Characteristics of two-phase gas-particle flows in circular pipes and bends

Heng, Jinliang

2018

Heng, J. (2018). Characteristics of two-phase gas-particle flows in circular pipes and bends.
Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/88958>

<https://doi.org/10.32657/10220/46063>

CHARACTERISTICS OF TWO-PHASE GAS-PARTICLE
FLOWS IN CIRCULAR PIPES AND BENDS

HENG JINLIANG

2018

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

SINGAPORE

UNIVERSITY OF
Southampton

CHARACTERISTICS OF TWO-PHASE GAS-PARTICLE
FLOWS IN CIRCULAR PIPES AND BENDS

HENG JINLIANG

SCHOOL OF MECHANICAL AND AEROSPACE

ENGINEERING, NTU

AND

FACULTY OF ENGINEERING AND THE ENVIRONMENT, UoS

2018

This page is intentionally left blank.

**CHARACTERISTICS OF TWO-PHASE GAS-PARTICLE
FLOWS IN CIRCULAR PIPES AND BENDS**

HENG JINLIANG

School of Mechanical and Aerospace Engineering, NTU
Faculty of Engineering and the Environment, UoS
A thesis submitted to the Nanyang Technological
University and University of Southampton
in partial fulfilment of the requirement for the joint-degree of
Doctor of Philosophy

2018

This page is intentionally left blank

Acknowledgements

I would like to thank my two supervisors Asst. Prof. New Tze How, Daniel from Nanyang Technological University (NTU), Singapore and Prof. Philip A. Wilson from University of Southampton (UoS), United Kingdom for their patience and guidance for me throughout my doctoral studies.

I am also grateful to both institutions, NTU and UoS, for providing a conducive and encouraging environment for my studies. In addition, I would like to express my gratitude to NTU for providing me with the NTU Research Scholarship and UoS for the use of its IRIDIS High-Performance Computing Facility in the completion of this work.

Next, I would like to thank my family members for their encouragement and support throughout all these years.

Lastly, I would like to express my utmost gratitude to a special friend, Isaac Ng, for the immense support, reassurances and encouragements for those times when things seemed too much to handle. Thank you.

Table of contents

Acknowledgements	i
Table of contents.....	ii
List of figures.....	iv
Abstract	x
Nomenclature.....	xii
Chapter 1 Introduction	1
1.1 Background.....	1
1.2 Literature review.....	8
1.3 Research motivation and significance.....	15
Chapter 2 Numerical method	18
2.1 Numerical model	18
2.2 Numerical setup.....	29
2.2.1 Geometry.....	29
2.2.2 Mesh generation	29
2.2.3 Solution strategy	33
2.2.4 Convergence and mesh dependency check.....	34
2.3 Model validation.....	36
2.3.1 Numerical setup	36

Table of contents

2.3.2 Turbulent dispersion model	38
2.3.3 Vertical pipe section	41
2.3.4 Horizontal pipe section.....	45
2.3.5 Effects of particle diameter.....	48
Chapter 3 Pipeline bulk conveying characteristics	52
3.1 Numerical setup.....	52
3.2 Effects of pipeline inlet pressure	56
3.3 Equivalent length for pipe bends	59
3.4 Conveying characteristics	61
Chapter 4 Flow characteristics of the industrial pipeline	63
4.1 Particle rope.....	63
4.2 Secondary flows.....	68
Chapter 5 Effects of pipe geometry on particle roping	73
5.1 Numerical setup.....	73
5.2 Bend radius	75
5.3 Bend connecting length	108
Chapter 6 Conclusion.....	133
List of publications arising from this study	138
References	139

List of figures

Figure 1 Typical positive pressure pneumatic conveying system.	2
Figure 2 Industrial pneumatic conveying pipeline at Jurong Port Pte. Ltd. cement conveying terminal	3
Figure 3 On-site photos of the cement conveying pipeline	3
Figure 4 Calculated versus experimental pressure drop	13
Figure 5 Summary of the Eulerian granular model	30
Figure 6 O-grid blocking strategy at the pipe bend used to generate ordered hexahedral cells.....	31
Figure 7 Hexahedral cells generated from O-grid strategy.....	32
Figure 8 Scaled residual for the continuity equations during a transient calculation	35
Figure 9 Outlet particle mass flow rate and air velocity plot at 0.1m after the bend.....	35
Figure 10 Pipeline dimensions for model validation, flow direction indicated by dashed arrows.....	37
Figure 11 Effect of different model constant for turbulent dispersion model on air velocity and particle concentration	39
Figure 12 Comparison of CFD results with experimental data of particle concentration along various vertical sections after the pipe bend as indicated.	42
Figure 13 Comparison of CFD results with experimental data of particle velocity along various vertical sections after the pipe bend as indicated.	44
Figure 14 Comparison of CFD results with experimental data of RMS particle fluctuation along various vertical sections after the pipe bend as indicated.	45

List of figures

Figure 15 Comparison of CFD results with experimental data of particle concentration, air velocity and RMS velocity fluctuation along various horizontal sections before the pipe bend as indicated.....	48
Figure 16 Particle concentration distribution within the horizontal pipe section when particle diameter increased to 100 μ m.	49
Figure 17 Pneumatic conveying pipeline dimensions	53
Figure 18 Meshing for the industrial scale pneumatic conveying pipeline	54
Figure 19 Pressure drop along the pipeline for different conveying pressures.	56
Figure 20 Pressure drop comparison between calculated values and numerical simulation.....	58
Figure 21 Pressure drop ratio of pipe bend to horizontal straight pipe for various conveying pressures.	60
Figure 22 Pipeline conveying characteristics for different solids loading ratio	62
Figure 23 Iso-surface plot of 0.2 cement volume fraction at bend number two	64
Figure 24 Cement volume fraction plots upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated.....	65
Figure 25 Illustration of (a) bend positions and (b) rope angular position	67
Figure 26 Angular position of particle rope and a fitted underdamped curve .	67
Figure 27 Secondary flow structure within the bend of a (a) single bend and (b) combination of bends.	69
Figure 28 Secondary flow structures upstream of the bend at various multiples of pipe diameter (d) and within the bend at various locations as indicated.....	71
Figure 29 Pipeline dimension for geometry investigations	74

Figure 30 Cement volume fraction plots for BR 2.5, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated.....	76
Figure 31 Cement volume fraction plots for BR 5, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated.....	78
Figure 32 Cement volume fraction plots for BR 10, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated.....	80
Figure 33 Cement volume fraction plots for BR 14, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated.....	82
Figure 34 Cement volume fraction plots for BR 20, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated.....	84
Figure 35 Cement volume fraction plots for BR 25, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated.....	86
Figure 36 Particle rope angular position for various bend ratio	88
Figure 37 Particle rope angular position for individual bend ratio test cases ...	89
Figure 38 Head loss of 90° radiused bends.....	90
Figure 39 Pressure loss across pipe bend for different bend ratios after accounting for pipe bend length and air velocity.....	93
Figure 40 Particle rope angular position, comparing between the truncated pipeline and full-scale pipeline	94

List of figures

Figure 41 Secondary flow vector plots for BR 2.5 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	96
Figure 42 Secondary flow vector plots for BR 5 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	98
Figure 43 Secondary flow vector plots for BR 10 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	100
Figure 44 Secondary flow vector plots for BR 14 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	102
Figure 45 Secondary flow vector plots for BR 20 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	104
Figure 46 Secondary flow vector plots for BR 25 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	106
Figure 47 Particle rope angular position for various connecting length	109
Figure 48 Particle rope angular position for individual connecting length test cases	110
Figure 49 Cement volume fraction plots for CL 0 at various locations within the bend as indicated.....	111

Figure 50 Cement volume fraction plots for CL 4.5 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	113
Figure 51 Cement volume fraction plots for CL 10 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	115
Figure 52 Cement volume fraction plots for CL 15 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	117
Figure 53 Cement volume fraction plots for CL 20 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	119
Figure 54 Pressure loss across pipe bend for different vertical pipe connecting length after accounting for pipe bend length and air velocity.	121
Figure 55 Particle rope dispersion in the vertical pipe section for CL 20 test case.....	122
Figure 56 Secondary flow vector plot at one diameter before the pipe bend entrance for different CL as indicated	123
Figure 57 Secondary flow vector plots for CL 0 at various locations within the bend as indicated.....	123
Figure 58 Secondary flow vector plots for CL 4.5 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	125

List of figures

Figure 59 Secondary flow vector plots for CL 10 , upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	127
Figure 60 Secondary flow vector plots for CL 15, upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	129
Figure 61 Secondary flow vector plots for CL 20, upstream of the bend at various multiples of pipe diameter (<i>d</i>) and at various locations within the bend as indicated.....	131

Abstract

Pneumatic conveying systems are employed across different industries with its routing flexibility being one of the reasons it is favoured over other systems. In pneumatic conveying, material fed into the system at the inlet goes through a series of closed piping in a very clean operation. However, the use of pipe bends for routing can cause a phenomenon called particle roping which may decrease the overall efficiency of the system. Due to inertial forces acting on the conveyed material about a bend, the material tends to cluster into a rope which decreases its travel velocity. Such phenomena can be studied through experiments and numerical simulations to develop ways to mitigate its detrimental effects. However, in the case of industrial scale pipelines already in operation, such pipelines are often optically opaque and runs for hundreds of metres at one time. Experimental observations of the flow characteristics within the pipeline are impossible. By employing numerical simulations, the cement conveying throughput of 500 tonnes per hour is well matched. The carrier gas is also matched at $12,000 \text{ m}^3/\text{hr}$. The flow characteristics from the simulation results such as velocity and pressure drop are replicated to within $\pm 10\%$ of values calculated from empirical formulas. With the results, much more insights can be gained on the flow characteristics. In particular, the region after a vertical-to-horizontal pipe bend displays a sloshing flow characteristic. This is primarily caused by the presence of another horizontal-to-vertical bend at close proximity upstream. This combination of bends results in a flow characteristic that is much different from a single pipe bend flow. Literature regarding the flow characteristics in pipelines with a combination of bends is sparse. Therefore, by employing numerical simulations, the sloshing characteristics can be

documented in detail for different cases. By varying the bend radius and the length of the vertical pipe connecting the two bends, different flow characteristics can be observed. Varying the bend ratio (BR) ranging from 2.5 to 25, three classifications of flow characteristic can be seen. The particle rope either has sufficient inertia to make complete loops along the walls of the pipe wall ($BR \lesssim 2.5$) or insufficient inertia to do so; and either splits up into two ropes before merging back at the bottom of the pipe ($5 \lesssim BR \lesssim 10$) or disperses within the bend and later on forms again at the bottom of the pipe due to gravity effects ($BR \lesssim 14$). Next, the connecting length is varied from 0 to 20 times the pipe diameter. It is found that the connecting length has a lesser effect on the particle rope compared to the bend ratio. A longer connecting length facilitates the dispersion of the particle rope formed from the previous bend. In the longest connecting length test case, results from the numerical simulation are able to shed light on the oscillating dispersion characteristics of a particle rope in a vertical pipe. In all, this study is able to shed light on the flow characteristics in an industrial pipeline that which the results could be used to further increase the operating efficiency. Also, the knowledge on the effects of a combination of pipe bends in close proximity is expanded for the current pipeline layout studied. Though, more future studies are needed to completely characterise the effects of a combination of bends in other orientations.

Nomenclature
Latin alphabets

C_D	Drag function
C_{TD}	Turbulent dispersion model constant
d	Particle diameter
D	Dispersion scalar
e_{ss}	Coefficient of restitution for particle collision
$\vec{F}$	Force
$\vec{g}$	Gravitational acceleration
g_0	Radial distribution function
$\bar{I}$	Identity tensor
k_{Θ_s}	Diffusion coefficient
K	Interphase momentum exchange coefficient
p	Pressure
$\vec{R}$	Interphase momentum exchange
Re	Relative Reynolds number
S_m	Mass source term
t	Time
V	Volume
v	Velocity

Greek alphabets

α	Volume fraction
γ_{Θ_s}	Collisional dissipation of energy
Θ_s	Granular temperature
λ	Bulk viscosity / Second viscosity
μ	Dynamic viscosity
μ_t	Eddy viscosity
ρ	Density
σ	Turbulent Prandtl number
$\bar{\tau}$	Stress tensor

Subscripts

g	Gas phase
q	Arbitrary phase
s	Solid phase

Chapter 1 Introduction

1.1 Background

The use of pneumatic conveying systems to transport bulk materials over extended distances is a commonplace of many industries. They had been utilized in power generation industry for the transport of pulverized coal; food industry for the transport of powdery ingredients such as wheat, flour or sugar; and construction industry for the transport of cement powder, to name a few. It stems from the many benefits that a pneumatic system poses over conventional belt conveying [1]. Apart from its versatility to be applied across different materials, the availability of different fittings allows for an extremely flexible system. The closed set-up of the system also means that the operation is clean, and that continuous operation is possible. One example is the pneumatic conveying of bulk cement powder that will be the focus of this study. Dry cement particles, having very small particle diameters, tend to produce dusty conditions if transported in conventional belt conveyors. Being able to be easily entrained in the air also makes pneumatic conveying a suitable choice of transport mode for bulk cement powder.

Pneumatic conveying systems typically consist of an air mover that can be installed near the inlet or the outlet for positive or negative pressure operation, respectively. The material to be conveyed is fed into the system by a feeder and is transported to the destination usually using air as the conveying medium. At the outlet, the material is separated from the conveying medium and discharged. Figure 1 shows a typical positive pressure pneumatic pipeline layout. The use of air as conveying medium means that the capacity of the system lies in the

Figure 1 Typical positive pressure pneumatic conveying system. Image source: [1]

capabilities of the air mover, feeding device, delivery pipes and the discharge mechanism. In order to optimize the system, a variety of factors must be considered. Mainly, the routing of the pipeline has to be determined in order to account for the pressure losses along the pipeline due to the number of bends. This information is required to correctly size the air mover and thereafter the capacity of the discharge mechanism. The appropriate feeding device has to be selected too. For example, in the current case of cement powder conveying, a screw feeder is used. However, such selections cannot be easily made without an understanding of the flow characteristics within the pipeline.

Figure 3 shows a schematic layout of the industrial pneumatic pipeline currently under study. The pipeline has five 90° bends, where bends 1 and 5 are labelled. Figure 2 shows some on-site photos of the pipeline where only one of the pipeline shown in the photos is being investigated. Bends 1 and 2 are shown on the right figure, bends 3 and 4 are shown on the bottom left figure, while the top left figure

Figure 3 Industrial pneumatic conveying pipeline at Jurong Port Pte. Ltd. cement conveying terminal

Figure 2 On-site photos of the cement conveying pipeline

shows another onsite pipeline to illustrate how the pipeline will discharge into a silo after bend 5. Bends 1 and 2 will be the focus of investigations in the later part of the thesis.

As the different industries continue to look for ways to improve their conveying systems, either for safety concerns [2] or for increasing throughput, pneumatic conveying has already been under research for many decades. Numerical simulations, or computational fluid dynamics (CFD), have emerged over the years as a reliable method to study the mechanisms of pneumatic conveying. Dense or dilute flow regimes such as particle-laden flows, two-phase or multiphase flows, and moving bed flows, etc., can all be investigated by numerical simulations. Dense and dilute phase conveying are terms used to identify under which regime the flow is classified. Dense phase conveying generally encompass flows where the material is transported as a distinct phase from its carrier gas at the bottom of the channel. Whereas, dilute conveying generally mean that the particulates are entrained in the carrier gas phase. A more detailed classification of the types of flow regime is available in textbooks on pneumatic conveying [3]. The book is also a useful start for interested readers to gain deeper insights into the theory of pneumatic conveying which are not critical to the understanding of this thesis.

Due to its complexities, experimental studies are still being conducted to study pneumatic conveying processes. Some examples will include work by Khan et al. [4] on the pick-up velocity of particles and de Moraes et al. [5] on head loss coefficient for pipe fittings, both of which deal with properties fundamental to pneumatic conveying processes. While experimental studies form a significant portion of pneumatic conveying investigations, the number of numerical studies are increasing as well. There are a few numerical approaches to study pneumatic conveying flows that build on top of established methods in calculating single-

phase flows. Following the continuum assumption that discrete molecules can be treated as a continuous phase in the derivation of the Navier-Stokes equations, discrete solid particles can also be assumed to be a continuous phase which intertwines with a carrier fluid. As it uses the Eulerian frame of reference, such method is referred to as an Eulerian approach. If the particles are assumed to be discrete in the numerical model, it is referred as a Lagrangian approach as it is applying a Lagrangian frame of reference in solving the equations of motion. Apart from direct numerical simulations and solving the complete equations of motion for each individual particle, there is no one unified numerical model to use for a gas-particle flow in which current computers can handle easily. In most cases, only the most appropriate modelling is being applied to reduce computational cost. The selection of either an Eulerian or Lagrangian mathematical treatment of the solids phase is based on the flow regime [6]. Also, a two-way or four-way coupling between the phases has to be chosen depending on the solids loading ratio (SLR) [7] and Stokes number [8]. A suitable turbulent flow model will also need to be selected and this can be decided through past literature as some models have consistently shown more accuracy over other models [9].

In each modelling approach, researchers have come up with many different numerical models catering to different conditions or area of interest in the flow. Constitutive relations are usually derived for certain flow conditions and may not be adequate beyond the range that it was originally derived for [10]. Therefore, this thesis first aims to demonstrate the ability of an Eulerian scheme in predicting the flow characteristics of a granular phase in a pneumatic pipe configuration

including a 90° bend. Both the upstream and downstream portions of the pipe bend will be looked at, with a focus on the particle distribution and both the gas and particle velocities. Also, the application of a Favre-Averaged Drag (FAD) model [11] towards a gas-solid multiphase flow is not common in the literature and this thesis intends to show and assess the effects of the FAD model on a granular flow. Specifically, the empirical model constant will be varied to observe its effects on the flow and then select a suitable model constant. It is expected that this model can be applied to gas-solid flows as its universality is concluded by the original authors. The effects of the particle diameter used within the model will also be briefly touched on.

Another purpose of this study is to look into the numerical simulation of a large, full-scale industrial pneumatic-based cement conveying system currently in active operation. Numerical investigations into pneumatic conveying require some form of experimental data to which the numerical results can be compared against. As such, numerical simulations are mostly only applied to experimental-scale pipelines. Furthermore, gas-solid flows are influenced by a multitude of factors such as the physical property of the material, inlet loading ratios, pipe geometry, etc. and experiments may not be able to effectively isolate each of their contribution to the flow characteristics. Numerical simulation can then serve as a complementary tool to individually study each factor [12] and look into the more complicated aspects of the flow. However, even after the confidence in the accuracy of numerical methods, they are rarely applied in full-scale studies. In fact, there may be of little incentive to spend a huge amount of computing resource in simulating a full-scale flow as scale-up procedures can be applied

during the pipeline design process. However, further research has shown that they are far from perfect and further improvements could be made [13]. Furthermore, to apply numerical simulation to only certain sections of an entire pipeline, it requires defining inlet and outlet boundary conditions which are difficult to obtain [12]. Therefore, by applying numerical simulation on a full-scale industrial pneumatic pipeline, it can serve to bridge the numerous experimental results to something that is already implemented as a working system.

The study of the full-scale pneumatic conveying system shown in Figure 3 will be as follows. Firstly, quantitative data such as the pressure drop and output capacity provided by Jurong Port Pte Ltd. (JPPL) will be used for the steady-state simulations. The results of these simulations will bridge theoretical and practical applications to simplify pipeline designing procedures. Secondly, transient simulations will be performed on a selected test case to illustrate and gain specific insights into the flow characteristics. In particular, particle concentrations within the pipe and secondary flow structures will be investigated to attain a better understanding of the flow behaviour within the pipeline, which would not be available from the actual industrial system. Further on, the effects of pipe geometry on the conveying characteristics can be looked at.

Particle ropes, a region of the pipe cross-sectional area where there is an increased concentration of particles, are caused by pipe bends [14]. When particles enter a pipe bend, inertial effects will tend to cluster the particles into a region of higher particle concentration. This area is usually located on the outer wall of the pipe bend. Particle ropes are sometimes seen as an unwanted occurrence in operations,

such as pulverized coal conveying where a well-dispersed particle phase is desired [15]. The region of higher particle concentration also means that more particle collisions could occur, and if the material conveyed is abrasive, the pipe walls will be subjected to wear and tear [16]. Mitigation actions, such as using wear resistant materials for pipe walls or reinforcements at the back of the pipe bends, will then need to be taken. Special pipe fixtures can also reduce abrasions on the pipe walls [17]. However, the location and characteristics of the particle rope must be known in order to apply the proper remedial actions. Particle ropes for a single pipe bend are well investigated in all orientations but not when there is a combination of two bends at close proximity [12] where the roping effects of the first pipe bend have a spillover effect on the roping at the second bend. In the latter part of this work, a pipeline geometry for a combination of a horizontal-to-vertical (HV) and a vertical-to-horizontal (VH) bend will be studied. Two parameters will be varied, namely the pipe bend ratio (bend diameter/pipe diameter) and the length of the connecting vertical pipe section between the two bends. The location of the particle rope within the pipeline will be characterized and the effects of these two parameters on the flow characteristics will be documented.

1.2 Literature review

The most commonly investigated aspect of pneumatic conveying is the pressure drop within the pipeline. Some examples of which, McGlinchey et al. [18] used the Eulerian formulation and predicted pressure drops in 90° pipe bends. Their results demonstrated qualitative agreement with experimental data. In another case, Makwana et al. [19] looked into the effects of dune formation on pressure

drops. Dunes, formed at the bottom of the pipe during some conveying processes, were numerically found to induce large pressure fluctuations. In other aspects of the pneumatic conveying system, Li et al. [20] predicted solid deposition characteristics using the Lagrangian approach. As the Lagrangian approach was used, only fairly large particles were modelled within their computations. Nevertheless, the study was able to shed light on the interaction between particles. They described how a particle plug would interact with the layer of deposited material left by the previous plug, and how the plug leaves some material behind as it traverses along the pipe. Hidayat and Rasmuson [21] investigated the gas-solid flow characteristics within a U-bend in attempt to optimize a pneumatic conveying dryer system. They detailed the effects of the bend radius on both the distribution of particles and the gas velocity and further concluded the importance of these two parameters on the efficiency of the dryer.

Furthermore, McGlinchey et al. [22] looked at the effects of an expanding pipe on the flow behaviour and identified the increased aeration of the abrupt expansion to be beneficial to the conveying process. Behera et al. [23] studied the inclusion of particle size variations in their numerical simulations of fine-particles conveying and observed good agreements between the pressure drop data between experiments and numerical simulations. They also found that segregation of different particle diameters happens in the pipeline, with different slip velocities for the bottom dense phase region and the top dilute phase region. In a study closely relevant to this thesis, Ibrahim et al. [24] considered the effects of pipe bend orientation on pneumatic conveying flow behaviour. They looked at each orientation of the pipe bends individually and used numerical simulations

to predict the particle trajectories. Schallert and Levy [25] performed an investigation on the flow within a combination of pipe bends with different orientations. In particular, they studied the flow changes associated with a horizontal-to-horizontal (HH) pipe bend followed by an HV pipe bend. One interesting observation made in that study was that the material spiral along the vertical pipe wall after exiting the second pipe bend. Such investigations demonstrated the applications of numerical simulations in the understanding pneumatic conveying flow characteristics in different situations.

While designing pneumatic pipelines, under or over-prediction of flow parameters such as pressure drop can have real consequences to the efficiency and cost of the system [26]. With numerical simulations as a tool for designing pipelines, there was a demand for more accuracy. As researchers work towards developing a more robust and accurate numerical model to study particle-laden flows, most of them have turned towards using a Lagrangian particle treatment for dilute flows [27]. One of the reasons arguing for solving the complete equations of motion for the particles is to obtain highly accurate simulations of particle-laden flows as all discrete characteristics of the solid phase are retained [28]. However, the primary downside of the Lagrangian scheme is that with current computing architectures, it is simply not feasible to scale up the computations to study real-life industrial applications due to computational costs [29]. For studies with more complex geometry or higher solids loading ratio, the Eulerian scheme remains the only practical approach in terms of computational cost and time. Though, one of the concerns surrounding the Eulerian scheme is its assumption of a continuum. This means that the actual dynamics of the flow

starts to deviate from the assumptions as the particle phase gets more dilute, i.e. the distance between particles increases [30]. Van Deemter and Van der Laan [17] performed the pioneering work of deriving the balance equations of an Eulerian scheme by integration methods. Anderson and Jackson [31] later on used the equations of motion for a particle, together with the Navier-Stokes equations for fluid motion, and came up with the set of governing equations that are still used today in many commercial computing software. However, many constitutive relations could not be obtained directly by their method. A large part of this initial gap has since been filled with models either derived with another method or by empirical correlations. These include, for example, the interface drag term [32-34], granular temperature [35] and solid shear stresses [36].

One concern that surfaces frequently in terms of CFD methods is its accuracy in modelling the behaviour of the materials within the pipe. It is with the pursuit of more accurate models that there is now a trove of studies within the literature that aims to improve the accuracy of numerical simulations of pneumatic conveying. Chu and Yu [37] employed a Lagrangian method to replicate earlier published experimental data. They obtained results in a VH pipe bend that were qualitatively similar to the experiments. This was observed in terms of particle distributions, velocities and conveying rates within the pipe. With that, they were able to attempt deeper analysis in terms of the gas-particle, particle-particle and particle-wall interactions. These are quantities that are non-measurable in experiments. Likewise in a Lagrangian modelling study, Quek, Wang and Ray [38] were able to quantitatively match, against published experimental results,

the particle velocity and concentrations. This paved way for further analysis into particle roping and deposition phenomena.

Ratnayake et al. [39] performed experiments on cement and ilmenite flows in a 90° HH pneumatic conveying system. The numerical study was done with the Eulerian granular approach and standard k- ϵ turbulence model, taking into account the drag and lift forces, particle slip, electrostatic forces and particle-particle and particle-wall interactions. Pressure drop, particle velocity and particle concentrations data collected from experimental procedures were within $\pm 15\%$ of numerical simulation results. Behera et al. [23] performed experimental and numerical simulations on alumina, a material similar to cement. The numerical study was done with an Eulerian approach and dispersed k- ϵ turbulence model, taking into account drag forces, particle-particle, particle-air and particle-wall interactions. Particle size distribution was also taken into account. The numerical model was validated against pressure drop data from experimental procedures. Particle velocity, particle concentration and pressure variation data obtained from the numerical simulation were averaged $\pm 20\%$ of experimental results. Figure 4 shows the data from the respective studies that presents calculated pressure drop versus the pressure drop measured in experiments. The greater deviations seen in [23] can be attributed to experimental uncertainties caused by using pressure transmitters which measures pressure near the wall where the turbulent intensity is much higher [40]. Pu et al. [29] were able to match their own experiments with the Eulerian model accurately. From there, they noted the effects of solids concentration and transport pressure in vertical pipes and slug flow characteristics within horizontal pipes.

Figure 4 Calculated versus experimental pressure drop of (a) [39] and (b) [23].

(Source: respective publications)

One of the closure relations of the Eulerian granular model that should be highlighted here is the FAD turbulent dispersion model. Because the distribution of particles (even in areas with relatively low particle concentrations) has been shown to affect the flow structures of the gas phase [41-43], it is essential to accurately model the particle distributions. Turbulent dispersion is the mechanism for the particle phase to be transported from regions of high concentration to lower concentrations [11] and this is especially so after a pipe bend. Burns et al. [11] utilized Favre averaging of the interphase drag term to obtain the additional terms in the momentum equations that account for turbulent dispersion. The authors had named it the FAD model. This model finds itself most commonly used in scenarios pertaining to gas-liquid flows (i.e. droplets or bubbly flows). Of the currently known publications that either referred to or utilized the FAD model in their numerical models, a majority of it sees no application towards a gas-solid flow. This is despite the original authors concluding that the model is a generalization of other turbulent dispersion models and it should hold true for a wide range of multi-phase flow scenarios.

Furthermore, to the best of the authors' knowledge, there were no known restrictions raised in the existing literature in terms of applying this particular turbulent dispersion model to a dilute gas-solid flow.

Also, in designing a pneumatic pipeline, it would be highly desirable if favourable design aspects of experimental pipelines can be scaled-up directly. In the commonly cited pneumatic conveying design guide, Mills [1] provided procedures to scale up pipe length and diameters on the basis of similar pressure drop and air velocities but as mentioned in the introduction, such procedures are not perfect. Therefore, Pan [44] and Mallick and Wypych [45] worked on trying to improve such scaling-up procedures. While the results are promising, the conclusions drawn are that the order of inaccuracy is only acceptable in practical design practice. When applying such techniques, a suitable numerical or theoretical model should be applied and more appropriate resulting scaling-up techniques have to be sought if better predictions are to be achieved. As such, the direct application of numerical simulation to the conveying pipeline appears to be a viable methodology to do that.

Lastly, moving on to the available pneumatic conveying literature that dealt with pipe bends, Vashisth and Grace [46] used the Lagrangian scheme to look at the conveying of different classification of granular materials. The pipe bend was an HV configuration and they concluded that secondary flow disperses the particle rope that was formed by the preceding bend. Also, the particle density and diameter was found to affect the flow. Lee et al. [47] also drew the same conclusion that different classifications of particles exhibit different flow

characteristics in a same HV pipe configuration. More importantly, they highlighted an annulus flow structure in the vertical pipe section where the particles are mainly conveyed upwards along the pipe walls. Kruggel-Emden and Oschmann [48] extended the study of pneumatic conveying to non-spherical particles to investigate the effects of the particles' shape on the flow behaviour. While some differences were documented, the general flow development is still a particle rope forming within the HV bend and an annular structure forming downstream inside the vertical pipe. There are many more studies on pipelines with a single pipe bend of different orientations. However, only the work of Schallert and Levy [25] discussed earlier was found to touch on a combination of two bends. Each orientation of pipe bend, or flow direction in the vertical pipe section, produces different flow characteristics. It may be the reason why there has not been much more studies that dealt with a combination of bends. With a combination of bends, the inlet of the second bend is influenced by the first bend and will produce different flow characteristics in the second bend. Therefore, it presents an opportunity to build on the literature for two-phase flows in pipelines with more than one bend.

1.3 Research motivation and significance

In summary, the motivation of the current work comes from both the research community and industry. This thesis evolved from an industrial research collaboration between NTU and JPPL of which the primary focus is to study the cement flow characteristics in a pneumatic conveying pipeline. The results will allow for further exploration into potentially beneficial engineering changes that may offer higher cement delivery rates and to enhance operational and energy

efficiency. Results obtained from the study which falls within the scope of this thesis is presented here. One of which is the application of numerical simulations directly on the full-scale pipeline model, motivated by the fact that interim flow data extraction from the pipeline is not possible. Although numerical simulation of the full-scale pipeline may be argued as an impractical approach, the process is necessary as the only means to gain insight into the flow within the pipelines. It is eventually found to be not as daunting as initially thought once a reasonable compromise is found between simulation accuracy and computational resources required. Also, scale-up design procedures have been concluded to be far from perfect and it presents an opportunity to demonstrate the ability of full-scale simulations as an alternative tool for the study of pneumatic conveying.

Moving on, a few gaps have been uncovered in the process of literature review. Firstly, the application of FAD turbulent dispersion model towards gas-solid flows is not common and its suitability is briefly discussed in this work. Secondly, the literature on the flow characteristics in a pipeline with a combination of two bends in close proximity is lacking. As pneumatic conveying boasts of the flexibility of the pipeline through various fixtures, multiple bends in close proximity will certainly appear in practical applications as evidently seen in JPPL's pipeline. However, the flow characteristics within such configurations are still not well-documented. This thesis aims to expand the currently limited literature through the use of numerical simulations for a combination of an HV and VH bend. Also, the effects of pipe bend radius and the length of the connecting pipe between the two bends are postulated to have a significant effect

on the flow characteristics. Such effects will be studied and results will cater for future research and industrial purpose.

Chapter 2 Numerical method

2.1 Numerical model

The study will use the Eulerian granular model from commercial software ANSYS Fluent 13.0 (ANSYS, Inc). The Eulerian scheme treats both the gas and solid phase as continuums and allows them to be interpenetrating. Therefore, the governing equations of the model start from the general continuum equations.

For a single-phase fluid flow, the mass conservation equation is:

$$\frac{\partial}{\partial t}(\rho) + \nabla \cdot (\rho \vec{v}) = S_m \quad (1)$$

where t is time, ρ is density, $\vec{v}$ is velocity and S_m is the mass source term where mass can be gained or lost.

The momentum conservation equation for a single-phase fluid is:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \bar{\bar{\tau}} + \rho \vec{g} + \sum \vec{F} \quad (2)$$

where p is pressure, $\vec{g}$ is gravitational acceleration and $\sum \vec{F}$ is the sum of external forces acting on the fluid. The stress tensor $\bar{\bar{\tau}}$ is given by:

$$\bar{\bar{\tau}} = \mu(\nabla \vec{v} + \nabla \vec{v}^T) + \left(\lambda - \frac{2}{3}\mu\right) \nabla \cdot \vec{v} \bar{\bar{I}} \quad (3)$$

where μ is dynamic viscosity, λ is bulk viscosity or also known as the second viscosity and $\bar{\bar{I}}$ is an identity tensor.

In a two-phase flow, the idea of volume fraction is introduced. The Eulerian model applies the continuum hypothesis for the solid phase. It assumes all the solid particles can be mathematically formulated like that of a fluid. The solids,

together with the fluid, is treated as two interpenetrating continuums. This interpenetrating effect is captured by a dimensionless variable called the volume fraction α ($0 \leq \alpha \leq 1$). In a certain (fixed) space, the volume occupied by a certain phase q is the sum (or integral) of the volume fraction multiplied by the volume:

$$V_q = \int_V \alpha_q dV \quad (4)$$

Then, the sum of all volume fractions must be equal to one:

$$\sum_{q=1}^n \alpha_q = 1 \quad (5)$$

The effective density $\hat{\rho}$ of phase q will then be:

$$\hat{\rho}_q = \alpha_q \rho_q \quad (6)$$

By using the effective density, a separate set of conservation equations can be written for the gas (subscript g) and solid (subscript s) phase. The mass conservation equation of the gas phase, following the form of Equation 1, is:

$$\frac{\partial}{\partial t} (\alpha_g \rho_g) + \nabla \cdot (\alpha_g \rho_g \vec{v}_g) = S_g + \sum_{s=1}^n (\dot{m}_{sg} - \dot{m}_{gs}) \quad (7)$$

where the additional term on the right represents the mass gained from solid phase $\dot{m}_{sg}$ and mass lost to the solid phase $\dot{m}_{gs}$. Other mass source terms that are not associated with the solid phase remains in the S_g term. For the current study, no mass exchange mechanisms are present between air and cement, and that the air does not gain or lose mass in other forms. Therefore, the entire right-hand side is zero.

Similarly, the momentum conservation of the gas phase, following the form of equation 2, is:

$$\begin{aligned} \frac{\partial}{\partial t}(\alpha_g \rho_g \vec{v}_g) + \nabla \cdot (\alpha_g \rho_g \vec{v}_g \vec{v}_g) \\ = -\alpha_g \nabla p + \nabla \cdot \bar{\bar{\tau}}_g + \alpha_g \rho_g \vec{g} + \sum \vec{F} + \sum_{s=1}^n \vec{R}_{sg} \end{aligned} \quad (8)$$

where $\bar{\bar{\tau}}_g$ is now:

$$\bar{\bar{\tau}}_g = \alpha_g \mu_g (\nabla \vec{v}_g + \nabla \vec{v}_g^T) + \alpha_g \left(\lambda_g - \frac{2}{3} \mu_g \right) \nabla \cdot \vec{v}_g \vec{I} \quad (9)$$

Comparing with equation 2, the additional term on the right-hand side of equation 8 is due to interactions between different phases. Momentum changes due to mass exchange are left out following the same arguments made for equation 7. The forces in the $\vec{F}$ term for the gas momentum equation represents an external body force $\vec{F}_g$. In this current study, there are no external body forces. Therefore, the last two terms on the right-hand side of equation 8 is zero. The bulk viscosity of the gas phase in equation 9 is trivial as there are no rapid expansion or compression of air in pneumatic conveying and is left out of the model. As seen, the governing equations for the gas phase in the Eulerian scheme is in fact not much different from the usual single-phase flow equations. The differences and complexities lies within the set of equations for the solid phase.

The mass conservation equation of the solid phase, again following the form of equation 1, is:

$$\frac{\partial}{\partial t}(\alpha_s \rho_s) + \nabla \cdot (\alpha_s \rho_s \vec{v}_s) = 0 \quad (10)$$

with the irrelevant terms on the right-hand side left out following the same arguments presented for equation 7.

The momentum conservation of the solid phase, following equation 2, is:

$$\begin{aligned} \frac{\partial}{\partial t}(\alpha_s \rho_s \vec{v}_g) + \nabla \cdot (\alpha_s \rho_s \vec{v}_s \vec{v}_s) \\ = -\alpha_s \nabla p_g - \nabla p_s + \nabla \cdot \bar{\bar{\tau}}_s + \alpha_s \rho_s \vec{g} + \sum \vec{F} \\ + \sum_{s=1}^n \vec{R}_{gs} \end{aligned} \quad (11)$$

where $\bar{\bar{\tau}}_s$ is:

$$\bar{\bar{\tau}}_s = \alpha_s \mu_s (\nabla \vec{v}_s + \nabla \vec{v}_s^T) + \alpha_s \left(\lambda_s - \frac{2}{3} \mu_s \right) \nabla \cdot \vec{v}_s \vec{I} \quad (12)$$

In equations 11 and 12, there are a number of unknowns that would require mathematical closure before the equations can be solved. Namely, the solids pressure p_s , sum of forces $\vec{F}$, sum of interphase momentum exchange $\vec{R}$, solids bulk viscosity λ_s and solids viscosity μ_s . However, a short discussion on the multiphase conservation equations is required. While the mathematical manipulations appear to be a matter of simply replacing the density with an effective density, no proof is presented herein. Nevertheless, it does not mean that derivations are not possible. Anderson and Jackson [31] derived, by local averages, the same set of equations using the equations of motion of a single particle and the Navier-Stokes equations. They provided a postulated form of the constitutive equations which paved way for rigorous research leading to many different models suitable either in general form or in selected cases. The complete set of closure relations applicable to this study is presented.

The solids pressure is an additional pressure term resulting from assuming the form of the stress tensor analogous to a Newtonian fluid. Similar to gas pressure, the solids pressure should in some way be related by a temperature term and volume term, since the pressure of an ideal gas depends on such factors ($pV = nRT$). Lun et al. [36] utilized a collisional particle pair-distribution function, which was obtained by simple kinetic analysis by Savage and Jeffrey [49], to derive the pressure tensor which consists of a kinetic part and a collisional part. The expression obtained is:

$$p_s = \alpha_s \rho_s \Theta_s + 2\rho_s(1 + e_{ss})\alpha_s^2 g_{0,ss} \Theta_s \quad (13)$$

where Θ_s is granular temperature, e_{ss} is coefficient of restitution for collision (taken as 0.9 in this study [50]) and $g_{0,ss}$ is the radial distribution function. The first term represents the contributions from the kinetic part and the second term from the collisional part. The interpretation of granular temperature is obvious. However, unlike in gases where the temperature is easily quantifiable, the granular temperature is proportional to the particles' fluctuation:

$$\Theta_s = \frac{1}{3} \overline{u_{t,s} u_{t,s}} \quad (14)$$

Ding and Gidaspow [35] later on derived the transport equation for granular temperature as follows:

$$\begin{aligned} \frac{3}{2} \left[\frac{\partial}{\partial t} (\rho_s \alpha_s \Theta_s) + \nabla \cdot (\rho_s \alpha_s \vec{v}_s \Theta_s) \right] \\ = (-p_s \bar{I} + \bar{\tau}_s) : \nabla \vec{v}_s + \nabla \cdot (k_{\Theta_s} \nabla \Theta_s) - \gamma_{\Theta_s} + \phi_{gs} \end{aligned} \quad (15)$$

where the diffusion coefficient k_{Θ_s} is provided by Syamlal et al. [51] who neglected the kinetic contribution from the expression by Lun et al. [36], because it tends to an unphysical value at low volume fractions:

$$k_{\Theta_s} = \frac{15d_s\rho_s\alpha_s\sqrt{\Theta_s\pi}}{4(41-33\eta)} \left[1 + \frac{12}{5}\eta(4\eta-3)\alpha_sg_{0,ss} + \frac{16}{15\pi}(41-33\eta)\eta\alpha_sg_{0,ss} \right] \quad (16)$$

with $\eta = 0.5(1 + e_{ss})$ and the collisional dissipation of energy γ_{Θ_s} is that of the expression by Lun et al. [26] based on the assumption of slight inelasticity:

$$\gamma_{\Theta_s} = \frac{12(1 - e_{ss}^2)g_{0,ss}}{d_s\sqrt{\pi}} \rho_s \alpha_s^2 \Theta_s^{\frac{3}{2}} \quad (17)$$

and $\phi_{gs} = -3K_{gs}\Theta_s$ is the energy exchange between the phases as given by Gidaspow et al. [52], with the form of the momentum exchange coefficient K to be discussed later.

What remains in the solids pressure and granular temperature formulation is the radial distribution function $g_{0,ss}$. The function is a correction factor to account for the changes in the probability of collision between two particles when the solid phase gets ‘denser’, i.e. the local solid phase volume fraction increases, resulting in higher possibilities of collision. Although not directly analogous, it can be argued that the radial distribution function serves the same purpose of the volume term in ideal gas theory. This function should then tend towards 1 and ∞ when the solid volume fraction tends towards 0 and the packing limit respectively. The radial distribution function $g_{0,ss}$ as presented by Ogawa et al. [53] through a kinetic model of collision of particles, is:

$$g_{0,ss} = \left[1 - \left(\frac{\alpha_s}{\alpha_{s,max}} \right)^{\frac{1}{3}} \right]^{-1} \quad (18)$$

where $\alpha_{s,max}$ is the packing limit of the solids phase. For regular spheres of constant diameter, this packing limit is 0.63 [54].

The force $\vec{F}$ in equation 11, unlike in equation 8, contains forces other than body forces. Such forces may include the lift force $\vec{F}_{lift,s}$, wall lubrication force $\vec{F}_{wl,s}$, virtual mass force $\vec{F}_{vm,s}$, and/or turbulent dispersion force $\vec{F}_{td,s}$. As cement particles are small, with diameters in the order of $10^{-5}m$, the lift force on the cement particles is insignificant compared to the drag. Wall lubrication force is only pertinent to bubbly multiphase flows and virtual mass force is appropriate when the density of the secondary phase is much lesser than that of the primary phase. This leaves the solid phase momentum equation to be:

$$\begin{aligned} \frac{\partial}{\partial t}(\alpha_s \rho_s \vec{v}_g) + \nabla \cdot (\alpha_s \rho_s \vec{v}_s \vec{v}_s) \\ = -\alpha_s \nabla p + \nabla \cdot \bar{\bar{\tau}}_s + \alpha_s \rho_s \vec{g} + \vec{F}_{td,s} + \vec{R}_{gs} \end{aligned} \quad (19)$$

The turbulent dispersion force $\vec{F}_{td,s}$ is modelled with the Favre-averaged drag (FAD) model [11] that is included in the software. Two other models, namely the Simonin model [55] and Lopez de Bertodano model [56] are available. The Simonin model essentially estimates the dispersion scalar D_s in Equation 20 by the turbulent viscosity, similar to the FAD model. The Lopez de Bertodano model is shown to be mathematically equivalent by Burns et al. Hence, the other two models should display the same accuracy, the only difference being the model constant which will be determined later on. The FAD model was developed by time averaging the conservation equations that were derived by ensemble averaging, giving:

$$\vec{F}_{td,s} = C_{TD} K_{sg} \frac{D_s}{\sigma_{gs}} \left(\frac{\nabla \alpha_g}{\alpha_g} - \frac{\nabla \alpha_s}{\alpha_s} \right) \quad (20)$$

where C_{TD} is a modifiable constant in the software, K_{sg} is the interphase momentum exchange coefficient, σ_{gs} is the turbulent Prandtl number and the dispersion scalar D_s is estimated by:

$$D_s = \frac{\mu_{ts}}{\rho_s} \quad (21)$$

The momentum exchange between phases or, interphase momentum exchange term, $\vec{R}$ obeys the following constrain that:

$$\overrightarrow{R_{sg}} = -\overrightarrow{R_{gs}} \quad (22)$$

and is modelled using the differences in velocities between the two phases:

$$\overrightarrow{R_{sg}} = K_{sg}(\vec{v}_s - \vec{v}_g) \quad (23)$$

where K is an interphase momentum exchange coefficient. The mechanism dominant within this term is the drag force. The coefficient used here follows the suggestion of Gidaspow [34]. It is a combination of the Ergun [32] model at solid volume fractions larger than or equal to 0.2 and the Wen and Yu [33] model at solid volume fractions lower than 0.2. Ergun's model was determined by the pressure drop of an experiment for fixed liquid-solid beds at packed conditions whereas Wen and Yu's model was correlated from solid particle settling experiments over a wide range of volume fractions. The gas-solid exchange coefficient K_{sg} is:

$$K_{sg} = \frac{3}{4} C_D \frac{\alpha_s \alpha_g \rho_g |\vec{v}_s - \vec{v}_g|}{d_s} \alpha_g^{-2.65} \quad (24)$$

when $\alpha_g > 0.8$. d_s is the particle diameter and the drag function C_D is:

$$C_D = \frac{24}{\alpha_g Re_s} \left[1 + 0.15 (\alpha_g Re_s)^{0.687} \right] \quad (25)$$

with Re_s being the relative Reynolds number:

$$Re_s = \frac{\rho_g d_s |\vec{v}_s - \vec{v}_g|}{\mu_g} \quad (26)$$

For when $\alpha_g \leq 0.8$,

$$K_{sg} = 150 \frac{\alpha_s^2 \mu_g}{\alpha_g d_s^2} + 1.75 \frac{\rho_g \alpha_s |\vec{v}_s - \vec{v}_g|}{d_s} \quad (27)$$

The bulk viscosity λ_s obtained from the total stress tensor by Lun et al. [36] is:

$$\lambda_s = \frac{8}{3} \alpha_s^2 \rho_s d_s g_{0,ss} \left(\frac{\Theta_s}{\pi} \right)^{\frac{1}{2}} \quad (28)$$

for perfectly elastic particles. For inelastic particles, the bulk viscosity is:

$$\lambda_s = \frac{8}{3} \alpha_s^2 \rho_s d_s g_{0,ss} \left(\frac{\Theta_s}{\pi} \right)^{\frac{1}{2}} \times \frac{1}{2} (1 + e_{ss}) \quad (29)$$

The solids pressure (equation 13), which arose from the normal components of the stress, contained both a kinetic and collisional part. Similarly, for the shear stress, these two components should be present. However, in the case when the solid volume fraction approaches the packing limit, the solid phase may transit from a viscous (fluid-like) medium to a plastic (solid-like) medium. Such a transition can be captured by an additional frictional component for the shear viscosity. The solids shear viscosity μ_s can now be written as consisting of three parts, collisional, kinetic and frictional part:

$$\mu_s = \mu_{s,col} + \mu_{s,kin} + \mu_{s,fr} \quad (30)$$

The collisional part obtained by Lun et al. [36] is:

$$\mu_{s,col} = \frac{4}{5} \alpha_s \rho_s d_s g_{0,ss} (1 + e_{ss}) \left(\frac{\Theta_s}{\pi} \right)^{\frac{1}{2}} \alpha_s \quad (31)$$

and is the same expression as obtained by Gidaspow [34].

The kinetic part originally obtained by Lun et al. [52] is:

$$\mu_{s,kin} = \frac{5}{96} \rho_s d_s \sqrt{\Theta_s \pi} \left[\frac{1}{\eta(2-\eta)} \right] \left[\frac{1}{g_{0,ss}} + \frac{8}{5} \eta \alpha_s g_{0,ss} \right] \left[1 + \frac{8}{5} \eta \alpha_s g_{0,ss} (3\eta - 2) \right] \quad (32)$$

where $\eta = \frac{1}{2}(1 + e_{ss})$. However, Gidaspow [34] argued that a correctional factor of $1/[\eta(2 - \eta)]$ is insignificant for a coefficient of restitution of 0.9, with less than a 0.3% difference. Hence, the kinetic part is:

$$\mu_{s,kin} = \frac{5}{96} \rho_s d_s \sqrt{\Theta_s \pi} \left[\frac{1}{\eta g_{0,ss}} \right] \left[1 + \frac{8}{5} g_{0,ss} \alpha_s \eta \right]^2 \quad (33)$$

Note that equation 33 is not obtained by simply removing the correction factor from equation 32. Nevertheless, both expressions have no discernible mathematical difference at $e_{ss} = 0.9$ [10].

The frictional part of solids shear viscosity follows the work of Johnson and Jackson [57] and is:

$$\mu_{s,fr} = 0.1 \alpha_s \frac{(\alpha_s - \alpha_{s,min})^2}{(\alpha_{s,max} - \alpha_s)^5} \sin \phi \quad (34)$$

where ϕ is the internal angle of friction. The frictional component of solid shear stress is not crucial in the current study as the flows simulated are in the dilute flow regime. The model only comes into the picture when the solid volume fraction reaches a critical value $\alpha_{s,min}$ and the solid volume fraction is not expected to reach $\alpha_{s,min} = 0.5$. The model is, however, defined in the numerical setup to capture any unexpected regions of dense flow. If no such regions exist, having the model defined would not increase computational costs.

The standard k- ϵ turbulence model was used for the gas phase turbulence and standard wall functions are used. For the Eulerian granular model, a ‘per-phase’

turbulence modelling was applied, where a set of separate $k-\epsilon$ turbulence equations was solved for each phase. Equations 35 and 36 shows the standard $k-\epsilon$ turbulence model applied to the solid phase where the interphase turbulence interaction is included.

$$\begin{aligned}
 \frac{\partial}{\partial t}(\alpha_s \rho_s k_s) + \nabla \cdot (\alpha_s \rho_s \vec{U}_s k_s) \\
 = \nabla \cdot \left(\alpha_s \left(\mu_s + \frac{\mu_{t,s}}{\sigma_k} \right) \nabla k_s \right) + \alpha_s G_{k,s} - \alpha_s \rho_s \epsilon_s \\
 + K_{gs} (C_{gs} k_g - C_{sg} k_s) - K_{gs} (\vec{U}_g - \vec{U}_s) \cdot \frac{\mu_{t,g}}{\alpha_g \sigma_g} \nabla \alpha_g \\
 + K_{gs} (\vec{U}_g - \vec{U}_s) \cdot \frac{\mu_{t,s}}{\alpha_s \sigma_s} \nabla \alpha_s
 \end{aligned} \tag{35}$$

$$\begin{aligned}
 \frac{\partial}{\partial t}(\alpha_s \rho_s \epsilon_s) + \nabla \cdot (\alpha_s \rho_s \vec{U}_s \epsilon_s) \\
 = \nabla \cdot \left(\alpha_s \left(\mu_s + \frac{\mu_{t,s}}{\sigma_\epsilon} \right) \nabla \epsilon_s \right) \\
 + \frac{\epsilon_s}{k_s} \left[C_{1\epsilon} \alpha_s G_{k,s} - C_{2\epsilon} \alpha_s \rho_s \epsilon_s \right. \\
 + C_{3\epsilon} \left(K_{gs} (C_{gs} k_g - C_{sg} k_s) - K_{gs} (\vec{U}_g - \vec{U}_s) \right. \\
 \left. \left. \cdot \frac{\mu_{t,g}}{\alpha_g \sigma_g} \nabla \alpha_g + K_{gs} (\vec{U}_g - \vec{U}_s) \cdot \frac{\mu_{t,s}}{\alpha_s \sigma_s} \nabla \alpha_s \right) \right]
 \end{aligned} \tag{36}$$

Coupling between pressure and velocity is achieved by the Phase Coupled SIMPLE (PC-SIMPLE) algorithm. The software provides two options for the pressure-velocity coupling, namely the phase-coupled SIMPLE (PC-SIMPLE) and multiphase coupled. However, in the current case, no mass transfer occurs between the two phases to which the multiphase coupled approach is more suited. SIMPLEC and PISO approaches are available only for the mixture multiphase model where only one set of conservation equation is solved. However, the mixture multiphase model trades a little accuracy for computational efficiency. Both models are equally valid for the current methodology but the Eulerian granular model is chosen for its ability to incorporate varying particle diameters

in future studies. Second-order spatial discretization is applied for density, momentum, turbulent kinetic energy and dissipation rate, while first order discretization is applied for the volume fraction. Higher order discretization schemes were favoured for better accuracy. However, the two suitable schemes available for the volume fraction are first-order discretization and the QUICK scheme. Although the mesh is generated as a hexagonal mesh, it is in fact an unstructured grid. Therefore, the QUICK scheme was not chosen.

Figure 5 shows a summary of the Eulerian granular model used in this study, along with the selected closure relations. The energy conservation equations are left out in the current model as the flow is assumed to be incompressible.

2.2 Numerical setup

2.2.1 Geometry

Geometries of each conveying pipeline used in numerical simulations will be presented at the start of each relevant sections. In total, four sets of pipelines are used. Each pipe geometry is drafted using commercial software SOLIDWORKS (Dassault Systèmes) and exported to the meshing software.

2.2.2 Mesh generation

Commercial software ANSYS ICEM CFD (ANSYS, Inc) is used for the mesh generation. When generating the mesh, an O-grid multi-block strategy is applied as it is able to provide an appropriate level of control over the cells. Although such a strategy is usually employed for the generation of structured meshes, the output mesh is exported as an unstructured mesh as the numerical solver used is

Figure 5 Summary of the Eulerian granular model

an unstructured solver. The advantage of this method is as follows. As a structured mesh generation strategy, it allows the generation of a relatively uniform hexahedral mesh at the pipe bend. By utilizing an ordered hexahedral

Figure 6 O-grid blocking strategy at the pipe bend used to generate ordered hexahedral cells.

mesh, the majority of the computational cells' face is orientated perpendicular to the flow direction, hence, increasing the stability of the calculations. Especially at the pipe bends, the blocking strategy is able to provide cells that are aligned to the flow in a consistent manner. Across the stream wise cross-section, the pipe is divided into five blocks, one in the middle and four at the wall sides. Figure 6 shows an illustration of the blocking strategy at the bend. At the bend, the geometry is sectioned into four roughly equal parts over the whole 90° to capture the curvature of the pipe. However, the number of segments will depend on the bend radius, which is a varying parameter in one of the sets of geometry.

To address two-phase flows with numerical simulation, the size of the mesh should be much larger than the particle diameter and much smaller when compared to the system. A small enough mesh, as per best practices in numerical simulations, was required to capture the gas-phase flow details. However, since the model treats the solid-phase as a continuum, the cells have to be much larger

than the particle diameter. From a physical point of view, only when the amount of solid particles in a certain volume is sufficient, can the continuum hypothesis hold. The appropriate size of the computational cells can be obtained from the literature. Table 1 lists some of the previous two-phase gas-solid studies who had published their mesh cells' data.

Table 1 List of studies and their reported mesh size

Year	Model treatment	Flow	$d_s / \mu m$	Volume of cells $/m^3$	Ref.
2005	Eulerian	Dilute	450^+	$\mathcal{O} 10^{-9}$ to $\mathcal{O} 10^{-8}$	[58]
2005	Lagrangian	Dilute	85^+	$\mathcal{O} 10^{-9}$ to $\mathcal{O} 10^{-7}$	[38]
2008	Lagrangian	Dilute	$2.8mm$	$\mathcal{O} 10^{-8}$ to $\mathcal{O} 10^{-7}$	[37]
2013	Eulerian	Dense	$10 - 61$	$\mathcal{O} 10^{-8}$ to $\mathcal{O} 10^{-7}$	[23]
2013	Lagrangian	Dilute	40^+	$\mathcal{O} 10^{-7}$	[59]
2017	Eulerian	Dilute	14^+	$\mathcal{O} 10^{-6}$	[60]*

⁺ - Mean diameter

^{*} - Larger cell size due to a compromise for the large domain size. Mesh dependency study conducted.

Figure 7 Hexahedral cells generated from O-grid strategy

From these studies, the volume of a single mesh cell fell within the order of 10^{-9} to $10^{-7}m^3$ regardless of whether the Eulerian or Lagrangian method was used, with the exception of [60]. However, a mesh dependency study was conducted and the particle diameter used in the numerical model was one of the smallest. Using such orders of magnitude of mesh cell volume, both dense and dilute phase conveying had been satisfactorily computed. Hence, this study will attempt to utilize hexahedral meshes with cell volumes in the order of 10^{-9} to $10^{-7}m^3$. When the hexahedral mesh was generated from the blocks, the approximate dimensions applied to the cells was $0.002 \times 0.002 \times 0.004m$. The longer dimension in the stream wise direction helps to reduce the total number of cells for the domain. The y^+ value of the first cell at the walls is 150. The maximum orthogonal skew and aspect ratio of the mesh is 0.259 and 4.84 respectively. Figure 7 shows one of the hexahedral mesh that was generated.

2.2.3 Solution strategy

The equations of the Eulerian granular model are highly coupled. This is because they share the same pressure (p_g appearing in both equation 8 and 11) and also have a constrain on the volume fraction term (equation 5). Another issue with the volume fraction term is that the solid volume fraction could in theory diminish indefinitely approaching zero. Physically, the smallest solid volume fraction would be just the fraction volume of one solid particle in a control volume. However, this might not be the case in numerical simulations. As such, the set of multiphase equations would sometimes be difficult to obtain a solution. In addition to those aforementioned, the model consists of much more equations and

closure relations than a single-phase flow would have. In this case, a few solution strategies can be applied.

Firstly, a single-phase solution can be obtained by disregarding the volume fraction through the software's interface. The solver can then solve for a steady-state solution of the gas flow with only one set of conservation equation. This solution is later on assigned as an initial condition before solving the full set of multiphase equations. By doing this, the computational resources required are reduced. Secondly, an initial condition is applied for the solid volume fraction. This is done by applying a constant value throughout the entire computational domain on top of the single-phase steady-state solution. Doing so reduces the occurrence of instabilities in the volume fraction term during calculation. Thirdly, the under-relaxation constraints are reduced a little, as compared to common values applied in single-phase calculations, to aid in the stability of the calculation. Lastly, for transient simulations, small time steps of the order $10^{-4}s$ is applied.

2.2.4 Convergence and mesh dependency check

The convergence characteristic of the Eulerian scheme and mesh dependency check is done using the geometry and test case in the numerical model validation process. Details of the geometry, mesh and problem setup can be found later in Section 2.3. Figure 8 shows the scaled residuals of the continuity equations. Due to a large number of iterations, it is not possible to present the entire convergence behaviour. Nevertheless, part of it is shown here. Residual for the continuity equations is seen to rapidly decrease within the first few time steps and thereafter,

increased a little to attain a rather constant value and steady behaviour. While the scaled residual does not go below the widely-practised value of 10^{-3} , it stays within the same order of magnitude. Such display of consistency would be sufficient in a problem that advocates the application of minute time steps in the order of 10^{-7} to 10^{-5} s [61]. Although the current time step used is larger, at 10^{-4} s, it is a necessary compromise. As later on, analysis will be done on an industrial scale pneumatic conveying pipeline with a considerably large domain size and smaller time steps are not possible due to computational resource limitations.

Figure 8 Scaled residual for the continuity equations during a transient calculation

Figure 9 Outlet particle mass flow rate (left) and air velocity plot at 0.1m after the bend (right)

The outlet particle mass flow rate behaviour could be used to further ensure that the solution is satisfactory. Two other meshes, a coarser and a finer mesh, were also generated to perform a mesh dependency test. The problem was calculated in transient state simulations to obtain a pseudo-steady result. Figure 9 (left) shows the outlet particle mass flow rate over time. The spike in the outlet flow rate plot represents transient characteristics of the flow where accumulated particles during the start-up acceleration phase are ejected collectively when the solution approached pseudo-steady state. The areas under all three graphs calculated with the plotting software are similar, signifying a consistent mass output for all three meshes used, as long as the simulation is allowed to proceed until the pseudo-steady state. The particle output is seen to reach a steady value beyond two seconds of flow time. Figure 9 (right) plots the air velocity across the pipe diameter at a location after the pipe bend. The coarser mesh presents a slight difference in the air velocity but the current mesh shows no discernible disagreement with that from the finer mesh. Hence, the current mesh sizing (labelled 'Fine mesh' in Figure 9) is deemed to be adequately small to achieve mesh independence and is selected to be used in succeeding calculations.

2.3 Model validation

2.3.1 Numerical setup

Numerical model validation will be based on a previous experimental work performed by Huber and Sommerfeld [62], where detailed descriptions of the experimental setup could be found. Experimental results provided by them will be used to compare against numerical simulation results arising from the present study. In their experimental study, spherical glass beads were conveyed by air

Figure 10 Pipeline dimensions for model validation, flow direction indicated by dashed arrows.

through a pipe bend. The diameter of the glass beads was sampled by phase Doppler anemometry and found to vary as a skewed distribution within the range of $1\mu\text{m} < d < 100\mu\text{m}$. However, the particle diameter is set to a particle size of $42\mu\text{m}$ in the current numerical simulations, following the number mean diameter the original authors had documented to allow more direct comparisons. By affixing the particle diameter to take a single value, it reduces numerical complexities by not having to define more than one Eulerian solid phase. The material density is defined as $2500\text{kg}/\text{m}^3$. The pipe diameter is set to 0.08m , similar to that used by the experimental study, and consists of a 90° HV bend. The length of the straight pipe section before and after the bend is 5.5m and 3m respectively. Figure 10 shows the pipeline that is used for the numerical model validation. The mesh generated is shown previously in Figure 7.

A mass flow rate inlet boundary condition is applied to match the original experiments; in which they had controlled the amount of material introduced into the pipe by screw feeders. The outlet boundary condition is set to a pressure outlet to atmospheric pressure. At the walls, different treatments are applied for both of the phases. A no-slip condition is applied for the gas phase and a specular coefficient of 0.6 is applied for the solids phase. The specular coefficient is used for a rough wall and it defines the collision characteristics of a particle with the wall, with a value of 0 being a perfectly smooth wall. For a wall roughness height that is comparable to the particle diameter, the specular coefficient will be large [63]. However, this value is not easily determined with an explicit expression but has to be compared against experiments [64]. Hence, the value of 0.6 is provided as an educated guess. In order to balance between consumption of computational resources and numerical stability, the time step size is set to $1 \times 10^{-4}s$ as the solution has no issues with convergence using the established time step size. Furthermore, the application of initial conditions listed in Section 2.2.3 aided in the computational stability. From the experimental data, the particle mass concentration is mostly between 1 to $2kg/m^3$. Using the expression *particle mass concentration = volume fraction $\times$ particle density*, it is possible to deduce that the actual particle volume fraction within the pipe is in the order of 0.001. Consequentially, the value of 0.001 is used as an initial condition for the solid volume fraction.

2.3.2 Turbulent dispersion model

One of the closure relations that should be highlighted during the model validation process is the turbulent dispersion model. Because the distribution of

Figure 11 Effect of different model constant for turbulent dispersion model on air velocity (left) and particle concentration (right).

particles (even in areas with relatively low particle concentrations) has been shown to affect the flow structures of the gas phase [41-43], it is essential to accurately model the particle distributions. Turbulent dispersion is the mechanism for the particle phase to be transported from regions of high concentration to lower concentrations [11] and this is especially so after a pipe bend. Burns et al. [11] utilized Favre averaging of the interphase drag term to obtain the additional terms in the momentum equations that account for turbulent dispersion. The authors had named it the Favre Averaged Drag (FAD) model. This model finds itself most commonly used in scenarios pertaining to gas-liquid flows (i.e. droplets or bubbly flows). Of the currently reviewed publications that either referred to or utilized the FAD model in their computational models, none had applications dealing with gas-solid dilute phase flows. This is despite the original authors concluding that the model is a generalization of other turbulent dispersion models and it should hold true for a wide range of multi-phase flow scenarios. Furthermore, to the best of the authors' knowledge, there are no known restrictions raised in the existing literature in terms of applying this particular turbulent dispersion model to a dilute gas-solid flow.

Here, the model constant of the turbulent dispersion model is varied while all other parameters remained constant. Figure 11 shows the simulation results and it emphasizes on the chosen model constant of 25 (solid black line) and experimental data (dashed black line). The results are compared with experimental data for air velocity and particle concentrations right after the bend. The left of Figure 11 plots air velocity and the right is particle concentration, for different model constants as presented. While it seems that the model constant does not affect the air velocity drastically, there is a significant effect on the results of particle concentration. Indeed, the turbulent dispersion term is present only in the momentum equation of the solids phase. A change in particle volume fraction changes the flow behaviour of the gas phase. Therefore, with particle volume fractions in the order of 10^{-3} , the air velocity is not subject to drastic changes. From the particle concentration plot, a model constant of 20 – 30 presents reasonable match with experimental data. This is determined by a rough evaluation of the area under the curve through a trapezoidal method instead of comparing absolute values because the particle mass flux has to be taken into consideration. Or rather, roughly the same amount of material should flow through a given cross-sectional area in both the experimental case and numerical simulations and this can be obtained from a combination of particle concentration and particle velocity data. Hence, the model constant 15 is not considered even though its peak value coincides with experimental data because the area under the curve will then be significantly more than that of the experimental data. As the air velocity plots are the closest match when the turbulent dispersion model constant is 25, this value is being adopted for the model.

2.3.3 Vertical pipe section

Figure 12 shows the plot of particle concentration in kg/m^3 at various sections of the vertical pipe section after the bend, as labelled. Dashed lines represent published experimental data and solid lines are the numerical simulation results. The particle concentration is derived using the value of particle volume fraction and the material density of the glass beads ($= volume\ fraction \times 2500kg/m^3$). At $y = 0.1, 1.1$ and $1.9m$, numerical simulations have reasonable representation of the experimental case. The concern lies with $y = 0.6m$ where the match is rather incongruent. It is believed that the discrepancy might be due to a combination of factors. One of the postulated contributing factor is that the $y = 0.6m$ vicinity is where the concentrated particle rope (a region within the pipe cross sectional area that more particles are concentrated) starts to disperse, thereby being a transitional zone that is inherently difficult to capture accurately. The peak distribution represents the particle rope and when the particles enter a pipe bend, inertial effects will tend to cluster the particles into a region of higher particle concentration. This area is usually located at the outer wall of the pipe bend. Furthermore, the curved pipe bend also generates a secondary flow in the form of a pair of counter-rotating vortex pair that should aid in the dispersion of particle ropes. Together with the turbulent dispersion force, the secondary flow is expected to transport the particle rope towards the middle of the pipe and disperse it thereafter. Such a mechanism is postulated to be the cause behind the offset in the peak particle distribution from the walls (i.e. at $y=0.1m$). Also, while the particle concentration plots differ, especially at $y=0.6m$, the particle mass flux is consistent across other locations.

Figure 12 Comparison of CFD results with experimental data of particle concentration along various vertical sections after the pipe bend as indicated.

The computational result is seen to have already dispersed a fraction of the particle rope at $y = 0.6m$ as compared to the experimental data. Such occurrence can appear when the cohesive nature of the particles is not taken into consideration. In numerical simulations, there are no cohesive forces between the particles, which allows them to disperse much easier and faster. As compared to the experimental case where cohesive forces between particles tend to agglomerate them into bigger lumps, thereby having more momentum to resist dispersion. This is also evident from the $y = 1.9m$ plot, where the particles are more evenly distributed across the pipe cross-section as compared to experimental results. Such could also be due to the possibility of an unsuitable

closure relation being used and it cannot be totally ruled out. This includes the suitability of the turbulence model, turbulence dispersion or wall friction, amongst others. However, it is not definitively identified that these are the causes of the simulation accuracy in this work and further studies are needed to justify these issues.

Figure 13 is the plot of the velocity of air and particles arising from the present numerical simulations, together with experimental data for particle velocity where they are compared at locations where the experimental data were taken. Note that an inconsistency can be discerned in the plot at $y = 0.1m$. Recall that the original experiments made use of particles that vary in particle diameter and comprise a considerable amount of very small particles ($<10\mu m$). Such particles, will follow very closely to the air flow which results in particles being detected across the pipe cross-section (with non-zero air velocities), albeit at a very small concentration that might not be reflected in the particle concentration plot. In the current numerical simulations however, a constant particle diameter of $42\mu m$ is used, in which resulted in areas with no particles (as compared to very minute particles in the experiments). Hence, the numerical model predicted zero particle velocity in areas where there are no particles present.

As with the particle concentration, $y = 0.6m$ for particle velocity presents a little deviation from the experimental data and the reasons could be attributed to disregarding the effects of cohesiveness as detailed above. $y = 1.1$ and $1.9m$ plots present a reasonable match. For $y = 0.6, 1.1$ and $1.9m$, the air and particles' velocity differs only a little from each other. This is because the Stokes number,

Figure 13 Comparison of CFD results with experimental data of particle velocity along various vertical sections after the pipe bend as indicated.

a measure of the responsiveness of the particle to the changes in its carrier gas, of the $42\mu\text{m}$ particle in this flow is small. Hence, the particle follows the air closely albeit with a small slip velocity present [65].

For completeness, the RMS velocity is plotted in Figure 14. In the original experimental study, RMS velocity fluctuation of particles in the axial direction was measured. However, in the numerical simulation, only the turbulent kinetic energy (k) values of the air-particle mixture is available. The RMS velocity fluctuations is therefore obtained by $u' = \sqrt{2k/3}$. This expression assumes the velocity fluctuations to be isotropic and the resulting graphs shows a lower

Figure 14 Comparison of CFD results with experimental data of RMS particle fluctuation along various vertical sections after the pipe bend as indicated.

absolute value than that measured in the experiments. This suggests that the velocity fluctuations are anisotropic with dominance in the stream wise direction. While the model under-predicts the velocity fluctuations, the agreement is still acceptable in terms of the trends. As the numerical model did not take into account different particle sizes, the particle size distribution plots are not discussed.

2.3.4 Horizontal pipe section

Figure 15 top row plots the particle concentration distribution in the horizontal pipe section at various locations as indicated by z , where the z locations are

measured from the pipe inlet. For $z = 2.5$ and $4.5m$, the plots are superimposed on the same graph. The disagreement between simulation results and experimental data is substantial. In the experiments, the bulk of the particles are seen to concentrate at the bottom of the pipe. However, it seems that the particles are fairly evenly distributed in the simulation results and does not move towards the bottom of the pipe as seen in the experiments. This is the same issue that some previous studies had reported. For instance, Hidayat and Rasmuson [21] had a similar issue where their numerical results did not match the experimental data in a horizontal pipe section. They had attributed this to the misinterpretation, by the original authors, of the experimental data which they compared against, using the concept of particle terminal velocity to support their claim. They utilized the terminal velocity of a particle in free fall to calculate the time it would take for the particle to travel from the top to the bottom of the pipe. Subsequently, they used the absolute particle axial velocity to calculate how far would the particle travel downstream in this period of time before reaching the bottom of the pipe. They concluded that the particle should reach the bottom of the pipe twenty times further downstream as compared to the experiments, which represents a significant discrepancy.

However, it should be noted that they had neglected the downward-acting velocity induced by the secondary flows of the carrier gas, which will hasten the movement of the particles towards the bottom of the pipe further. Including this velocity component in the analysis will provide a more accurate assessment of the flow scenario. Another plausible contributing factor could be due to the exclusion of the particle diameter variations during the numerical simulations and

results presented in Section 2.3.5 appear to support this claim. While the model prediction differs from the experimental data, it can be seen that the particle concentration distributions do not differ significantly between these two locations ($z = 2.5$ and $4.5m$) in both experimental and numerical results. The simulation results display the same consistency as compared to experimental results, even though good accuracy has not been achieved in the current simulation.

For the particle velocity plots shown in Figure 15 middle row, only the $z = 0.5m$ location display a discrepancy to experimental data which suggests a possibly undeveloped velocity profile for the simulation. This is because in the experiment, $z = 0m$ is the last honeycomb structure which were used to ensure even distribution of particles. These honeycomb structures might not have produced its intended function perfectly, contributing to the mismatch in numerical results and experimental data at both the particle concentration distribution and particle velocity plots.

The RMS velocity fluctuations are shown in Figure 15 bottom row and are derived from the same expression used in the vertical pipe section. Like the particle velocity plots, numerical results at $z = 2.5$ and $4.5m$ shows a decent match to experimental data. Again, $z = 0.5m$ does not match well for the same reason stated earlier. The honeycomb structures are believed to have induced additional turbulence in the flow, hence, the higher velocity fluctuations measured in the experimental data.

Figure 15 Comparison of CFD results with experimental data of particle concentration (top row), air velocity (middle row) and RMS velocity fluctuation (bottom row) along various horizontal sections before the pipe bend as indicated.

2.3.5 Effects of particle diameter

Considering the sizable difference between numerical results and experimental data for the particle concentration present within the horizontal pipe section, it is essential that the possibility of an unsuitable model reasoning be ruled out. Otherwise, the integrity of this study may be compromised. A potential cause is the numerical model not taking into consideration the particle diameter. One,

Figure 16 Particle concentration distribution within the horizontal pipe section when particle diameter increased to $100\mu\text{m}$.

particle diameter variations will be present in practical applications of material transported by pneumatic conveying. Such variations are naturally accounted for in the experiments but not in the current numerical work. Two, the multitude of other factors will contribute to variations in the actual particle diameter present within the pipeline. Such factors include the effects of humidity, cohesiveness of the material or electrostatic forces that may contribute to agglomeration of the particles, resulting in effectively larger particle diameters. For that reason, the simulation was performed again with an arbitrary particle diameter of $100\mu\text{m}$ (as compared to the initial $42\mu\text{m}$) to simulate the presence of larger diameter particles within the conveying process. Figure 16 graphs the results of assuming larger particle diameter in the horizontal pipe section. Here, it shows that inertial and gravitation forces come into greater effect with increasing particle diameter resulting in the concentration of particles increasing near the bottom of the pipe. From this, it can be seen that once the variation of particle diameter is taken into consideration, the precision of the numerical model increases dramatically even

though the approach taken here is just a simple increment in the particle diameter. The better results seen in the use of a larger particle size means that the effects of particle agglomeration are being captured by the simulation and strongly suggests that the particles used in the experiments have a tendency to stick to each other while being transported. To improve prediction of the flow characteristics, it is possible to define multiple ‘Eulerian’ phases for the solids phase using different particle diameters, based on the actual distribution, for each ‘Eulerian’ phase defined.

However, care has to be taken when utilizing this method as each additional phase introduces its individual set of conservation equations and might prove to be computationally expensive in applications of large domain sizes. Also, defining more phases for different particle diameters still does not account for agglomeration effects. To integrate the effect of particle diameter variations due to other factors (humidity, cohesiveness of the material or electrostatic forces) within a single numerical model, it will require relations that deal with coalescence and break-up of the particles. These are very specific situations that may even touch upon the chemistry of the individual particles itself and is, therefore, out of the scope of the current study. Nevertheless, these problems are being tackled by researchers. Korevaar et al. [66] and Pei et al. [67] for example, used the Lagrangian scheme with appropriate modelling to investigate contact electrification effects of particles. Experimental investigations on electrostatic effects were also performed by Yao et al. [68]. Humidity and effects from particle properties were also covered by Karde and Ghoroi [69] and Shah et al.

[70] respectively. These examples serve as a starting point for future works proposing to incorporate such effects into the Eulerian granular model.

The numerical model validation is done by comparing against an experimental study that looked at spherical particles. However, actual conveying processes involve materials of non-spherical nature. Capturing non-spherical particle's effects in numerical simulation is an entire subject matter in itself. While it has been concluded in a few studies that the particle shape does influence flow properties, the accuracy of capturing such effects still needs to be improved in the future [71]. However, the general conveying characteristics are largely similar. For example, the particle distribution for materials of different shapes is similar compared to spherical particles at the bend inlet, within the bend and at the bend outlet. Also, the particle velocity deviates only slightly between different particle shapes, with significant difference seen only in particles that are of very distinctive shapes (cubes and pyramids) [48]. Lastly, there is also a concern for the numerical model to capture the effects of particle roughness. Such effects will come into the numerical model at the collisional dissipation of energy term γ_{Θ_s} in equation 15 [31], where it is affected by the value of coefficient of restitution e_{ss} . While the coefficient of restitution commonly describes the elasticity of the inter-particle collisions, it can be modified to include the rotational collision effects between particles. However, for slightly frictional spheres, the modification is not significant [72]. Therefore, the value of 0.9 established previously is sufficient to capture inter-particle collision effects.

Chapter 3 Pipeline bulk conveying characteristics

3.1 Numerical setup

The geometry used in this numerical study is modelled directly after one of the pneumatic pipelines at JPPL cement terminal. There are two pipelines that discharge into a single silo and they share the same dust collector system. As both pipelines are similar and work independently of each other, only one of the pipeline will be studied. A perspective projection of the single pipeline that is studied here is shown earlier in Figure 2. All straight pipe sections are orthogonal to each other and the flow direction is as indicated by the arrows. Due to physical and operational constraints, it is not possible to obtain actual flow data along various sections of the pipeline as it cannot be modified or have measuring probes inserted. Therefore, a breakup of the computational domain into smaller sections is not possible, since the boundary conditions of the broken-up pipe sections could not be defined. The cast iron pipeline consists of five 90° bends of bend radius 2.5m. Its inner diameter is 0.3556m (i.e. 14 inches) everywhere and the total pipe length (as measured along its axis) is 156.185m. Details on the various pipe section dimensions are provided in Figure 17 – note that the pipe diameter is not drawn to scale. The air mover back pressure at the inlet side was measured as 1.8bars. Maximum rated capacity of the dust collector system downstream of the pipe is 24,000m³/h. The dust collector is running at maximum capacity and is being shared between two pipelines. Therefore, when both pipelines are assumed to be conveying at the same rate, this means that the maximum airflow that could be achieved in one of the pipelines can be reasonably assumed to be approximately 12,000m³/h. Currently, only operational bulk flow data could be obtained by averaging throughout the entire conveying operation. When both

Figure 17 Pneumatic conveying pipeline dimensions

pipes are working together, the system is able to convey cement at a rate of approximately $800\text{tons}/\text{hour}(tph)$ or $400tph$ for each pipeline. However, it has to be highlighted that these values are inclusive of the start-up/power-down phases and down times in between. The ideal flowrate is estimated to be about $500tph$ and the cement is discharged into a silo at the end of the conveying system that is at atmospheric pressure.

The same blocking strategy is applied to the full-scale industrial pipeline to obtain a hexahedral mesh. Due to the large domain size, it is very computationally expensive even during the mesh generation stage. The earlier practice of limiting the cell size to roughly $0.002 \times 0.002 \times 0.004m$ (or just $0.002m$ for simplicity) is not practical. Instead, the cells are now set at about $0.004 \times 0.004 \times 0.008m$ ($0.004m$), the same size used as the “coarser” mesh. Earlier, the mesh dependency test was conducted with three different meshes of

Figure 18 Meshing for the industrial scale pneumatic conveying pipeline

size 0.001, 0.002 and 0.004m (finer, fine and coarser) with the same numerical model. It is found that the 0.001m and 0.002m mesh does not differ much while the 0.004m mesh only differs a little from the 0.002m mesh (Figure 9). With that, the coarser mesh is applied in this study for its massive computational resource advantage and the cell volumes ranges between 4.24×10^{-8} to $6.73 \times 10^{-7}m^3$. Inaccuracy is projected to be acceptable based on the mesh dependency test. Figure 18 shows the hexahedral mesh generated. The mesh required about 400 CPU hours for each steady-state simulation. Also, coupled with the applied time-step size, the mesh required about 1,500 CPU hours for each second of transient flow time simulated. The transient test case is simulated for 35 seconds and requires over 50,000 CPU hours.

To begin with, the first set of simulations is performed under steady-state conditions to study the conveying process at a macro level. The pressure and flowrates of each phase (i.e. air and cement flows) are compared against the bulk

flow data to ensure that the model is able to simulate the flow problem to a reasonable degree of accuracy. The inlet is a pressure inlet set at the air mover back pressure of 1.8bars and the outlet is a pressure outlet to atmospheric pressure (i.e. 0Pa gauge). Material properties of ordinary Portland cement are obtained from [73]. The material density is defined in the simulation as 3200kg/m^3 and the particle diameter as $21\mu\text{m}$. Following the numerical model that was validated earlier, different treatments are applied for both of the phases at the wall. A no-slip condition is applied for the gas phase and a specular coefficient of 0.6 is applied for the solid phase. Next, the conveying characteristics of the pneumatic conveying system under different conveying conditions are investigated. Conveying pressure is reduced from 1.8bars to 1.5bars , in steps of 0.1bar , to look into the impact on cement conveying in situations where the air pressure provided by the unloading system may not be as high. For each different conveying pressure, the inlet solids volume fraction is also varied. This effectively varied the solids loading ratio (kg cement/kg air) within the range of 20 to 110. All other parameters remain constant. Lastly, a transient simulation is performed to gain insights to the cement flow characteristics within the pipe. The flow conditions are identical to the first steady-state case. The time step size is set to $1 \times 10^{-3}\text{s}$. To save computational time and improve convergence behaviour, a numerical solution from a single-phase gas flow is applied to the whole domain as an initial condition prior to all the simulations. For the steady state simulations, an arbitrary solids volume fraction is applied to the whole domain as an additional initial condition.

Figure 19 Pressure drop along the pipeline for different conveying pressures

3.2 Effects of pipeline inlet pressure

Figure 19 plots the pressure drop against the pipe physical distance, where pressure values along the various location of the pipe axis are extracted from the steady-state simulation solutions. This set of data is obtained from four different inlet pressures with all the same inlet cement volume fraction of 0.1. This equates to an SLR of about 45.12. The four lines represent different pressures applied at the inlet boundary condition, from 1.5 to 1.8bars. The inlet of the pipe is labelled 0m and the outlet is 156.2m, with the greyed columns indicating the 90° bends locations. The bends are numbered one to five for easy reference. Results show that there is a distinctly bigger pressure drop along each of the pipe bends as compared to straight pipe sections and that the orientation of the pipe bend also affects the extent of the pressure drop. In the current setup, only bend 3 consists of a HH bend and the pressure drop along this pipe bend is smaller. The other four pipe bends are either a HV or VH bend. This behaviour is similar

to the conclusion drawn by Ibrahim et al. [24] previously. However, the current simulations also further revealed that there is practically no difference between HV and VH pipe bends with regards to the pressure drop.

Table 2 presents the pipe bend pressure drop data in numerical form. The pressure drop is also more pronounced when it is a vertical pipe section (between bends 1 and 2, and between bends 4 and 5) due to the additional driving force needed to overcome gravitational effects acting on the material being conveyed.

Theoretically, the magnitude of the pressure drop across the bends can be calculated. Chambers and Marcus [74] presented empirical formulations to calculate pressure drops along a horizontal pipe, vertical pipe or pipe bends, although the basis for its derivation was not presented. Even so, other researchers [75-77] had used the model to success. Later models, such as that proposed by Pan [44] and Pan and Wypych [78], provided derivations and experiments were used to determine the exponents. Calculation results from these models are found to have minimal differences between each other. Figure 20 shows the comparison

Table 2 Numerical details of pressure drop per unit length along pipe bends

Conveying pressure	Bend 1 (HV)	Bend 2 (VH)	Bend 3 (HH)	Bend 4 (HV)	Bend 5 (VH)
1.8bars	2948Pa	2930Pa	2051Pa	2968Pa	2846Pa
1.7bars	2842Pa	2804Pa	1951Pa	2858Pa	2804Pa
1.6bars	2729Pa	2681Pa	1849Pa	2751Pa	2663Pa
1.5bars	2648Pa	2594Pa	1718Pa	2640Pa	2600Pa

Figure 20 Pressure drop comparison between calculated values and numerical simulation

between pressure drop calculated from the empirical formulations against the present numerical simulation results. Both empirical calculations and numerical simulation results match up closely, with the pipe bend and vertical pipe section pressure drops to be within 5% discrepancies. One possible reason with the horizontal pipe section having a larger but nevertheless reasonable $< 10\%$ discrepancy is that the empirical formulation is dependent on the pipe roughness, and pipe roughness effects are likely to be more considerable in horizontal pipe sections. In this case, note that a roughness height of $0.001m$ is used in the calculations. Furthermore, the pressure drop will have been dependent on the specular coefficient used in the numerical simulations that determines the shear forces associated with the solid phase at the wall. These preceding factors are believed to result in the above discrepancies.

3.3 Equivalent length for pipe bends

It will be useful for industrial applications if the pressure drops across the 90° pipe bends can be related to the pressure drop across a straight pipe section. For single-phase fluid flows, this has been well-established as the “equivalent pipe length” concept. However, this is not the case for gas-particle flows such as the ones encountered here. This value of pressure drop per unit length is convenient for designers to gain a sense of the type of conveying mechanism (i.e. dense or dilute flow) the pipeline is operating in [79]. It is also an important parameter to consider when designing pneumatic conveying pipelines [80, 81]. Figure 21 represents the ratio of pressure drop per metre of pipe bend against pressure drop per metre of straight pipe. This ratio is observed to be a little higher than three for conveying pressure of 1.5bar. This means that for every metre of pipe bend, it is equivalent to three metres of straight pipe in terms of pressure drop. The equivalent pipe length is seen to be dependent on the conveying pressure which is inadvertently caused by increased conveying velocities at higher conveying pressures.

With the current dataset, it reveals that the ratio varies linearly with the inlet conveying pressure, thereby allowing for the possibility of extrapolating pressure losses across pipe bends for different conveying pressures. However, note that the extent to which this linear relationship holds beyond the current conveying pressures is not investigated. At some point, this linear relationship will have to break down as the ratio should approach the value of 1 as the conveying pressure decreases. However, conveying cases at too low a conveying pressure are impractical in any case. The ratio would also be expected to be dependent on the

Figure 21 Pressure drop ratio of pipe bend to horizontal straight pipe for various conveying pressures.

material being conveyed. Nevertheless, the equivalent length results obtained from the current simulations is similar to results obtained from experiments conducted by Spedding and Benard [82] for gas-liquid flows, where their results also show a monotonically increasing ratio. However, it should be reminded again that the current conclusion of the pressure drop ratio being a little over 3 is only determined for the present pipeline and flow scenarios and the exact pressure drop ratio will vary depending on the actual situation. Nevertheless, the current pipe geometry is that of an actual pipeline in operation within the industry and the present results provide insights into their designs. Hence, these findings will be useful for individuals or organizations involved with full-scale pneumatic conveying pipelines, be it research or system design.

3.4 Conveying characteristics

The pipeline conveying characteristics are now investigated by varying the inlet material feed rate in addition to the different conveying pressures. Inlet solids volume fractions are varied from 0.05 to 0.2 and Figure 22 presents the air and cement flowrate at the outlet. As the maximum capacity of the dust collector at the outlet is $12,000\text{m}^3/\text{h}$, the horizontal dashed line at the left denotes this limit. With the numerical results presented in this way, it is possible to ascertain if the dust collector can handle the conveying operation at a certain conveying pressure and SLR. At the same pressure and SLR, it is possible to read off the maximum conveying capacity of the pipeline. Apart from applying this data directly to operational situations, the data here provides a glimpse of the conveying characteristics that might be typical to a system like this. It is seen that at all conveying pressures, the amount of air used decreases with increasing SLR (blue graphs). One immediate observation is that the relationship is not linear. Also, the amount of cement conveyed increases with increasing SLR (red graphs) and this relationship is also observed to be non-linear (though less pronounced than the air flowrate). Therefore, it will not be the most efficient way to simply increase the inlet SLR, where the power requirements for the conveying system increases significantly. This will also be physically limited by the cement properties, such as the maximum packing limit or the saltation velocity. The maximum packing limit is how much cement that can actually fit into the pipe cross-section. The saltation velocity is the minimum velocity of the conveying medium that is required to pick up the cement. Any velocities lower than the saltation velocity will result in the cement depositing at the bottom of the pipe.

Figure 22 Pipeline conveying characteristics for different solids loading ratio

While this set of data is not able to predict these limits, the general trends of the conveying characteristics can be inferred.

In all, the results of the steady-state simulations presented so far provide an optimistic view that the actual pneumatic conveying process is well simulated. Simulation results match with both the bulk flow data and theoretical calculations. Arguably, the results from steady-state simulations are extremely handy only in terms of practical operations. However, the primary importance highlighted in this work is that the steady-state simulations here manage to bridge the gap between theoretical research and industrial applications. It is believed that with decreasing computational cost and increasing robustness in numerical simulations, such a method of directly investigating an industrial scale system will become increasingly feasible.

Chapter 4 Flow characteristics of the industrial pipeline

4.1 Particle rope

Results of unsteady-state simulations presented here will illustrate the intricate details of the solids flow behaviour within the pipe. Figure 23 shows the iso-surface plot of 0.2 cement volume fraction at pipe bend number two which is a VH bend. The iso-surface plot can be approximated to be the particle rope within the pipe. The particle rope, which is a concentration of particles at certain regions of the pipe, is seen to navigate the pipe bend adhering to the outer part of the pipe bend. This is expected due to inertial forces of the cement particles. When exiting the pipe bend, the particle rope is seen to follow one side of the pipe wall (farther side, in this case) to the bottom of the pipe. This is in contrast to other studies which saw the particle rope settle down in the middle, through the axis of the pipe. The difference here is due to the pipe orientation prior to the bend. For the current geometry, note that an HV pipe bend precedes the VH pipe bend. Such an arrangement of pipe bends changes the position of the particle rope entering the pipe bend and also alters the secondary flows within the pipe (see Section 4.2). For a better understanding, Figure 24 shows the cross-sectional contour plots of cement volume fraction. The position of the particle rope will then be characterized by its angular position. Figure 26 illustrates the location of the different angles and how the particle rope angular position is measured.

In Figure 24a, $-4d$ to 0° shows the various plots for the vertical pipe section, where cross-sections of a distinct particle rope on the right side of the pipe can be observed. This particle rope was formed by the first bend. With the very short downstream section of the first bend, the particle rope does not disperse and

Figure 23 Iso-surface plot of 0.2 cement volume fraction at bend number two

enters the second bend (0°) with the structure of the particle rope being more or less preserved. In 0° to 75° , after the particle rope enters the bend, inertial forces and secondary flows start to alter its structure. After 30° , the particle rope is seen to start dispersing due to secondary flow structures. The particles also begin to move towards the pipe wall and shift their locations from the right side of the pipe towards the outer bend due to inertial forces. After the bend (Figure 24b) the concentration of particles did not make a complete revolution over to the left side of the pipe before falling down to the bottom of the pipe. Instead, it traces its path back on the right side of the pipe before falling to the bottom of the pipe. This is largely due to the structure of the secondary flows. By observing the sequence of cross-sectional plots downstream of the bend, the particles are seen to slosh around the bottom of the pipe before the sloshing damps down and the particles traverse to the bottom of the pipe. This sloshing motion is not evident when

Figure 24a Cement volume fraction plots upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

presented in a three-dimensional view (Figure 23). By simple measurements, the location of higher particle concentrations can be represented in a graphical format. The area of highest particle concentration is determined by the thickest location

Figure 24b Cement volume fraction plots downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

of the 0.2 volume fraction contour, as measured from the pipe wall. By assigning

Figure 26 Illustration of (a) bend positions and (b) rope angular position

Figure 25 Angular position of particle rope (solid line) and a fitted underdamped curve (dashed line).

the top of the pipe as 0°, the solid line plot in Figure 25 shows the location of the rope. The grey region refers to the region of pipe bend and the region to the right of the grey box being the horizontal pipe section up to 100 pipe diameters downstream.

The particle rope is seen to enter the pipe bend at an angle of about 90° formed due to the upstream pipe bend. Within the pipe bend, it shifts its position towards 0° , being the outside of the pipe bend. Then, after exiting the pipe bend, it falls rapidly to the bottom of the pipe and displays an oscillatory motion. This motion is the sloshing of the particles at the bottom of the pipe, seemingly highly reliant on gravitational effects. It is interesting to note that the angular position of higher concentration of particles can be fitted with an under-damped graph (dashed line in Figure 25) starting at $0D$ after the bend. The equation of this under-damped curve is provided in the figure. By applying theories from damped harmonic oscillations, system characteristics such as the angular frequency and damping ratio can be obtained. At this point, these characteristics are postulated to be linked to the viscous forces of the material conveyed.

4.2 Secondary flows

Secondary flow structures of a fluid inside a simple pipe bend are well documented in the literature. Figure 27(a) shows the typical secondary flow structure that is formed in a pipe bend. The dashed lines show the direction of flow of fluid in the plane that is perpendicular to the flow direction. As inertial forces take over when navigating a pipe bend, the fluid tends to move towards the outside of the pipe bend. It normally forms a symmetrical pair of counter-rotating vortices and sometimes two pairs of counter-rotating vortices in 180° bends [83]. A snapshot of secondary flow structure in the current flow case is shown in Figure 27(b). It is a velocity vector plot of velocities tangential to the displayed plane. While the general behaviour of fluid moving out towards the outside of the pipe bend is still present, the secondary flow structure is no longer

Figure 27 Secondary flow structure within the bend of a (a) single bend (Image source: [83]) and (b) combination of bends (current case). Inside of the bend is on the right.

symmetric. Also, there exists a weaker pair of counter-rotating vortices alongside a dominating pair. The secondary flow structures are indisputably different in both cases.

In Figure 28a ($-4d$ to 0°) where the secondary flow structures in the upstream section is shown, the symmetrical pair of counter-rotating vortices formed by the previous pipe bend is seen to slowly dissipate until it reaches the entrance of the next pipe bend ($0d$). They are, however, not fully dissipated and the velocity profile of the fluid does not approach that of a uniform turbulent flow profile. Hence, after it enters the pipe bend, the residual secondary flow from the previous pipe bend interferes with the flow producing non-symmetrical secondary flows. Figure 28a (0° to 75°) shows the secondary flows within the pipe bend, reveals that the pair of weaker counter-rotating vortices that is only vaguely present from 30° onwards. They are then seen to strengthen and take shape where it is

distinctly visible at the 60° location. Thereafter, it merges with the stronger pair to produce a disorganized secondary flow pattern. Figure 28b shows the secondary flow structures downstream of the pipe bend. After exiting the pipe bend, the secondary flow vortices start to merge into a single clockwise vortex. It is evident that this clockwise secondary flow is one of the possible reasons as to why the particle rope does not complete a revolution around the wall of the pipe (see Figure 23 and Figure 24b). Also, by comparing the secondary flow structures with the volume fraction contour plots, particularly for $10d - 30d$, the left-right sloshing of the material conveyed in the horizontal pipe section is accompanied by a change in the direction of the secondary flow vortex. In addition to that, it can be seen from $20d - 30d$ that the change in sloshing direction in the volume fraction contour plot actually lags the direction change of secondary flow in the tangential velocity vector plot. This suggests that, apart from gravitational forces, secondary flow patterns also play a part in determining the flow behaviour of the conveyed material. Beyond $50d$, secondary flow patterns are almost non-existent and the sloshing behaviour is entirely dominated by gravitational forces.

It should be highlighted that the transient flow results obtained through the present study expand on the limited knowledge associated with a combination of pipe bends. Apart from the work of Schallert and Levy [25], literature detailing detailed flow behaviour when more than one pipe bend is used remains limited. Hence, the present findings broaden the current understanding of cement flows within a pipeline with several different pipe bends.

Figure 28a Secondary flow structures upstream of the bend at various multiples of pipe diameter (d) and within the bend at various locations as indicated. Top side being the outside of the bend.

Figure 28b Secondary flow structures downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Chapter 5 Effects of pipe geometry on particle roping

5.1 Numerical setup

In this section, the effects of two parameters on the flow are observed, namely the pipe bend ratio (radius of pipe bend/pipe radius) and the length of the connecting pipe between both bends. As seen earlier, when the pneumatic conveying system has multiple bends that are connected by a rather short section of pipe, the flow characteristics at the outlet of the second pipe bend is much different from if it were going through just one pipe bend. The focus here will be the outlet of the second pipe bend which connects to a long straight horizontal pipe section in the JPPL pipeline. Therefore, the JPPL geometry is truncated at the long horizontal straight pipe section to save up much computing resources needed for many iterations of different pipeline geometry. Figure 29 shows the portion of the JPPL pipeline which is used. Pipe bend portions are demarcated by faint lines for clearer dimensioning. After the second bend, the straight section measures 200 times of the pipe diameter. Before, the entire pipeline could not be broken down into smaller sections as there are no intermediate flow data along the pipeline to use as boundary conditions. The total pressure drop of this pipeline can now be determined from Figure 19 (page 56) and hence, applied as boundary conditions here. The inlet pressure is now set at $0.8bar$, interpolated from Figure 19. The bend ratio (BR) will be varied firstly, from 2.5 times the multiple of pipe radius to 25 times (original BR is 14 times pipe radius). Both bends will have the same BR in each case. Next, withholding the original JPPL bend radius of $2.5m$, the connecting pipe length is varied from 0 times the multiple of pipe diameter to 20 times (original connecting pipe length is 4.5 times pipe diameter).

Figure 29 Pipeline dimension for geometry investigations

The mesh generated is similar to Figure 18 (page 54) , however, the blocking at the pipe bends are adaptive to the pipe geometry. Instead of dividing the pipe bend into four sections (Figure 6, page 31), the number of sections depends on the BR. With a higher BR, the pipes are physically longer and hence may require more blocks to ensure the mesh cells are not “compressed” at the inner side of the bend. For BR 14 and connecting length $4.5d$, the mesh volume ranges from 1.87×10^{-8} to $5.70 \times 10^{-7} m^3$. Transient simulation is performed on each case for 25s of flow time. Each case requires about 20,000 CPU hours, and the total computing resource needed for this portion of the study is over 200,000 CPU hours. With the amount of computing resources required, it is not possible to apply design of experiment (DOE) techniques to investigate the combined effects of both the BR and connecting length (CL, presented in a later section).

5.2 Bend radius

Figure 30 Figure 35 first shows the volume fraction contour plots extracted from each of the test cases. Upstream of the bend, all six test cases show an undispersed particle rope at the right side of the pipe wall. The particle rope is due to the preceding HV bend and it stays undispersed because the vertical pipe connecting the two bends is not long enough for dispersion to occur. Upon entering the bend, the particle rope can display different behaviour, influenced by its BR. These particle rope behaviours are better described by the angular position later in Figure 36. However, the similarity across all the BR test cases is that the particle rope is always near the pipe walls. This is due to the dominance of a single rotating vortex in the secondary flow. After exiting the bend, the particle rope sloshes around the bottom of the pipe in a motion that was described earlier in Section 4.1.

As the particle rope is consistently seen to be at the pipe walls, the location of the particle rope within the pipe can be appropriately quantified in the same way as in Figure 25 (page 67) by measuring its angular position on the pipe wall. Figure 36 shows the particle rope's angular position for all of the simulated BR of 2.5, 5, 10, 14, 20 and 25. For different BR, the actual length of the pipe bend is different. Therefore, the data points at the pipe bend are plotted against the pipe bend position for easy comparison. Figure 26 shows an illustration of the pipe bend locations. As a recap, the outside of the pipe bend is assigned 0° angular position and the inside of the bend as 180° . The shaded portion is the pipe bend. The straight horizontal pipe section after the pipe bend is identified by multiples

Figure 30a Cement volume fraction plots for BR 2.5, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 30b Cement volume fraction plots for BR 2.5, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 31a Cement volume fraction plots for BR 5, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 31b Cement volume fraction plots for BR 5, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 32a Cement volume fraction plots for BR 10, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 32b Cement volume fraction plots for BR 10, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 33a Cement volume fraction plots for BR 14, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 33b Cement volume fraction plots for BR 14, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 34a Cement volume fraction plots for BR 20, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 34b Cement volume fraction plots for BR 20, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 35a Cement volume fraction plots for BR 25, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 35b Cement volume fraction plots for BR 25, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 36 Particle rope angular position for various bend ratio

of pipe diameter. Granted, such a presentation method as in Figure 36 does not allow for much useful analysis. However, it is left here should the reader want to make a comparison between different cases on the same graph. The angular position data is much more useful when studied individually. Figure 37 shows the plots for each individual test cases.

The flow characteristics can be broadly classified into three scenarios. For BR 2.5, the particle rope loops around the perimeter of the pipe and then after exiting the pipe bend follows through and completes the loop towards the left side of the pipe. BR 5 and 10 have the particle rope splitting into two distinct ropes while in the bend. They then take different paths, along the left and right side of the pipe wall, before merging back at the bottom of the pipe. BR 14 – 25 shows the

Figure 37 Particle rope angular position for individual bend ratio test cases

particle rope enters the bend and disperse within the pipe bend itself. In BR 14 and 20, the particle rope is not fully dispersed and the bulk of it falls down towards the bottom of the pipe the moment it enters the pipe bend. This is due to a combination of a gradual bend, gravitational and secondary flow effects. The rope is slow to disperse, however, after dispersing, inertial and secondary flow effects propel the particles to start forming another rope at the top of the pipe bend. This second rope then follows the right side of the pipe wall down. When the bend is made even more gradual, BR 25 has sections of the pipe where there is no distinct particle rope present. The dispersion is much more rapid than in BR

Figure 38 Head loss of 90° radiused bends. (Image source: [1])

20. In all cases except BR 25, the particle rope, after falling to the bottom of the pipeline, displays the sloshing motion before damping down.

This three distinctly different flow types can be related to the pipe bend pressure loss. For theoretical calculations of the pressure loss across pipe bends, a head loss coefficient k is usually used. This coefficient is determined by the pipe bend ratio. Figure 38 shows a graph that can be used to select the value of k to be used in theoretical calculations. Relating it to the current data, one can see that BR 2.5 (and lower) corresponds to a high k value, or higher pressure loss. In such situation, the momentum of the particles and the strength of the secondary flows propels the particle rope to swirl vigorously within the pipe. By extrapolating the trend of the particle rope's motion, it would not be entirely impossible for the particle rope to have enough momentum to make multiple revolutions along the pipe wall. At BR 5 – 10, it corresponds to a region towards the left of the minimum point on the head loss coefficient graph. In this range, the entire particle rope does not gain enough momentum to make the complete revolution. Instead,

it splits up and only a portion of the rope makes a complete revolution. The location where the particle rope splits into two is dependent on the BR too, with the particle rope splitting earlier into the bend at higher BRs. Also, the smooth flow of the two particle ropes would be disturbed when they collide at the bottom of the pipe. This collision of the two ropes will possibly result in some additional pressure loss in getting the unified particle rope to flow smoothly again. In addition, more particle collisions will also result in higher momentum loss. Increasing the BR further, eventually, the entire particle rope does not gain enough momentum from the secondary flow and no splitting occurs. Concurrently, it is observed that the particle starts to disperse after entering the bend. At the region where $BR > 14$, the pipe bend is so gradual that it contributes to the dispersion of the particle rope. In theory, it is most efficient in terms of pressure loss for the conveyed material to be well dispersed in the conveying medium. However, when the pipe bend is made too gradual, one has to keep in mind that the longer pipe bend sustains the secondary flows for a longer distance. This would be disadvantageous for the pressure loss and the graph of k value starts to rise slowly for BR 20 and 25.

To the best of the author's knowledge, the pipe bend head loss has never been directly correlated to the motion of the conveyed solids in this manner before. Probably due to the fact that much of the research done was on a single bend. In this case, the combination of pipe bends resulted in non-symmetrical secondary flows that drive the motion of the particles. As a result, particle ropes swirl around the pipe walls in a three-dimensional motion that can be studied qualitatively.

Table 3 provides some numerical data for the different pipe bend ratio test cases. The inlet and outlet pressure and outlet air velocity values are obtained at the centreline of the pipeline as they do not vary much across the cross-section of the pipe. The pressure loss per unit length across a pipe bend is a function of the pipe diameter d , density ρ and square of the velocity v^2 . By taking into account the difference in pipe lengths due to different bend ratio and the velocity, the pressure drop across the pipe bend is divided by the its length and square of the air velocity. A value, being a function of the product of the pipe diameter and the density, is obtained. While the pipe diameter is constant across all test cases, the density is difficult to establish because the particles are not evenly dispersed in the pipe cross-section. Hence, the effective density (equation 6) is difficult to determine. However, the numerical simulations have reached a steady mass flowrate at the outlet and the density value should not affect analysis when values are compared against each other. Nevertheless, an analysis of the trend can be done. Figure 39 plots the pressure loss, after accounting for the bend length and air velocity, against the bend ratios. Keeping in mind that the results are from transient simulations, a curve is fitted for the data. Immediately recognisable, the graph mirrors the graph used for determination of the bend head loss coefficient of a

Table 3 Numerical data for bend ratio test cases

BR	Pipe bend inlet pressure /Pa	Pipe bend outlet pressure /Pa	Air velocity /ms ⁻¹	$\frac{\Delta P}{v^2 L} / \text{kgm}^{-4}$
2.5	42904	31149	22.9	32.1
5	43576	35306	17.6	19.1
10	43753	32137	19.0	11.5
14	44518	26898	18.1	13.7
20	41887	17846	18.6	12.4
25	38179	14753	14.9	15.1

Figure 39 Pressure loss across pipe bend for different bend ratios after accounting for pipe bend length and air velocity.

single-phase flow which is a well-established theory. As the particle diameter of cement used in this study is $14\mu\text{m}$, the particles will follow the conveying medium very closely. Hence, it is expected that the optimal bend ratio (about 10 – 15 from Figure 39) would be very similar to a single-phase flow (Figure 38). The small particle diameter does not affect the gas flow. In flows with large particulates that affect the gas flow significantly, the same method discussed here could be employed to determine the optimal pipe bend ratio.

For the sake of consistency, it is mentioned here that the current numerical simulation of a truncated pipeline with the same bend radius as the JPPL pipeline differs a little from the previous data presented in section 4.1. Figure 40 illustrates this difference. The primary difference is between its amplitudes where the current truncated pipe shows a less rigorous sloshing motion. This is suggested

Figure 40 Particle rope angular position, comparing between the truncated pipeline and full-scale pipeline

to be due to inlet pressure applied at the inlet. While the current inlet pressure is directly obtained from the steady-state simulation, it is actually too low. On hindsight, the pressure loss effects of the pipe outlet were entirely disregarded while selecting the inlet pressure to be used. Due to this, the effective conveying pressure is reduced and the material moves at lower velocities. This results in less momentum for the sloshing motion and the lower amplitude. It also contributes to the lower frequency of the sloshing motion in the straight horizontal pipe section. Lastly, the particle rope in the current truncated pipeline dispersed in the bend instead of the continuous particle rope seen in the full pipeline. This is also due to less momentum of the particles, allowing dispersion effects to take over.

Figure 41 to Figure 46 shows the vector plots of the different test cases. Looking at the vertical pipe section connecting the two bends ($-4d$ to $-1d$), BR 2.5 (Figure 41a) clearly shows a pair of symmetrical counter rotating vortices which

weakens downstream (towards $-1d$). Since the BR is small, the secondary flows induced are stronger and its effects can stay longer downstream of the bend. This is evident in the different BR test cases where the strength of the secondary flow in the vertical pipe section weakens when the BR is increased. In the pipe bend, (0° to 75°), strong secondary flows are seen for the tightest bend and its strength decreases if the BR is increased. The strength of the secondary flow structure for BR 2.5 is dominated by a single vortex which keeps the particle ropes close to the pipe walls. The strength of the secondary flow starts to reduce immediately after exiting the pipe bend. Further downstream as the secondary flow structure weakens, there is less mixing of the particles with the carrier gas. As seen earlier in the volume fraction contour plots, particles will start to fall towards the lower part of the pipeline.

Figure 41a Secondary flow vector plots for BR 2.5, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 41b Secondary flow vector plots for BR 2.5, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 42a Secondary flow vector plots for BR 5, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 42b Secondary flow vector plots for BR 5, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 43a Secondary flow vector plots for BR 10, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 43b Secondary flow vector plots for BR 10, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 44a Secondary flow vector plots for BR 14, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 44b Secondary flow vector plots for BR 14, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 45a Secondary flow vector plots for BR 20, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 45b Secondary flow vector plots for BR 20, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 46a Secondary flow vector plots for BR 25, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 46b Secondary flow vector plots for BR 25, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

5.3 Bend connecting length

The effects of the connecting length (CL) of the vertical pipe between the HV and VH pipe bend is discussed here. Figure 47 shows the particle rope's angular position for varying CLs which were obtained by the same method as in Figure 25. The position of the particle rope is measured, in degrees, relative to the outside of the bend and plotted against the position within the pipeline. It is immediately obvious that the plot is not as overwhelming as in the one for pipe bend ratio (Figure 36). The reason is twofold. Firstly, it gives a broad picture of what extent does the CL affect the particle rope flow characteristics and, in this case, the effects of varying CLs are not as strong as varying BRs. Secondly, there are more sections (in CL $15d$ and $20d$) that do not have a distinct particle rope and are seen as the discontinuities on the graph. This is caused by the longer connecting pipe which facilitates the dispersion of the particle rope induced by the previous bend.

Figure 48 shows the plots individually. For CL $0d$, i.e. outlet of the first pipe bend joins the inlet of the second pipe bend, the splitting of the particle rope into two can be seen. Having roughly the same flow characteristic as BR 10, it could suggest that this configuration would be one that is acceptable for use in an actual conveying system. For CLs 4.5 and 10, there is little difference. The particle rope starts to disperse upon entering the pipe bend and then forms a new particle rope which swirls down through the right side of the pipe. Based on the bend ratio test cases, this flow characteristic is also a desirable one as the pressure loss is minimal. When $CL > 15$, the dispersion of the particle rope becomes apparent. In the BR test cases, it is seen that dispersion of the rope within the bend is

Figure 47 Particle rope angular position for various connecting length

somewhat favourable. Therefore, based on the flow characteristics seen, the CL does not matter in the efficiency of the pneumatic conveying process. Figure 49 to Figure 53 shows the volume fraction contour plots for each test case.

Table 4 provides some numerical data for the different CL test cases, similar to that presented for the BR cases. Likewise, Figure 54 shows the pressure drop across the pipe bends after accounting for the pipe bend length and the air velocity. Based on the data, it seems like a linear relationship is possible and a line is fitted to the data. When the CL between an HV and VH bend is minimal, the pressure loss across the VH bend is also minimal. In the BR test cases, the reason provided for higher pressure losses in higher BRs is due to the secondary flow being sustained for longer distances by the pipe bend. However, in the CL test cases, the BR is held constant. The higher pressure losses seen now is due to the dispersion of the particle rope in the vertical pipe section.

Figure 48 Particle rope angular position for individual connecting length test cases

Figure 56 picks out the vector plots at one pipe diameter before the pipe bend for different CLs, which shows the secondary flow structure. For short connecting lengths (CL 4.5 and 10), there is an absence of any strong secondary flows before the pipe bend. In $CL > 15$, the vector plots are significantly more perturbed. This convoluted flow pattern right before the pipe bend extends into the pipe bend, increasing the complexity of the flow in the pipe bend. This causes the trend of increase in pressure loss across the pipe bend in Figure 54. As the vertical pipe section is lengthened, the particle rope disperses within the vertical pipe section. It is this dispersion mechanism that introduces the unfavourable flow patterns. Figure 57 to Figure 61 shows the complete set of secondary flow

Figure 49a Cement volume fraction plots for CL 0 at various locations within the bend as indicated. Top being the outside of the bend.

vector plots. Figure 55 shows the particle rope dispersion mechanism for CL 20 test case. The snapshots of volume fraction plot are 0.1s apart, from 12.1 to 18.0s. The particles within the particle rope, closest to the pipe wall, loses enough momentum in the vertical pipe section that it starts to flow backwards. When this happens, a small region of circulating flow within the particle ropes causes the rope to break apart (from 13.6 – 14.1s and 15.8 – 16.2s). Although the particle distribution after the particle rope breaks is fairly even, the rope breaking mechanism is violent, causing a very disturbed and turbulent flow within that portion of the pipe. This increased turbulence extending into the pipe bend causes more pressure loss across the pipe bend.

Figure 49b Cement volume fraction plots for CL 0, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 50a Cement volume fraction plots for CL 4.5, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 50b Cement volume fraction plots for CL 4.5, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 51a Cement volume fraction plots for CL 10, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 51b Cement volume fraction plots for CL 10, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 52a Cement volume fraction plots for CL 15, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 52b Cement volume fraction plots for CL 15, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 53a Cement volume fraction plots for CL 20, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 53b Cement volume fraction plots for CL 20, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Table 4 Numerical data for connecting length test cases

CL	Pipe bend inlet pressure /Pa	Pipe bend outlet pressure /Pa	Air velocity / ms^{-1}	Cement velocity / ms^{-1}	$\frac{\Delta P}{v^2 L}$ / kgm^{-4}
0d	46630	33744	18.2	18.1	9.9
4.5d	44518	26898	18.1	18.0	13.7
10d	34557	20865	14.1	14.0	17.5
15d	25907	13304	13.4	13.4	17.9
20d	26668	13625	12.0	12.0	20.3

Figure 54 Pressure loss across pipe bend for different vertical pipe connecting length after accounting for pipe bend length and air velocity.

Figure 55 Particle rope dispersion in the vertical pipe section for CL 20 test case

Figure 56 Secondary flow vector plot at one diameter before the pipe bend entrance for different CL as indicated

Figure 57a Secondary flow vector plots for CL 0 at various locations within the bend as indicated. Top being the outside of the bend.

Figure 57b Secondary flow vector plots for CL 0, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 58a Secondary flow vector plots for CL 4.5, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 58b Secondary flow vector plots for CL 4.5, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 59a Secondary flow vector plots for CL 10, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 59b Secondary flow vector plots for CL 10, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 60a Secondary flow vector plots for CL 15, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 60b Secondary flow vector plots for CL 15, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Figure 61a Secondary flow vector plots for CL 20, upstream of the bend at various multiples of pipe diameter (d) and at various locations within the bend as indicated. Top being the outside of the bend.

Figure 61b Secondary flow vector plots for CL 20, downstream of the bend at various multiples of pipe diameter (d) as indicated. Top side being the outside of the preceding bend.

Chapter 6 Conclusion

The application of a multiphase numerical model with Eulerian treatment for the solids phase is presented in Chapter 2, where validation of the model is mainly based on previously published experimental results. Using the validated numerical model, an industrial scale pipeline is investigated in Chapter 3 where the conveying characteristics are matched with the current operation of the pipeline. Later on in Chapter 4, one section of the pipeline, consisting of a VH bend preceded by an HV bend is studied in detail. The sloshing motion of the solid phase is documented. Thereafter, the effects of varying the bend radius and the connecting pipe length between the two bends are presented in Chapter 5.

The computational mesh was sized initially with reference to other multiphase simulation studies. Thereafter, a mesh dependency test was performed. In applying the FAD turbulent dispersion model, its model constant is varied to determine the most appropriate value for such dilute flows. The model constant is observed to have a greater effect on the particle concentration distribution than on the gas phase velocity, and a most appropriate model constant was selected for to compare with earlier experimental results. Note that while numerical results show certain discrepancies, they otherwise captured the flow characteristics adequately after addressing the underlying cause of inaccuracies.

Gas-phase velocity is well predicted for both vertical and horizontal pipe sections. However, the model is less accurate in predicting particle concentration distribution in certain areas of the pipe. In the vertical pipe section, the inaccuracy results from the difficulty of capturing the transitional zone where the particle

Conclusion

rope starts to disperse. Outside this dispersion zone, the model was able to predict the particle concentration distributions. For the horizontal pipe section, initially, it appears that the model is erroneous in predicting particle concentrations. However, particle diameter variations appear to account for the mismatch. The simulation is repeated with an increased particle diameter. The model is then able to capture the particle concentration distribution in the horizontal pipe section with increased accuracy, which demonstrated the suitability of employing the Eulerian treatment for two-phase gas-particle flows.

While the Eulerian treatment of the solids phase presents an advantage over a Lagrangian treatment in terms of computational resources needed, it would still have been too demanding, on an industrial system scale, to incorporate the effects of the inherent variations in particle diameter within the material being conveyed. As such, variations in particle diameter was not included in the scope of this study. This would have meant that agglomeration of particles and break-up of agglomerates could not be taken into account within the model. Such limitations are the main cause of inaccuracies seen for the computational model. Barring this, the suitability of employing the Eulerian treatment for particles is well demonstrated.

Next, the validated numerical model was applied to an industrial scale pneumatic conveying pipe to study the flow characteristics of the material being conveyed. Steady-state numerical results are observed to agree well with bulk flow data that was obtained from JPPL, indicating an appropriate application of the numerical model. Subsequently, the conveying pressures and inlet loading were varied to

observe the relationships between the parameters and conveying output capacity. The relationship between output and both inlet pressure and loading was found to be non-linear, due to the physical limitations of the conveying system. Such findings will be useful to assess the impact of these parameters upon the operational capabilities of the pneumatic conveying pipe. In particular, these results also demonstrate that numerical simulations of a full-scale, operational pneumatic conveying pipe are feasible and sufficiently accurate for practical purposes.

Transient simulations were also performed on one selected flow configuration. The flow behaviour in a VH bend, preceded by an HV bend, was evaluated in terms of the particle rope and flow structures. Results show that, instead of passing through the centre axis of the pipeline, the particle rope stayed on the sides of the pipe wall after exiting the bend. This is due to the influences exerted by the secondary flow structures produced by the upstream bend. Also, the interesting sloshing behaviour of the conveyed material is observed and the angular position of the concentrated particles mimic that of an under-damped system. Collating with the transient result, the preceding flow observations are partly due to the secondary flow structures, on top of gravitational effects.

The effect of pipe bend ratio investigated next. With the steady-state simulation of the full JPPL pipeline conducted, the pipe geometry can now be reduced to a shorter section to conserve computational resources. By utilizing the same methodology as the transient simulation of the full pipeline, the bend ratio is found to affect the flow behaviour. Three different flow behaviours were

Conclusion

identified. One, the particle rope could swirl a complete revolution around the pipe wall due to a tight bend. Two, the particle rope may not possess enough momentum as in the first case and may split into two distinct ropes while inside the bend. These two ropes then take different paths to eventually meet at the bottom of the pipe in a region with increased particle collisions. Three, the particle rope disperses in the pipe bend which would otherwise be the ideal scenario if not for the gradual pipe bend prolonging the presence of non-symmetrical secondary flows.

Finally, the length of the vertical pipe connecting the two pipe bends is varied. Simulation results favour a shorter connecting pipe length in terms of the pressure drop across the second pipe bend. This is due to the dispersion mechanism of the particle rope increasing turbulence going into the pipe bend. This increase in turbulence causes additional pressure loss across the pipe bend. The dispersion mechanism of a particle rope in a vertical pipe section is also briefly described.

Based on the current findings, certain areas that could be built upon by future work have been uncovered. Flow characteristics, mainly the sloshing motion after a VH bend preceded by a HV bend, is presented in the current thesis. Together with flow characteristics of an HV preceded by a VH bend by previous researchers, it forms only a small subset of different combinations two bends can take and future studies can take the path of documenting flow characteristics involving different combinations. Also, the effects of two parameters, mainly the bend radius and the pipe length connecting two bends is studied individually. Additional work by employing tools from design of experiments (DOE) is certainly required to relate these two parameters on how they work together to

affect the flow characteristics. The effects of particle diameter used in the numerical simulation also require much more specialized studies to ascertain its relations within the numerical models. Notwithstanding these, there is also an incentive to discuss the numerical model closures in greater detail. The application of different turbulence models such as the k-omega model and/or other pressure-velocity coupling methods such as SIMPLEC or PISO could be an area of interest for future investigations. These areas should certainly be visited to achieve a more robust and accurate numerical modelling of pneumatic conveying processes.

List of publications arising from this study

J. Xiang, J. Heng, and T. H. New, "A numerical parametric and optimization study of an industrial air-slide conveyor system," *Powder Technology*, vol. 315, pp. 367-378, 2017

J. Heng, T. H. New, and P. A. Wilson, "On the application of an Eulerian granular model towards dilute phase pneumatic conveying," *Powder Technology*, vol. 327, pp. 456-466, 2018.

J. Heng, T. H. New, and P. A. Wilson, "Application of a numerical model to an industrial scale pneumatic conveying pipeline," Submitted to *Advanced Powder Technology* in July 2017.

J. Heng, T. H. New, and P. A. Wilson, "Effects of pipe geometry on pneumatic conveying process consisting of two pipe bend in close proximity," In preparations for submission to *Powder Technology*.

References

- [1] D. Mills, *Pneumatic conveying design guide*. Butterworth-Heinemann, 2003.
- [2] A. Yilmaz and E. K. Levy, "Roping phenomena in pulverized coal conveying lines," *Powder Technology*, vol. 95, no. 1, pp. 43-48, 1998.
- [3] G. E. Klinzing, *Pneumatic conveying of solids: A theoretical and practical approach*. Springer, 1997.
- [4] T. S. Khan, Y. Dai, M. S. Alshehhi, and L. Khezzar, "Experimental flow characterization of sand particles for pneumatic transport in horizontal circular pipes," *Powder Technology*, vol. 292, pp. 158-168, 2016.
- [5] M. S. de Moraes *et al.*, "Experimental quantification of the head loss coefficient K for fittings and semi-industrial pipe cross section solid concentration profile in pneumatic conveying of polypropylene pellets in dilute phase," *Powder Technology*, vol. 310, pp. 250-263, 2017.
- [6] A. Adeniji-Fashola and C. Chen, "Modeling of confined turbulent fluid-particle flows using Eulerian and Lagrangian schemes," *International Journal of Heat and Mass Transfer*, vol. 33, no. 4, pp. 691-701, 1990.
- [7] B. Vreman, B. J. Geurts, N. Deen, J. Kuipers, and J. Kuerten, "Two-and four-way coupled Euler–Lagrangian large-eddy simulation of turbulent particle-laden channel flow," *Flow, Turbulence and Combustion*, vol. 82, no. 1, pp. 47-71, 2009.
- [8] S. Elghobashi, "On predicting particle-laden turbulent flows," *Applied Scientific Research*, vol. 52, no. 4, pp. 309-329, 1994.

References

- [9] M. Yin, F. Shi, and Z. Xu, "Renormalization group based κ - ε turbulence model for flows in a duct with strong curvature," *International Journal of Engineering Science*, vol. 34, no. 2, pp. 243-248, 1996.
- [10] B. van Wachem, J. Schouten, C. van den Bleek, R. Krishna, and J. Sinclair, "Comparative analysis of CFD models of dense gas-solid systems," *American Institute of Chemical Engineers Journal*, vol. 47, no. 5, pp. 1035-1051, 2001.
- [11] A. D. Burns, T. Frank, I. Hamill, and J. Shi, "The Favre averaged drag model for turbulent dispersion in Eulerian multi-phase flows," in *5th International Conference on Multiphase Flow*, 2004, vol. 4, pp. 1-17.
- [12] E. Manjula, W. H. Ariyaratne, C. Ratnayake, and M. C. Melaaen, "A review of CFD modelling studies on pneumatic conveying and challenges in modelling offshore drill cuttings transport," *Powder Technology*, vol. 305, pp. 782-793, 2017.
- [13] S. S. Mallick and P. Wypych, "Evaluation of scaleup procedures using "system" approach for pneumatic conveying of powders," *Particulate Science and Technology*, vol. 28, no. 1, pp. 41-50, 2010.
- [14] H. Akilli, E. Levy, and B. Sahin, "Gas-solid flow behavior in a horizontal pipe after a 90 vertical-to-horizontal elbow," *Powder Technology*, vol. 116, no. 1, pp. 43-52, 2001.
- [15] A. Yilmaz and E. K. Levy, "Formation and dispersion of ropes in pneumatic conveying," *Powder Technology*, vol. 114, no. 1, pp. 168-185, 2001.

-
- [16] G. C. Pereira, F. J. de Souza, and D. A. de Moro Martins, "Numerical prediction of the erosion due to particles in elbows," *Powder Technology*, vol. 261, pp. 105-117, 2014.
- [17] H. K. Lee *et al.*, "Anti-abrasion pipe fittings for high-speed particle-laden flow," Patent U.S. Patent No. 6,422,608, July 23, 2002.
- [18] D. McGlinchey, A. Cowell, E. Knight, J. Pugh, A. Mason, and B. Foster, "Bend pressure drop predictions using the Euler-Euler model in dense phase pneumatic conveying," *Particulate Science and Technology*, vol. 25, no. 6, pp. 495-506, 2007.
- [19] A. B. Makwana, A. Patankar, and M. Bose, "Effect of Dune Formation on Pressure Drop in Horizontal Pneumatic Conveying System," *Particulate Science and Technology*, vol. 33, no. 1, pp. 59-66, 2015.
- [20] J. Li *et al.*, "Solids deposition in low-velocity slug flow pneumatic conveying," *Chemical Engineering and Processing: Process Intensification*, vol. 44, no. 2, pp. 167-173, 2005.
- [21] M. Hidayat and A. Rasmuson, "Some aspects on gas–solid flow in a U-bend: numerical investigation," *Powder Technology*, vol. 153, no. 1, pp. 1-13, 2005.
- [22] D. McGlinchey, A. Cowell, and R. Crowe, "CFD investigation of dense phase pneumatic conveying at a pipeline enlargement," *Particuology*, vol. 10, no. 2, pp. 176-183, 2012.
- [23] N. Behera, V. K. Agarwal, M. G. Jones, and K. C. Williams, "CFD modeling and analysis of dense phase pneumatic conveying of fine particles including particle size distribution," *Powder Technology*, vol. 244, pp. 30-37, 2013.
-

References

- [24] K. Ibrahim, M. El-Kadi, M. Hamed, and S. M. El-Behery, "Effect of bend orientation and flow direction on the behaviour of gas–solid flow," in *Proceedings 8th International Conference of Fluid Dynamics and Propulsion*, 2006, vol. 136, pp. 14-17.
 - [25] R. Schallert and E. Levy, "Effect of a combination of two elbows on particle roping in pneumatic conveying," *Powder Technology*, vol. 107, no. 3, pp. 226-233, 2000.
 - [26] B. Kaur, A. Mittal, P. Wypych, S. S. Mallick, and S. Jana, "On developing improved modelling and scale-up procedures for pneumatic conveying of fine powders," *Powder Technology*, vol. 305, pp. 270-278, 2017.
 - [27] N. Huber and M. Sommerfeld, "Modelling and numerical calculation of dilute-phase pneumatic conveying in pipe systems," *Powder Technology*, vol. 99, no. 1, pp. 90-101, 1998.
 - [28] X. Chen and J. Wang, "A comparison of two-fluid model, dense discrete particle model and CFD-DEM method for modeling impinging gas–solid flows," *Powder Technology*, vol. 254, pp. 94-102, 2014.
 - [29] W. Pu *et al.*, "Numerical simulation on dense phase pneumatic conveying of pulverized coal in horizontal pipe at high pressure," *Chemical Engineering Science*, vol. 65, no. 8, pp. 2500-2512, 2010.
 - [30] C. T. Crowe, *Multiphase flow handbook*. CRC Press, 2014.
 - [31] T. B. Anderson and R. Jackson, "A fluid mechanical description of fluidized beds. Equations of motion," *Industrial & Engineering Chemistry Fundamentals*, vol. 6, no. 4, pp. 527-539, 1967.
 - [32] S. Ergun, "Fluid flow through packed columns," *Chemical Engineering Progress*, vol. 48, pp. 89-94, 1952.
-

-
- [33] C. Wen and Y. Yu, "Mechanics of fluidization," in *Chemical Engineering Progress Symposium Series*, 1966, vol. 6, pp. 100-101.
- [34] D. Gidaspow, *Multiphase flow and fluidization: continuum and kinetic theory descriptions*. Academic Press, 1994.
- [35] J. Ding and D. Gidaspow, "A bubbling fluidization model using kinetic theory of granular flow," *American Institute of Chemical Engineers Journal*, vol. 36, no. 4, pp. 523-538, 1990.
- [36] C. Lun, S. B. Savage, D. Jeffrey, and N. Chepurniy, "Kinetic theories for granular flow: inelastic particles in Couette flow and slightly inelastic particles in a general flowfield," *Journal of Fluid Mechanics*, vol. 140, pp. 223-256, 1984.
- [37] K. Chu and A. Yu, "Numerical simulation of the gas– solid flow in three-dimensional pneumatic conveying bends," *Industrial & Engineering Chemistry Research*, vol. 47, no. 18, pp. 7058-7071, 2008.
- [38] T. Y. Quek, C.-H. Wang, and M. B. Ray, "Dilute gas-solid flows in horizontal and vertical bends," *Industrial and Engineering Chemistry Research*, vol. 44, no. 7, pp. 2301-2315, 2005.
- [39] C. Ratnayake, M. C. Melaaen, and B. K. Datta, "Pressure drop prediction in dense phase pneumatic conveying using CFD," in *Fourth International Conference on CFD in the Oil and Gas, Metallurgical & Process Industries SINTEF/NTNU Trondheim, Norway*, 2005, pp. 6-8.
- [40] W. Willmarth, "Pressure fluctuations beneath turbulent boundary layers," *Annual Review of Fluid Mechanics*, vol. 7, no. 1, pp. 13-36, 1975.
- [41] S. Lee and F. Durst, "On the motion of particles in turbulent duct flows," *International Journal of Multiphase Flow*, vol. 8, no. 2, pp. 125-146, 1982.
-

References

- [42] D. Modarress, S. Elghobashi, and H. Tan, "Two-component LDA measurement in a two-phase turbulent jet," *American Institute of Aeronautics and Astronautics Journal*, vol. 22, no. 5, pp. 624-630, 1984.
 - [43] Y. Tsuji, Y. Morikawa, and H. Shiomi, "LDV measurements of an air-solid two-phase flow in a vertical pipe," *Journal of Fluid Mechanics*, vol. 139, pp. 417-434, 1984.
 - [44] R. Pan, "Improving scale-up procedures for the design of pneumatic conveying systems," Department of Mechanical Engineering, University of Wollongong, New South Wales, Australia, 1992.
 - [45] S. S. Mallick and P. W. Wypych, "On improving scale-up procedures for dense-phase pneumatic conveying of powders," *Particulate Science and Technology*, vol. 29, no. 5, pp. 407-427, 2011.
 - [46] S. Vashisth and J. R. Grace, "Simulation of granular transport of Geldart type-A,-B, and-D particles through a 90° elbow," *Industrial & Engineering Chemistry Research*, vol. 51, no. 4, pp. 2030-2047, 2011.
 - [47] L. Y. Lee, T. Yong Quek, R. Deng, M. B. Ray, and C. Wang, "Pneumatic transport of granular materials through a 90 bend," *Chemical Engineering Science*, vol. 59, no. 21, pp. 4637-4651, 2004.
 - [48] H. Kruggel-Emden and T. Oschmann, "Numerical study of rope formation and dispersion of non-spherical particles during pneumatic conveying in a pipe bend," *Powder Technology*, vol. 268, pp. 219-236, 2014.
 - [49] S. Savage and D. Jeffrey, "The stress tensor in a granular flow at high shear rates," *Journal of Fluid Mechanics*, vol. 110, pp. 255-272, 1981.
-

-
- [50] M. Ehsani, S. Movahedirad, S. Shahhosseini, and M. Ashtiani, "Effects of Restitution and Specularity Coefficients on Solid - Liquid Fluidized Bed Hydrodynamics," *Chemical Engineering & Technology*, vol. 38, no. 10, pp. 1827-1836, 2015.
- [51] M. Syamlal, W. Rogers, and T. J. O'Brien, "MFIx documentation: Theory guide," *National Energy Technology Laboratory, Department of Energy, Technical Note*, 1993.
- [52] D. Gidaspow, R. Bezburuah, and J. Ding, "Hydrodynamics of circulating fluidized beds: kinetic theory approach," Illinois Institute of Technology, Chicago, IL (United States). Department of Chemical Engineering, 1991.
- [53] S. Ogawa, A. Umemura, and N. Oshima, "On the equations of fully fluidized granular materials," *Journal of Applied Mathematics and Physics (ZAMP)*, vol. 31, no. 4, pp. 483-493, 1980.
- [54] R. Brown and P. Hawksley, "Packing of regular (spherical) and irregular particles," *Nature*, vol. 156, no. 3962, pp. 421-422, 1945.
- [55] O. Simonin and P. Viollet, "Modelling of turbulent two-phase jets loaded with discrete particles," *FG Hewitt, et al., Phenomena in multiphase flow*, vol. 259, 1990.
- [56] d. B. M. Lopez, "Turbulent bubbly two-phase flow in a triangular duct," 1993.
- [57] P. C. Johnson and R. Jackson, "Frictional–collisional constitutive relations for granular materials, with application to plane shearing," *Journal of Fluid Mechanics*, vol. 176, pp. 67-93, 1987.
-

References

- [58] R. Rundqvist, C. Ljus, and B. Van Wachem, "Experimental and numerical investigation of particle transport in a horizontal pipe," *American Institute of Chemical Engineers*, vol. 51, no. 12, pp. 3101-3108, 2005.
 - [59] S. Laín and M. Sommerfeld, "Characterisation of pneumatic conveying systems using the Euler/Lagrange approach," *Powder Technology*, vol. 235, pp. 764-782, 2013.
 - [60] J. Xiang, J. Heng, and T. New, "A numerical parametric and optimization study of an industrial air-slide conveyor system," *Powder Technology*, vol. 315, pp. 367-378, 2017.
 - [61] H. Li and S. A. Vasquez, "Numerical simulation of steady and unsteady compressible multiphase flows," in *ASME 2012 International Mechanical Engineering Congress and Exposition*, 2012, pp. 2239-2251: American Society of Mechanical Engineers.
 - [62] N. Huber and M. Sommerfeld, "Characterization of the cross-sectional particle concentration distribution in pneumatic conveying systems," *Powder Technology*, vol. 79, no. 3, pp. 191-210, 1994.
 - [63] T. Li and S. Benyahia, "Revisiting Johnson and Jackson boundary conditions for granular flows," *American Institute of Chemical Engineers Journal*, vol. 58, no. 7, pp. 2058-2068, 2012.
 - [64] C. Loha, H. Chattopadhyay, and P. K. Chatterjee, "Euler-Euler CFD modeling of fluidized bed: Influence of specular coefficient on hydrodynamic behavior," *Particuology*, vol. 11, no. 6, pp. 673-680, 2013.
 - [65] W. C. Hinds, "Aerosol technology: properties, behavior, and measurement of airborne particles," *New York, Wiley-Interscience*, 1982. 442 p., vol. 1, 1982.
-

-
- [66] M. Korevaar, J. Padding, M. A. Van der Hoef, and J. Kuipers, "Integrated DEM–CFD modeling of the contact charging of pneumatically conveyed powders," *Powder Technology*, vol. 258, pp. 144-156, 2014.
- [67] C. Pei, C. Y. Wu, D. England, S. Byard, H. Berchtold, and M. Adams, "Numerical analysis of contact electrification using DEM–CFD," *Powder Technology*, vol. 248, pp. 34-43, 2013.
- [68] J. Yao *et al.*, "Experimental investigations of granular shape effects on the generation of electrostatic charge," *Particuology*, vol. 15, pp. 82-89, 2014.
- [69] V. Karde and C. Ghoroi, "Fine powder flow under humid environmental conditions from the perspective of surface energy," *International Journal of Pharmaceutics*, vol. 485, no. 1, pp. 192-201, 2015.
- [70] U. V. Shah, V. Karde, C. Ghoroi, and J. Y. Heng, "Influence of particle properties on powder bulk behaviour and processability," *International Journal of Pharmaceutics*, vol. 518, no. 1-2, pp. 138-154, 2017.
- [71] W. Zhong, A. Yu, X. Liu, Z. Tong, and H. Zhang, "DEM/CFD-DEM modelling of non-spherical particulate systems: theoretical developments and applications," *Powder Technology*, vol. 302, pp. 108-152, 2016.
- [72] J. Sun and F. Battaglia, "Hydrodynamic modeling of particle rotation for segregation in bubbling gas-fluidized beds," *Chemical Engineering Science*, vol. 61, no. 5, pp. 1470-1479, 2006.
- [73] H. F. Taylor, *Cement chemistry*. Thomas Telford, 1997.
- [74] A. Chambers and R. Marcus, "Pneumatic conveying calculations," in *Second International Conference on Bulk Materials Storage, Handling*
-

- and Transportation: 1986; Preprints of Papers*, 1986, pp. 49-52: Institution of Engineers, Australia.
- [75] S. S. Mallick, "Modelling of fluidised dense-phase pneumatic conveying of powders," Ph.D. Dissertation, Centre for Bulk Solids and Particulate Technologies, Faculty of Engineering, University of Wollongong, New South Wales, Australia, 2009.
- [76] M. Jones and K. Williams, "Solids friction factors for fluidized dense-phase conveying," *Particulate Science and Technology*, vol. 21, no. 1, pp. 45-56, 2003.
- [77] G. Setia, S. S. Mallick, R. Pan, and P. W. Wypych, "Modeling solids friction factor for fluidized dense-phase pneumatic transport of powders using two layer flow theory," *Powder Technology*, vol. 294, pp. 80-92, 2016.
- [78] R. Pan and P. Wypych, "Dilute and dense phase pneumatic conveying of fly ash," in *Proceedings of the 6th International Conference on Bulk Materials Storage and Transportation*, Wollongong, NSW, Australia, 1998, pp. 183-189.
- [79] F. A. Zenz, "Two-phase fluid-solid flow," *Industrial & Engineering Chemistry*, vol. 41, no. 12, pp. 2801-2806, 1949.
- [80] M. Bradley, "Pressure losses caused by bends in pneumatic conveying pipelines, effects of bend geometry and fittings," *Powder Handling and Processing*, vol. 2, no. 2, pp. 315-321, 1990.
- [81] Y. Morikawa, Y. Tsuji, K. Matsui, and Y. Jittani, "Pressure drops due to pipe bends in air-solids two phase flows; circular and elliptical bends," *International Journal of Multiphase Flow*, vol. 4, no. 5, pp. 575-583, 1978.
-

- [82] P. Spedding and E. Benard, "Gas–liquid two phase flow through a vertical 90 elbow bend," *Experimental Thermal and Fluid Science*, vol. 31, no. 7, pp. 761-769, 2007.
- [83] M. Rowe, "Measurements and computations of flow in pipe bends," *Journal of Fluid Mechanics*, vol. 43, no. 04, pp. 771-783, 1970.