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4000 rpm</td>
<td>6000 rpm</td>
</tr>
<tr>
<td>Truncated CH1</td>
<td>1.1</td>
<td>1.9</td>
</tr>
<tr>
<td>Truncated CH2</td>
<td>0.5</td>
<td>1.6</td>
</tr>
<tr>
<td>Truncated CH3</td>
<td>0.6</td>
<td>0.9</td>
</tr>
<tr>
<td>Truncated CH4</td>
<td>0.4</td>
<td>0.4</td>
</tr>
<tr>
<td>Truncated CH5</td>
<td>0.1</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Table 4.9: Summary of the numerically predicted peak tangential velocity along a radial line inside the collection hoppers
Figure 4.14: Normalized tangential velocity profile along the radial line shown in Fig. 4.3 for the five collection hopper geometries on the dynamic Truncated cyclone, for (A) $\omega = 4000$ rpm and (B) $\omega = 6000$ rpm.
4.4.4.2 Predicted Collection Efficiencies

The Full-size and Intermediate cyclone geometries with the benchmark collection hopper yielded 100% TCE of the simulated Portland cement when the classifier rotational speed is 4000 rpm. The TCE of the Truncated cyclone with the benchmark collection hopper is 86% when the classifier rotational speed is 4000 rpm, and decreases to 75% when the classifier rotational speed is increased to 6000 rpm. The predicted TCE of the simulated Portland cement for all numerical cases are summarized in Table 4.10. The FCE curves are presented in Fig. 4.15 for the three cyclone geometries and Fig. 4.16 for the five collection hopper geometries. Both Fig 4.15 and Fig. 4.16 show that any particle larger than 7 μm will be captured for every case except for the Truncated cyclone with the benchmark collection hopper and classifier rotational speed of 6000 rpm.

The number of times the particle passed between the cyclone and the collection hopper was recorded to monitor particle re-entrainment from the collection hopper. The simulations of the Full-size and the Intermediate cyclones yielded no particle re-entrainment. The Truncated cyclone with the benchmark collection hopper had a small amount (1% of injected particle mass) of re-entrainment when the classifier rotational speed is 4000 rpm. However, re-entrainment increased to 17% when the classifier rotational speed was increased to 6000 rpm. Increasing the diameter of the collection hopper or adding a vortex stabilizer to the collection hopper effectively eliminated particle re-entrainment.

Table 4.10: Summary of the numerically predicted total collection efficiency.

<table>
<thead>
<tr>
<th>Cyclone</th>
<th>TCE [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4000 rpm</td>
</tr>
<tr>
<td>Full-Size CH1</td>
<td>100</td>
</tr>
<tr>
<td>Intermediate CH1</td>
<td>100</td>
</tr>
<tr>
<td>Truncated CH1</td>
<td>86</td>
</tr>
<tr>
<td>Truncated CH2</td>
<td>89</td>
</tr>
<tr>
<td>Truncated CH3</td>
<td>90</td>
</tr>
<tr>
<td>Truncated CH4</td>
<td>87</td>
</tr>
<tr>
<td>Truncated CH5</td>
<td>88</td>
</tr>
</tbody>
</table>
Discussion

Table 4.11 shows the percent difference \( (P_D) \) between the numerically predicted and experimentally measured performance metrics, where the percent difference is defined as:

\[
P_D = \frac{|C_{exp} - C_{num}|}{C_{exp}}
\]  

(4.6)

with \( C_{exp} \) being an experimentally measured value for a given parameter and \( C_{num} \) being the numerically predicted value for the same parameter.

Table 4.11: Summary of the percent difference between the numerically predicted and experimentally measured total collection efficiency, pressure drop, pressure coefficient.

<table>
<thead>
<tr>
<th>Cyclone Geometry</th>
<th>Collection Hopper</th>
<th>( \omega ) [rpm]</th>
<th>( PE ) of TCE [%]</th>
<th>( PE ) of ( \Delta P ) [kPa]</th>
<th>( PE ) of ( \Delta c_p ) [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full-Size</td>
<td>CH1</td>
<td>4000</td>
<td>16</td>
<td>27</td>
<td>27</td>
</tr>
<tr>
<td>Intermediate</td>
<td>CH1</td>
<td>4000</td>
<td>18</td>
<td>27</td>
<td>27</td>
</tr>
<tr>
<td>Truncated</td>
<td>CH1</td>
<td>4000</td>
<td>1</td>
<td>31</td>
<td>31</td>
</tr>
<tr>
<td>Truncated</td>
<td>CH1</td>
<td>6000</td>
<td>7</td>
<td>20</td>
<td>20</td>
</tr>
</tbody>
</table>

Figure 4.15: Predicted fractional collection efficiencies of the three cyclone geometries in standard and dynamic configuration (\( \omega = 4000 \) rpm).
Figure 4.16: Predicted fractional collection efficiencies for the five collection hopper geometries on the dynamic Truncated cyclone, for (A) $\omega = 4000$ rpm and (B) $\omega = 6000$ rpm.
The $PE$ of the TCE was less than or equal to 18% for the four cases examined. For each cyclone shown in Table 4.5, the numerical simulations overpredicted the TCE compared to the experimentally measured TCE. The overpredicted TCE is common in the literature on dynamic cyclones [4, 5]. The overprediction of the TCE may be the result of neglecting collision between particles. For the $PE$ of $\Delta P$ and $\Delta c_p$, the Truncated cyclone geometry yielded the lowest (20%) and the highest (31%) values. The lowest $PE$ of $\Delta P$ and $\Delta c_p$ was for a rotational speed of 6000 rpm, and the highest $PE$ of $\Delta P$ and $\Delta c_p$ was for a rotational speed of 4000 rpm. The magnitude of the $PE$ for $P$ and $\Delta c_p$ is consistent with values found in literature, which range between 15% and 36% [22-25]. The underprediction in the pressure drop obtained from the CFD simulations are at least in part due to the smooth walls and perfect transitions that exist in the numerical model while the physical cyclone have a certain degree of wall roughness and bolted joints.

The dynamic cyclone results presented in this Chapter (Sections 4.3.4 and 4.4.4) are compared to the standard cyclone with collection hopper results from Chapter 3. Table 4.12 summarize the experimental and numerical results from Chapter 3.

By adding the rotating classifier to the Truncated cyclone geometry with the benchmark collection hopper (CH1), the experiments and numerical studies showed initial improvements to the TCE. However, with increasing the rotational speed beyond 4000 rpm results in a significant drop in the TCE.

Table 4.12: Summary of numerical modeling and experimental (shown in parentheses) results obtained in Chapter 3 for standard cyclones with a collection hopper.

<table>
<thead>
<tr>
<th>Cyclone</th>
<th>$\Delta P$ [kPa]</th>
<th>$\Delta c_p$ [%]</th>
<th>TCE [%]</th>
<th>$\frac{U_{\theta,max}}{U_{in}}$</th>
<th>Short-cutting Flow [%]</th>
<th>Short-cutting Particles [%]</th>
<th>Particle Re-entrainment [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full-size</td>
<td>0.6 (0.9)</td>
<td>6.1 (9.1)</td>
<td>98 (83)</td>
<td>1.9</td>
<td>26</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>Intermediate</td>
<td>0.7 (1.1)</td>
<td>7.4 (11.1)</td>
<td>94 (85)</td>
<td>2.2</td>
<td>29</td>
<td>2</td>
<td>0</td>
</tr>
<tr>
<td>Truncated</td>
<td>0.8 (1.2)</td>
<td>8.4 (12.1)</td>
<td>82 (82)</td>
<td>2.5</td>
<td>36</td>
<td>3</td>
<td>0</td>
</tr>
</tbody>
</table>
The numerical study in this Chapter showed that adding a rotating classifier not only eliminates any flow from shortcutting under the vortex finder, as Jiao et al. [3] stated, but diverts flow entering the classifier from below back towards the cyclone walls. The elimination of shortcutting flow directly under the vortex finder also prevented particles from shortcutting. The numerical study in this Chapter also showed that adding a rotating classifier increased the strength of the tangential velocity component, examined by the peak normalized tangential velocity, and impacted the amount of shortcutting. The increased tangential velocity of the flow inside the cyclone was responsible for both the initial improvement (added by elimination of particle shortcutting) and the significant drop in the TCE.

The strong tangential flow created by the rotating classifier was able to extend further down the cyclone and into the collection hopper. Thus the rotating classifier introduced particle re-entrainment when using the benchmark collection hopper. With the classifier rotating at 4000 rpm a small amount of the injected powder was re-entrained from the collection hopper, but as the rotational speed increased so did the amount of particles being re-entrained. The decreases in the TCE seen when the classifier was rotating at 6000 rpm could also be attributed to particle attrition (fragmentation) [26]. The strength of the vortex in the dynamic cyclone with a classifier rotating at 6000 rpm would be equivalent to a standard cyclone with an inlet speed of 25-30 ms$^{-1}$, which is within the normal operation range of cyclones [2, 22, 24, 27, 28].

The numerical study showed that the effect the rotating classifier had on the Truncated cyclone was similar for the Full-size and Intermediate cyclones. The rotating classifier improved the TCE for the Full-size and Intermediate cyclones. However, the effect of the classifier on the collection efficiency achieved by the Truncated and Full-size cyclones was not fully realized, as both predicted 100% collection efficiency (the physical limit) when the classifier rotational speed is 4000 rpm.

The addition of the rotating classifier was not without drawbacks. The classifier requires additional power input to rotate. The power input from the numerical simulations is based on the torque the pressure forces create on the blades of the classifier and, thus, does not
include power associated with rotating the mass of the classifier, or losses due to friction or motor efficiency. The power requirements of the classifier in the numerical simulation for the Full-size cyclone was the highest of the three dynamic cyclones operating at 4000 rpm. Each scale reduction in the cyclone height reduced the required power, with a more considerable power reduction coming from the shortening of the barrel height. The additional power imparted to the flow by the rotating classifier was also observed in the pressure drop across the cyclone. The rotating classifier pulls air from above and below it, where the air is exiting the classifier, and forces it radially outwards to the cyclone walls. Thus, a higher pressure drop across the cyclone is required to maintain the same mass flow rate. The radial outward flow created by the classifier was evident in the experimental setup. With the vacuum truck disconnected from the cyclone and a classifier rotational speed of 4000 rpm, the airflow in the inlet pipe was reversed, and the air was exiting what was typically the inlet. In both the experiments and numerical simulations, the pressure drop of the dynamic cyclones were similar between the three cyclone geometries. The increase in the pressure drop for the Full-size cyclone with the classifier rotating at 4000 rpm was the same as the 0.5 kPa pressure drop increase found in Zhou et al. [5] for an inlet speed of 15 ms\(^{-1}\) and a rotational speed of 4000 rpm.

Increasing the rotational speed of the classifier further emphasized its drawbacks. The energy requirements of the classifier increased by a factor of 3 when the rotational speed was increased from 4000 rpm to 6000 rpm in the truncated cyclone geometry. Additionally, the pressure drop across the cyclone increased by a factor of 2 with the increase in the rotational speed.

Adding the rotating classifier to the truncated cyclone showed the potential to recover the collection performance lost in the scale reduction of the cyclone height. However, at high rotational speeds, particle re-entrainment mitigated any benefit of the classifier. The design of the collection hopper was altered to reduce the effect of particles being re-entrained from the collection hopper. The numerical study showed that adding a flat or cone vortex stabilizer to CH1 (CH2 and CH3, respectively) or doubling / quadrupling the diameter of CH1 (CH4 and CH5, respectively) slightly increases the TCE compared to the TCE achieved by CH1, when the classifier is rotating at 4000 rpm. When the rotational speed
was increased to 6000 rpm, the effect of the alternative collection hoppers (CH2-CH5) on the predicted TCE was more prominent. The alternative collection hoppers, at both 4000 rpm and 6000 rpm, prevented particles from being re-entrained. Thus, the significant increase in the TCE seen in the alternative collection hoppers with the classifier rotating at 6000 rpm was attributed to the high particle re-entrainment present with CH1. The increase in the TCE, and FCE curves with the increasing rotational speed of the classifier seen with the alternative collection hoppers coincide with the increase in the FCE curves found in the literature on dynamic cyclones [3-5].

When the rotational speed was increased from 4000 rpm to 6000 rpm, the vortex that extended into the collection hopper also increased in strength.

The increases in the tangential velocity in the collection hopper prevented the particles from settling out of the bulk flow and thus re-entrained them into the cyclone body. The elimination of particle re-entrainment obtained by the alternative collection hoppers (CH2-CH5) was achieved in two different ways. Collection hoppers CH2 and CH3 both maintain a strong tangential velocity within the collection hopper. Additionally, the magnitude of the peak tangential velocity is dependent on the rotational speed of the classifier. The peak tangential velocity in CH2 with a rotational speed of 6000 rpm was greater than the peak tangential velocity in the smaller CH1 with a rotational speed of 4000 rpm, and yet had no particle re-entrainment. The elimination of particle re-entrainment in CH2 and CH3 is, thus, attributed to the physical barrier created by the vortex stabilizers used in these two collection hopper designs. By doubling the diameter of the benchmark collection hopper (CH4), the peak tangential velocity dropped by 60% and was not affected by the rotational speed of the classifier. Increasing the diameter of the collection hopper further (CH5) resulted in almost complete elimination of the tangential velocity component in the collection hopper.

Of the four alternative collection hoppers CH4 provided the smallest improvement from the TCE yielded by CH1 at rotational speeds of 4000 rpm and 6000 rpm. For the simulations at 4000 rpm, the collection hopper that gave the highest TCE was CH5, followed by CH2 and CH5. In contrast, for the simulations at 6000 rpm, the collection
hopper that gave the highest TCE was CH5, followed by CH3 and CH2. Although the four alternative collection hoppers yielded different TCE predictions, the span from the lowest to largest TCE was only 3%, regardless of rotational speed. The effect of the alternative collection hopper design on the dynamic cyclones was primarily limited to the collection performance. The pressure drops across the Truncated cyclone with the alternative collection hoppers and the classifier rotating at 4000 rpm or 6000 rpm were similar magnitude compared to the benchmark collection hopper. A similar trend was seen for the power added to the flow by the classifier.

The increase in the TCE seen in the dynamic Truncated cyclones resulted from the combination of the rotating classifier and alternative collection hopper. Changing the benchmark collection hopper (CH1) to one of the alternate collection hoppers (CH2-CH5) was insufficient to increase the collection performance. Simulation of the classifier rotating at 4000 rpm in the Truncated cyclone demonstrated that the benchmark collection hopper (having minimal particle re-entrainment) could achieve a similar TCE to the alternative collection hoppers. To further show that the alternative collection hoppers are insufficient at increasing the TCE, the truncated cyclone with CH4 was simulated without a rotating classifier. This simulation yielded a TCE of 81%, which is 1% less than the TCE achieved by the Truncated standard cyclone with the benchmark collection hopper. This shows that the rotating classifier is responsible for the improved TCE, provided that particle re-entrainment is not present.

4.6 Conclusion

The performance of three dynamic cyclone particle separators was investigated experimentally and by using computational fluid dynamics (CFD) simulations to understand the impact that a rotating classifier and alternate collection hoppers have on the collection performance. The experimental results showed that adding a classifier that rotates at 4000 rpm increased the Total Collection Efficiency (TCE) by 3% compared to a Truncated cyclone without a classifier. Further increases to the rotational speed resulted in reductions to the TCE, with 4500 rpm and 6000 rpm yielding reductions in the TCE of 2% and 12%. When compared to a Truncated cyclone without a classifier, the pressure drop increased by 0.4 kPa when the classifier was rotating at 4000 rpm and by 1.2 kPa when
the classifier was rotating at 6000 rpm.

The numerical results showed a similar trend to the experiments; both showed a slight increase in the TCE when the classifier was rotating at 4000 rpm and higher rotational speeds, yielding a reduction in the TCE. The critical flow features leading to the increase in the TCE observed at a rotational speed of 4000 rpm are the increase in the vortex strength and the elimination of flow and particle shortcutting under the vortex finder. When compared to a Truncated cyclone without a classifier, the vortex strength increased by 0.5 when the classifier was rotating at 4000 rpm and by 2.1 when the classifier was rotating at 6000 rpm. The feature attributed to the reduction in TCE at higher rotational speeds was particle re-entrainment from the collection hopper. When the classifier was rotating at 4000 rpm, there was minimal particle re-entrainment, whereas when the classifier was rotating at 6000 rpm, 17% of the injected particles entered, then exited the collection hopper and then the cyclone.

The numerical investigation into five collection hopper designs allowed for examining particle re-entrainment in the dynamic Truncated cyclone. The collection hopper designs did not significantly impact the internal flow structure when examining the pressure drop or vortex strength. However, changing the collection hopper design from CH1 to any other designs (CH2-CH5) eliminated particle re-entrainment. As a result, the predicted TCE for the dynamic Truncated cyclones was minimally impacted (less than 5% change) by the choice of collection hoppers when the classifier was rotating at 4000 rpm. When the classifier was rotating at 6000 rpm, the TCE was significantly increased (greater than 19% change) by switching from CH1 to any of the other collection hoppers.

The dynamic Truncated cyclone with a CH3 and a classifier rotating at 6000 rpm achieved the highest TCE (97%) and nearly recovered the TCE of the Full-size cyclone without a classifier. The dynamic Truncated cyclone in this configuration fell short of the Full-size TCE (98%) by 1% in the numerical simulations.
4.7 References


Chapter 5

5 Summary, Contributions and Suggestions for Future Research

The following chapter will summarize the contributions made in Chapters 2-4 and relate them to the objectives laid out in Chapter 1. Potential future research based on the contributions made in this thesis is then discussed, followed by a final summary.

5.1 Summary and Contributions

The work presented in this thesis discussed methods of achieving high collection efficiency for a scale reduced cyclone particle separator using experimental and numerical evaluations. The primary method used to improve the collection efficiency of the scale reduced cyclone was to add a rotating classifier to the cyclone, yielding a dynamic cyclone. The three cyclone geometries examined were Full-size, Intermediate and Truncated cyclones. The Full-size resembled a Stairmand cyclone design. The Intermediate cyclone maintained the same cross-sectional dimensions as the Full-size except for the barrel height being halved. The Truncated cyclone also maintained the same cross-sectional dimensions as the Full-size, except that both the barrel height and the cone height were halved.

The specific contributions made in the thesis were:

1. (Objective 1) development of the Aggregate STEady random Particle (A-STERP), approach for injecting particles into a Computational Fluid Dynamic (CFD) domain that accurately represents a dilute pneumatic transport injection of particles while being computationally efficient compared to a fully transient injection method (requiring less than 0.05% of the computational time for cyclones),

2. (Objective 2) identification of increased flow and particle shortcutting due to the scale reduction of a cyclone particle separator,

3. identification of the need for including the collection hopper in CFD simulations to account for particle re-entrainment and

4. (Objective 3) improvement of the collection performance of a scale reduced
cyclone particle separator by adding a rotating classifier to eliminate shortcutting and increasing the collection hopper diameter or adding a vortex stabilizer to prevent particle re-entrainment, thus, allowing for the industrial partner (DBM) to integrate a scale reduced cyclone into their vacuum trucks without sacrificing collection performance.

In Chapter 2, the A-STERP injection approach was developed to reduce the computational time required by the Discrete Phase Model (DPM) to model random cloud injection in simulations of particle motion inside a cyclone particle separator. The injection method commonly used in the literature on cyclone particle separators was a *surface injection*, which depends on the grid representing the numerical domain. The constant injection locations yielded by the surface injection method were an unrealistic representation of a dilute pneumatic transport of particles entering the domain. However, a surface injection method can be used with steady or transient particle tracking, and when steady particle tracking is used, it can be very computationally efficient. The alternative to the surface injection was to use a randomized transient (unsteady) injection method where the injection location of each particle changed with time. However, this method required transient tracking of particles, necessitating a significant amount of computational resources. The unsteady injection method with transient tracking would have potentially taken six months to complete particle tracking for each cyclone simulated in Chapters 3 and 4, based on the time and computation resources used by the cyclone considered in Chapter 2. The A-STERP injection method uses a series of generated particle injection patterns containing a set number of randomly positioned injection locations, with the results from each pattern cumulatively averaged. The A-STERP injection method predicted a similar collection efficiency as the unsteady injection method and required a similar amount of computational resources as the surface injection method. Thus, the A-STERP injection method reduces the computational time to predict the collection efficiency for each cyclone from months to hours.

In Chapter 3, the CFD and DPM (using A-STERP injection) investigation on the Full-size, Intermediate, and Truncated cyclones showed that the collection performance decreased with each scale reduction. The collection performance of these three cyclones is shown in
Fig. 5.1 A-C. The reduction in collection performance with scale reduction was also found in the experiments on the three cyclones. The numerical simulation of the three cyclones also showed that, with each scale reduction, the degree of shortcutting flow increased. The Full-size cyclone experienced 26% of the inlet mass flow rate shortcutting. When the barrel section was reduced in height (Intermediate cyclone), shortcutting increased to 29%, whereas reducing the height of both the barrel and conical sections increased the shortcutting to 36%. The shortcutting flow results in less mass of the continuous phase moving in the vortex contained by the cyclone body. The shortcutting flow also pulls particles with it, preventing some particles from making sufficient rotations around the cyclone where they have a greater chance of being collected.

Also, in Chapter 3, simulations of the Truncated cyclone geometry, as well as the Full-size and Intermediate cyclones, were conducted with and without a collection hopper being a part of the numerical domain. The flow field and collection efficiency for the Full-size and Intermediate cyclones were found to be negligibly impacted by the exclusion of the collection hopper in the CFD domain. In contrast, the Truncated cyclone geometry...
experienced a drop in the predicted total collection efficiency of 20% when the collection hopper was not included (shown in Fig 5.1 C and D) and including the collection hopper in the numerical domain had the added benefit of allowing for numerically predicting particle re-entrainment from the collection hopper.

By neglecting the collection hopper in the CFD domain, any particle that collided with the bottom of the cyclone (where they would have entered the collection hopper) was considered trapped and tracking of the specific particle was, then, terminated. In contrast, when the collection hopper was included in the numerical domain, particles were allowed to enter and exit the collection hopper, allowing for particle re-entrainment to be simulated and, thus, quantified. The importance of accounting for particle re-entrainment was shown in Chapter 4, where changing the collection hopper design affected the amount of particle re-entrainment and the overall collection efficiency of the Truncated dynamic cyclone. The change in collection efficiency between different collection hoppers for the dynamic Truncated cyclone is also shown in Fig. 5.1 E-I.

In an effort to increase the collection performance of the Truncated cyclone by eliminating the increased shortcutting flow, a rotating classifier was added to the cyclone. The experiments and the numerical simulations in Chapter 4 showed that with the classifier rotating at 4000 rpm, the collection efficiency of the Truncated cyclone was improved from 82% to 85%, and particle re-entrainment was minimal (1%). However, when the rotational speed of the classifier was increased to 6000 rpm, the collection efficiency was reduced to 75% (70% from the experiments) and particle re-entrainment was increased to 17%.

The collection hopper required the addition of a vortex stabilizer or an increase in size to prevent particle re-entrainment induced by the rotating classifier. The addition of the vortex stabilizer created a physical barrier to the particles, ensuring that once the particles entered the collection hopper and moved past the vortex stabilizer, they were unable to re-enter the cyclone. Increasing the diameter of the collection hopper weakened the vortex inside it, thereby preventing the vortex from re-entraining the collected particles.

In Chapter 4, the addition of a rotating classifier and ensuring that the collection hopper design did not allow for particle re-entrainment, a scale reduced cyclone particle separator
was made to achieve a similar collection efficiency to the Full-size cyclone particle separator. The addition of the classifier resulted in the drawback of requiring additional energy input. Not only did the classifier require a motor to power it, but the rotation of the classifier resulted in a larger pressure drop across the cyclone. The pressure drop across the Full-size cyclone without a rotating classifier was 0.6 kPa. In comparison, the pressure drop of the Truncated cyclone with the largest collection hopper and a classifier rotating at 6000 rpm was 2.6 kPa. The increased pressure drop for the Truncated dynamic cyclone equates to the device used to move the air through the cyclone having to work harder to maintain the same flow rate.

## 5.2 Suggestions for Future Research

### 5.2.1 Classifier Blade Design

The classifiers used in this work and those presented in the literature on dynamic cyclones have had simple designs where the blades are flat and rectangular [1-5]. Zhou et al. [1] and Jiao et al. [2] examined the effect of angling the blades of the classifier by ± 20° and ± 30°, respectively. By angling the blades in a negative direction, the collection efficiency was found to increase, and the pressure drop across the cyclone was found to decrease [1, 2]. Thus, the classifier blade design has been shown to impact the performance of dynamic cyclones. However, the shape of the blades in both Zhou et al. [1] and Jiao et al. [2] were still flat and rectangular in both studies. The performance of a dynamic cyclone has the potential to be further improved by using more complex blade designs, such as curved blades or airfoil-shaped blades. Such blade designs are often used in centrifugal blowers [6-8]. Improving the classifier blade design would allow for reducing the additional power required by the scale reduced dynamic cyclone.

### 5.2.2 The Particle Location Distribution

The location of each particle injection used with A-STERP was uniformly randomly distributed across the injection area. Thus, the chance of a particle being located towards the centre of the injection area is just as likely as towards any corner or edge. The effect of having a uniform distribution of injection locations was not tested against other distributions, such as a normal distribution or a skewed distribution. A normal distribution
would have concentrated the injected particles towards the centre of the injection area where the flow velocity is the greatest. In comparison, a skewed distribution could increase the number of injected particles towards the bottom of the injection location, where gravity could have potentially concentrated them. Additionally, a combination of distributions could be implemented, for example a uniform distribution could be used for the horizontal coordinate, and a normal distribution could be used for the vertical component. Using a different distribution that is more realistic of dilute pneumatic transport [9, 10] might have an impact on the predicted performance of the cyclone particle separator or any particle separating device, or the effect might be mitigated by tracking the particles through a short section of the pipe leading to the device.

5.2.3 Development of Dynamic Cyclone Design Guidelines
One of the common design guidelines for cyclone particle cyclones, Stairmand [11], dates back to 1951 and provides designs for both high throughput and high collection efficiency. With the addition of the rotating classifier to a cyclone, creating a dynamic cyclone, this thesis has shown that the cyclone can be scale reduced while maintaining a similar collection efficiency. Thus, a set of geometry and operating guidelines specifically for dynamic cyclones could aid in the future design and implementation of dynamic cyclones in industry.

5.3 Summary
The final chapter of this thesis outlined the main contributions made to the cyclone and numerical particle tracking research fields and provided potential avenues for future research in these fields. The chapter linked and summarized chapters 2-4, showing how A-STERP had a critical role in determining that the collection performance of a scaled reduced cyclone particle separator can be recovered with the addition of a rotating classifier, provided re-entrainment is prevented.

5.4 References


# Curriculum Vitae

<table>
<thead>
<tr>
<th>Name:</th>
<th>Mark Parker</th>
</tr>
</thead>
<tbody>
<tr>
<td>Post-secondary Education and Degrees:</td>
<td>University of Ontario Institute of Technology Oshawa, Ontario, Canada 2012-2017 B.A.</td>
</tr>
<tr>
<td></td>
<td>The University of Western Ontario London, Ontario, Canada 2017-2019 M.A.</td>
</tr>
<tr>
<td></td>
<td>The University of Western Ontario London, Ontario, Canada 2019-2023 Ph.D.</td>
</tr>
<tr>
<td>Honours and Awards:</td>
<td>Best Graduate Student Presentation 2023</td>
</tr>
<tr>
<td></td>
<td>Mitacs Accelerate 0219-2023</td>
</tr>
<tr>
<td></td>
<td>Province of Ontario Graduate Scholarship 2021-2022</td>
</tr>
<tr>
<td>Related Work Experience</td>
<td>Professor</td>
</tr>
<tr>
<td></td>
<td>MECH-6003 Advanced Theory of Mechanical Design</td>
</tr>
<tr>
<td></td>
<td>MECH-6009 SPC &amp; Metrology</td>
</tr>
<tr>
<td></td>
<td>MACH-3057 CNC Applications &amp; Intro to CAD/CAM</td>
</tr>
<tr>
<td></td>
<td>Fanshawe College 2019-2023</td>
</tr>
<tr>
<td></td>
<td>Teaching Assistant</td>
</tr>
<tr>
<td></td>
<td>MME Graduate Seminar Series</td>
</tr>
<tr>
<td></td>
<td>MME-3381 Kinematics and Dynamics of Machines</td>
</tr>
<tr>
<td></td>
<td>MME-4452 Robotics and Manufacturing</td>
</tr>
<tr>
<td></td>
<td>MME-2204 Thermodynamics I</td>
</tr>
<tr>
<td></td>
<td>MME-4459 Advanced CAE: Manufacturing Technologies</td>
</tr>
<tr>
<td></td>
<td>MME-3307 Heat Transfer II</td>
</tr>
<tr>
<td></td>
<td>MME-2202 Mechanics of Materials</td>
</tr>
<tr>
<td></td>
<td>The University of Western Ontario 2017-2023</td>
</tr>
</tbody>
</table>
Publications:


