April 2014

**Numerical Simulation on Dilute Phase Pneumatic Transport**

Ethan T. Doan
*The University of Western Ontario*

Supervisor
Anthony G. Straatman
*The University of Western Ontario*

Graduate Program in Mechanical and Materials Engineering

A thesis submitted in partial fulfillment of the requirements for the degree in Master of Engineering Science

© Ethan T. Doan 2014

Follow this and additional works at: [http://ir.lib.uwo.ca/etd](http://ir.lib.uwo.ca/etd)

Part of the [Computer-Aided Engineering and Design Commons](http://ir.lib.uwo.ca/computer-aided_engineering_and_design_commons)

**Recommended Citation**

[http://ir.lib.uwo.ca/etd/1984](http://ir.lib.uwo.ca/etd/1984)

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 tadam@uwo.ca.
NUMERICAL SIMULATION OF DILUTE PHASE PNEUMATIC TRANSPORT

(Thesis format: Monograph)

by

Ethan Doan

Graduate Program in Mechanical and Materials Engineering

A thesis submitted in partial fulfillment
of the requirements for the degree of
Masters of Engineering Science

The School of Graduate and Postdoctoral Studies
The University of Western Ontario
London, Ontario, Canada

© Ethan Doan 2014
Abstract

Pneumatic conveying is a technique that is widely used in many industrial mechanical and chemical applications. In the case of cement manufacturing pneumatic conveying is a large scale operation moving several kilograms of material per second which consumes electrical energy (operation of fans) and money (replacement of filters to remove particles from the air). At St Mary’s Cement the pneumatic conveying line was studied with a CFD model. The treatment of the secondary solid phase was done with the DPM formulation in ANSYS Fluent and turbulence was modelled with k-ω SST. Some modifications and alterations to the system are suggested to improve the overall pressure drop. It was found that simple geometric alterations could reduce the pressure drop significantly while larger alterations such as the addition of a cyclone separator could increase the pressure drop over 50% and achieve a monetary savings by the increasing the life of the filters.

Keywords

CFD simulation, dilute phase pneumatic conveying, pressure drop
Acknowledgments

I would like to thank my advisor, Dr. Straatman for his continual assistance and guidance over the last 2 years. I thank my parents, Linda and Trent, and my siblings, Jeremy and Jessica, for giving me the motivation and confidence I need to be successful. Lastly, I would like to thank St Mary’s Cement Co. and Mitacs for providing me with the opportunity of writing this thesis and obtaining a graduate degree.
# Table of Contents

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

Acknowledgments...................................................................................................................... iii

Table of Contents ...................................................................................................................... iv

List of Tables ............................................................................................................................ v

List of Figures ............................................................................................................................ vi

Nomenclature .............................................................................................................................. xi

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

1 Intro to Problem .................................................................................................................... 1

1.1 Pneumatic conveying: ....................................................................................................... 4

1.1.1 Dilute Phase: ............................................................................................................... 5

1.1.2 Dense Phase: .............................................................................................................. 5

1.2 Goal of the project: .......................................................................................................... 6

1.3 Outline of Thesis .............................................................................................................. 7

Chapter 2 .................................................................................................................................... 9

2 Literature Review ................................................................................................................... 9

2.1 Pneumatic Transport ........................................................................................................ 10

2.2 Secondary Phase Numerical Approach .......................................................................... 11

2.3 Summary .......................................................................................................................... 14

Chapter 3 .................................................................................................................................. 15

3 Numerical Setup .................................................................................................................. 15

3.1 Governing equations: ...................................................................................................... 15

3.2 Gas-Solid Multiphase Flow: .......................................................................................... 17

3.2.1 Euler-Lagrange ......................................................................................................... 17

3.3 Turbulence: ...................................................................................................................... 18
List of Tables

Table 4.1: Qualitative and mathematical descriptions of the boundary conditions .......... 32

Table 4.2: Grid convergence for pneumatic transport duct ........................................ 37

Table 4.3: Grid convergence for knockout chamber ..................................................... 37

Table 5.1: Pressure drop comparison from a straight pipe test with the influence of particles ................................................................. 39

Table 5.2: Summary of results of the parametric study modifications for air-pressure drop, loaded-air pressure drop and percent change in pressure drop as compared to the original geometry ............................................. 53

Table 5.3: Summary of results of the simple modifications for air-pressure drop, loaded-air pressure drop and percent change in pressure drop as compared to the original geometry .... 56

Table 5.4: Cyclone specifications ................................................................................. 64

Table 5.5: Results for the analysis of the implementation of the cyclone separator ........ 66

Table 5.6: Summary of three cases of suggested modifications on the pneumatic transport system .................................................................................................................. 67
List of Figures

Figure 1.1: Energy Consumed per tonne of Cement produced in Canada [3] ..................... 1

Figure 1.2: Cement manufacturing process schematic ......................................................... 2

Figure 1.3: St Mary’s pneumatic conveying schematic .......................................................... 3

Figure 1.4: FBD of a particle in horizontal flow ................................................................. 4

Figure 1.5: Illustration of the particle loading and distribution in dilute phase conveying ..... 6

Figure 1.6: Illustration of the particle loading and distribution in dense phase conveying ..... 6

Figure 2.1: State diagram showing the boundaries of dense and dilute flow over various
velocities and system pressure drops [6] .................................................................................. 9

Figure 4.1: St Mary’s pneumatic conveying schematic ......................................................... 23

Figure 4.2: Roller mill interior schematic ............................................................................... 24

Figure 4.3: Side and bottom view of the idealized geometry of the roller mill at St Mary’s . 25

Figure 4.4: Simplified duct geometry ...................................................................................... 27

Figure 4.5: Turning vanes in the upper bend of the simplified duct geometry ....................... 27

Figure 4.6: Bottom segment of the simplified duct geometry including the cavity ............... 28

Figure 4.7: Knockout chamber geometry ............................................................................... 29

Figure 4.8: Lower half of the baghouse modelled with only a portion of each chamber, the
drawn portion shows a sample of how the geometry would look if the whole baghouse was
modelled .................................................................................................................................. 30

Figure 4.9: Internal geometry of an individual chamber in the baghouse ......................... 31

Figure 4.10: Top looking down view of the filter layout of an individual chamber in the
baghouse cut by a symmetry plane .......................................................................................... 32
Figure 4.11: Simplified roller mill geometry shown with coarse hexahedral mesh .............. 33

Figure 4.12: Knockout chamber with coarse hexahedral mesh ........................................ 34

Figure 4.13: Cross section of hexahedral mesh through the duct .................................... 34

Figure 4.14: Coarse tetrahedral mesh from the baghouse inlet plenum and chamber cross section .................................................................................................................................................. 35

Figure 4.15: Baghouse inlet plenum near wall fine prism mesh ....................................... 36

Figure 5.1: X, Y, and Z velocities at the outlet of the roller mill to be imposed as the inlet boundary condition for the duct .................................................................................................................................................. 41

Figure 5.2: Long inlet section of the pneumatic transport duct where the red cross-section corresponds to the uniform velocity and the blue cross-section is the actual inlet of the duct 42

Figure 5.3: Simplified duct geometry (blue to blue) with extended inlet and outlet sections (red to red) to accommodate imposition of boundary conditions ................................. 43

Figure 5.4: Illustration of the definition of r and D for its application to minor losses in the duct .................................................................................................................................................. 44

Figure 5.5: Showing the portions of the duct that are inside and outside of the buildings with the constrained locations highlighted in red .................................................................................. 45

Figure 5.6: Velocity vectors colored by magnitude for the original bend configuration ...... 46

Figure 5.7: Velocity vectors colored by magnitude for the r/D bend of 0.5 ....................... 47

Figure 5.8: Velocity vectors colored by magnitude for the r/D bend of 1 ......................... 48

Figure 5.9: Velocity vectors colored by magnitude for the r/D bend of 1.5 ....................... 48

Figure 5.10: Velocity vectors colored by magnitude for the modified r/d bend of 1 ........... 49

Figure 5.11: Flow Secondary flow vectors for the cross section of the duct immediately after the original 90 degree bend ............................................................................................................. 50
Figure 5.12: Secondary flow vectors for the cross section of the duct immediately after the modified 90 degree bend

Figure 5.13: Velocity vectors colored by magnitude for the flat plate modification 1

Figure 5.14: Velocity vectors colored by magnitude for the flat plate modification 2

Figure 5.15: Particle tracks of injected particles coloured by residence time inside the domain

Figure 5.16: Filter sections to measure accretion in different locations inside the baghouse

Figure 5.17: Particle accretion on different sections of filters

Figure 5.18: Baghouse existing inlet

Figure 5.19: Baghouse extended duct with 45 degree plate inlet

Figure 5.20: Baghouse fully extended duct inlet

Figure 5.21: Particle accretion for geometric modifications vs the normal case

Figure 5.22: A top down view of a cyclone separator showing a neutral vane

Figure 5.23: A top down view of a cyclone separator without a neutral vane
Nomenclature

**Roman Letters**

- $C_1, C_2, C_3$: constants for the dissipation rate equation
- $C_D$: drag coefficient
- $d$: diameter, m
- $F_d$: drag force 1/s
- $F$: body force, N/m$^3$
- $g$: gravitational acceleration, m/s$^2$
- $G$: generation
- $I$: Identity matrix
- $k$: turbulent kinetic energy, m$^2$/s$^2$
- $R_e$: Reynolds number
- $R$: gas constant, J/kg'k
- $S$: source term
- $t$: time, s
- $\overline{T}$: stress tensor
- $u$: velocity, m/s
- $\vec{v}$: velocity, m/s
- $\vec{v}^T$: transposed velocity, m/s

**Greek Letters**

- $\Delta$: change in a property
- $\nabla$: gradient
- $\varepsilon$: dissipation rate, m$^2$/s$^3$
\( \omega \) specific dissipation rate, \( 1/\text{s} \)

\( \mu \) dynamic viscosity of a fluid, \( \text{kg/m}\cdot\text{s} \)

\( \mu_T \) eddy viscosity of a fluid, \( \text{kg/m}\cdot\text{s} \)

\( \rho \) density, \( \text{kg/m}^3 \)

\( \sigma \) turbulent Prandtl number

\( \Gamma_k \) diffusivity of turbulent kinetic energy, \( \text{kg/m}\cdot\text{s} \)

\( \Gamma_\omega \) diffusivity of specific dissipation rate, \( \text{kg/m}\cdot\text{s} \)

**Subscripts**

\( k \) turbulent kinetic energy

\( p \) particle

\( s \) solid

\( \varepsilon \) dissipation rate

\( \omega \) specific dissipation rate

**Abbreviations**

DEM Discrete Element Method

DPM Discrete Particle Method

FBD Free Body Diagram

PBC Periodic Boundary Condition

RNG Re-Normalization Group

SST Shear Stress Transport
Chapter 1

1 Intro to Problem

Every day new infrastructure is being built worldwide using cement as a key component. Buildings, roads, bridges, and sidewalks can all use some type of cement as a building material. For this reason the cement industry as a whole produced more than 3.4 billion tonnes of cement in 2011. As worldwide development has only been increasing, this number is also growing, up 47% from the reported 2005 values [1]. Going hand-in-hand with its massive scales of production are its massive scales of energy consumption. Roughly 2% of the world’s energy production is consumed by the manufacturing of cement [2]. On average this equates to between 4 and 5 GJ of energy consumed per tonne of cement produced. In Canada the energy consumption is below the average hovering around 3.8 GJ per tonne. The reason for this could include many factors such as the availability and type of fuel used, and worldwide prices. Using cheaper fuel offers less incentive to reduce energy consumption. Most of this energy consumption comes from the burning of natural gas or some other fuel during the process of calcination and sintering, however, 12-15% of the 4-5 GJ consumption is reported to be electrical energy consumption [3]. Statistics from the Cement Association of Canada show there has been no improvement in electrical energy efficiency in cement production in over 20 years. Figure 1.1 shows the total energy consumed per tonne of cement produced and its breakdown into thermal and electrical energy.

![Energy per Tonne of Cement](image)

Figure 1.1: Energy Consumed per tonne of Cement produced in Canada [3]
The manufacturing of cement is a complicated process with many steps and sequences. A complete process diagram can be found in Figure 1.2. The following is a rough outline of the steps involved (see numbered boxes in figure 1.2):

1: Acquisition of raw materials
2: Grinding, drying and conveying
3: Blending and storage
4: Preheating, calcination, sintering and cooling
5: Finish milling, storage and dispatch

![Cement manufacturing process schematic](image)

**Figure 1.2: Cement manufacturing process schematic**

The largest consumer of thermal energy in this process is the calcination and sintering, done in the kiln roller, whereas the largest consumers of electrical energy are the grinding, drying, conveying, and finish milling.

A St Mary’s Cement production facility located in St Mary’s Ontario reports that approximately 40 percent of its total operating costs go towards the purchase of fuel and electrical energy. As previously stated, the electrical costs come from the grinding, drying, and conveying of raw material. An area suitable to make improvements to the electrical efficiency would be the pneumatic conveying system that moves raw meal from
the roller mill to storage before it is further processed. Figure 1.3 shows a schematic of the pneumatic transport configuration at the St. Mary’s plant. The sketch is not to scale. Although some approximate dimensions are given below, the detailed geometry will be discussed in subsequent sections.

Roller mill: diameter = 5.5 m, height = 10.5 m

Transport duct: height = 25 m, cross section = 2x2 m

Knockout chamber: height = 10 m, length = 5 m depth = 7.5 m

Baghouse: height = 12.5 m, length = 20 m, depth = 10.5 m

Waste hot air from the kiln is injected into the roller mill via radial nozzles that create a swirling flow. The swirling air entrains fine particles of limestone that are crushed by the mill. The now loaded air enters the transport duct where it is carried to the knockout chamber and through the filters. In the knockout chamber and the baghouse particles are filtered and fall to the bottom conveyor to be taken away to storage while the air continues out of the baghouse.
1.1 Pneumatic conveying:

Pneumatic conveying, or transport, is the use of flowing gases to move solid materials. When a particle is moving through a fluid it has a number of forces acting upon it. It can experience lift, drag (friction and form), gravity, and buoyancy. Lift and drag forces depend on the shape and orientation of a particle relative to the flow. Lift acts perpendicular to its motion while drag acts opposite to motion. The gravity force is constant, depends on particle mass and always acts downward. The buoyancy force opposes gravity and depends on the density of the fluid and the volume of the particle. In pneumatic conveying we can ignore the buoyancy effects since the density of air is small. An FBD of a particle in horizontal flow is shown in figure 1.4.

![Figure 1.4: FBD of a particle in horizontal flow](image)

The lift and drag force between the air and the particle are the forces that make pneumatic conveying possible. When a fluid moves over a particle the net drag force causes it to move in the direction of the flow while a lift force will move it from sitting on a surface to being fully entrained in the flow. Lift on an spherical particle is primarily caused by a reaction force between the particle and the air. When the shape of a particle deflects air a certain way the reaction force on the particle is known as lift. If considering spherical particles this type of lift cannot exist but there are two others that can be taken into account. The Magnus lift force arises due to the spin of a particle while the Saffman force arises due to a velocity gradient causing a pressure differential. The Magnus lift force is typically important for large particles with diameters on the order of millimetres or larger [4]. The Saffman lift force is negligible except in cases when the particle Reynolds
number is less than one [5]. There is more complicated physics that can be explored but these fundamental forces are the main factors in effect during pneumatic conveying and this explanation is sufficient to understand the subsequent chapters. Two main types of transport are discussed; dilute and dense. Both can be done with a negative pressure (vacuum) or a positive pressure system. In each case a pressure differential is achieved between the beginning and end of the system which causes the movement of fluid from high to low pressure. A vacuum system will have fewer problems with leaks since, in the event of an opening, the vacuum pressure will draw outside air in keeping the entrained particles inside the system. With a positive pressure system a hole will discharge the conveying fluid and/or particles since the pressure inside is greater than atmospheric. In a vacuum system a limitation of 1 atmosphere is placed on the system differential pressure, while in a positive pressure system the differential can be several times greater.

1.1.1 Dilute Phase:

Dilute phase conveying has particles fully suspended in the air. In dilute conveying depending on the material properties the designer must know a choking velocity, the velocity at which particles become unsuspended in vertical transport, and a saltation velocity, the velocity at which particles become unsuspended in horizontal transport. To avoid unwanted particle drop out the air velocity must always be greater than the lower limit velocity. For this reason dilute phase transport usually has relatively high gas velocities and relatively low differential pressures. It is more likely a dilute phase conveying system will use vacuum pressure because it is safer and the limitation of 1 atmosphere of differential pressure is enough for the desired output. This type of transport is good at moving and drying material at the same time.

1.1.2 Dense Phase:

Dense phase conveying, or pulse conveying, moves large amounts of particles in waves. The dense clusters of material are moved along with high pressure blasts and low air velocities. To obtain the high differential pressures required, this type of conveying will typically use a positive pressure system. The nature of dense phase conveying limits it to
horizontal movements. Figures 1.4 and 1.5 illustrate the difference between dilute and dense phase conveying.

![Figure 1.5: Illustration of the particle loading and distribution in dilute phase conveying](image)

There are a number of pros and cons to each method. However, due to the conditions at St Mary's the pulse conveying method is not an option. They want to be able to move a massive amount of particles up a large vertical distance and this simply cannot be done with the pulse conveying. They use a dilute phase vacuum system. The vacuum option was chosen because it is safer and cleaner since they are less likely to have the hot air and particles (~100°C) discharge at every opening.

1.2 Goal of the project:

The goal of this project is to find ways to lower the pressure drop in the pneumatic transport system at St Mary's to improve their efficiency. In order for St. Mary’s to increase their efficiency they must either use less energy to produce the same amount of cement or produce more cement with the same amount of energy. Increasing the efficiency of the pneumatic system is a practical way to do either. By reducing the pressure drop between the raw mill, where the raw materials are ground and entrained in
air, and the bag house, where the particles are filtered from the air and collected for blending, St Mary’s will either be able to reduce the power consumed by their fans or move more particles for the same amount of power. Their fans are currently operating at a pressure differential of over 4 kPa moving, on average, 180 tonnes per hour of material and over 110 cubic meters per second of air. The volume fraction of particles is on the order of $10^{-4}$ which makes this system a dilute system. The pneumatic system is more than 25 meters of roughly 2x2 meter duct which contains, two bends, a diffuser, a bypass channel junction that when closed creates a cavity in the wall, a knockout chamber, and a final filtering chamber called the baghouse. To study this problem commercial CFD code will be used to model the current geometry and flow conditions. Through simulation the geometry will be optimized to give the lowest pressure drop with a monetary cost of the required alterations kept in mind.

1.3 Outline of Thesis

The remaining chapters of the thesis are as follows:

- **Chapter 2**

  The literature review shows the validity of using CFD for the study of pneumatic transport. The secondary phase numerical modelling options are compared for general accuracy and specifically the accuracy of the prediction of pressure drop.

- **Chapter 3**

  The numerical modelling of fluid flow is shown, and the secondary phase modelling options as well as turbulence modelling options are explored in more detail.

- **Chapter 4**

  The pneumatic transport system is broken down into individual components the function and geometry of each is discussed as well as the creation and meshing of the geometry.
• **Chapter 5**

  The results for all the CFD simulations are presented and a summary of the results is given for a number of cases.

• **Chapter 6**

  A summary of the present work is given along with the contributions made and some recommendations for future work.
Chapter 2

2 Literature Review

This literature survey will give a general review on the use of CFD in the simulation of pneumatic conveying. The focus will then shift to the use of CFD for dilute phase pneumatic conveying and the calculation of pressure drop. This chapter will serve to validate the use of CFD to model pneumatic transport and explore the options available for the multiphase treatment.

As previously stated, pneumatic transport can be performed in two ways, dilute and dense phase. The implementation of either system has been useful in industry for many years. A state diagram for pneumatic conveying is given in figure 2.1.

![State diagram showing the boundaries of dense and dilute flow over various velocities and system pressure drops](image)

**Figure 2.1**: State diagram showing the boundaries of dense and dilute flow over various velocities and system pressure drops [6]

The above figure shows a general trend for the behavior of a system for various loading ratios ($W_s$). This type of diagram is not applicable to all scenarios and must be reproduced for a specific system to be of use in determining the system pressure drop. As a result of the lack of generally applicable analytical or empirical correlations, designers of a system have historically relied on experienced guesses, empirical correlations, and rules of thumb to make their estimates. These come from handbooks such as the Pneumatic Conveying Handbook [7] or other studies done to give empirical correlations for specific situations such as Klinzing et al. [8] who specified a phase diagram approach.
to calculating a pressure drop and Yang [9] who correlated a solid friction factor to apply
to a pressure loss equation for straight pipes. Historically, the trend was to oversize
equipment to make up for the large uncertainty factor in the calculations. A more modern
approach for studying pneumatic transport is the use of CFD. Recent years have seen the
use of CFD becoming more widely applied to studying both dilute and dense phase
systems.

2.1 Pneumatic Transport

The emergence of CFD in the late 1980s and 1990s brought about considerable change in
the level of sophistication to which pneumatic transport could be analyzed. Kuang et al.
[10] studied the general characteristics of dense phase conveying with the application of a
periodic boundary condition (PBC). Their case is focused on fully developed flow in a
long straight pipe. A short section of the pipe was modelled with the implementation of
periodic boundary conditions, and a long section of pipe was also studied to ensure the
developed profile. An in-house code was used for the implementation of a Discrete
Element Method (DEM) treatment of particles. The results from the small pipe with the
PBC agreed with the results from a section of the long pipe and it was determined the
application of the PBC could be confidently used to model a system and drastically
reduce the computational resources and time. Behera et al. [11] used a one dimensional
CFD model to investigate the effect that particle shape and size has on the pressure drop
in dense phase conveying of fly ash. An equation of state was used for the gas properties
and one-dimensional conservation of mass and momentum equations were implemented.
The solid phase was treated through the use of a solid friction factor, an area factor that
accounted for particle-particle and particle-wall interactions, and a void fraction for use
with the gas phase conservation equations. They used the solids friction factor as well as
the area factor to create a correlation for pressure drop which could be used for any
system geometry. Four cases were tested and four different friction factors and area
factors were obtained. The use of these factors to determine pressured drop is restricted
to use with systems of identical gas and solid properties to the test case. Since it was a
one dimensional model the results are considered highly ideal. Lain and Sommerfeld
[12] performed a study on the characterization of flow for the dilute phase conveying in
horizontal and vertical lines. An Euler-Lagrange formulation was used for the treatment of the particles and the turbulence was modelled with the k-ε model. The domain was a long thin circular pipe of 10 m in length and 150 mm in diameter. The pipe was turned upward to transition the flow from horizontal to vertical with a 90 degree bend placed halfway through the pipe. A test was performed analyzing the particle trajectories near the bend for a case with two way coupling vs four way coupling (particle-particle interactions). It was found the four-way coupling heavily impacted the flow structure of the particles around the bend as well as the resultant pressure drop. These recent examples chosen from a wide collection of relevant literature show the use of CFD for the simulation of pneumatic transport is conducive to obtaining results for many different characteristics of conveying. As shown from the literature cited above studying the particles present in pneumatic transport can be done in many different ways. The different numerical techniques used for the treatment of the solid secondary phase are explored and summarized in the next section.

2.2 Secondary Phase Numerical Approach

To ensure the reliability of the CFD calculation the most important factor that must be modelled correctly is the treatment of the secondary phase. The two options available for multiphase modelling are Euler-Euler and Euler-Lagrange. The Euler–Euler formulation considers all phases as a continuum; both phases are observed and solved in an Eulerian reference frame. This means the flow fields are resolved by looking at a point in space and time and solving the conservation equations for that point. The second way is by considering the secondary phase as a discrete phase instead of a continuum (Euler-Lagrange). This way the primary phase is solved with the same Eulerian reference frame but the secondary phase is solved within a Lagrangian frame. The Lagrangian frame follows a particle (DEM) or a group of particles (DPM) as they move through the domain tracking the path line as they move. A comparison of these two Lagrangian set ups will be discussed later. The Lagrangian formulation uses Newtonian laws of motion to compute trajectories instead of the mass and momentum conservation approach used for a
continuum. Most CFD simulations of pneumatic transport use the Euler-Lagrange formulation.

Ebrahimi et al. [13] used a Lagrangian treatment of particles to investigate the relationship between particle size, particle loading, the inclusion of a lift force, and the resulting particle distribution. The lift forces were added to the particle trajectory equation via a source term. Fluent was coupled with a commercial DEM solver to conduct this study. The solid phase was modelled as spherical glass beads with diameters ranging from 0.8mm – 2mm for each case. It was concluded that the particle loading ratios and diameters had a significant effect on the particle velocity and that the inclusion of the Magnus lift force played a large part in the particle trajectories. Li et al. [14] have performed a study on the effect of conveying velocity on the transition from dilute to dense phase conveying. A commercial CFD-DEM package was used for the study. A PBC was used in conjunction with a short pipe to produce results that showed that a change in velocity leads to a change in particle friction. This change in the particle friction affects the type of particle flow that is observed i.e. dilute or dense flow. It was concluded that for a range of solids loading ratio depending on the conveying velocity the system could either have dilute or dense phase conveying. Mezhericher et al. [15] and Kloss et al. [16] both investigated the differences in results between DEM and DPM. DEM tracks every particle so it can model the particle-particle and particle wall interaction, DPM only models the particle-wall interaction since it does not model every single particle. It was concluded that DPM was acceptable to use with low loading ratios, such as pneumatic transport, while DEM was much better, but more computationally expensive, for high loading ratios, such as a fluidized bed. This is due to the fact that particle-particle collisions become highly important in a high loading case where the volume fraction is much higher. The above examples from the literature do not directly relate to the pressure drop in pneumatic conveying but serve to show how using the Euler-Lagrange method of multiphase modelling is a generally valid technique regardless of application. One example of an Euler-Lagrange simulation set up to study pressure drop is done by Henthorn [17]. A good correlation was found with experimental data however, this relationship is reported to break down when the particles become highly aspherical.
Although the use of the Euler-Lagrange framework is prevalent in the literature, the Euler-Euler framework can also be found. Two main ways of solving a pneumatic transport simulation with the Euler-Euler multiphase model is the mixture method and the Eulerian method. The mixture method blends both phases together by creating a new ‘mixed’ fluid with average properties. The mixture properties are evaluated by a weighted average using a volume fraction to compute the volume of each phase in different locations. The conservation equations are solved once for the mixture. The interaction between the two phases is modeled with a slip velocity which is derived from drag force. The Eulerian method keeps the phases separate and solves the conservation equations for both individually. This method is computationally expensive, but is accurate, since no mixing assumptions have to be made. Some more relevant literature examples dealing with pressure drop calculations for the dilute phase for both formulations are explored. Mcglinchey et al. [18] employed the use of both the Eulerian and the Mixture model in Fluent to evaluate the pressure drop through a long cylindrical tube and a 90 degree bend with various loading ratios. The turbulence model used was the mixture turbulence model which is an extension of the standard k-ε model specifically altered for two phase flows. The study produced results which correlated poorly to experimental data for high loading ratios and reasonably for low loading ratios. Patro and Dash [19] also used a two fluid model and incorporated the kinetic theory for granular particles in the secondary phase. Pressured drop was tested for fully developed flow for a wide range of particle characteristics and they found good agreement with experimental values. It was determined the granular temperature model given by Ding and Gidaspow [20] as well as the particle-wall collisions played an important role in the velocity profiles. Wang et al. [21] used the Euler-Lagrange formulation in a similar setup to Mcglinchey. The study was performed with the DPM model in Fluent and used the k-ε RNG turbulence model. The study produced a good accordance to experimental pressure drop for low loading ratios. Since dilute phase pneumatic conveying by definition has a low loading ratio it appears we can obtain reasonable pressure drop results with DEM, DPM or an Eulerian formulation. However, for collecting other information, the DPM Lagrangian treatment of particles appears to be the best option for pneumatic transport in terms of accuracy and computational time.
2.3 Summary

CFD has been proven valid for the simulation of pneumatic transport. The choice of multiphase model depends on the application of the simulation but the most prevalent formulation is the Euler-Lagrange. For the Lagrangian treatment of particles the two equation turbulence models appear to be exclusively used for the turbulence in the gas phase. For the Eulerian treatment of particles turbulence models similar to the two equation models which have been specifically tuned to include a secondary phase are used. With the knowledge that the flow to be studied is well within the dilute range, the Euler-Lagrange framework will be used with a DPM setup and a 2 equation turbulence model.
Chapter 3

3 Numerical Setup

3.1 Governing equations:

In order to solve a fluid flow problem computationally, the conservation of mass and momentum equations must be discretized and solved. As a problem becomes more complex, additional transport equations must be introduced to account for heat transfer, turbulence, multiple phases, or porous media conditions.

The conservation of mass equation is given as:

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

where \( \rho \) is density, \( t \) is time, \( \vec{v} \) is the three-dimensional velocity vector and \( S_m \) is a source term. The \( \frac{\partial \rho}{\partial t} \) term accounts for the change in mass inside a control volume over time. The next term \( \nabla \cdot (\rho \vec{v}) \) accounts for the mass passed between adjacent control volumes. The final term is a source term used to model a mass source or sink, such as addition of solid phase particles into a gas stream.

The conservation of momentum equation is given as:

\[
\frac{\partial \rho \vec{v}}{\partial t} + \nabla \rho \cdot \vec{v} + \nabla \cdot \mathbf{\bar{T}} = -\nabla p + \rho \vec{g} + \vec{F}
\]

where \( \vec{g} \) is gravitational acceleration, \( \vec{F} \) is a source term and \( \mathbf{\bar{T}} \) is the stress tensor given by:

\[
\mathbf{\bar{T}} = \mu \left[ \left( \nabla \vec{v} + \nabla \vec{v}^T \right) - \frac{2}{3} \nabla \cdot \vec{v} \mathbf{I} \right]
\]

where \( \mu \) is the dynamic viscosity and \( \mathbf{I} \) is the identity matrix. The conservation of momentum equation is derived from Newton's second law and is applicable to a continuum. The first term on the left hand side describes the loss or gain of momentum
in a control volume over time. This term is zero in a steady-state flow, but is always included in the formulation and discretization due to the necessity to approach a steady-state solution by taking steps in time. The next term is the advection of momentum into or out of a control volume. On the right hand side is the pressure gradient and the viscous diffusion of momentum, respectively. The last two terms account for body forces due to gravity and other body forces, \( \bar{F} \). Other body forces might include the effect of particles in a multiphase flow, or the influence of a porous region. To incorporate turbulence into the momentum balance, the transport equation must be time-averaged, and then closed using one of a variety of methods, which will be discussed later in this chapter.

In the commercial software Fluent (ANSYS), there are two main ways to model a gas flow. The energy equation can be utilized and the flow can be considered compressible using an equation of state to specify properties at every location, or the flow can be considered incompressible with constant properties. The latter case is simpler in terms of computational time and resources but is an idealization of the real case, thus creating a trade-off of accuracy for speed. To determine which model to choose, information is required for the state of the air flowing through the pneumatic duct system. If there is little change in pressure and temperature through the system, then the benefit of using the incompressible approach outweighs the minimal decrease in accuracy. St. Marys constantly observes checkpoints in their pneumatic conveying system to monitor temperature, pressure, and flow rate. The average values of pressure and temperature at each end of the system were sampled over several hours of normal operation. It was found the difference in temperature from the roller mill to the knockout chamber was on the order of 2 °C, while the pressure difference averaged 2 kPa. Using these values combined with the gas constant for air (\( R = 0.287 \text{ J/kgK} \)), the ideal gas law yields 0.877 kg/m³ and 0.873 kg/m³ for the maximum and minimum densities, respectively. Since the difference in the system is small, it made sense to use the assumption of incompressible flow. Thus, air in all simulations was given constant properties that correspond to the average temperature and pressure from these two extremes. A density of 0.875 kg/m³ and a viscosity of 2.1 E-5 Pa-s were used in all simulations. The energy equation was not
solved since it is not required in the incompressible gas model. Furthermore, heat transfer through the walls and interaction with the outside environment was deemed negligible.

### 3.2 Gas-Solid Multiphase Flow:

Multiphase modeling in CFD can be done in many ways. Most commercial software allows you to model any combination of solid, liquid, and gas in pairs or all three at once. It is also not uncommon to have more than one constituent of a phase present in a single simulation an example being two different solid materials being transported by a liquid or gas. As stated, multiphase modelling can be performed in an Euler-Euler or Euler-Lagrange framework. An outline of the Euler-Lagrange formulation is given since it was chosen to via the supporting literature to be implemented.

#### 3.2.1 Euler-Lagrange

The main model used in Fluent under the Euler-Lagrange umbrella is called the Discrete Phase Model or DPM. The DPM solves the primary phase flow field individually; then in a separate sequence, it injects particles into the domain. The particles flow through the domain based on the previously solved flow field. Each particle is tracked until it runs out of ‘steps’ or hits a boundary, which then stops the calculation. The numbers of steps a particle can take is specified by the user to ensure that particles trapped in a recirculation zone don’t create an infinite loop in the solver. Once the particles have been tracked their influence on the flow is taken into account by the addition of a source term in the conservation of momentum equation. When the DPM iteration is finished the flow field resolves itself again with the new source terms added. This is done through many iterations until a converged solution is obtained. The equations used to compute the trajectory of the discrete phase particles is given as:

\[
\frac{du_p}{dt} = Fd(u - u_p) + \frac{g(\rho_p - \rho)}{\rho_p} + F
\]
Where $u_p$ and $u$ are the particle and fluid velocities, respectively, and $\rho_p$ and $\rho$ are the particle and fluid densities, respectively. The drag force, $Fd$, acting on a spherical particle is:

$$Fd = \frac{18\mu C_D Re}{\rho_p d_p^2 24}$$

where $Re$ is the Reynolds number, $\mu$ is the dynamic viscosity of the fluid, and $C_D$ is the drag coefficient based on the shape of the particle. A number of options exist for the injection of particles. The most applicable in pneumatic transport modelling is to assume a uniform injection over a surface area. Other options include injection from a single point or line and a conical ‘spray’ injection. The surface injection means that each face on a particular surface will release a particle with a specified velocity during the DPM iteration. The particle properties that can be specified are as follows: shape, size, and material properties such as density. For this study we have considered inert spherical particles with size and density specified from a study provided by St. Marys.

A few benefits arise to tracking particles in the Lagrangian frame, one of which is the accretion on surfaces. Accretion is the particle build up on a surface where the particle boundary condition is set to trap incident particles. Once a particle collides with a trap surface, the trajectory calculations are complete and the particle is removed. Other boundary conditions that are used are: reflect, where particles bounce off the surface, and escape where particles are free to leave the domain and end the calculation.

### 3.3 Turbulence:

For internal (duct) flow, Reynolds numbers higher than 2300 indicates turbulence is present and this must be accounted for in the momentum balance. The air in the pneumatic duct system has a high velocity and a low viscosity yielding Reynolds numbers much greater than 2300 everywhere. In addition to the high Reynolds number, the flow never has a chance to develop its velocity profile due to the swirling nature at the beginning, the constant bending and changing cross sectional area in the duct, and the geometry of the particle filtering sections.
There are many ways to account for the effect of turbulence in the momentum balance. Turbulence can be modeled with simple 1-equation models to complex 7-equation models. Large-Eddy Simulation (LES) models can also be used where part of the turbulence is directly computed. The apex of these models is direct numerical simulation (DNS), wherein all of the turbulence is directly computed and no modeling needs to be performed. The trade-off to moving from a simple turbulence model to a more sophisticated model is computational time and resources. Two-equation models are almost exclusively used in industrial flow simulations due to their favorable trade-off between accuracy and computational time. Their method of accounting for turbulence in the momentum balance is to Reynolds- and time-average the instantaneous transport equations. Closure models must then be introduced for the new terms that arise from the Reynolds/time-averaging. The averaging process for deriving the turbulence transport equations is described fully in Wilcox [22] and is not carried out here, since no refinements or enhancements are introduced. The Reynolds- and time-averaged forms of the mass and momentum transport equations are

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

\[3.6\]

and

\[
\frac{\partial \rho \vec{v}}{\partial t} + \nabla \rho (\vec{v} \cdot \vec{v}) = \nabla \cdot \vec{p} + \nabla \cdot \vec{F} + \nabla \rho \left( \vec{v}' \cdot \vec{v}' \right) + \rho \vec{g} + \vec{F}
\]  

\[3.7\]

where the instantaneous variables from the general equations are now interpreted as time-averaged variables in both equations. As a result of averaging, a new term called the Reynolds-stress term appears in the conservation of momentum equation, and the momentum equation in this form is often referred to as the Reynolds-Averaged Navier-Stokes (RANS) equation. The Reynolds stress term accounts for the influence of turbulence in the momentum balance, and takes the form of a tensor with six unique components. This term must be modelled in terms of known quantities to obtain closure. One approach to modelling this term is to introduce the Boussinesq approximation, which models the Reynolds stress term as:
where $\mu_T$ is the local eddy viscosity, which is derived from a local turbulent velocity scale and a local turbulence length scale. Within the scope of two-equation turbulence models, there are two main approaches for obtaining these scales: the $k$-$\varepsilon$ approach and the $k$-$\omega$ approach, where $k$ represents the local turbulent kinetic energy from which the velocity scale is derived, and $\varepsilon$ and $\omega$ are the dissipation rate and the specific dissipation rate, respectively, from which the length scale can be derived.

Taking the $k$-$\varepsilon$ approach requires the solution of the transport equations:

$$ \frac{\partial \rho k}{\partial t} + \nabla \rho (k \vec{v}) = \nabla \cdot \left( \mu + \frac{\mu_T}{\sigma_k} \nabla k \right) + G_k + G_b + \rho \varepsilon + Y_M + S_k $$

$$ \frac{\partial \rho \varepsilon}{\partial t} + \nabla \rho (\varepsilon \vec{v}) = \nabla \cdot \left( \mu + \frac{\mu_T}{\sigma_\varepsilon} \nabla \varepsilon \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon $$

where $Y_M$ accounts for compressibility effects on turbulence, $G_k$ is the generation of turbulence due to the velocity gradients (local strain), $G_b$ is the generation of turbulence due to buoyancy effects, and $\sigma_k$ and $\sigma_\varepsilon$ are the turbulent Prandtl numbers for $k$ and $\varepsilon$. $C_{1\varepsilon}, C_{2\varepsilon},$ and $C_{3\varepsilon}$ in the dissipation rate equation are constants that are derived by calibration with experiments, and $S_k$ and $S_\varepsilon$ are user defined source terms. The local solution for $k$-$\varepsilon$ enables calculation of the local eddy viscosity, which takes the form: $\mu_T = \rho C_{\mu} k^2 / \varepsilon$. The eddy viscosity appears in the momentum equations and in the $k$ and $\varepsilon$ equations so the solution procedure is strongly coupled.

Taking the $k$-$\omega$ approach, the following transport equations must be solved:

$$ \frac{\partial \rho k}{\partial t} + \nabla \rho (k \vec{v}) = \nabla \cdot (\Gamma_k \nabla k) + G_k - Y_k + S_k $$

$$ \frac{\partial \rho \omega}{\partial t} + \nabla \rho (\omega \vec{v}) = \nabla \cdot (\Gamma_\omega \nabla \omega) + G_\omega - Y_\omega + S_\omega $$
where \( \omega \) is the specific dissipation, \( G_\omega \) is the generation of \( \omega \), and \( \Gamma_k \) and \( \Gamma_\omega \) are the diffusivity of \( k \) and \( \omega \). By this approach, the local eddy-viscosity is calculated as
\[
\mu_T = \rho k / \omega.
\]

There are pros and cons associated with all the 2 equation turbulence models and neither has been deemed superior in all situations. It is widely accepted that \( k-\varepsilon \) gives good predictions of flow away from walls while the \( k-\omega \) gives good predictions close to walls. Both models have had additions and revisions made to them over years of research to make them more accurate than their original formulations. A few key revisions are outlined below.

3.3.1 \( k-\varepsilon \) RNG:

The \( k-\varepsilon \) formulation has an RNG model, which stands for Re-Normalization Group. It is derived from the instantaneous conservation equations using the renormalization statistical technique. The RNG model differs from the standard model in a few ways, the addition of a term in the \( \varepsilon \) transport equation helps give better prediction of rapidly strained and swirling flows, and an analytical expression for the Prandtl numbers improves the previously used constant numbers in the standard model. These additions make the RNG model generally more reliable than the standard \( k-\varepsilon \) model.

3.3.2 \( k-\varepsilon \) Realizable:

The \( k-\varepsilon \) Realizable model changes the formulation of \( \mu_T \), the eddy viscosity, by exchanging the constant \( C_\mu \) with an expression. It also replaces the epsilon transport equation with one derived from an exact equation for vorticity fluctuation.

3.3.3 \( k-\omega \) SST:

The \( k-\omega \) SST model includes the shear stress caused by turbulence in the eddy viscosity formulation and uses a blending of the \( k-\omega \) formulation and the \( k-\varepsilon \) formulation to have good predictions near the walls using \( k-\omega \) and good predictions away from the walls using \( k-\varepsilon \). The \( k-\omega \) SST model switches between logarithmic wall functions and linear wall functions based on the local \( Y+ \) value. \( Y+ \) is a dimensionless quantity based on the
velocity gradient and viscosity which take the form of shear stress, density, and distance from the wall used to define the law of the wall. The law of the wall shows where a linear approximation and a logarithmic approximation are valid for the fluid velocity specification.

In order to determine which turbulence model to choose for this problem a simple test was performed by calculating the pressure drop in pipe flow. A straight pipe was constructed in ANSYS ICEM. The pipe was made of hexahedral cells using the blocking method, an o-grid was implemented and the cells were refined in the axial and radial direction until a grid independent solution was found. The flow was modeled in Fluent with a fully developed inlet boundary condition. The fluid was isothermal air at atmospheric conditions. The pressure drop given by the three different turbulence models was compared to the well-known empirical solution given from the Darcy Weisbach equation. The results for this straight pipe were similar therefore the choice of turbulence model between the three was not going to play a large part in the results of the final model. Ultimately the k-ω SST model was chosen for a few reasons: its superior wall treatment, performance in adverse pressure gradients, and its better convergence behavior.
Chapter 4

4 Computational Domain:

The pneumatic transport system at St. Marys is a large system with many components. All components are necessary to model to establish the total pressure drop, and to find the regions where the most significant savings could be realized. Due to computational limitations the whole system cannot be modeled as a continuous geometry. In order to accurately study the system with a sufficient level of detail it must be divided into its individual components. The function and geometry of each of the components in the schematic from figure 4.1 are discussed in detail and in sequence in this section.

![Figure 4.1: St Mary’s pneumatic conveying schematic](image)

4.1.1 Roller Mill

The pneumatic conveying line begins at the roller mill where limestone is crushed into a fine dust that can be entrained in a fast-moving airstream. An internal schematic of a typical roller mill is shown in figure 4.2. The roller mill at St Mary’s is functionally the same with only a few minor differences.
During normal operation, the three large grinding rollers shown in red near the bottom of the illustration in Fig 4.1 rotate about the central axis on a table. The external pull rods keep the rollers compressed against the hard table such that they crush the feed rock into a fine dust that can be entrained in air. Particles that are small enough will be pushed to the outside and picked up by a swirling air flow emerging from peripheral nozzles. Larger pieces will remain on the table to be crushed again. The classifier in the upper region will filter and remove any impurities. The classifier also serves to deflect any larger particles that have become entrained in the airstream back down to the grinding wheels. The roller mill in figure 4.2 shows the air exiting at the top, whereas the roller mill at St Mary’s has an extra compartment above the classifier where the air leaves tangentially.

The function and geometry of the roller mill does not require detailed modelling since it is not intended to be modified. Instead, the roller mill was studied to obtain an inlet condition (velocity and turbulence profiles) to the pneumatic transport duct that closely mimics the actual operating condition. The alternative would be to impose uniform or arbitrarily skewed velocity profiles that do not necessarily represent the true condition.
In addition, it would be impossible to estimate the level of turbulence or the distribution of turbulence intensity across the pneumatic duct inlet without simulating the flow through the roller mill. In view of what was required of the simulations, the roller mill was modeled in such a way that the main features of the flow were preserved without worrying about details that would not significantly impact the structure of the outlet flow.

The roller mill was modeled based on details and measurements provided by St. Mary's cement. Features like grinding wheels and classifier were not included, as it was assumed that they would not significantly modify the structure of the flow leaving the mill in the top chamber. The simplified geometric model of the roller mill is shown from two views in figure 4.3.

![Idealized geometry of the roller mill at St Mary’s](image)

**Figure 4.3: Side and bottom view of the idealized geometry of the roller mill at St Mary’s**

The inlet is the red ring on the bottom surface and the outlet is on the end of the upper rectangular protrusion. The rest of the surfaces are walls which surround an open interior. Computations showing velocity and turbulence profiles are given in the next chapter following a description of the modelling approach.
4.1.2 Pneumatic Conveying Duct

Exiting from the roller mill air is passed horizontally into the transport duct. This duct provides a path between the roller mill and the knockout chamber. The inlet of the duct is 2.13 x 2.13 m (7’ x 7’). The flow immediately turns 75 degrees upward and undergoes a cross sectional area change to 2.13 x 1.83 m (7’ x 6’) before moving past a bypass junction. The bypass is opened when the roller mill is not in operation to keep the air moving through the system from the previous processes past the roller mill. When the bypass is closed it creates a large cavity in the duct wall. After the cavity the flow goes through a small bend of 15 degrees to complete is transition from horizontal to vertical. At this point the flow moves vertically for 12.2 m (40’) with a slight sideways translation to be in-line with the downstream junction to the knockout chamber. After the vertical segment the flow reaches a diffuser changing the cross sectional area to 3 x 2.6 m (9'10” x 8'6”). Immediately after the diffuser the air moves through a tight 90 degree bend with the help of turning vanes which precedes the entrance to the knockout chamber. The duct has expansion joints and wall seams at regular intervals.

While the duct between the roller mill and the knockout chamber has many detailed features, a simplified geometry was used for the preliminary work. This geometry was not straight forward to create in ICEM, so to save time it was generated in SolidWorks and imported for meshing into ICEM. This simplified duct left out features such as expansion joints, wall seems, wall roughness, and the junction cavity which will be studied in detail on its own. The simplified duct is shown in figure 4.4.
Figure 4.4: Simplified duct geometry

The turning vanes in the upper 90 degree bend were modeled as two dimensional surfaces since the thickness was negligible. The turning vanes are shown in figure 4.5.

Figure 4.5: Turning vanes in the upper bend of the simplified duct geometry

To study the bypass cavity a truncated duct model was used. It is a reasonable assumption that the cavity will not affect the flow far upstream therefore the use of a
truncated duct served to save computational time. The truncated duct geometry was cut from the full duct geometry and the cavity was added with dimensions from the given blueprints. The cavity is modeled as 5 walls with the outer facing walls being flush with the duct walls. The inlet is the face on the left the outlet is the face on top, all other surfaces are walls. The geometry is shown in figure 4.6.

**Figure 4.6: Bottom segment of the simplified duct geometry including the cavity**

### 4.1.3 The knockout Chamber

When the air has completed its transit through the duct the particles have reached their intended destination and must now be “unloaded” from the air. The first device in this process is the knockout chamber. The chamber was designed to lighten the load experienced by the filters in the baghouse. The chamber slows air down, with the help of baffle plates, by expanding it in the large open area. As the air velocity drops below the saltation velocity the particles will drop out of the air and collect at the bottom where a secondary conveyor will take them to a blending and storage silo. This type of gravity settling chamber would cost less in terms of a pressure drop than a traditional cyclone separator. Upon installation and use, the baffle plates quickly became eroded by the impinging particles. They were removed and not replaced before this study began. The effectiveness of the chamber without the baffle plates is unknown. The chamber was modeled as is, not as it was designed, consequently the baffle plates were omitted.
The chamber was generated using details provided from St Marys. Geometry of the chamber is shown in figure 4.7. The inlet and outlet openings are on opposite walls to each other. The outlet on the bottom of the tapered sections is for particles only while the larger outlet above is for both air and particles, all other surfaces are walls.

![Image of Knockout Chamber Geometry]

**Figure 4.7: Knockout chamber geometry**

### 4.1.4 The Baghouse

After the knockout chamber the flow travels horizontally briefly before entering the baghouse. The baghouse consists of a geometrically tapered inlet plenum with nozzles leading to 6 separate chambers on each side. The high volume of flow and particles being filtered warrants the use of multiple chambers. Inside each of the 12 chambers are 285, 5.5 m (18') long cylindrical filters that remove all the remaining particles from the air before allowing it to enter the exhaust plenum. The filters are periodically cleaned via an air pulse. The pulse shakes the particles loose from filters causing them to fall toward the bottom. The majority of particles do not re-entrain in the air stream due to the large mass of particles falling at once and the reduced velocity in the chamber. The frequency of occurrence of the pulse depends on the differential pressure inside the individual chambers. As the filters become congested with particles the pressure drop through the filter increases, and when a threshold value is reached, the filters are pulsed and the
process repeats. The geometric model shown in figure 4.8 is the lower half of the baghouse. The inlet plenum is modeled with 12 nozzles leading into 12 separate chambers. This model does not include the filters, the upper portion of the individual chambers or the exit plenum. The outlet of this geometry is the cross section inside the chambers before the air interacts with the hanging filters. The drawn portion shows a sample of what the full chambers would look like with 1 of the 285 bags shown. To leave the chambers the air uses a small straight section of duct which leads into the exhaust plenum. It is an identical and reversed version of the inlet plenum shown. The benefit to using this simplified geometry is being able to model all the chambers together to study how the incoming flow profile from the knockout chamber and the geometric taper effect the flow entering each of the individual chambers. The inlet is shown in blue, the red faces are the outlet and the remaining surfaces are walls. The air travels a sample path shown by the green arrows.

Figure 4.8: Lower half of the baghouse modelled with only a portion of each chamber, the drawn portion shows a sample of how the geometry would look if the whole baghouse was modelled
In order to properly see the effect of the filters on the pressure drop, air flow, and particle paths, a detailed model of one of the chambers was created. The 12 chambers are geometrically identical; consequently, only one model was required. Figure 4.9 shows the inside view of the detailed baghouse geometry cut through the symmetry plane.

![Diagram](image)

**Figure 4.9: Internal geometry of an individual chamber in the baghouse**

The inlet of this domain is the cross section where the nozzle from the inlet plenum discharges the air and particles into the chamber. The filters hang from a perforated separation wall and are modelled as a 2 dimensional porous jump boundary condition. The separation wall serves to allow only the air which has passed through the filters to reach the outlet. The exhaust plenum and the conduit leading to the exhaust plenum are not modelled. The outlet of this geometry is considered to be a plane located slightly above the separation wall. Figure 4.10 shows the filter configuration from the top looking down where the darker region is the inlet cut off by the symmetry plane.
4.2 Boundary Conditions

The boundary conditions used in these simulations and their effect on the conservation equations are shown in table 4.1

Table 4.1: Qualitative and mathematical descriptions of the boundary conditions

<table>
<thead>
<tr>
<th>Description</th>
<th>Mathematical Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass Flow inlet</td>
<td>$w = \frac{\dot{m}}{\rho A}, u = v = 0$</td>
</tr>
<tr>
<td>Pressure Outlet</td>
<td>Gradient of $(u,v,w,k,\omega) = 0$</td>
</tr>
<tr>
<td></td>
<td>Gauge pressure = 0</td>
</tr>
<tr>
<td>No Slip Wall</td>
<td>$(u,v,w) = 0$</td>
</tr>
<tr>
<td>Porous Jump</td>
<td>$\Delta P = \left[ \frac{\mu}{\alpha} + \frac{1}{2} C_2 \rho \nu^2 \right] m$</td>
</tr>
</tbody>
</table>
4.3 Grid Generation

For most of the geometry described above the use of the blocking method was favorable. This method creates blocks that can be associated to geometry points, curves, and surfaces. Once associated, the blocks are populated with hexahedral volume elements and quadrilateral surface elements. The elements created from these blocks are easy to refine and coarsen in any areas where it is required, and the blocking method makes it straightforward to control the number of elements used. For similar element size, hexahedral elements not only have a lower cell count but also better control of aspect ratios, skewness, and cell size transition than other element types. The improved quality of elements mean better specification of flux between faces since there are fewer corrections required and fewer factors need to be treated explicitly. In general it is always preferred to use hexahedral elements when the geometry allows it.

The roller mill, the simplified duct, the duct with the cavity, and the knockout chamber were all discretized using hexahedral elements. A geometric spacing of elements was used to better capture the near-wall gradients while having a coarse mesh in the mean flow areas to not over burden the computation. With the exception of near wall mesh all other elements were made to be as uniform as possible in all directions. Figures 4.11 and 4.12 show examples of the hexahedral mesh generated. The mesh shown is coarse in order to better see the layout strategy.

![Hexahedral mesh](image)

Figure 4.11: Simplified roller mill geometry shown with coarse hexahedral mesh
Figure 4.12: Knockout chamber with coarse hexahedral mesh

All computational grids used in simulations were refined near the walls, as mentioned above. Figure 4.13 shows the detailed mesh for near wall treatment.

Figure 4.13: Cross section of hexahedral mesh through the duct

In more complicated geometric models, such as the baghouse model, the blocking method is no longer useful and we must make use of the ICEM meshing algorithms. ICEM has a few built in meshing algorithms that populate the geometry with tetrahedral volume elements, and triangular surface elements. It is common practice to use the Robust Octree method in order to create good quality surface elements and then use the Delaunay method to replace the volume elements. The nature of the Delaunay algorithm creates elements with lower aspect ratios which is good for computational elements.
These algorithms make meshing fast but there is not much control in element size and location. Figure 4.14 shows a cross section of the baghouse mesh with coarse tetrahedral cells.

![Coarse tetrahedral mesh from the baghouse inlet plenum and chamber cross section](image)

**Figure 4.14:** Coarse tetrahedral mesh from the baghouse inlet plenum and chamber cross section

In order to create a useable mesh that is fine and coarse in the proper areas we must make use of the mesh density and the part mesh functions. The mesh density function will allow us to create local areas anywhere in the geometry where mesh refinement or coarsening is required. The part mesh setup allows any surfaces or individual fluid areas to have its own size specification. Near wall treatment can be done by replacing the tetrahedral elements near the wall with triangular prism elements. This gives the mesh a better distribution of nodes close to the surface which helps resolve the near wall gradients. Figure 4.15 shows prisms near the wall of the baghouse inlet plenum.
The quality of computed results is based on many things, which include: the types of physical models selected, the numerical schemes chosen, the order of the discretization, the convergence, and the grid independence, among other things. While factors related to the numerical model are addressed in the next chapter, the convergence and grid-independence are described here. Convergence means to the level to which the equations are forced to conservative. As an illustrative example, the energy equation expresses a balance between transient, transport and source effects, which must sum to zero. When discretized, the equation sums to a small departure from zero due to the numerous numerical approximations made in the discretization procedure. The departure from zero is called a residual, and these are typically used to judge the convergence of a solution. Convergence on all of the present simulations was based on the scaled residuals given in Fluent, by measuring mass flux, and the pressure drop. Simulations were deemed converged when the net mass flux residual was on the order of $10^{-6}$, when the pressure drop between inlet and outlet no longer fluctuated and when the residuals of the other transport equations were all below $10^{-3}$.

Grid independence tests need to be performed when doing analysis on any discretized geometry. These tests ensure that the influence of element size is not a significant factor in the results. By comparing results with different mesh sizes we can see how the simulations are changing with increasing mesh density. It is common practice to start
with a coarse mesh and refine by roughly doubling the number of elements. When the percent change between two mesh sizes is small (typically less than 2-5%), it can be confirmed that the discretization no longer has a significant effect on the results and the result is grid-independent.

The mesh for all geometries was refined and tested until a grid independence of 5% (or less) was achieved. Pressure drop was used as independence criterion since it is the quantity of interest in these simulations. Tables 4.1 and 4.2 give sample values for grid independence for the simplified duct and the knockout chamber respectively.

Table 4.2: Grid convergence for pneumatic transport duct

<table>
<thead>
<tr>
<th># Cells</th>
<th>Pressure Drop</th>
<th>Percent change</th>
</tr>
</thead>
<tbody>
<tr>
<td>174915</td>
<td>429.14</td>
<td>--</td>
</tr>
<tr>
<td>354125</td>
<td>499.34</td>
<td>16.36</td>
</tr>
<tr>
<td>626688</td>
<td>514.49</td>
<td>3.03</td>
</tr>
</tbody>
</table>

Table 4.3: Grid convergence for knockout chamber

<table>
<thead>
<tr>
<th># Cells</th>
<th>Pressure Drop</th>
<th>Percent Change</th>
</tr>
</thead>
<tbody>
<tr>
<td>399170</td>
<td>84.74</td>
<td>--</td>
</tr>
<tr>
<td>850060</td>
<td>104.54</td>
<td>23.37</td>
</tr>
<tr>
<td>1720880</td>
<td>105.48</td>
<td>0.90</td>
</tr>
</tbody>
</table>

4.5 Summary:

In this chapter we have discussed the individual components that make up the pneumatic transport system at St Mary's. The function and geometry are described in detail as well as the meshing strategy for each component. The differences in geometry lead to different mesh strategies and in some cases different element types but the mesh on each geometry was laid out to give the highest quality possible. Grid independence studies were performed for all geometries and the results were shown for two cases. Going forward the mesh which achieved a grid independence of 5% was used for all the future simulations which are discussed in the results section.
Chapter 5

5 Results

The components in the pneumatic conveying line were analyzed separately, starting at the roller mill with information passed downstream from one geometric model to the next. This chapter will discuss results from the multiple CFD simulations ran on each of the previously introduced geometries. A summary of all results and suggestions for improvement can be found at the conclusion of this chapter.

5.1 Estimating the effect of particles

In some cases the use of the DPM in Fluent was not practical due to the computational resources required, or not necessary, due to the availability of correlations that could be used to estimate the influence of particles. In this situation, we can resort to an empirical relationship found in the literature. The pressure drop due to the particles-only for dilute phase transport can be calculated by a relationship given as [9]

\[ \Delta P_p = f_s \times (1 - \nu_f) \times \frac{L \times \rho V^2}{D \times 2} \]

where \( f_s \) is equal to:

\[ \frac{1 - \nu_f}{\nu_f^3} \times 0.126 \left( 1 - \nu_f \right) \times \frac{Re_t}{Re_p}^{-0.979} \]

\( Re_t \) is the terminal Reynolds number, \( Re_p \) is the particle Reynolds number and \( \nu_f \) is the void fraction or the ratio of particle volume occupancy to total volume in a system. This pressure drop needs to be added to the pressure drop calculated for the air to obtain the total pressure drop. It should be noted that this relationship is only valid for predicting the losses in straight sections of duct; any additional minor losses due to the influence of bends, expansions, etc., are not captured by this expression. Another simple relationship
that can be used comes from a pneumatic conveying design handbook [7] and is considered as a ‘rule of thumb’ for design engineers. This relationship is given as:

\[
\Delta P_{\text{loaded}} = (slr + 1) \times \Delta P_{\text{air}}
\]

where \(\Delta P_{\text{loaded}}\) is the total pressure drop including the air and particles, \(\Delta P_{\text{air}}\) is the pressure drop through the system for air only, and \(slr\) is the solid loading ratio and is equal to \(\dot{m}_{\text{solid}} / \dot{m}_{\text{air}}\). This relationship is also limited by its inability to capture the minor losses of the particles. However, since it is based on the air pressure drop, which includes minor losses, it should perform slightly better in cases containing minor losses. Accordingly, the use of these relationships for this project should be seen as a lower limit or a best case scenario for the ‘loaded’ pressure drop. The empirical relationship, the rule of thumb, and the DPM from Fluent were compared by considering flow through a straight duct 10 meters in height with a circular cross section of 1 square meter. Fully-developed flow of atmospheric air with an average velocity of 20 m/s was imposed at the duct inlet. The particles used were spherical particles of diameter 22 \(\mu\)m and density of 2400 kg/m\(^3\), which corresponds to particles seen in the cement industry. For the numerical calculation, the pipe was discretized with hexahedral elements and an o-grid radial meshing strategy with uniform nodes in the axial direction. A summary of the results of the testing is given in table 1.

<table>
<thead>
<tr>
<th></th>
<th>Empirical Eq. 5.1</th>
<th>Rule of Thumb Eq. 5.3</th>
<th>DPM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Drop [Pa]</td>
<td>207.5</td>
<td>207.4</td>
<td>205.5</td>
</tr>
</tbody>
</table>

The results show little difference for the three cases, therefore, going forward we will be using Eq. 5.3 to estimate the pressure drop because it can capture some effects of the minor losses due to the particles. This rule of thumb will be used as a lower limit because of this limitation. It will be used in the results and recommendations only when directly calculating the pressure drop with the DPM is not an option.
5.2 Inlet Conditions

In order to analyze the duct and remaining elements of the transport system an inlet condition must be determined. The mass flow rate of air through the system will be 98.8 kg/s a daily average as measured by St Mary’s. Two options exist to determine the inlet boundary condition: using the outlet of a simulation of the roller mill, and using a long straight inlet section to create some quasi-developed flow from a uniform velocity inlet.

5.3 Roller Mill

As stated, the useful result from the roller mill simulation is the outlet turbulence and velocity profiles which can be imposed on the duct inlet to give a realistic flow profile in the main duct. To conduct this simulation the inlet of the roller mill was specified as a velocity flow inlet with a swirling inlet pattern to mimic the nozzle ring in the real roller mill. Only mass flow rate and velocity were known so a turbulence test needed to be performed to determine the inlet turbulence properties. The inlet was tested with a variety of turbulence intensity values from 2% (low) to 8% (very high). The changes seen in the turbulence properties at the outlet profile between these two extreme cases were less than 0.5 %. This shows that even if the inlet conditions for the roller mill are not exact, since the exit conditions are similar for different cases, it can be assumed to be a suitable representation of the real case. The turbulence intensity at the outlet of the roller mill is approximately 6%; the x, y and z velocities can be seen in figure 5.1.
Figure 5.1: X, Y, and Z velocities at the outlet of the roller mill to be imposed as the inlet boundary condition for the duct

The X and Z velocity contours show the swirling nature of the flow parallel to the outlet, the Y velocity shows the velocity perpendicular to the outlet.

5.4 Quasi-Developed Flow

The second inlet option uses a long inlet section with a uniform velocity condition. When the flow reaches the actual inlet of the simulation it will have reached a quasi-developed state with boundary layers growing off the walls. An illustration of the long inlet is shown in figure 5.2.
Figure 5.2: Long inlet section of the pneumatic transport duct where the red cross-section corresponds to the uniform velocity and the blue cross-section is the actual inlet of the duct

This option is less realistic than using the roller mill condition and must be used when studying elements near the inlet and around the first bend in the duct. Since convergence of simulations with the long-inlet is more rapid, this condition was used when studying components further downstream where the inlet conditions have little effect.

5.5 90 degree bend

At an initial glance of the whole system there was one area which introduces many complications to the flow that could be conveniently addressed. The connecting duct between the roller mill and the knockout chamber, specifically the upper region immediately upstream of the knockout chamber is the location where a reduction in pressure drop could be realized. The diffuser, the tight bend, and the turning vanes were all studied together to find a better solution for moving the flow towards the knockout chamber and altering the cross section of the duct. The section in question is far downstream of the inlet so the quasi-developed inlet conditions were used to facilitate convergence. The real outlet of the duct is immediately after the 90 degree bend, and for this reason reversed flow at the outlet was a problem due to some larger swirls that formed past the bend. A long section similar to the extended inlet section was added onto the outlet to facilitate convergence by eliminating this reversed flow. Figure 5.3 shows a
representation of the duct where the blue surfaces are the real inlet and outlet and the red surfaces are the extended inlet and outlet.

![Figure 5.3: Simplified duct geometry (blue to blue) with extended inlet and outlet sections (red to red) to accommodate imposition of boundary conditions](image)

5.5.1 Turning vanes

The use of turning vanes in a duct line can be beneficial but do not completely correct the flow. The drawback to using turning vanes is they introduce impedance to the flow which will require a higher pressure differential to overcome. Their use in pneumatic transport introduces an additional complication in that the vanes are constantly being eroded by the impinging particles and need to be frequently replaced to remain effective. The benefit to using turning vanes comes in the reduced turbulence of the flow moving through and after a bend. If used properly, turning vanes can lower the pressure drop in a sharp corner by reducing the turbulence through and after the corner by guiding the flow smoothly around the corner. When the turbulence through and after a bend is lowered this results in a lower pressure drop. If the bend is designed properly to facilitate smooth
flow, turning vanes are not required because excess turbulence will not be generated. This presents the best case scenario because it gives low pressure drop due to less turbulence and does not obstruct the flow in such a way that turning vanes would. Since we are intending to give the best case scenario with a properly designed smooth bend we will not further consider the turning vanes.

5.5.2 Bend and diffuser

The 90 degree bend leading into the knockout chamber presents the most significant pressure drop in the duct system and is addressed first. The 90 degree bend introduces two complications into the flow: first that the flow must turn 90 degrees around a nearly square corner, and second that the bend is preceded by a small diffuser section to change the cross section of the duct. Both of these geometric influences are addressed in modifications to the duct system.

While it is well known that a larger radius bend gives a smaller minor loss – and smaller pressure drop -- than a sharp corner, there are limits to how large the bend can be before the pressure drop begins to increase due to other effects. Figure 5.4 shows a schematic diagram of the bend region of the duct system extended to give more rounded features.

![Figure 5.4: Illustration of the definition of r and D for its application to minor losses in the duct](image)

The \( r/D \) ratio is the radius of the bend, \( r \), divided by the diameter of the duct, \( D \); since the duct is of rectangular cross-section, we use the hydraulic diameter \( 4A_c/P \), where \( A_c \) is the cross sectional area and \( P \) is the wetted perimeter. While the pressure drop usually
reaches a minimum with an \( r/D \) ratio greater than 5, for the present case where the hydraulic diameter at the duct outlet is 2.7 m, this leads to a duct system that is structurally unfeasible. The portion of the duct in question at the St. Mary’s facility lies outside of the building. For the current investigation, to keep recommended modifications simple and cost effective, only the unconfined portion of the duct (i.e. that which is outside the building) was modified; all other features remained the same. With this condition applied it means the outlet cross section and location leading into and out of the buildings are a design constraint. Figure 5.5 is an illustration of the geometry of the duct in relation to the buildings.

Figure 5.5: Showing the portions of the duct that are inside and outside of the buildings with the constrained locations highlighted in red

To study the bend, a parametric study was performed where the \( r/D \) ratio of the upper bend was altered from 0.5-1.5 beyond which it became structurally unfeasible. The results obtained are compared to simulation results from original geometry. The diffuser section was merged into the alteration so the new bend designs contain a smooth change
in cross section rather than an abrupt diffuser. Figures 5.6-10 show velocity vectors colored by magnitude plotted at the central cross section for the original bend and each of the new r/D ratios.

Figure 5.6: Velocity vectors colored by magnitude for the original bend configuration

In figure 5.6 a number of interesting features can be observed. We see the velocity has a large spike in magnitude where it rounds the inside of the sharp corner. This is due to the pressure gradient in the radial direction of the bend. The pressure is high at the outer wall and low near the inner wall due to a centripetal force caused by the change of velocity. This pressure gradient pushes more fluid towards the inside to take the path of least resistance near the inner wall. A large recirculation zone is present in the upper corner of the bend. This is due to a pressure gradient in the axial direction along the outer wall. The pressure gradient acts in the same direction of the flow and causes the slower flow close to the wall to change direction and start to recirculate this is known as flow separation. The final significant feature we can see is the air impingement on the upper surface. When the air collides with the surface and rapidly changes direction additional turbulence is generated. All of these features are detrimental to obtaining a low pressure
drop and will try to be reduced or eliminated in the following modifications. The pressure drop in this simulation for the simplified duct with the original bend is 374 Pa.

Figure 5.7: Velocity vectors colored by magnitude for the r/D bend of 0.5

In figure 5.7 we see the bend where r/D equals 0.5. It is immediately evident that progress has been made from the original case. The inner radius velocity spike has been reduced which means there is less of a radial pressure difference. This is due to the smooth features of the bend. The flow impingement on the upper surface is gone but a large recirculation zone still exists. The pressure drop across the duct in this case is 234 Pa, a significant reduction from the original case.
In figure 5.8 we now have the bend with r/D ratio equal to 1. The recirculation zone has diminished and the velocity spike near the inner wall is almost eliminated. The pressure drop in this case is 225 Pa, a small improvement from the r/D equal to 0.5 bend.

Figure 5.9: Velocity vectors colored by magnitude for the r/D bend of 1.5
In figure 5.9 the r/D equal to 1.5 bend is shown. The upper portion of the bend is showing the best results but due to the geometric constraints mentioned earlier a second bend is formed where the duct must bow out to accommodate the large r/D ratio. This new bend is now a problem having a large recirculation zone and a higher velocity spike. The pressure drop in this case is 279 Pa. This is better than the original case but is worse than both other cases tested. Further efforts will be focused on the r/D equals 1 bend since it has the lowest pressure drop.

The major problem remaining with the r/D=1 bend is the small recirculation zone. To eliminate this we can move the outer wall boundary in to cut away the recirculation zone as described by the lattice Boltzmann method which gives direction in the shape optimization of bends in fluid flow [23]. The trade-off for doing this is increasing the velocity because of the smaller cross sectional area. Figure 5.10 shows the modified r/D 1 bend.

Figure 5.10: Velocity vectors colored by magnitude for the modified r/d bend of 1

In this figure we see a very small recirculation and a reasonable peak velocity in the inner corner. This is the optimal shape where any more constriction would increase velocity.
further while any less constriction would allow a larger recirculation. The pressure drop for the modified r/D=1 bend is 192 Pa a significant drop from the r/D=1 bend and almost half the pressure drop from the original bend. Figures 5.11 and 5.12 show the in plane vectors for the duct cross section after the bend of the original and the modified geometry.

Figure 5.11: Flow Secondary flow vectors for the cross section of the duct immediately after the original 90 degree bend
Figure 5.12: Secondary flow vectors for the cross section of the duct immediately after the modified 90 degree bend

The vectors from the original case show a chaotic pattern resulting from the high turbulence and chaotic flow through the bend. The vectors from the modified bend show a typical secondary flow pattern of the two counter rotating vortices that one would expect from flow after a bend. The vortices are formed because the pressure gradient pulls the fluid toward the inner radius of the bend. This comparison shows that the velocity is behaving better in the cross sectional direction to give a lower total pressure drop.

The pressure drops reported are for the airflow alone. The pressure drop for air loaded with cement particles is estimated from the air pressure drop using Eq. 5.3. For cases with minimal recirculation, this estimate is relatively accurate. The results and method shown in Appendix A can be used to create a pressure loss factor for the additional effects of the particles in the bend, but were not used in the present study.
To verify the accuracy of the predicted results, a comparison is made to measurements made at the St. Mary’s cement plant. The measurements indicate that in the existing duct system, a pressure drop of 2000 Pa is present for the same airflow and loading conditions as simulated above, which is considerably larger than that predicted. This study focused on the general shape of the duct system and did not include many of the features upstream on the 90 degree bend. Such features, which include a pocket in the lower bend (bypass), imperfections at fittings and weldments, and other lesser features, will all contribute to increases in the predicted pressure drop across the duct system. These simulations also used the quasi-developed inlet condition which could also be a factor in the magnitude of the pressure drop. That the predicted pressure drops are lower than the measured quantity is an indication that the error is on the correct side of the actual value.

An effective way to use the present simulation results is to consider the percent change that was brought about by the geometric change at the 90 degree bend in the duct system. Table 5.2 shows estimates of the pressure drop for the loaded airflow. Table 5.2 also gives the % change in pressure drop between the new cases and the original case with the sharp bend. The table indicates that significant improvement can be made to the duct system by making a straightforward modification to the duct geometry between the roof of the lower duct and the inlet to the knockout chamber. To be conservative, the approximately 290 Pa pressure drop reduction (case \( r/D = 1.0 \) mod.) could be applied to the measured total pressure drop of 2000 Pa indicating that a 13% improvement can be made in the duct system by modifying the geometry of the bend. This would be the lower limit on the gains made by these improvements. The actual improvement would likely be closer to the percentages seen in the table.
Table 5.2: Summary of results of the parametric study modifications for air-pressure drop, loaded-air pressure drop and percent change in pressure drop as compared to the original geometry

<table>
<thead>
<tr>
<th>r/D</th>
<th>Pressure Drop (air) [Pa]</th>
<th>Pressure Drop (loaded) [Pa]</th>
<th>% change in pressure drop compared to original geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>374</td>
<td>589</td>
<td>-</td>
</tr>
<tr>
<td>.5</td>
<td>234</td>
<td>368</td>
<td>-38%</td>
</tr>
<tr>
<td>1</td>
<td>225</td>
<td>354</td>
<td>-40%</td>
</tr>
<tr>
<td>1.5</td>
<td>279</td>
<td>439</td>
<td>-25%</td>
</tr>
<tr>
<td>1 (mod)</td>
<td>192</td>
<td>302</td>
<td>-49%</td>
</tr>
</tbody>
</table>

The above modifications require a complete overhaul of the duct elbow. Some simpler, and inexpensive, modifications are also considered. These modifications will make the use of flat plates that can be hung or welded inside the existing geometry to eliminate some of the recirculation in the original bend configuration.

5.5.3 Flat plate modification 1:

Modification 1 replaces the upper bend of the elbow with a flat plate that is attached at a 45 degree angle 1 meter up and downstream of the current bend. Figure 5.13 shows the new geometry as well as the velocity vectors for the upper bend.
Figure 5.13: Velocity vectors colored by magnitude for the flat plate modification 1

In comparison to figure 5.6 we see less air impingement on the upper wall and a slightly smaller recirculation zone. The pressure drop for this case is 323 Pa an approximate 50 Pa decrease from the original case.

5.5.4 Flat plate modification 2:

Modification 2 increases the size of the plate in modification 1 to be attached an additional 1 meter up and down stream. This is done to eliminate more of the upper recirculation without too much restriction of the flow. Figure 5.14 shows the geometry as well as the velocity vectors.
Figure 5.14: Velocity vectors colored by magnitude for the flat plate modification 2

By extending the flat plate modification 1 we have eliminated most of the upper recirculation but have now introduced a stronger recirculation zone after the bend in the lower portion. The pressure drop in this case is 313 Pa a marginal improvement from modification 1. Other modifications were considered such as opening up the lower radius with another flat plate or using more than one plate to decrease the 45 degree contact angle. The small, or in some cases no savings, were not worth the extra implementation cost. Table 3 shows a summary of the pressure drops for the new modifications in a similar way as Table 2.
Table 5.3: Summary of results of the simple modifications for air-pressure drop, loaded-air pressure drop and percent change in pressure drop as compared to the original geometry

<table>
<thead>
<tr>
<th></th>
<th>Pressure Drop (air) [Pa]</th>
<th>Pressure Drop (loaded) [Pa]</th>
<th>% change in pressure drop compared to original geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>374</td>
<td>589</td>
<td>-</td>
</tr>
<tr>
<td>Flat plate mod 1</td>
<td>323</td>
<td>506</td>
<td>-14.1%</td>
</tr>
<tr>
<td>Flat plate mod 2</td>
<td>313</td>
<td>491</td>
<td>-16.6%</td>
</tr>
</tbody>
</table>

The best way to reduce the pressure differential in the duct quickly would be to install the larger plate shown in modification 2 in the upper region of the duct. The results shown in table 3 can be used in the same way as the results given in table 2 for the full redesign. The lower limit reduction would be 100 Pa, the absolute difference between the original and the modification 2, (5% of the 2000 Pa) while the more realistic change would be the reduction of 16.6 percent shown in the table.

This modification was introduced into the duct by St. Mary’s during a plant maintenance shut down. Upon use it was reported that the measured pressure reduction was about 12% which fits into the limits provided by the study.

5.6 Duct junction cavity

To test the duct cavity a simulation using the truncated simplified duct geometry was compared with and without a cavity. The roller mill inlet condition for the duct was used since the junction cavity is close to the inlet and the inlet effects are significant. The cavity simulation would not converge in the steady state solver due to some initial velocity oscillations near the end of the cavity. After some time in the transient solver the solution appeared to have reached steady state operation and the data was transferred back into a steady state solver to verify. The addition of the cavity to the simulation only
showed a very small increase in the pressure drop, 7 Pa. When compared relative to the total pressure drop from the original duct case this equates to about a 2 percent difference. The air in the cavity became a steady rotating flow being propelled by the air passing by through the main duct. The additional pressure drop is simply the energy taken from the main flow that is required to propagate the rotational flow inside the cavity. The simulation was run without the influence of particles. The particles injected in the DPM calculations became stuck in the rotating flow and could not escape to finish the calculation. The additional effect of the cavity on particle laden flow is unknown.

5.7 Knockout chamber

The knockout chamber efficiency is unknown to St Mary’s. Without the use of the internal baffle plates they cannot rely on the specifications given by the manufacturer. The knockout chamber is supposed to lighten the particle loading on the filters in the baghouse by removing some particles from the airstream before they enter the baghouse. St Mary’s has expressed a desire to find the pressure drop through this device and find an approximate number of how many particles are being removed from the air stream. This information will lead to a decision whether the pressure drop – filter savings trade-off for the system is favorable.

To save computational time and resources the knockout chamber was considered isolated from the rest of the system. The duct outlet flow was used as an inlet condition to the knockout chamber. The knockout chamber was tested using the outlet condition of the duct simulation as the inlet. The DPM model in Fluent was used to inject particles into the domain from the inlet. The mass flow of the secondary phase is 51.1 kg/s, a daily average as measured by St Mary’s. The particles were two-way coupled with the air which means the air can transfer its momentum to the particles and vice versa. The particle Reynolds number is approximately 7. This means we can neglect the Saffman lift force for the particle trajectory calculations. In addition to the pressure drop the particle knockout was also tested. The knockout rate was tested by setting the particle boundary condition on at the bottom of the domain to trap. The particle accretion was examined and dropout could be calculated in this way.
The particles injected were uniformly spherical with a diameter of 22 microns and a mass flow equal to the average mass flow obtained from the received process data sheets. The number of particles was truncated due to computational resources but was large enough to give a good statistical representation of a realistic case. The particles were injected uniformly over the inlet, since there was no way to fine the real particle distribution. The knockout chamber yielded a 100 Pa drop when measured with the influence of particles. When the particles were tracked through the chamber the dropout rate was calculated to be 8.5%. Figure 5.15 shows a sample of the particle tracks through the knockout chamber.

**Figure 5.15: Particle tracks of injected particles coloured by residence time inside the domain**

As can be seen in the figure the majority of the particles circulate around the domain impacting the walls and ceiling and then leave the domain through the outlet. Only 8-10% of the particles are trapped on the bottom surface which is the sole purpose of this component. These results can be verified by the fact that the erosion patterns given by the DPM result match with the erosion seen by St Mary’s on the chamber roof.
5.8 Baghouse

The pressure drop through the baghouse is a parameter that oscillates with time. As the filters become loaded the pressure drop rises and when they are cleaned the pressure drop falls. The maximum pressure drop can be specified by St. Mary’s. Accordingly, the objective of this study is not to improve the pressure drop through the chambers but to investigate the loading on the filters. St Mary’s has expressed a desire to extend the life of the filters by creating a more even particle accretion distribution among the filters.

5.8.1 Full geometry

Since there are 12 individual chambers the first task was to find information on the flow rates in each chamber. The baghouse lower half model was used with the outlet condition of the knockout chamber simulation imposed as the inlet condition for the baghouse. This geometry was used in order to see the effect that the geometric tapered inlet plenum and the chaotic inlet flow has on the mass flow distribution in each of the individual chambers. It was found there was only a slight variation in the mass flow entering each chamber therefore going forward modeling one chamber will be a representative case of all the chambers. Although the mass flow into each chamber at any given time is roughly equal we still must investigate the effect that a varying mass flow has on the flow inside the chamber since the mass flow of the air and particles changes on an hourly basis within the plant.

5.8.2 Individual Chamber

To determine the effect of varying mass flow rates, a comparison study was done. The particle accretion on the filter surfaces was measured for the average daily flow rate and +/- 10%. To study the accretion on different filters the filters were broken up into sections. An illustration of the sections is shown in figure 5.16.
The green section is labelled inside since it is close to the inlet the purple section is labelled outside since it is close to the far wall. The results of the comparison study are shown in figure 5.17.

Figure 5.17: Particle accretion on different sections of filters
This shows that for different flow rates the accretion across the various sections is similar and therefore we can use the average flow rate for further simulations.

As stated above, the goal of the simulations is to level out the particle distribution among the filters. This is equivalent to saying the bars seen in figure 5.17 for inside – outside sections should be equal. Some simple geometric modifications are used to try and even out the baghouse loading. The original geometry and modifications are shown in figures 5.18-20.
The first modification was an extended inlet section with a 45 degree plate added on shown in blue. This was done to try and push more flow toward the middle bags because they have the lowest particle accretion.

![Figure 5.20: Baghouse fully extended duct inlet](image)

The second modification took the first modification further by extending the inlet section even farther to include the 45 degree plate section.

The results for these three cases are shown in figure 5.21
The modifications to try and push the flow towards the middle of the chamber have actually made the accretion profile worse. The inside bags became more saturated in both modifications. A low pressure region was created with the addition of the inlet extensions that drew more particles back toward the inside section. A simple modification for the baghouse chamber accretion problem was not found and an alternative option was considered.

### 5.9 Cyclone separator

A cyclone separator is a device commonly found in pneumatic conveying applications that uses a rotating flow to separate solid particles from an air stream. A proposal was made to examine the implications of the implementation of a cyclonic separator after the roller mill. By introducing a cyclone separator after the roller mill the pressure drop...
through the duct system will be drastically reduced due to the reduction in particle loading, also the filter life would be increased since they would no longer be responsible for filtering the majority of the particle load. A comparison is made between a case with cyclones and the case without. The specifications for the cyclone are provided by a third party and are used as given. The cyclone specifications are given in table 5.4.

Table 5.4: Cyclone specifications

<table>
<thead>
<tr>
<th>Specification</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gas flow rate at the inlet of cyclones:</td>
<td>310,000 m$^3$/h (110°C, -9500 Pa, 8.8% O2)</td>
</tr>
<tr>
<td>Cyclone inlet velocity:</td>
<td>17 m/s</td>
</tr>
<tr>
<td>Cyclone no.:</td>
<td>2</td>
</tr>
<tr>
<td>Dia D m</td>
<td>4.50</td>
</tr>
<tr>
<td>Inlet ht a m</td>
<td>2.25</td>
</tr>
<tr>
<td>Inlet width b m</td>
<td>1.13</td>
</tr>
<tr>
<td>Outlet length S m</td>
<td>2.81</td>
</tr>
<tr>
<td>Outlet dia De m</td>
<td>2.25</td>
</tr>
<tr>
<td>Cylinder ht h m</td>
<td>9.00</td>
</tr>
<tr>
<td>Overall ht H m</td>
<td>18.01</td>
</tr>
<tr>
<td>Dust outlet dia B m</td>
<td>1.13</td>
</tr>
<tr>
<td>Efficiency:</td>
<td>85%</td>
</tr>
<tr>
<td>Pressure drop:</td>
<td>550 Pa without Neutral Vane, 260 Pa with Neutral Vane.</td>
</tr>
</tbody>
</table>

The cyclone separator that was quoted above has an efficiency of 85% and a pressure drop of 550 Pa without a neutral vane and 260 with a neutral vane. The 85 percent efficiency means it will knockout 85 percent of the entrained particles leaving only 15 percent to be carried by the pneumatic transport system and filtered with the bags. A neutral vane (shown in figure 5.22) is a simple extension of the gas stream inlet into the cyclone that reduces turbulence in the cyclone by directing the gas downward in the spiral pattern is it meant to travel. The swirling gas does not impact with the incoming gas, less turbulence in generated which means there is a smaller pressure drop across the device.
To conduct the analysis we will use the cyclone quoted at 550 Pa (no neutral vane) for the worst case and the cyclone quoted at 260 Pa for the best case. An efficiency of 80 percent for both cyclones will be used to be conservative. In the previous analysis of the duct geometry the pressure drop due to flowing air was found then a conservative assumption for adding the effect of particles using Eq. 5.3 was applied. For this analysis the measured pressure drop will be used as $P_{\text{Loaded}}$ and the air pressure drop will be derived from Eq.5.3. This air pressure drop will provide the basis to determine the new $P_{\text{loaded}}$ where $\dot{m}_{\text{solid}}$ is now 20 percent of what it was before the implementation of the cyclone. This analysis will provide an upper limit to the savings realized by the reduction in particle loading in the duct. Table 5.5 provides the results for the implementation of the cyclone.
Table 5.5: Results for the analysis of the implementation of the cyclone separator

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>No neutral vane</td>
<td>550</td>
<td>1416</td>
<td>1966</td>
<td>-1.7%</td>
</tr>
<tr>
<td>(worst case)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Neutral vane</td>
<td>260</td>
<td>1416</td>
<td>1676</td>
<td>-16.2%</td>
</tr>
<tr>
<td>(best case)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

This shows for the conservative method to determining the new pressure drop the implementation of a cyclone is beneficial even in the worst case. This drop includes the now completely obsolete effect of the knockout chamber and none of the other possible pressure drop reduction modifications. The other important fact to consider is the life of the bags in the baghouse. With the cyclones operating at 80 percent efficiency the bags will presumably last 5 times as long.

5.10 Summary:

The pneumatic system at St. Marys was broken down into components and were individually studied and optimized to produce a lower pressure drop. The values of pressure drop results produced by the CFD simulations do not match the magnitude of the measurements made at St. Mary’s but are useful if dealing with percentage changes and comparisons. Some cases are considered below which combine various proposed modifications to the system. The modifications considered are: neutral vane cyclone, best case modified upper bend, and removal of knockout chamber. Modifications not considered are alteration of the junction cavity because of its low yield.

5.10.1 Case 1:

The first case examined is the best case scenario this is where we look at the system with all the optimal modifications made. This case will include the implementation of the cyclone separator, the best case modified duct, and the replacement of the knockout chamber with an extension of the duct.
5.10.2 Case 2:

In the next case we will examine the implementation of a cyclone separator with the existing duct and no knockout chamber.

5.10.3 Case 3:

For the last case we will consider the modified duct and the removal of the knockout chamber with no addition of the cyclones. Table 5.6 gives a summary of the pressure drop and filter life information for the proposed cases.

Table 5.6: Summary of three cases of suggested modifications on the pneumatic transport system

<table>
<thead>
<tr>
<th>Case</th>
<th>Filter life increase</th>
<th>Effect of particle loading from cyclone</th>
<th>Modified duct</th>
<th>Effect of cyclone</th>
<th>Knockout chamber</th>
<th>New pressure drop</th>
<th>Percent Reduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>x5</td>
<td>-584 Pa</td>
<td>-49 %</td>
<td>+260 Pa</td>
<td>-100 Pa</td>
<td>854 Pa</td>
<td>-57.3</td>
</tr>
<tr>
<td>2</td>
<td>x5</td>
<td>-584 Pa</td>
<td>---</td>
<td>+260 Pa</td>
<td>-100 Pa</td>
<td>1576 Pa</td>
<td>-21</td>
</tr>
<tr>
<td>3</td>
<td>---</td>
<td>---</td>
<td>-49%</td>
<td>---</td>
<td>-100 Pa</td>
<td>880 Pa</td>
<td>-56</td>
</tr>
</tbody>
</table>
Chapter 6

6 Summary

The manufacturing of cement is an energy intensive process that has shown no gains in electrical energy efficiency in Canadian manufacturing plants for many years. For St. Mary's Cement Co, the pneumatic transport system used to move material around the plant is a large consumer of electrical energy. The transport system was separated into components and studied individually. The components included: a roller mill, transport duct (cavity, diffuser, bend), knockout chamber, and filter chamber. Each was modelled in ANSYS ICEM and studied with the commercial CFD code Fluent with the use of DPM where applicable for the treatment of the secondary solid phase. Each component was studied differently but the main theme of altering the geometry to optimize pressure drop remained constant.

The roller mill was modeled to provide an inlet condition to the duct system. The roller mill was not intended to be modified to the model was simplified since the only useful outcome of the simulation was the outlet boundary conditions which were applied to the duct as an inlet. The duct was modeled and two areas were examined in detail. The savings realized by the alteration of the junction cavity near the roller mill was determined insignificant to pursue further then some preliminary calculations. The upper 90 degree bend was the focus of a parametric study to lower the pressure drop as much as possible while maintaining geometrical and structural constraints. The final geometry considered showed a 49% reduction in pressure drop. The knockout chamber was studied with the use of the DPM model. The effectiveness of the knockout chamber was examined and the tradeoff between pressure drop and particle knockout rate was found to be unfavorable. Approximately 10 percent of the particles were removed from the air stream while the device burdened the system for 10-20 percent of the total pressure drop in the system. The filter chamber was studied crudely as a whole and in detail for the individual chambers. This study deviated from the general goal of lowering pressure drop since the pressure drop in the chamber is determined by the particle build up on the filters. Instead the goal of this simulation was to measure the particle accretion on the surfaces and test different inlet geometries in an attempt to even out the particle loading
experienced by the different filters. The DPM model was again employed and the particles were tracked until they became trapped on the filter surfaces. The particle accretion on the surface was measured for a number of inlet configurations and it was determined the best case scenario was the one given by the current geometry. Lastly, the installation of a new component in the conveying line, the cyclone separator, was also considered. The cyclone separator was not modeled but rather its net effect was used as new conditions to test the current geometry. Since the cyclone was specified to have an efficiency of 80 percent this means the new inlet condition to the duct system would be air with only 20 percent of the previous solid loading. Significant savings were found not only in the pressure drop but also in the increased filter life in the baghouse due to the reduced particle loading.

Most results showed that a geometrical change could be made to significantly improve the pressure drop such as the 90 degree bend in the transport duct. Other results however, revealed that the current geometry is the best case like the inlet plenum of the filter chamber. Finally some results determined some savings could be realized but they may not be worth the investment as in the case of the modification of the junction cavity. An extensive summary and recommendations for all the proposed improvements is given at the conclusion of the previous chapter.

Since the CFD results did not agree in magnitude to the experimental measurements made we have validated the results in another way. St. Mary’s installed the flat plate modification to the upper duct corner shown in table 5.3 during a plant maintenance shutdown. They reported an approximate 12% pressure drop reduction when the transport system began running again under the average daily loading conditions. This improvement fits in the limits given by the lower and upper limit predictions from the CFD simulations. Therefore, we can be confident that the other recommendations for lower pressure drop will also prove to be accurate.

6.1 Contributions

The contributions made to St Mary’s Cement from this study come in the form of increased efficiency and money saving. It was shown that simple geometric changes
made to the system would significantly reduce the pressure drop in the system. Such changes include the removal of the knockout chamber and the implementation of the smooth 90 degree bend. The particle accretion on the filters in the baghouse was also analyzed and it was shown that the current geometry gives the best results. Lastly, the effect of the installation of a cyclone separator was analyzed and it was shown that even the worst case scenario lead to money savings via the increased life of the filters in the baghouse.

6.2 Future Work

The results given by this study are a comprehensive analysis on the pressure drop of the pneumatic conveying system at St Mary's. The results given to St Mary's must be considered with a financial aspect to determine the best case to use. The other option that could be explored is a mechanical conveying system to completely replace the pneumatic conveying. Again this study would have to be performed with a financial aspect in order to determine the best scenario.

Once the conveying of particles is considered there are a number of other areas and processes in the cement manufacturing process which can be examined for energy savings. An area eligible for improvement could be the use of the waste hot air stream at the conclusion of the conveying line. The air used to convey the particles is discharged from the plant with a significant amount of thermal energy.
Bibliography


Appendix A

The purpose of this study is to determine the parameters required to characterize minor frictional losses due to the influence of a solid secondary phase in a gas flow in bends. By parametrically studying a bend in circular pipes of equal diameter and length we obtain a relationship between the pressure drop and the bend angle. The commercial software ANSYS Fluent is used with the Discrete Phase Model (DPM) to simulate the solid particles. The results form a simple expression that can be used to estimate pressure drop in pneumatic transport lines conveying fine particles (cement powder).

To validate the chosen models and results, we compare initial CFD simulations to well-known expressions for straight ducts. The pressure drop due to fluid motion in a straight duct is quantified using the Darcy-Weisbach equation [24]:

\[ \Delta P = f_D \frac{L}{D} \frac{\rho v^2}{2}, \]  

where \( f_D \) is the frictional loss coefficient that can be obtained from a Moody chart [24] using the Reynolds number of the flow and the dimensionless pipe roughness, \( e/D \), where \( e \) is the roughness coefficient and \( D \) is the pipe diameter; \( L \) is the pipe length and \( \rho v^2/2 \) is the dynamic head of the flow. To find the pressure drop due to minor losses, Eq. 1 is modified to replace the friction factor and dimensionless length \( L/D \) by a minor losses coefficient \( k_b \):

\[ \Delta P = k_b \frac{\rho v^2}{2} \]  

In this manner, the losses across different types of valves, fittings, and bends are all expressed as a function of the dynamic pressure head, making the process of obtaining the total pressure drop straightforward. The minor loss coefficient \( k_b \) is documented for many different pipe configurations and fittings [24].

In a similar manner, the effect of particle size and loading on the pressure drop in a straight duct section can be expressed as:
\[ \Delta P = f_s \cdot (1 - v_f) \cdot \frac{L}{D} \cdot \frac{\rho v^2}{2}, \]  

(3)

where \( v_f \) is the void fraction, and the solid friction factor \( f_s \) uses the particle Reynolds number \( Re_p = \frac{\rho v_p D_p}{\mu} \) to quantify the formation of vortex shedding behind each particle and its effect on the pressure drop [17]. Equation 3 includes the term \((1-v_f)\), such that the expression only accounts for the additional effect of the particles on the pressure drop. Since Eqs. 1 and 3 take a similar form, it is fitting that the minor loss equations should also take a similar form:

\[ \Delta P = k_{bs} \cdot (1 - v_f) \cdot \frac{\rho v^2}{2} \]

(4)

Here, \( k_{bs} \) is an additional minor loss due to the motion of particles through the valve, bend or other minor loss. Once again, the use of \((1-v_f)\) makes it clear that this additional pressure drop applies only to the presence of particles. While information does exist to quantify \( f_s \) for various particle sizes and loadings [9], little information is available to quantify \( k_{bs} \). The computational study presented below seeks to quantify \( k_{bs} \) for large circular ducts with bends of various angles using the DPM model for gas-solid flows.

The geometry considered was a circular duct of diameter 1m and length 10m to (loosely) replicate a pneumatic transport duct in a large-scale cement production facility. A blocking method was used in ANSYS ICEM to create regions that could be filled with hexahedral elements. As the duct is circular, a five-block o-grid was adopted. All simulations were conducted on a grid independent mesh of 20,000 control volumes.

The inlet boundary condition for the pipe was identical for all simulations. To obtain this condition, simulations were run on the straight pipe using a periodic condition between the inlet and outlet. In this manner a fully-developed profile (for velocity and turbulence quantities) was obtained that could be used as an inlet condition for all subsequent calculations to ensure that the effects of flow development were not present in the predicted pressure drop. For this simulation, the fluid was modeled as isothermal air at standard atmospheric conditions and the mass flow was 0.48 kg/s (\( Re_D = 33,108 \)).
Turbulence in the momentum balance was modelled using the $k-\omega$ SST turbulence model. While other models were tested, ($k-\varepsilon$ realizable model, and $k-\varepsilon$ RNG model), the differences across the duct were not significant. The particles in the DPM injection were given a density of 1550 kg/m$^3$ and a diameter of 22 micrometers. These parameters were held constant through all the simulations; the only parameters to be varied in this test is the angle of the bend.

The injection was specified to be 0.2 kg/s, which gives a mass loading of 0.417. Having a loading value under 0.5 is typical for dilute phase pneumatic conveying of cement particles. Using the mass loading, the density, and the size of the particles, the volume fraction can be computed to be on the order of $10^{-4}$. To validate the use of the DPM in Fluent, the sum of Eq. 1 and 3 for the total pressure drop in a straight pipe with particles was calculated. The DPM gave results deviating less than 1 percent from the analytical prediction from these equations.

In order to find a pressure drop in a gas-solid flow system with bends we can use and addition of the pressure drops from equations 1, 2, 3, and 4. The problem that arises is the lack of the loss coefficients to apply to Eq. [4]. The method used in this report to obtain the loss coefficients is to add the empirical pressure drops given by equations 1, 2, and 3 and subtract this value from the value given by the computational model. The resultant pressure drop will be the effect of the particles in the bend.

Table one shows the results of the full computational pressure drop and the empirical result given from the addition of Eqs. 1,2, and 3. Column three is the difference of columns one and two. Theoretically the difference for the straight section should be 0 but due to some numerical error this is not the case. The difference between empirical and computational is corrected by the difference seen between the empirical and computational results for the straight duct. This result is shown in column four. This helps to minimize the error when dealing with the bend only pressure drop. The corrected difference is the pressure drop that will be used in further calculations with Eq. (4b) to find the loss coefficient.
Table 1: Difference of empirical and computational results to give the minor loss of the particles due to the bend

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Straight</td>
<td>0.04610</td>
<td>0.04484</td>
<td>0.00126</td>
<td>0.0</td>
</tr>
<tr>
<td>30 Degree</td>
<td>0.06070</td>
<td>0.05765</td>
<td>0.00305</td>
<td>0.00255</td>
</tr>
<tr>
<td>60 Degree</td>
<td>0.08242</td>
<td>0.07347</td>
<td>0.00616</td>
<td>0.00566</td>
</tr>
<tr>
<td>90 Degree</td>
<td>0.09244</td>
<td>0.08628</td>
<td>0.00895</td>
<td>0.00769</td>
</tr>
</tbody>
</table>

Eq. [4] can be rearranged to solve for the loss coefficient.

\[ k_s = \frac{\Delta P}{(1-\nu_f)\frac{\rho v^2}{2}} \]  \hspace{1cm} (4b)

Using equation 4b we can obtain the loss coefficient seen in table two.

Table 2: The loss coefficient for various bend angles derived from the pressure drop with Eq.4b

<table>
<thead>
<tr>
<th>Degree of Bend</th>
<th>Pressure Drop</th>
<th>Loss coefficient ( k_{bs} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
<td>0.00255</td>
<td>0.039846</td>
</tr>
<tr>
<td>60</td>
<td>0.00566</td>
<td>0.088486</td>
</tr>
<tr>
<td>90</td>
<td>0.00769</td>
<td>0.12031</td>
</tr>
</tbody>
</table>

The loss coefficient \( K_{bs} \) is plotted with the loss coefficient \( k_b \) in figure 9 to see similarities in the trend. The \( k_b \) curve is a well-established curve created from values taken from Frank White’s 7th edition of Fluid Mechanics text.
Figure 1: The bend loss coefficient due to the solid phase, $k_{bs}$, and the gas phase, $k_b$, for various bend angles.

Both curves have similar trends showing an increase in pressure loss as the degree of bend is increased. The results presented show the expected correlation for the loss coefficient verses the degree of bend. A constant solid loading ratio with a constant particle diameter was tested; therefore, the bend loss coefficient calculated can only be applied to a scenario with similar conditions. Other losses can also be tested such as valves, sudden expansions, sudden contractions etc. The effect of particle size, mass and volume loadings should also be tested to create a comprehensive index of solid loss coefficients for a number of pipe features.
## Curriculum Vitae

<table>
<thead>
<tr>
<th>Name:</th>
<th>Ethan Doan</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Post-secondary Education and Degrees:</strong></td>
<td>The University of Western Ontario London, Ontario, Canada 2008-2012 B.ESc.</td>
</tr>
<tr>
<td></td>
<td>The University of Western Ontario London, Ontario, Canada 2012-2014 M.ESc.</td>
</tr>
<tr>
<td><strong>Honours and Awards:</strong></td>
<td>The University of Western Ontario Scholarship of Distinction 2008</td>
</tr>
<tr>
<td></td>
<td>ASHRAE Award 2012</td>
</tr>
<tr>
<td><strong>Related Work Experience:</strong></td>
<td>Teaching Assistant</td>
</tr>
<tr>
<td></td>
<td>The University of Western Ontario 2012-2014</td>
</tr>
</tbody>
</table>