Case Study on CFD Investigation of Dense Phase Pneumatic Conveying at a Pipeline Enlargement

MUHAMMAD RASHDAN BIN MOHAMMAD ROHIZAT

BACHELOR OF CHEMICAL ENGINEERING (GAS TECHNOLOGY)
UNIVERSITI MALAYSIA PAHANG

Thesis Access Form

No__________________________Location_____________________________

Author : ………………………………

Title : ………………………………

Status of access OPEN / RESTRICTED / CONFIDENTIAL

Moratorium period:_____________ years, ending___________/_______200____

Conditions of access proved by (CAPITALS): MISS SITI NORAISHAH ISMAIL
Supervisor (Signature)………………………………………………

Faculty: ………………………………

Author's Declaration: *I agree the following conditions:*

OPEN access work shall be made available (in the University and externally) and reproduced as necessary at the discretion of the University Librarian or Head of Department. It may also be copied by the British Library in microfilm or other form for supply to requesting libraries or individuals, subject to an indication of intended use for non-publishing purposes in the following form, placed on the copy and on any covering document or label.

*The statement itself shall apply to **ALL** copies:*

This copy has been supplied on the understanding that it is copyright material and that no quotation from the thesis may be published without proper acknowledgement.

Restricted/confidential work: All access and any photocopying shall be strictly subject to written permission from the University Head of Department and any external sponsor, if any.

Author's signature……………………………………….Date: ………………………………

users declaration: for signature during any Moratorium period (Not Open work):
I undertake to uphold the above conditions:

Date	Name (CAPITALS)	Signature	Address

Case Study on CFD Investigation of Dense Phase Pneumatic Conveying at a Pipeline Enlargement

MUHAMMAD RASHDAN BIN MOHAMMAD ROHIZAT
KC11040

BACHELOR OF CHEMICAL ENGINEERING (GAS TECHNOLOGY)
FAKULTI KEJURUTERAAN KIMIA DAN SUMBER ASLI
UNIVERSITI MALAYSIA PAHANG

SUPERVISOR'S DECLARATION

We hereby declare that we have checked this thesis and in our opinion, this thesis is adequate in terms of scope and quality for the award of the degree of Bachelor of Chemical Engineering (Gas Technology).

Signature	:
Name of main supervisor	:
Position	:
Date	:

STUDENT'S DECLARATION

I hereby declare that the work in this thesis is my own except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature :

Name :

ID Number :

Date :

DEDICATION

"To both my parents who have gave me courage and support for me to raise after falling down to the ground and to my supervisor for her unlimited guidance and supporting me to complete this case study with flying colours"

ACKNOWLEDGEMENT

I would like to show my greatest appreciation to;
My supervisor, Miss Siti Noraishah Ismail

- For her guide and support in completing this partial thesis flawlessly
- For her guide in helping me in rearranging my schedule to complete this case study
- For her guidance and support for me to manage my time for final year project and plant design

My supporters, Dr. Jolius Gimbun and Miss Woon Phui

- For their guidance in doing Gambit and Fluent
- For their time for improvising my CFD
- For their patients through all of my problems

My parents, Mohd Rohizat Aziz and Wan Aminah Salleh

- For their attention in keeping me well and hearing my problems
- For their support in keeping me back on my own two feet after countless of falling down to the ground
- For their love that let me to always remember my intention on choosing this way of life

ABSTRACT

Pneumatic conveying is being widely used by industry for their conveying system and the most critical problem that the system has is the corrosion of the pipeline. Some of the many engineers has develop a solution order to reduce the corrosion rate of the pipeline that is to increase the diameter of the pipe with objective to reduce the flow velocities as the flow velocities contribute the most in corroding the pipeline. By using Fluent 6.3.26 simulation program, 'Eulerian' Computational Fluid Dynamic (CFD) model, simulating the movement of the particles is possible and by varying the three different pipeline geometry; single bore, abrupt step, and gradual step, constructed using Gambit 2.4.6 from a pipe bore of 75-100 mm. The flow behaviour of plug of material passing through the pipeline is investigated. With 5x10-3 s time step, the solid volume fractions is recorded at 0.01 s of flow time at the point of enlargement and visualised throughout the pipe. Supported by 5 m/s air flow, the plug movement is illustrated showing that there is a potential of stagnant zone formation with the abrupt step enlargement geometry, and on the other hand, the gradual step shows a smooth dispersed particle flow without any potential of stagnant zone formation.

Key words: pneumatic conveying, Computational Fluid Dynamic (CFD), dense phase, dilute phase, enlargement geometry

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

$C\mu$, $C1\varepsilon$, $C2\varepsilon$	turbulence model constants
C_D	drag coefficient
d	particle diameter, m
e_{ss}, e_s	coefficient of restitution
g_0	radial distribution function
g	gravity vector, m/s^2
$G_{k,m}$	production of turbulent kinetic energy, $kg/(m\ s^3)$
I	identity matrix
I_{2D}	2nd invariant of the strain rate tensor, $1/s^2$
k	turbulence kinetic energy tensor, m^2/s^2
K_{gs}	gas–solid exchange coefficient, $kg/(m^3\ s)$
p	pressure, Pa
Re_s	particle Reynolds number
t	time, s
v	velocity vector, m/s
α	volume fraction
ρ	density, kg/m^3
τ_s	stress tensor, Pa
Θs	granular temperature, m^2/s^2
$\gamma\Theta s$	collisional dissipation of energy, $kg/(m\ s^3)$
μ	shear viscosity, Pa s
ε	turbulence dissipation rate, m^2/s^3
ϕ	angle of internal friction, degree
λs	bulk viscosity, Pa s
$\mu_{t,m}$	turbulent viscosity, Pa s

1 INTRODUCTION

1.1 Introduction and Problem Statement

A pneumatic conveying system transfers powders, granules, and other dry bulk materials through an enclosed horizontal or vertical conveying line. The motive force for this transfer comes from a combination of pressure differential and the flow of air (or another gas) supplied by an air mover, such as a blower or fan. By controlling the pressure or vacuum and the airflow inside the conveying line, the system can successfully convey materials. (Nol-Tec, 2014).

Pipeline enlargement in pneumatic conveying systems can be an advantages in reducing the pipeline erosion, product degradation and flow resistance (Klinzing, Rizk, Marcus, & Leung, 2010; Mills, 2004). This phenomenon is mainly due to the pipeline enlargement or increasing the pipeline cross sectional area at the same time may reduce the conveying gas flow velocity and increasing the pressure. The main approach of the system is of course the high pressure flow resulted from the conveying gas flow velocity reduction in the dilute phase conveying system (Zhang, Zhang et al., 2010), where a large portion of excess energy is used to overcome the friction experienced by the gas phase (conveying gas). This system can also be applied on the dense phase conveying system to prevent plug from reaching high velocity that has the potential in damaging the pipeline and system component.

1.2 Problem Statement and Motivation

The use of stepped pipeline enlargement in pneumatic conveying systems can be an advantages in reducing the pipeline erosion, product degradation and flow resistance (Klinzing, Rizk, Marcus, & Leung, 2010; Mills, 2004). This phenomenon is mainly due to the pipeline enlargement or increasing the pipeline cross sectional area at the same time may reduce the conveying gas flow velocity and increasing the pressure. The main approach of the system is of course the high pressure flow resulted from the conveying gas flow velocity reduction in the dilute phase conveying system which can be installed for over 1 km long (Zhang, Zhang et al., 2010), where a large portion of excess energy is used to overcome the friction experienced by the gas phase (conveying gas). This system can also be applied on the dense phase conveying system to prevent plug from reaching high velocity that has the potential in damaging the pipeline and system component.

However, interestingly that the note from early works from D.Mills, (2004) shows that there is a potential benefit of increased production rate by using lower velocities flow and higher material feed rate by using dense phase conveying system where proportionally less energy is required to overcome the resistance (D.Mills, 2004).

This research is to apply the same method of pipeline size increment onto the dense phase pneumatic conveying system in reducing the conveying gas flow velocity and at increase the flow pressure at the same time to reduce in the damage on the conveying system. The problem that currently occur when involving moving particle is that the corrosion of the pipeline due to high velocity of particles moving inside the pipeline due to the frictional forces that has been occur with the pipeline wall can damage the wall resulting in decreasing performances and frequent maintenance.

The treatment of a single (gas phase) passing through an abrupt or gradual enlargement is founded in standard texts, while some work has been published on gas-particle flow in the dilute phase region (Huang et al., 2009).

The detail of the flow behaviour of dense phase plugs in passing through a step may affect the overall pressure loss and there is the potential for the formation of 'stagnant' zones (blockage) at the location of the step or the plug may de-aerate which can potentially block the pipeline. These aspects did not seem to have been covered in the literature, and some initial work in CFD simulation using Fluent 6.3 (Fluent Inc., 2006) and Euler-Euler model applied to three pipeline geometries: single bore, abrupt step, and gradual step, is reported. Thus, this research is to find the dense phase plug behaviour by enlarging the pipeline by step in three different geometries. The findings are discussed in result and discussion section in terms of solids flow behaviour (solids volume fraction) and related velocities (Don McGlinchey et al., 2012).

1.3 Objective of the Case Study

The main objective of this case study is to reduce the velocity movement of a particle inside the dense phase pneumatic conveying by investigate the effect of enlarging the cross sectional area of the dense phase pneumatic conveyor by using three different geometry, that is; single bore, abrupt step, and gradual step, to illustrate and project the particle movement inside the conveying line, and the solid volume fraction at the end of the enlargement point of the pipeline under the same operating conditions.

1.4 Scope of the Research Study

The scope of this study are mainly to study and investigate the effect of pipeline enlargement against the particle movement by using three different enlargement geometry, that is; single bore, abrupt step, and gradual step. The method in running this research is by using CFD modelling work, which are by using the two computational softwares, Gambit 2.4.6 and Fluent 6.3.26. The operating conditions for all geometries are limited to single operating condition. The particle size is $2.5x10^{-5}$ m with density of 2500 kg/m^3 . The pipeline length is 3 m where the initial bore is 0.075 m and the final bore is 0.1 m. The time step of the simulation is $5x10^{-3}$ s which is the simulation results are obtained in every $1x10^{-2}$ seconds.

1.5 Hypothesis

This study is expected to improve the current form of pneumatic conveying by enlarging the pipeline diameter by using three different types of geometries by using CFD method in studying the fluid movement mechanics inside the pipeline of the conveyor to study in which type of enlargement can tackle the most effective problem solver of the plug inside the pipeline of the dense phase pneumatic conveying system. Gambit 2.4.6 is used as a medium to construct the pipeline geometry and mesh while Fluent 6.3.26 is used to transfer the constructed pipeline geometry to run the simulating program under programmed operating conditions.

1.6 Main Contribution on This Case Study

The following is the contributions I obtained along this semester doing this thesis:

- Main contribution was prior to my supervisor's guidance and support in handling and helping me in learning on how to make this thesis a successful thesis and deep study on using CFD simulation software, Gambit and Fluent.
- Study the different types of flow inside the dense phase pneumatic conveying pipeline and compare in between those three types of enlargement to be applied to future pneumatic conveyor.

1.7 Organization of the Thesis

The structure of the thesis from beginning to the end is outlined as follow:

Chapter 2 is the study on the pneumatic conveyor background and the previous research that has been done on this type of conveyor to further improve its performance for further usage. An overview about CFD software used in this project, such as Gambit and Fluent software, which are used to design geometry, mesh, calculate, simulate and finish this project and comparison between pneumatic conveying with other conveyors that has been used in the industries.

In Chapter 3 gives a review on the modelling method and procedure to work on this study**.** Most of the modelling method is being implemented in the simulation process where the geometrical design, mesh design and computational domain is being simulated. Results is compared with the different types of geometrical pipeline expansion and the flow of the particles inside the pipeline.

Chapter 4 illustrates the main findings of this study and the result outcome for the study and the results is supported by original researcher study. It gives out the images of particle flows inside the pipeline with their own boundary condition for the simulation programming.

2 LITERATURE REVIEW

2.1 Definition of Pneumatic Conveying

Pneumatic conveying system is mainly affected by controlling the pressure of the system. The differential pressure along a pipeline moves a bulk material along with the air as the air will likely moves towards the area from high pressure to lower pressure. This can be done with a vacuum inducer, or compressed air being injected into one end of the pipeline to create a pressure different inside the pipeline for continuous flow of the system (Steele, 2005).

Pneumatic conveying provides several advantages over the mechanical conveying. A pneumatic conveying system can be configured with bends to fit around existing equipment, giving it more flexibility than a mechanical conveyor with its typically straight conveying path. This also means the pneumatic conveying systems occupy less space than a comparable mechanical conveyor. The pneumatic conveying system is totally enclosed, unlike many mechanical conveyors, which enables the pneumatic system to contain dust. The pneumatic conveying system typically has fewer moving parts to maintain than a mechanical conveyor (Noc-Tel, 2014).

2.2 Dilute Phase and Dense Phase Pneumatic Conveying

Pneumatic conveying is divided into two types which is a dilute phase pneumatic conveying (low pressure) and dense phase pneumatic conveying (dense phase) systems. Dilute phase pneumatic conveying systems utilize the differential pressure which is less than 1 atmospheric pressure (atm). The system use either positive or negative pressure to either push or pull the martial through the pipeline (conveying line) at relatively high velocity of air flow. They are described as low pressure with high velocity systems which have high air to material ratio (Steele, 2005).

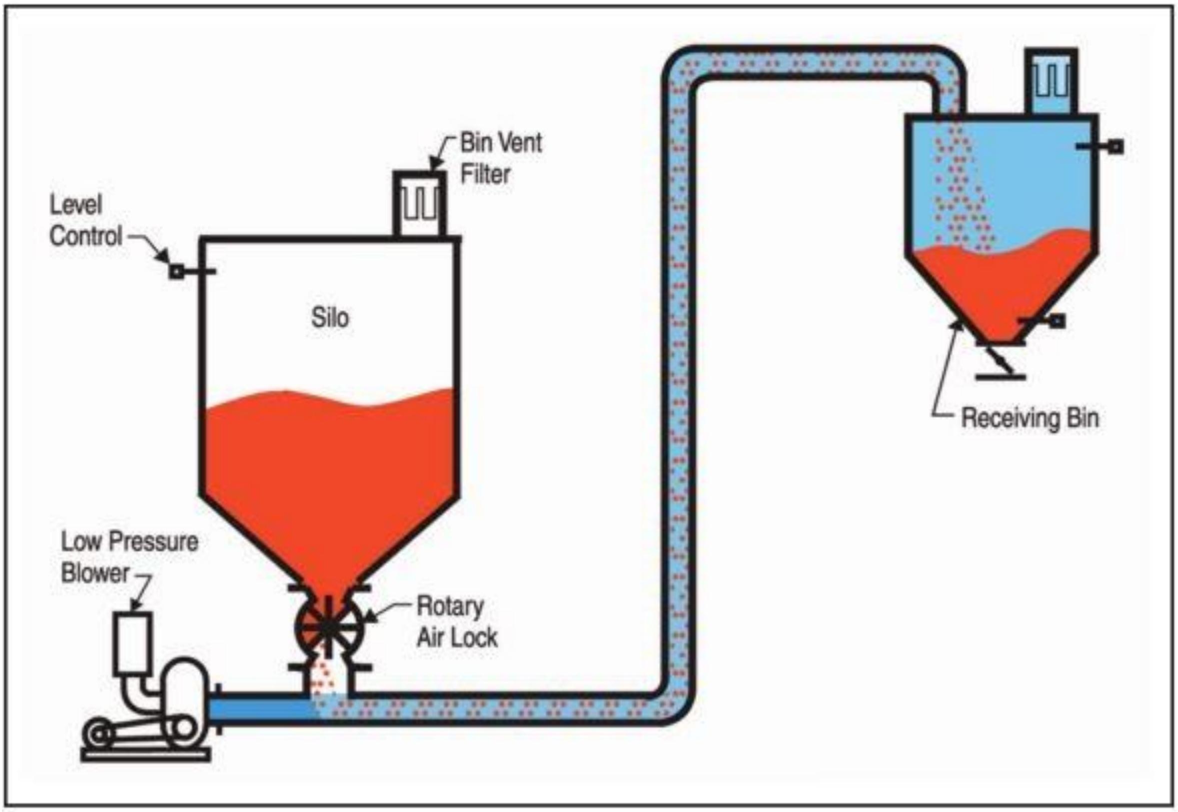

Figure 2.1: Dilute Phase Pneumatic Conveying

On the other hand, dense phase pneumatic conveying systems utilize the differential pressure which is more than 1 atmospheric pressure (atm). The system use positive pressure to push the material through the pipeline (conveying line) (Steele, 2005).

Dense phase pneumatic conveying system is a gentle way to convey or transfer difficult, abrasive, friable and mixed-batch materials by pushing the material through along the pipeline in a plug form at relatively low velocities. The advantages in using the pneumatic conveying systems is that it can reduce the rate of erosion, product degradation and flow resistance primarily due to the basic physics study by reducing the conveying gas velocity following an increase in the pipe cross sectional area (Nol-Tec, 2014).

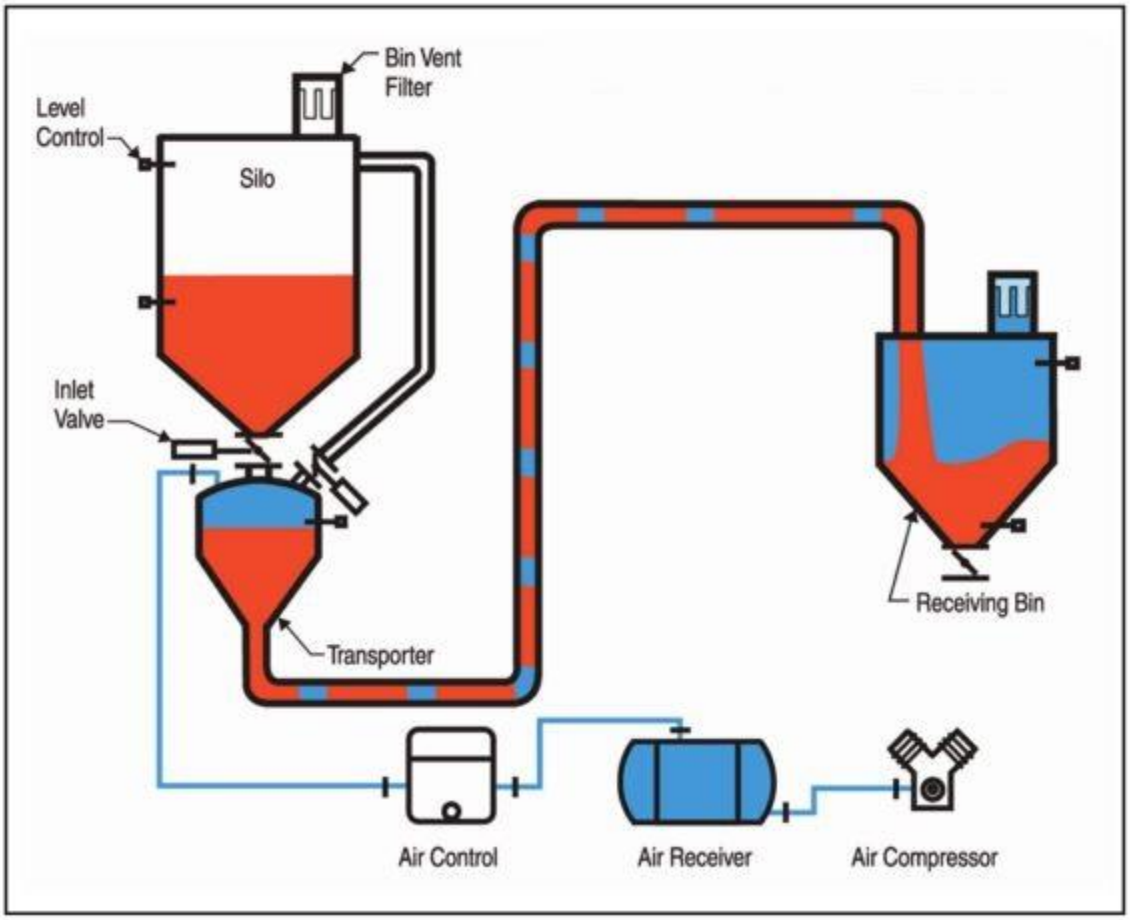

Figure 2.2: Dense Phase Pneumatic Conveying

The benefits of this approach are obvious for a high-pressure dilute phase conveying system which can be installed for over 1 km long inside a system. A large portion of energy available from the fan or pump is used to overcome the friction experience by the gas phase. This may be appropriate for dense phase system in order to prevent plugs or pipeline from reaching high velocity with the potential in damaging the components and support system. Pneumatic conveying system has the potential to increase the production rate at lower velocities and higher solids feed. For example, a dense phase conveying proportionally need less energy to overcome the resistances to flow (Zhang et al., 2010).

2.3 Pipeline Enlargement

The use of stepped pipeline enlargement in pneumatic conveying systems can be an advantages in reducing the pipeline erosion, product degradation and flow resistance (Klinzing, Rizk, Marcus, & Leung, 2010; Mills, 2004). This phenomenon is mainly due to the pipeline enlargement or increasing the pipeline cross sectional area at the same time may reduce the conveying gas flow velocity and increasing the pressure. This approach may also be appropriate for dense phase systems, for example, to prevent plugs from reaching high velocity with the potential of damaging system components and supports.

The benefits of this approach are obvious for high-pressure dilute phase conveying systems where a large portion of the available energy is used to overcome the friction experienced by the gas phase (Don, 2012). There is the potential benefit of increased product throughput at lower velocities and much higher solids loadings where proportionally less energy is required to overcome the resistances to flow (Mills, 2004).

2.4 Product Flow Rates and Air Mass Flow Rates

Figure 2.1 gradually explain the effect of product flow rates against air mass flow rates for a single bore pneumatic conveying system which lead to the solid lines show equal line pressures, and dashed lines show stated solids feed ratios, which is the ratio of mass flow rate of solid feed over mass flow rate of air. At points covering a broad range of conveying conditions are number which give the ratios of the mass flow rate of product found in a system with a step in pipeline bore (mp one step) over the mass flow rate of product measured in the same line pressure and solids feed or loading ratio in single bore line (mp single bore) (Mills, 2004). As shown in Figure 2.1 (reproduced from historical experimental data by D. Mills at Glasgow Caledonian University), the solids mass flow rate through the conveying system with a single step is approximately double that from a system with exactly the same layout and route but with a single bore pipeline, and this can increase the spans of the entire range of conveying conditions from dilute to dense phase covering the experimental data.

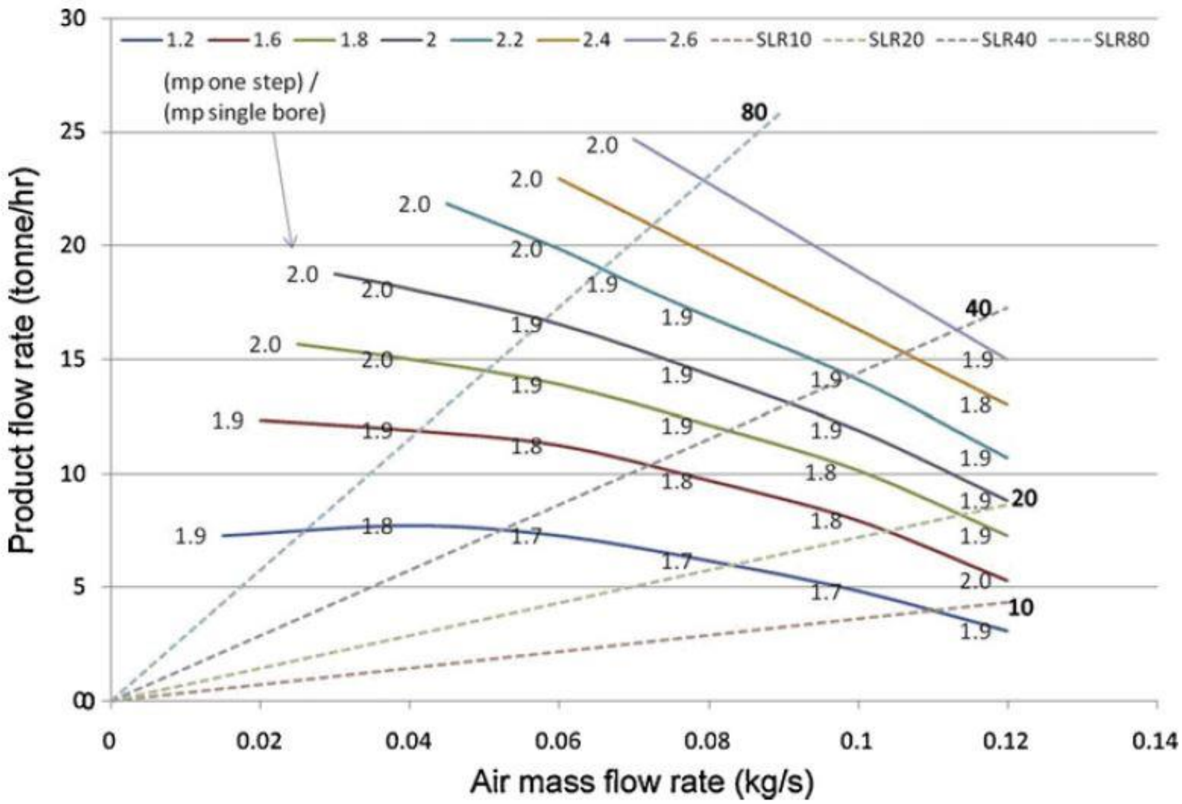

Figure 2.3: Effect on product mass flow rate of introducing a step in the conveying line (Mills, 2004). Solids lines of conveying line pressure drop (bar); dashed lines of constant solids loading ratio (-)

2.5 Why pneumatic conveyor?

2.5.1 Pneumatic Conveyor vs Screw Conveyor

Table 2.1 briefly discuss on the advantages of pneumatic conveyor over screw conveyor. These two conveyors are mainly used in industry as their transportation line system. Both have their own pros and cons but this research gives the mainly advantages in pneumatic conveyor (Flexicon Corporation, 2014).

Table 2.1: Advantages of Pneumatic Conveyor over Screw Conveyor

Pneumatic Conveyor	Screw Conveyor
• Can support long distance transport • High initial cost but low maintenance cost and cheap operating for long period of time • Can support multiple material sources • Can transfer to multiple material destinations • Conveyor routing can be organized and indirectly transfer material from source to destination • Material can be evacuated from conveying system • Can transport large amount of material • Can transport material without damaging the system and the material transported	• Can only support short and medium distance transport • Low initial cost but costly maintenance • Can only support one material source • Can transfer to multiple material destinations • Need a direct routing of material source and destination for installation • Material is transported directly to the destination and cannot be evacuated without shutting it down • Can transport limited amount of material • Can transport a large amount of material but very limited due to it can damage the system and the material transported

2.6 Previous Work on Pneumatic Conveying

The summary of published literature listed in Table 2.2 shows a few attempts of researcher in the study of pneumatic conveying system. A review by Marcus et al. (1990) provides an extensive list of ca. 300 types of materials that is suitable for pneumatic conveying with different particle properties, such as size, size distribution, shape, density and surface hardness. Several aspects of gas-solid suspension behaviour in pipes of different sizes and materials by varying the operating conditions are reported in the literature (Sankar and Smith, 1986; Laouar and Molodtsof, 1998; Molerus and Heucke, 1999; Costa et al., 2000).

Jiang et al. (1994) studied the influence of particle size on the fluid dynamic characteristics for the transport system by using low density polymeric particles (660 kg/m3) and a mean size ranging from 90 to 500 mm. For comparison, more experiments were carried out with Fluid Cracking Catalysts (FCC) (dp = 89 mm) and glass beads (dp=2000 mm) and with mixtures of different particles were also carried out. The results indicate in significantly wider operating range for the fast fluidization regime and enhancement of fine particle holdups in a bed with coarse particles. A mechanical model considering particle-particle collision was proposed (Jiang et al.; 1994) to explain the enhancement of fine particle holdups observed experimentally.

The transport of several types of coarse particles in horizontal tubes was studied by Molerus and Heucke (1999). In order to further study on how particle-fluid interactions affect flow regime and pressure loss in pneumatic transport, the authors carried out experiments in which several significant parameters were used as variable, including diameter of the transport tube, static pressure, particle and fluid densities, particle size and gas and solids flow rates. From all the parameters studied, particle size was found to be the least relevant.

However, interestingly that the note from early works from Mills (2004) shows that there is a potential benefit of increased production rate by using lower velocities flow and higher material feed rate by using dense phase conveying system where proportionally less energy is required to overcome the resistance (Mills, 2004).

Table 2.2: Summaries of Previous Literature Study on Pneumatic Conveying

Author	Previous Works/ Research	Parameter	Main findings
Marcus et al., 1990	Study on the effect of different types of materials, particle size, particle properties, size, distribution, shape, surface hardness and density on pneumatic conveying.	• Prediction/assumption based on no author has proposed a general model that involve all variables	Gas-solid transport is affected by the properties of the solids and by the riser characteristics
Sankar and Smith et al., 1986	Study on solid suspension behaviour of pipe with different sizes and material.	• Particle size = 96-637µm • Glass beads, sand, steel shots	Particle size has significant effect than particle density
Jiang et al., 1994	Influence of particle size on fluid dynamic characteristic for transport by using polymeric particles, fluid cracking catalysts, glass beads and mixtures.	• Particle size = 90-500µm • Particle densities = 660 kg/m^3 • Other particle type = FCC (dp=89 µm) and glass beads (dp=2000 µm)	Fast fluidization regime and enhancement of fine particle holdups in a bed with coarse particles
Molerus and Heucke, 1999	Coarse particles, flow regime, and pressure drop in horizontal tubes affected by particle-fluid interactions.	• Varying diameter of the transport tube, static pressure, particle and fluid densities, particle size and gas and solids flow rates	Particle size variable was found to be the least relevant
Mills, 2004	Effects in lowering the flow velocities can increase the production flow rate.	• Enlarging the pipeline diameter to reduce air flow rate	Reducing the air flow rate can increase the moving force pressure

2.7 Computational Fluid Dynamics (CFD)

Computational fluid dynamics, also known as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems involving fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. Computers with high processing speed may yield better results and solutions. Software is used to increase the accuracy and speed for complex calculation and simulation such as transient or turbulent fluid flow.

CFD can also predict the fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical and numerical equations which govern these processes using a numerical processes and iterations. Claude-Louis Navier and George Gabriel Stokes introduced viscous transport into the Euler equations, which resulted in the Navier–Stokes equation based on current CFD. Richardson (1991) developed the first numerical weather prediction system when he divided physical space into grid cells and used the finite difference approximations of Bjerknes's "primitive differential equations". The earliest numerical solution for fluid flow past a cylindrical pipe was carried out by (Thom et al., 1993).

Thus, CFD was developed from the pioneering efforts by (Richardson et al., 1991, Thom et al., 1993, Courant et al., 1928, Southwell et al.1940, Neumann at al., 1950), who in their endeavours to procure insight into fluid motion producing the development of powerful numerical techniques that can describe all types of fluid flow (Shang et al., 2004). The theoretical division of NASA contributed many numerical methods, and Spalding with his colleagues in developing many codes and numerical method algorithms (Runchal et al., 2003). Commercial CFD codes began to widely known and used from the early 1980s. During the last 30 years, a market for commercial CFD software began to grow quickly, and the commercial CFD software is used in almost all engineering working fields and calculations (Fluent et al., 2003). CFD is based on three principle numerical approaches – the Finite Difference Method (FDM), Finite Element Method (FEM) and Finite Volume Method (FVM). Finite difference (FD) discretization is known as the earliest method used and is based on the application of polynomial, Legendre polynomial, Fourier and Taylor series expansions to represent many ordinary differential equations (ODE) (Peiro et al., 2005).

This scheme motivated the use of an integral form of partial differential equations (PDEs) and automatically helping the development of the next numerical approaches, Finite Element (FEM) and Finite Volume techniques (FVM). Current CFD mainly uses the FEM and FVM method rather than the FDM, which has the limitation in handling complicated designs and geometries. Finite Element (FE) discretization divides up the region into a number of smaller regions which for the computational domain is based on a piece of wise approximation and assumption of the solution. The PDEs used in solving the numerical equations are typically obtained by restating the conservation equation in a weak formulation (Ferziger et al., 2002, Kumar et al., 2009). This solving process was established by the Galerkin method. Finite Volume (FV) discretization is based on an integral form of the PDE to be solved, with the values of the conserved variables averaged across the volume. The PDE is written in a form which can be solved for a given finite volume (or cell). The computational domain is discretized into finite volumes, and then for every volume the governing equations are solved (Ferziger et al., 2002, Ahmad N et al., 1998).

2.8 Fluent

This project uses Fluent 6.3.26 as the major simulation software used to calculate and simulate the entire simulation process of the process by calculation using iterations of finite numerical methods where the 3-Dimensional pipeline geometry construction and design mesh is constructed by using Gambit 2.4.6 computer software. The pipeline designed and mesh then extracted to be used by Fluent as their calculating medium in simulation. Fluent, one of the commercialized CFD software package, is based on a finite volume method approach. This software solver uses cell-centred finite volumes. In cell centred schemes, the flow variables are stored at the centres of the mesh elements (Fluent et al., 2003). Fluent focused in offering several solution approaches and the final results desired by the user (density-based as well as segregated and coupled pressure-based methods).

2.9 Gambit

Other than Fluent, this project require Gambit computer software as the main geometry design and generate mesh of the pipeline to be use as the medium inside Fluent, a computational software. Other than Gambit to generate the geometry, there are other software that can do the same such as AutoCAD and Google Sketch Up. Gambit is used in this project as a tool to generate or import geometry as it is widely used by engineer in engineering so that it can be used as a basis for simulations runs in Fluent. Thus, Gambit is used rather than other software as the main program for generating pipeline geometry design. With geometry in place it generates a mesh for the surface and volume of the geometry allowing it to be used for computational fluid dynamics. Fluent is a "Flow Modelling Software" that is used to model fluid flow within a defined geometry using the principles of computational fluid dynamics. Unlike Gambit, it utilizes a multi-window pane system for displaying various configuration menus and grids instead of a single window with several embedded sub-windows restricted within the space of the parent window. Fluent is able to read geometries generated in Gambit and model fluid flow within them. It can model various scenarios using computational fluid dynamics, including compressible and incompressible flow, multiphase flow, combustion, mass and heat transfer.

2.10 Mesh Design

Grid generation is a key issue in flow simulation as it governs the stability and accuracy of the flow predictions. For the present case, flow of plug through pipeline, is structured to three pipeline enlargement geometries; single bore, abrupt step, and gradual step. Figure 2.4 shows the example of pipeline mesh designed using Gambit to be use later in Fluent simulation.

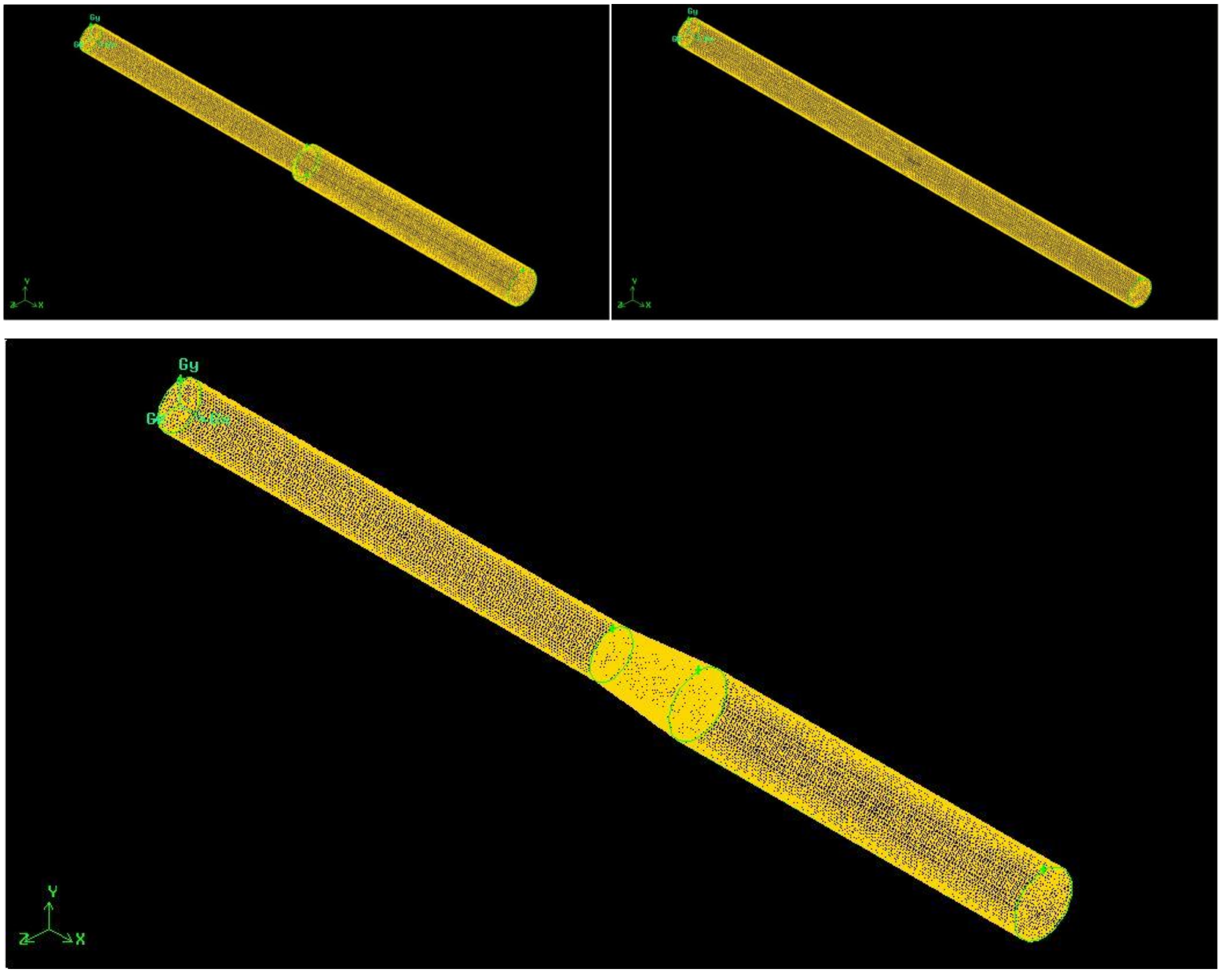

Figure 2.4: Sample of typical geometry, boundaries and unrefined mesh

3 MATERIALS AND METHODS

3.1 Overview

There are many approaches in modelling but the author has chosen to use mathematical model as the main CFD model of pneumatic conveyor modelling. There are a total of three types of modelling have to be made for this pipeline enlargement geometries project; single bore, abrupt step, and gradual step. The parameters used to run the fluid component inside the pipe is constant throughout the project and of the same for all three types of different pipeline geometries. The operating conditions for all geometries are limited to single operating condition. The particle size is 2.5×10^{-5} m with density of 2500 kg/m^3. The pipeline length is 3 m where the initial bore is 0.075 m and the final bore is 0.1 m. The time step of the simulation is 5×10^{-3} s which is the simulation results are obtained in every 1×10^{-2} seconds. Again, this project need both of the CFD modelling software that is Gambit and Fluent computational design and simulation software.

3.2 Mathematical Model

As the project conducted is experimented on gas phase and majority on gas flow rate a turbulence modelling is expected to be used in this project. One of the simplest of the multi-phase turbulence models, is an extension of the single phase k-ε model, provided in Fluent 6.3.24, was used in this case study project.

The k and ε equations in this model are:

Equation 3.1

$$\frac{\partial}{\partial t}(\rho_{\mathrm{m}} k) + \nabla \cdot (\rho_{\mathrm{m}} \vec{v}_{\mathrm{m}} k) = \nabla \cdot \left(\frac{\mu_{\mathrm{t,m}}}{\sigma_{\mathrm{k}}} \nabla k \right) + G_{\mathrm{k,m}} - \rho_{\mathrm{m}} \varepsilon,$$

and Equation 3.2

$$\frac{\partial}{\partial t}(\rho_{\mathrm{m}} \varepsilon) + \nabla \cdot (\rho_{\mathrm{m}} \vec{v}_{\mathrm{m}} \varepsilon) = \nabla \cdot \left(\frac{\mu_{\mathrm{t,m}}}{\sigma_{\varepsilon}} \nabla \varepsilon \right) + \frac{\varepsilon}{k}(C_{1\varepsilon} G_{\mathrm{k,m}} - C_{2\varepsilon} \rho_{\mathrm{m}} \varepsilon),$$

Where the mixture density, ρ_m, and the mixture velocity, $\bar{v}_m$, are given as:

Equation 3.3

$$\rho_{\mathrm{m}} = \sum_{i=1}^{2} \alpha_i \rho_i,$$

And Equation 3.4

$$\vec{v}_{\mathrm{m}} = \frac{\sum_{i=1}^{2} \alpha_i \rho_i \vec{v}_i}{\sum_{i=1}^{2} \alpha_i \rho_i}.$$

The turbulent viscosity for the mixture, $\mu_{t,m}$, and the production of turbulence kinetic energy, $G_{k,m}$, are derived as:

Equation 3.5

$$\mu_{\mathrm{t,m}} = \rho_{\mathrm{m}} C_{\mu} \frac{k^2}{\varepsilon},$$

And Equation 3.6

$$G_{\mathrm{k,m}} = \mu_{\mathrm{t,m}} (\nabla \vec{v}_{\mathrm{m}} + (\nabla \vec{v}_{\mathrm{m}})^{\mathrm{T}}) : \quad \nabla \vec{v}_{\mathrm{m}}.$$

The model constants were left as default values as there are no evidence to suggest any improvement in performance from any alternative values.

The Eulerian multi-phase calculations is solved by using the phase coupled SIMPLE algorithm. A density-based solver is used to solve for the components velocity of all phases simultaneously, a pressure correction equation is built based on total volume of the continuous flow and then the pressure and velocities are corrected to satisfy that continuous flow.

3.3 Simulation Modelling and Parameter

This thesis uses simulation modelling as one of the approach in pneumatic conveyor modelling although there are many other types of approaches. Simulation model can illustrate as pneumatic conveying of particulate solids, including 1-Dimensional models (Mason, Marjanovic, and Levy 1998), to combine with CFD-DEM models (Tsuji, Tanaka & Ishida, 1992; Xiang & McGlinchey, 2004; Xiang, McGlinchey, & Latham, 2010) and two fluid model (Mason & Levy, 2001).

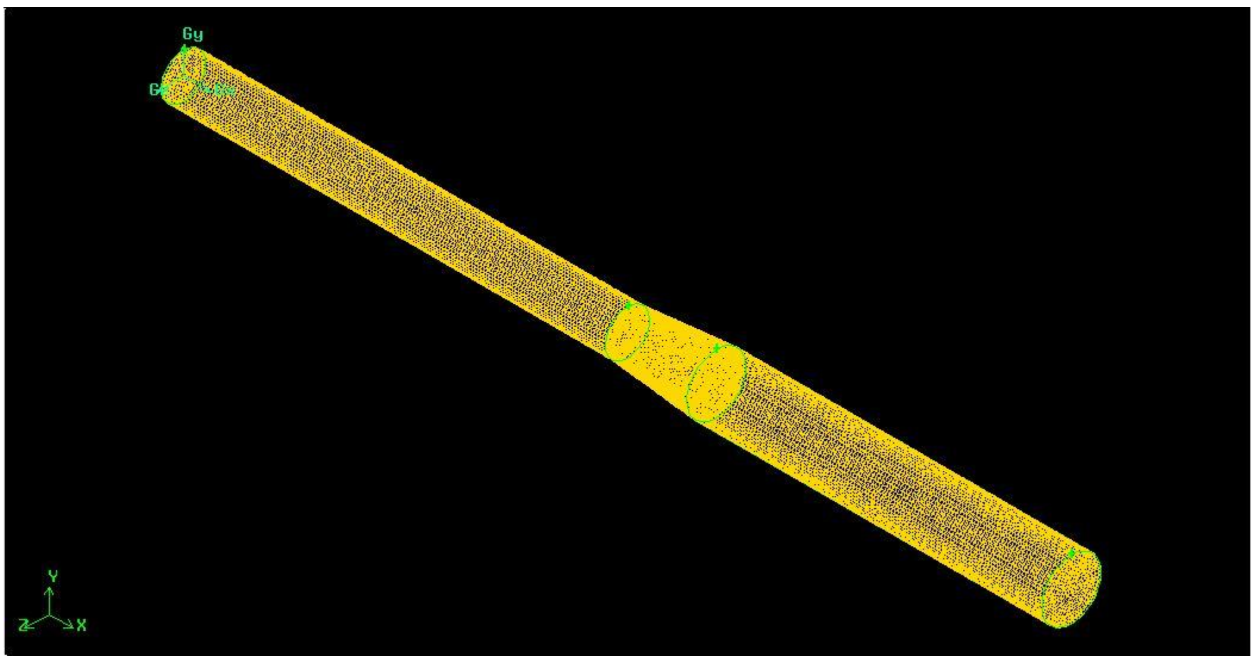

Figure 3.1: Illustration of 3-Dimensional Design of Gradual Step Pipeline Enlargement

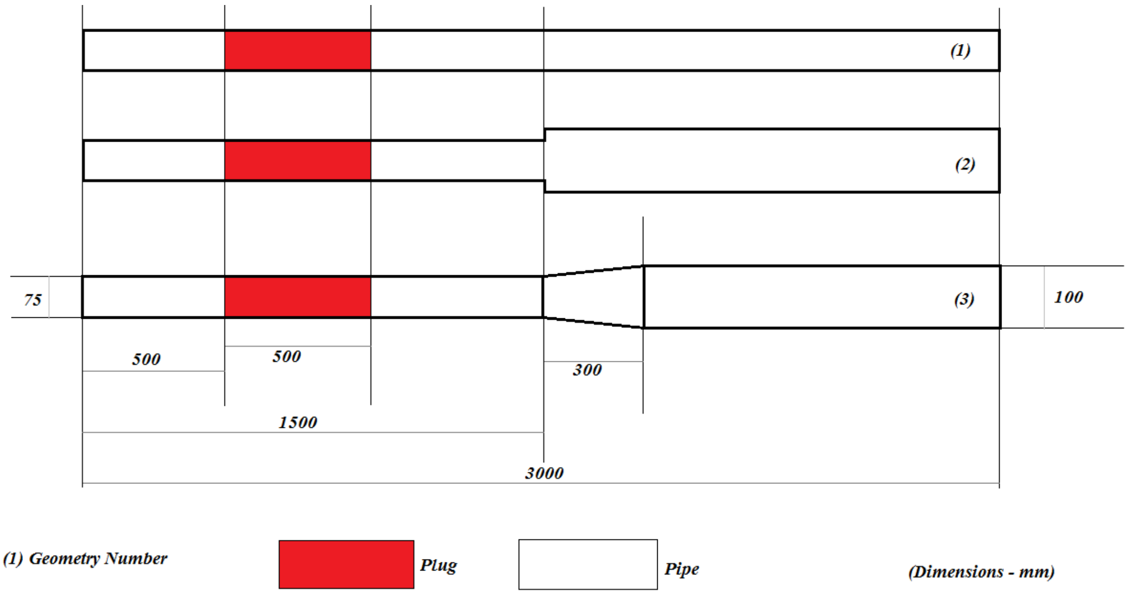

Figure 3.2: Illustration of Three Pipeline Geometries

Euler-Euler based model is used with the solid phase to represent the granular kinetics formation as coded in Fluent 6.3.26 (Fluent Inc., 2006). This approach has been used in van Wachem, Schouten, Krishna, and van den Bleek studies by using fluidised bed (Cammarata, Lettieri, Micale, & Colman, 2003; van Wachem, Schouten, Krishna, & van den Bleek, 1998) and it is proven that the model is appropriate for modelling dense phase pneumatic conveying (McGlinchey & Cowell, 2007, 2008; McGlinchey et al., 2007; Ratnayaka, Melaaen, & Datta, 2004; Zhang, Chen et al., 2010; Zheng, Pugh, McGlinchey, & Ansell, 2008; Zheng, Pugh, McGlinchey, & Knight, 2009).

Table 3.1: Simulation Models and Boundary Conditions

Particle size	$2.5x10^{-5}$ m
Particle density	2500 kg/m^3
Inlet boundary condition	Velocity inlet (5m/s)
Outlet boundary condition	Pressure outlet (0 Pa gauge)
Wall boundary condition	No slip for air, free slip for solids
Pipe bore initial	0.075 m
Pipe bore final	0.1 m
Initial bore length	1.5 m
Gradual length	0.3 m
Total pipe length	3 m
Initial plug length (full bore)	0.5 m
Initial volume fraction of solid phase	0.6
Initial solids velocity	3 m/s
Initial gas velocity	5 m/s
Viscous model	Standard k-ε
Wall treatment	Standard
Pressure velocity coupling	Phase coupled SIMPLE
Discretisation	First order upwind
Time step	$5x10^{-3}$ s

4 RESULTS AND DISCUSSIONS

4.1 Overview

This section propose the result outcome from the CFD simulation project that is conducted. The total amount of solid volume fraction is recorded for every 1×10^{-2} s where a total of 100 results is obtained for every types of pipeline geometries. The data is illustrated as a figures for every 0.1 s for easier interpretation and observations.

4.2 Results Outcome

The results from this study of the effect of enlargement geometry are mainly presented in the form of contour plots of solids volume fraction and velocity vector plots of solids velocity, and discussed in terms of the implications for pneumatic conveying system operation and design. The simulations begin with a ‘plug’ of particulate solids initialised as a patch in the geometry at a distance of 500 mm from the pipe inlet. The progression of the plug of material is followed at 0.1 s intervals. The phenomenon of plug collapse is well known for plugs of solids with coarse particle size, where the plug collapses to form a stationary layer in the pipe (Xiang & McGlinchey, 2004). The solids volume fraction was monitored at the centre of the pipe cross-section location at the start of the enlargement and the values of solids volume fraction against flow time for the sudden expansion and the gradual expansion over 300 mm is recorded. The table shows the data taken from this research and should be taken for all three geometrical types.

Table 4.1: Solids Volume Fraction over Flow Time for Single Bore

Flow Time (seconds)	Solids Volume Fraction
0.1	0.002669598
0.2	0.536962800
0.3	0.286464800
0.4	0.006514125
0.5	0
0.6	0
0.7	0

0.8	0
0.9	0
1.0	0

Table 4.2: Solids Volume Fraction over Flow Time for Abrupt Step

Flow Time (seconds)	Solids Volume Fraction
0.1	0.002183689
0.2	0.534836800
0.3	0.246278700
0.4	0.001505854
0.5	0.000731976
0.6	0.003305335
0.7	0
0.8	0
0.9	0
1.0	0

Table 4.3: Solids Volume Fraction over Flow Time for Gradual Step

Flow Time (seconds)	Solids Volume Fraction
0.1	0
0.2	0.000873616
0.3	0.516671400
0.4	0.426696800
0.5	0.036458890
0.6	0.000166430
0.7	0
0.8	0
0.9	0
1.0	0

From the CFD simulation that has been conducted, the main objective is to determine the solid volume fraction of the particle at the point after the enlargement of the pipeline. From that data of solid volume fraction, the velocity of the flow and the pattern of the particle movement inside the pipeline can be predicted.

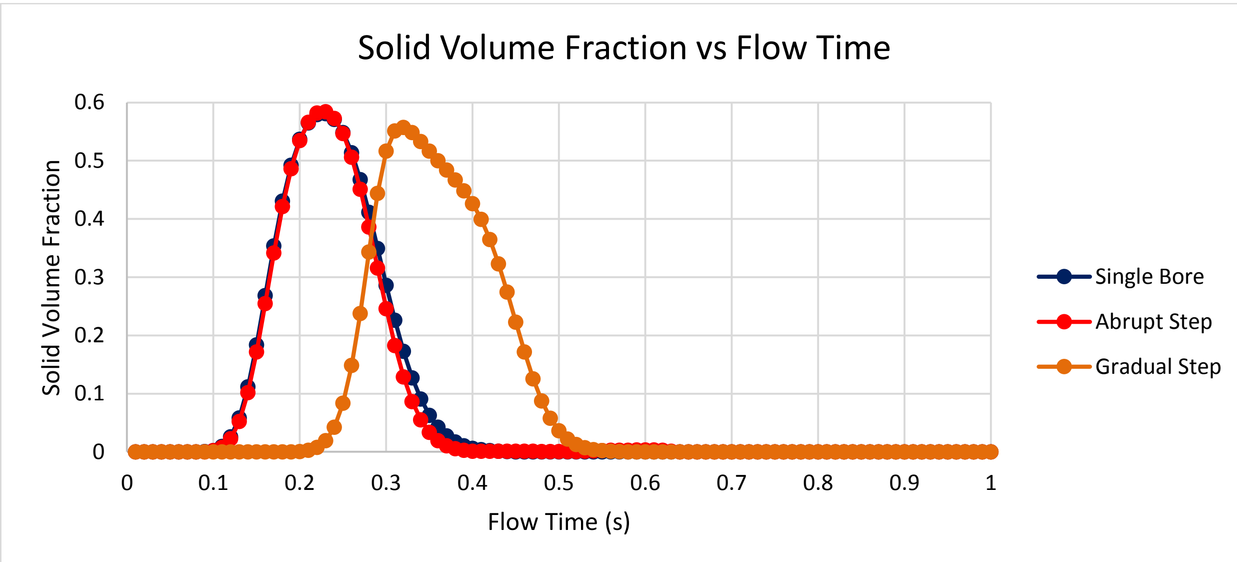

Figure 4.1: Solid Volume Fraction of Particles after the Enlargement Step against Flow Time (s)

Figure 4.1 shows the data recorded of solid volume fraction at the centre of the pipe at the location after the enlargement is clearly different abrupt step enlargement than for the gradual step enlargement. In an abrupt step enlargement, the solids volume fraction starts to rise from zero at a flow time of approximately 0.12 s. The distance from the plug to the enlargement step is consistent, 0.5 m and the plug initial velocity being set at 3 m/s. The solid volume fraction then rises to a value of 0.58 at 0.23 s, and declined. Before the solid volume fraction reaches completely zero, there is a step increase at 0.42 s in the solid volume fraction, shown in Figure 4.2 before it continues back declining. Figure 4.2 shows the disadvantage of the abrupt step enlargement which shows that the movement of particle is not very smooth after the enlargement. The graph shows that there is a sudden step increase of the solid volume fraction at the enlargement point which is supposed to be declining. This happen due to the sudden changes in the cross sectional area of the pipeline which created a cyclone in between the flow which traps some of the particle back to the enlargement point. The air intends to completely fill the pipeline and at the same time trap some of the particle with it showing a sudden increase in the solid volume fraction at the point of the enlargement. By showing a sudden increase of the volume fraction during the declining period, abrupt step enlargement shows a huge potential on accumulating solid volume fraction at the enlargement step and at the same time producing 'stagnant' zone which may block the flow of the particle inside the pipeline. The total time taken for the plug to pass the monitoring point is then seen to be 0.38 s. The plug flow pattern in the abrupt step enlargement shows almost identical result as the standard single bore pipeline. The initial relatively rapid rise over 0.11 s and steep decline for 0.14 s as the plug leaves the monitoring point is almost consistent with a plug shape which has a relatively long tail.

Gradual step enlargement shows a completely different flow pattern from the abrupt step enlargement. The solid volume fraction incline at a bit later than abrupt step due to its different monitor location. The solid volume fraction first incline at 0.23 s and continue to peak until 0.55 solid volume fraction at 0.33 s. The time taken for the solid volume fraction to peak is around 0.1 s and the value begins to decline at a slightly slower rate than abrupt step expansion. The solid volume fraction reaches almost zero at 0.54 s taking almost 0.21 s longer period than the abrupt step expansion.

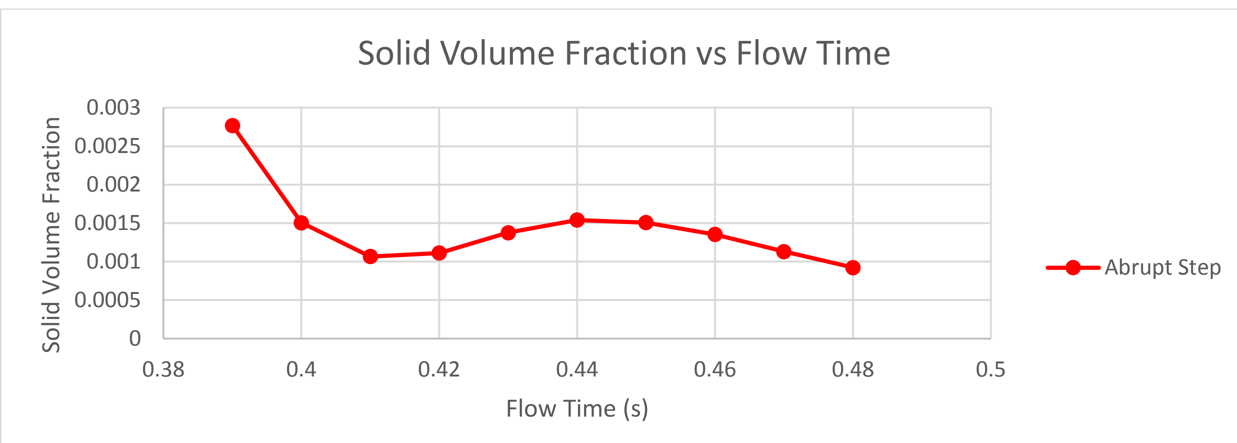

Figure 4.2: Solid Volume Fraction of Particles of Abrupt Step against Flow Time (s)

On the other hand, previous researcher (McGlinchey, 2012) recorded that the solid volume fraction for sudden expansion fluctuates before it peaked at 0.49 solids volume fraction at approximately 0.3 s. The gradual expansion shows a little fluctuation and lower peak value of 0.29 solid volume fraction approximately at 0.18 s. The main reason of the different in the results may be the material selection with the researcher and the number of iterations run on Fluent CFD software. The previous researcher might have run the iterations until the formula converge to obtain the most accurate result possible.

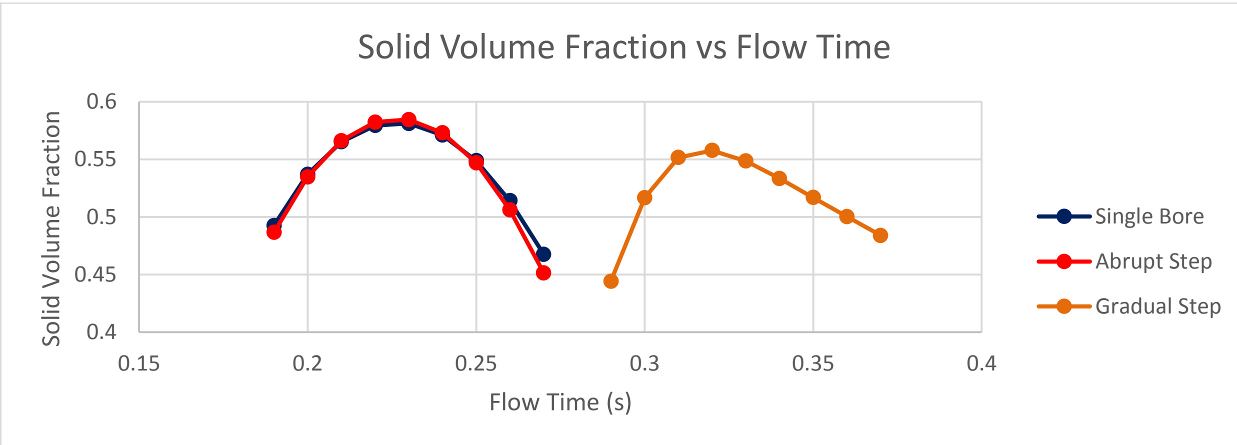

Figure 4.3: Peak point of the Solid Volume Fraction at the Enlargement Point against Flow Time (s)

Figure 4.3 shows the peak solid volume fraction of the particle at the point of enlargement against the flow time. From here, it shows that the abrupt step enlargement particle movement is almost identical to the standard pipeline of a single bore. On the other hand, the gradual step enlargement shows a different particles movement where the declining factor is slower than the one shown by the abrupt step enlargement and the standard single bore pipeline. Gradual step enlargement meet the main objective of this case study which is to increase the pressure inside the pipe and at the same time, reducing the velocity of the particles and the declining solid volume fraction did not show any sudden step increase like the abrupt step enlargement type. The peak point of the gradual step also shows a slight decrease of solid volume fraction which is around 0.55 from the other two which is approximately at 0.58 indicating that this type of step enlargement disperse the particle more effectively.

The result on the previous researcher, McGlinchey (2012), about the same experiment on the enlargement step of the pipeline. The results shows fluctuation in the solid volume fraction at the enlargement step and the most intense fluctuation is at the abrupt step enlargement where the solid volume fraction peak at approximately 0.49 where gradual step enlargement only peak at approximate 0.29. This shows that high solid volume fraction at the abrupt step enlargement has a high possibility on creating the 'stagnant' zone which may block the entire system of the conveyor. From the result of the previous research, it also shows that the abrupt step at a disadvantage to apply on the dense phase pneumatic conveying system.

4.3 Contour Display of Particle Movement

The results from this study of the effect of enlargement geometry are mainly presented in the form of contour plots of solids volume fraction. The simulations begin with a 'plug' of particulate solids initialised as a patch in the geometry at a distance of 500 mm from the pipe inlet.

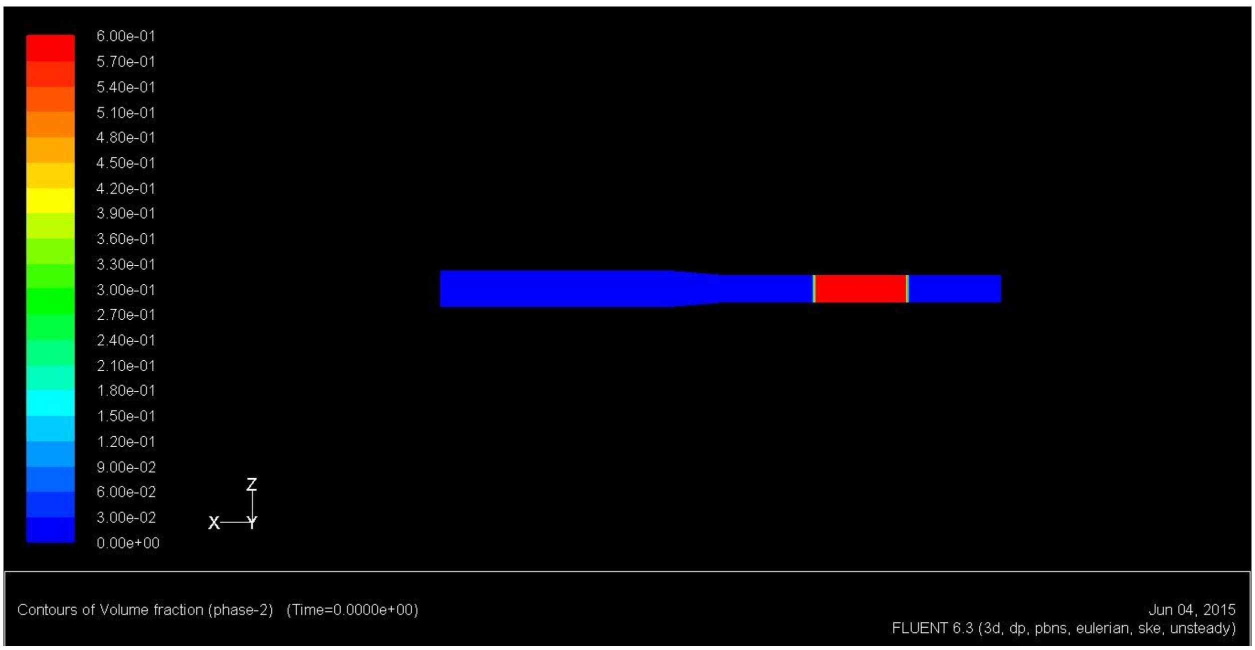

Figure 4.4: Abrupt Step Contour Display of Plug Location at time 0 s

Figure 4.4 shows the solids volume fraction in an abrupt expansion case at time = 0 s. The progression of the plug of material is followed at 0.1 s intervals as illustrated in Figure 3.2, which show the movement of the plugs in the no expansion, sudden expansion and gradual expansion cases respectively.

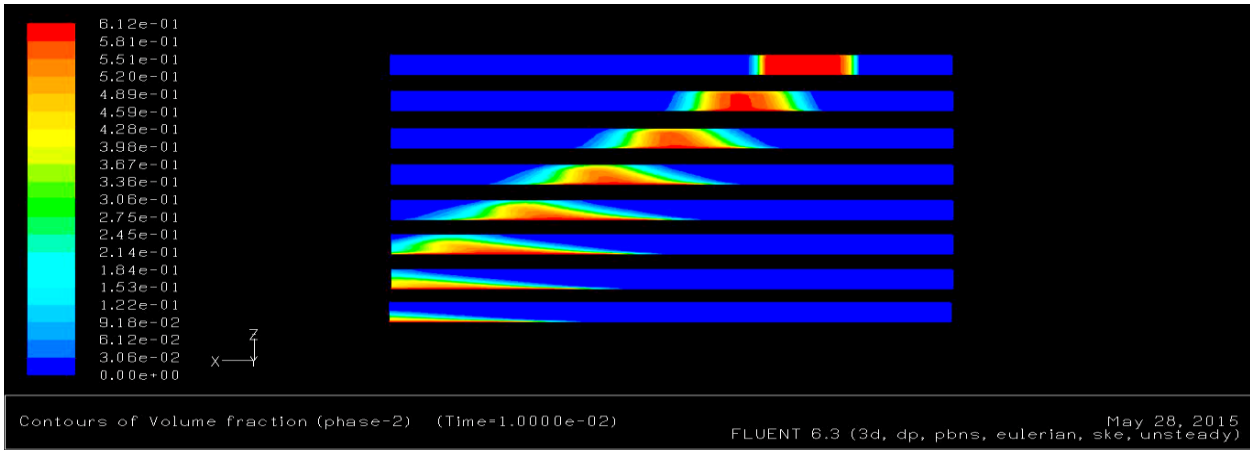

Figure 4.5 Contour Display of Movement Flow of Particles inside the Single Bore Pipeline

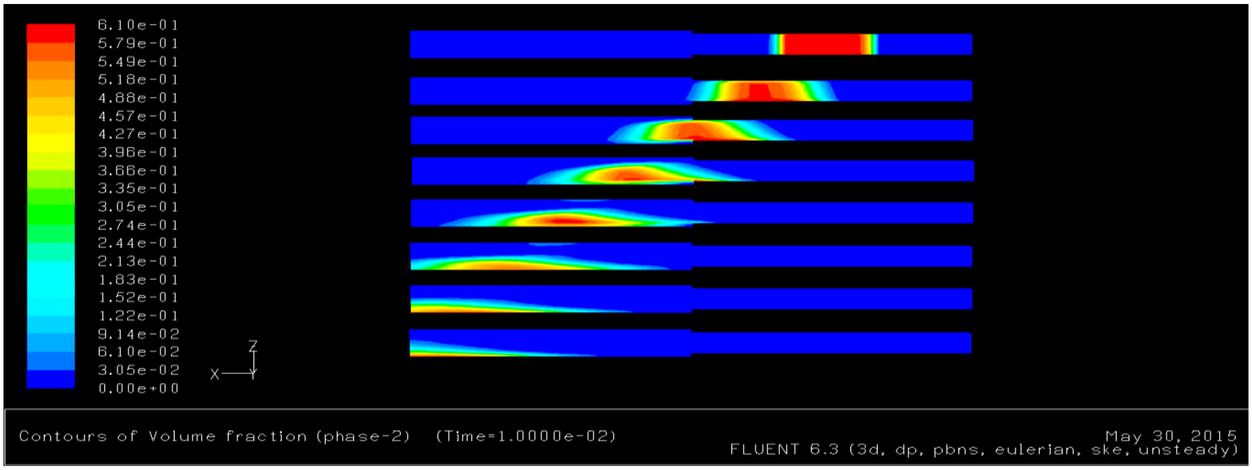

Figure 4.6: Contour Display of Movement Flow of Particles inside the Pipeline through Abrupt Enlargement Step

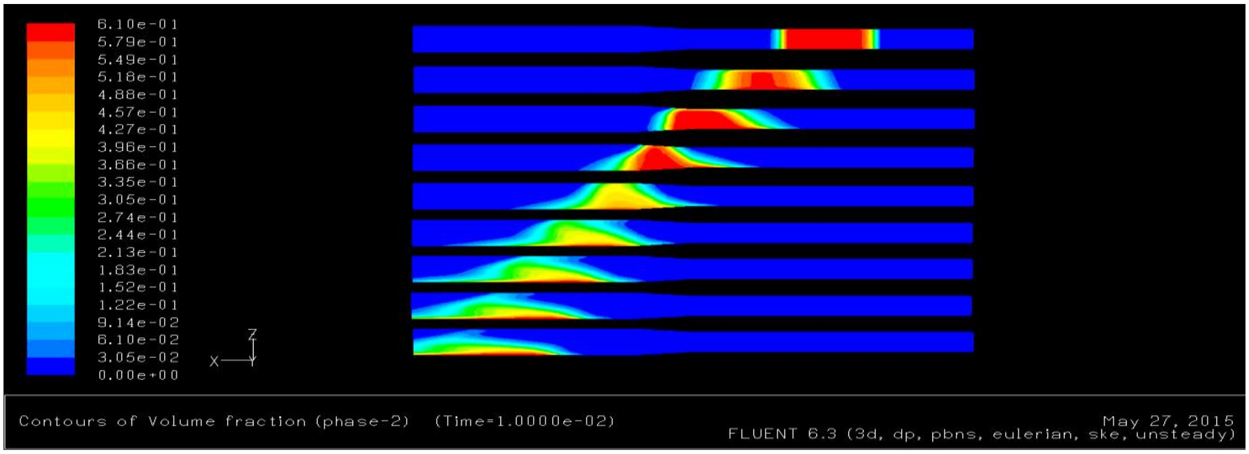

Figure 4.7: Contour Display of Movement Flow of Particles inside the Pipeline through Gradual Enlargement Step

The above figures illustrate the movement of the plug inside the pipeline at 0.1 s of flow time step. With a single bore pipe without any expansion, the plug appears to be collapsing while travelling along the pipe. This phenomenon of plug collapse is well known for plugs of solids with coarse particle size, where the plug collapses to form a stationary layer in the pipe (Xiang & McGlinchey, 2004). From observation of the contour display of the plug, particle tends to travel along a horizontal pipe as a ‘moving (fluidised) bed’, which is similar to what is being shown in Figure 4.5. The solids volume fraction was monitored at the centre of the pipe cross-section location at the end of the enlargement and the values of solids volume fraction against flow time for the sudden expansion and the gradual expansion over 300 mm is shown in Figure 4.1.

Figure 4.6 shows a contour display for sudden expansion of the pipeline. The plug flow at the pipeline expansion is a bit messy as a cyclone in between the flow which traps some of the particle back to the enlargement point. The air intends to completely fill the pipeline and at the same time trap some of the particle with it showing a sudden increase in the solid volume fraction at the point of the enlargement. The plug continue to flow with the same pattern as the single bore pipeline.

Figure 4.7 shows a contour display for gradual expansion pipeline which shows a smooth plug dispersion area at the enlargement step. Figure 4.1 also shows no sudden increase in the solid volume fraction at the monitor point and the declining period is a bit longer than the other pipeline geometry. The plug continue to flow in a bigger and longer form of plug where the volume fraction has been divided equally in every bore direction.

Contour image of Xiang & McGlinchey (2012) shows that there is a slight plug trapped at in between the wall of the sudden expansion (abrupt) enlargement of the pipeline. Differ from the result obtained showing the plug dispersed randomly resulting in solid volume fraction fluctuation at the point where the result is recorded. Both for the single bore pipeline and gradual step enlargement step shows almost identical plug flow from the researcher.

4.4 Conclusion

The objectives of this case study is to reduce the velocity of the particle movement inside the pipeline by increasing the cross sectional area of the pipeline under different types of enlargement geometries, illustrate the movement of particles and the solid volume fraction inside the pipeline, and compare which is the best enlargement step for the system. The key to this objective is to not allow any potential in creating a ‘stagnant’ zone which may block the entire system. The peak point of the gradual step also shows a slight decrease of solid volume fraction which is around 0.55 from the other two which is approximately at 0.58 indicating that this type of step enlargement disperse the particle more effectively. Gradual step enlargement meet the main objective of this case study which is to increase the pressure inside the pipe and at the same time, reducing the velocity of the particles and the declining solid volume fraction did not show any sudden step increase like the abrupt step enlargement type.

5 REFERENCES

- Ahmad N, Rappaz J, Desbiolles JL, Jalanti T, Rappaz M, Combeau H, et al. Numerical simulation of macro segregation : a comparison between finite volume method and finite element method predictions and a confrontation with experiments.MetallMaterTransA1998;29:617–30.
- Cammarata, L., Lettieri, P., Micale, G. D. M., & Colman, D. (2003). 2D and 3D CFD simulations of bubbling fluidized beds using Eulerian–Eulerian models. International Journal of Chemical Reactor Engineering, 1, ArtA48.
- Courant R, Friedrichs K, Lewy H. Über die partiellen Differenzengleichungen der mathematischen Physik. Math Ann1928; 100:32–74.
- Flexicon Corp,. 2014
- Fluent I. FLUENT user's guide. Fluent Inc; 2003.
- Fluent 6.2: Fluent Inc. Lebanon (NH, USA); 2005.
- Fluent Inc,. 2006
- Gidaspow, D. (1994). Multiphase flow and fluidization: Continuum and kinetic theory descriptions. Boston: Academic Press.
- Huang, W., Gong, X., Guo, X., Dai, Z., Liu, H., Cao, Z., et al. (2009). Study of the pressure drop of dense phase gas–solid flow through nozzle. Powder Technology, 189, 82–86.
- Jiang, P, Bi, H, Liang, S.C. and Fan, L.S., Hydrodynamic Behavior of Circulating Fluidized Bed with Polymeric Particles. AIChE Journal, vol. 40, no 2, pp. 193-205 (1994).
- Klinzing, G. E., Rizk, F., Marcus, R. D., & Leung, L. S. (2010). Pneumatic conveying of solids: A theoretical and practical approach (3rd ed.). London: Springer.
- Kumar A. CFD modelling of gas–liquid–solid fluidized bed. Rourkela, India: B. Tech, NIT; 2009.
- Marcus, R.D., Leung, L.S., Klinzing, G.E. and Rizk, F., Pneumatic Conveying of Solids: a Theoretical and Practical Approach. 1st ed. Great Britain, St Edmundsbury Press (Powder Technology Series) (1990).
- Mason, D. J., & Levy, A. (2001). A model for non-suspension gas–solids flow of fine powders in pipes. International Journal of Multiphase Flow, 27, 415–435.
- Mason, D. J., Marjanovic, P., & Levy, A. (1998). A simulation system for pneumatic conveying systems. Powder Technology, 95, 7–14.

- McGlinchey D, Cowell A, Crowe R. CFD investigation of dense phase pneumatic conveying at a pipeline enlargement. Particuology 2012;10(2):176-183
- McGlinchey, D., & Cowell, A. (2008). CFD visualisation of pneumatic conveying passive by-pass line. Bulk Solids & Powder Science & Technology, 3(3), 109–115.
- McGlinchey, D., & Cowell, A. (2007, March). Varying parameters of the fluent Eulerian model for predicting pressure drops in pneumatic transport. In PARTEC 2007, International Congress on Particle Technology Nuremberg, Germany, (pp. 27–29).
- Mills, D. (2004). Pneumatic conveying design guide (2nd ed.). Oxford: Butterworth-Heinemann.
- Molerus, O. and Heucke, U., Pneumatic Transport of Coarse Grained Particles in Horizontal Pipes. Powder Technology, vol. 102, pp. 135-150 (1999).
- Noc-Tel,. 2014
- Peiro J, Sherwin S. Finite difference, finite element and finite volume methods for partial differential equations. Hand book of materials modelling. 2415–46.
- Ravi Sankar, S. and Smith, T.N., Slip Velocities in Pneumatic Transport, part I. Powder Technology, vol. 47, pp. 167-177 (1986).
- Richardson L. The approximate arithmetical solution by finite differences of physical problems involving differential equations, with an application to the stresses in a masonry dam. Philos Trans R Soc A1911; 210:307–57.
- Runchal AK. Brian spalding: CFD & reality. Int J Heat Mass Transfer 2008; 10.
- Shang J. Three decades of accomplishments in computational fluid dynamics. Prog AerospSci2004; 40:173–97.
- Southwell RV. Relaxation methods in engineering science. London, UK: Oxford UniversityPress; 1940.
- Steele GJ. Why dense phase? What is pneumatic conveying? Pneumatic conveying systems for cement. 2005.
- Thom A. The Von Neumann J, Richtmyer R. A method for the numerical calculation of hydrodynamic shocks. J Appl Phys1950; 21:232–7.
- van Wachem, B., Schouten, J. C., van den Bleek, C. M., Krishna, R., & Sinclair, J. L. (2001). Comparative analysis of CFD models of dense gas–solid systems. AIChE Journal, 47, 1035–1051.

- Xiang, J., McGlinchey, D., & Latham, J.-P. (2010). An investigation of segregation and mixing in dense phase pneumatic conveying. Granular Matter, 12, 345–355.
- Zhang, C., Zhang, Y., McGlinchey, D., Du, Y., Wei, X., Ma, L., et al. (2010). Experimental and numerical study on power consumptions in a double-tube-socket pneumatic conveying system. Powder Technology, 204(2–3), 268–272.
- Zhang, Y., Chen, L., Zhang, C., Yu, L., Wei, X., Ma, L., et al. (2010). CFD analysis of pneumatic conveying in a double-tube-socket (DTS) pipe. Applied Mathematical Modelling, 34, 3085–3097.
- Zheng, Y., Pugh, J., McGlinchey, D., & Knight, E. A. (2009). Investigation of heat transfer coefficient in a gas–solids dense phase pneumatic pipe flow. Bulk Solids & Powder Science & Technology, 4(4), 169–174.

6 APPENDICES

6.1 GAMBIT

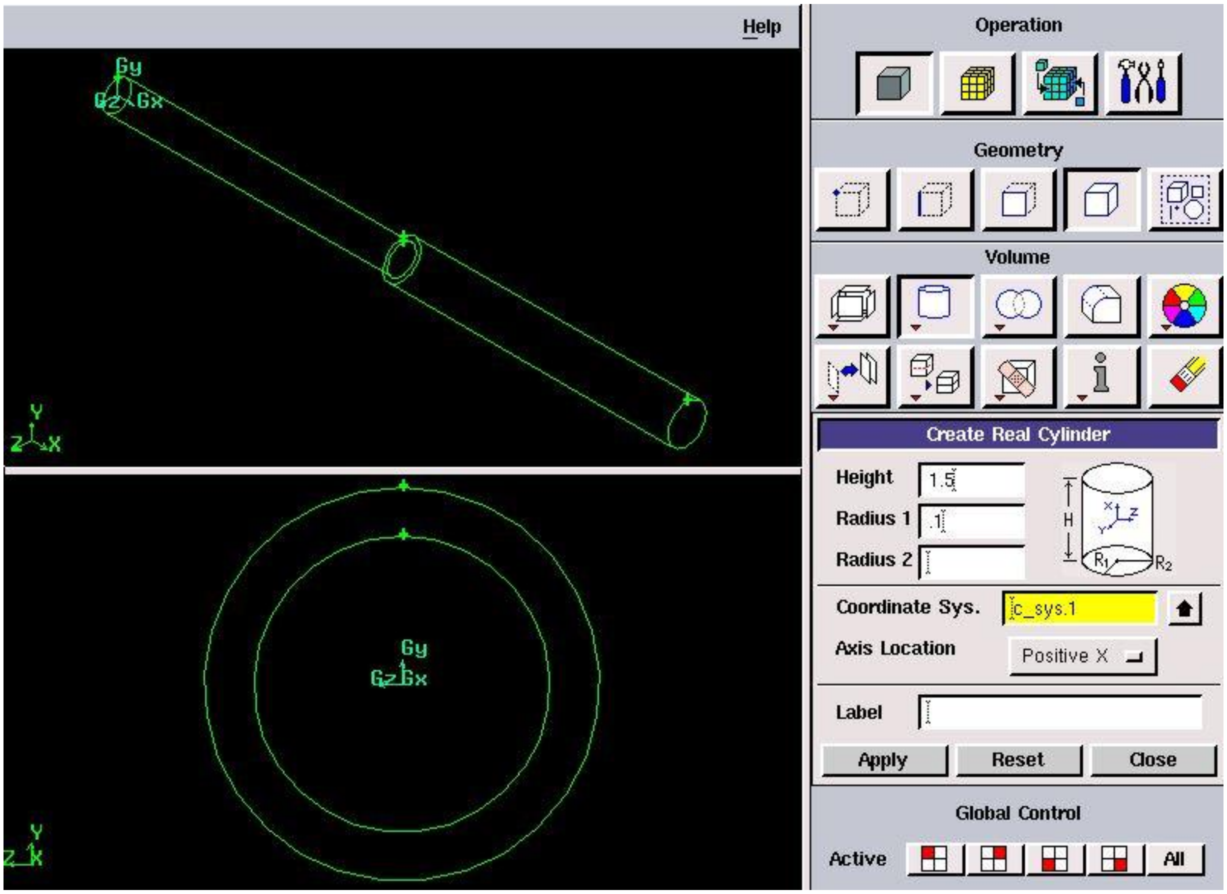

Figure 6.1: Geometry Design

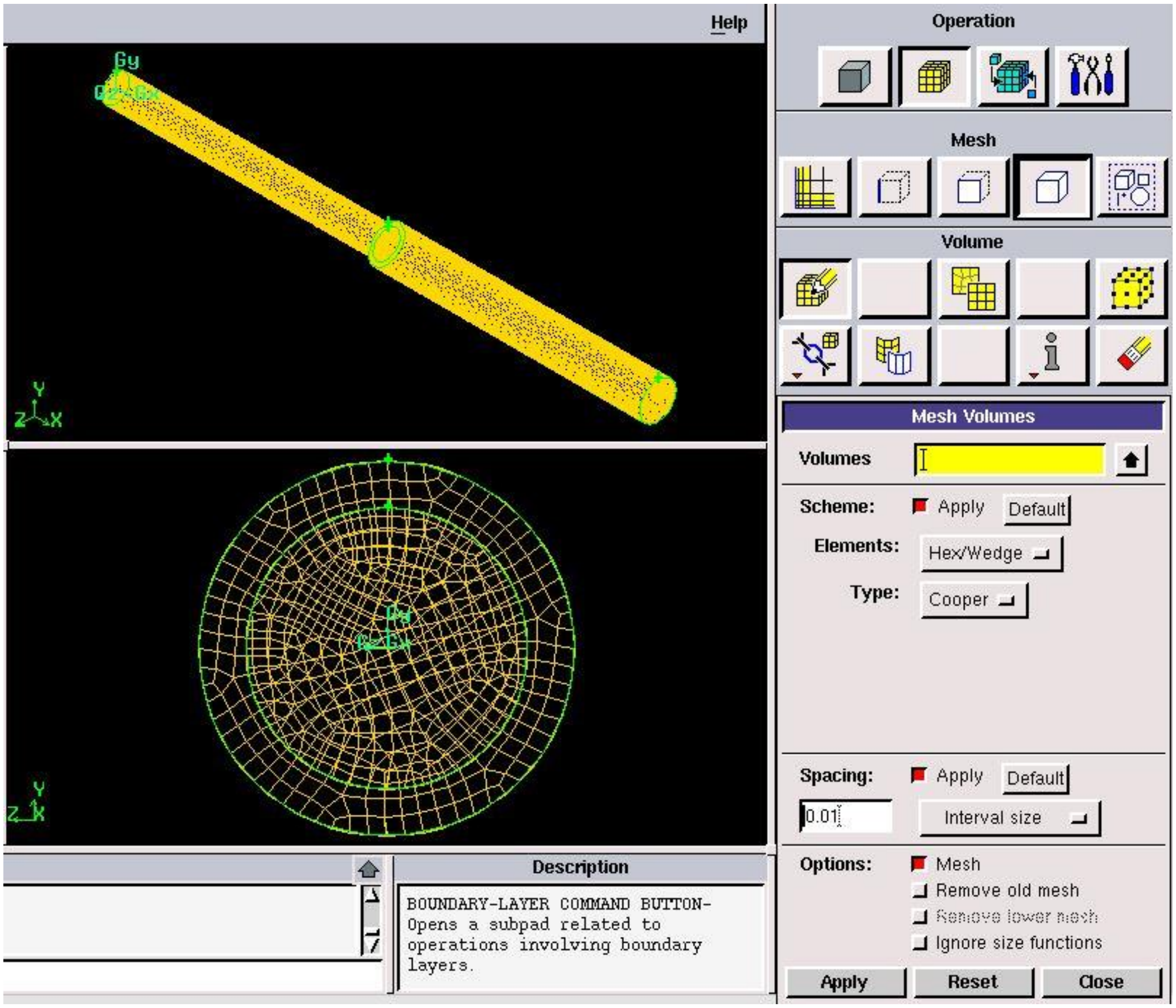

Figure 6.2: Mesh Design

6.2 FLUENT

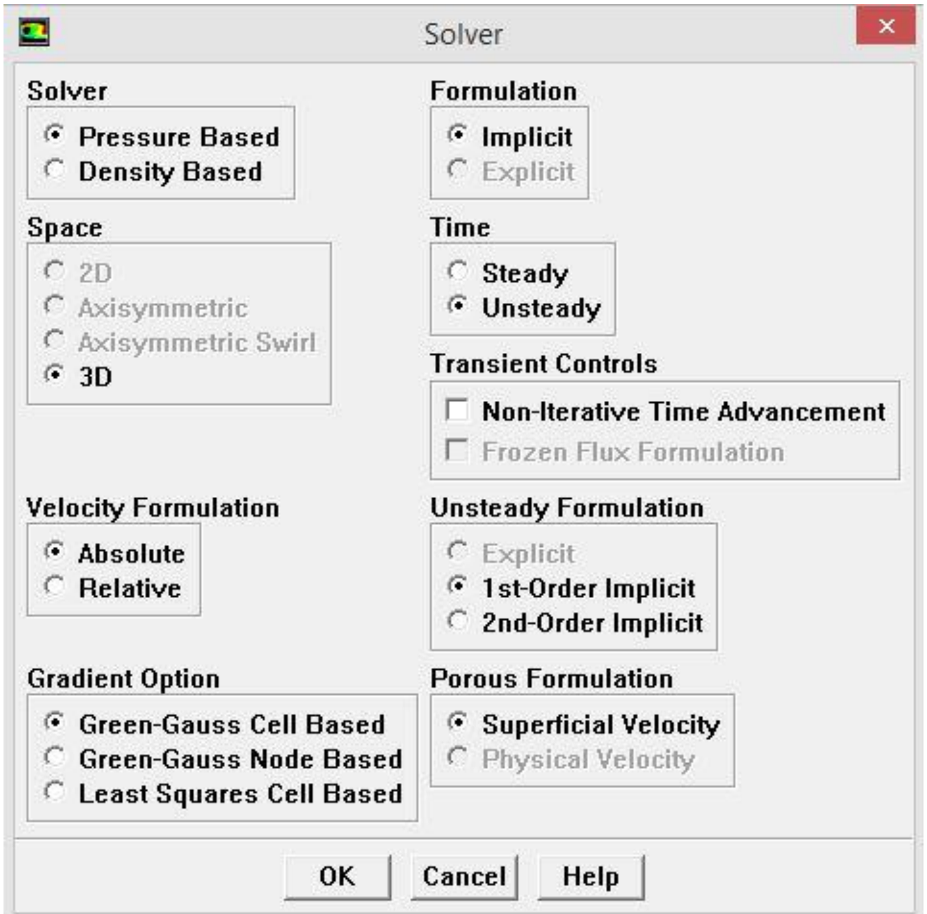

Figure 6.3: Fluent Solver Setting

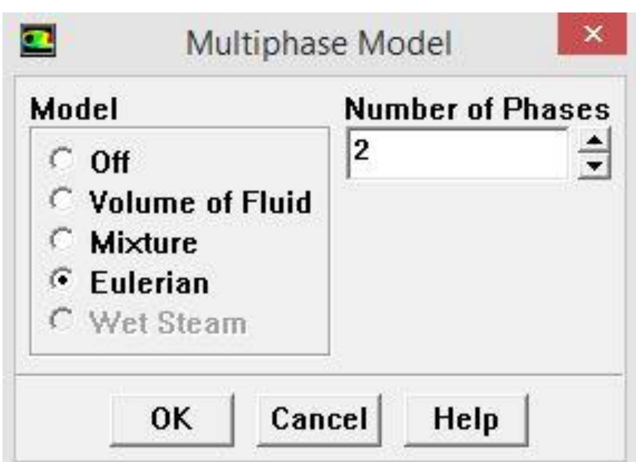

Figure 6.4: Fluent Multiphase Modelling

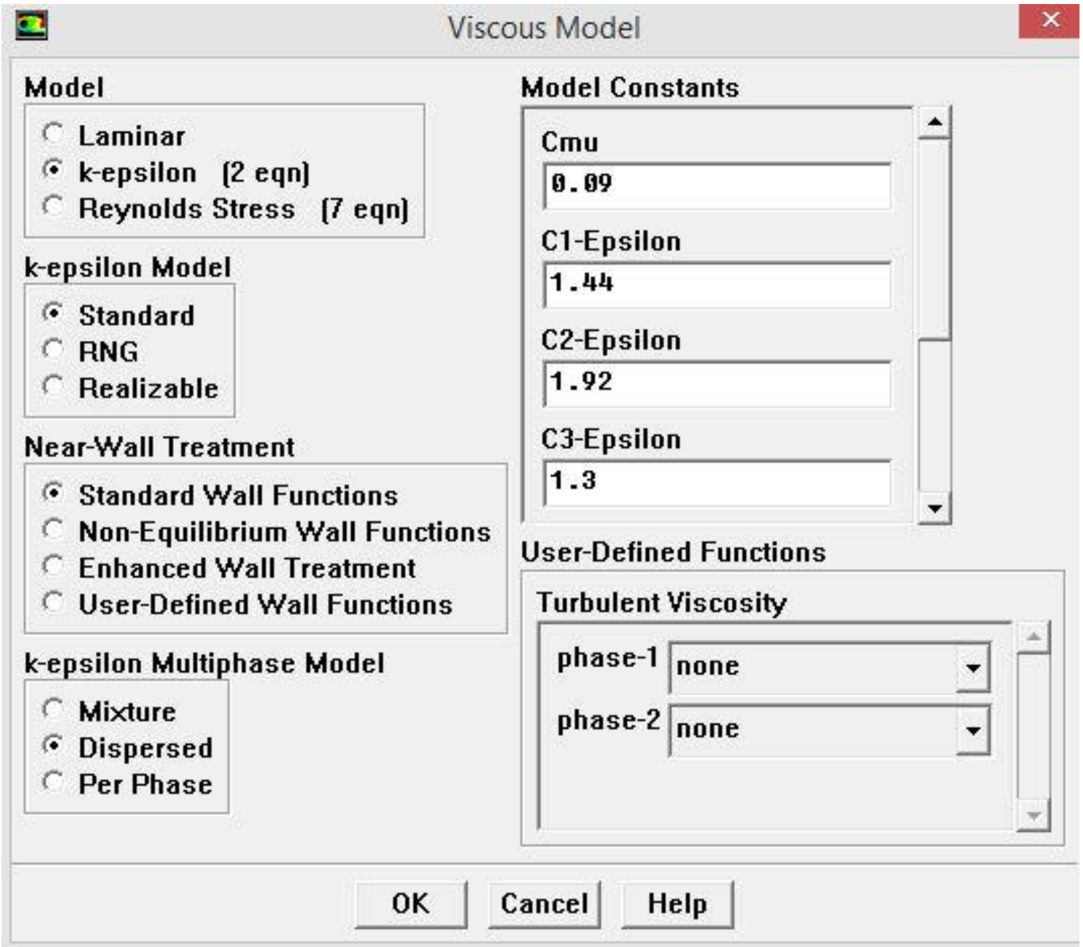

Figure 6.5: Fluent Viscous Modelling

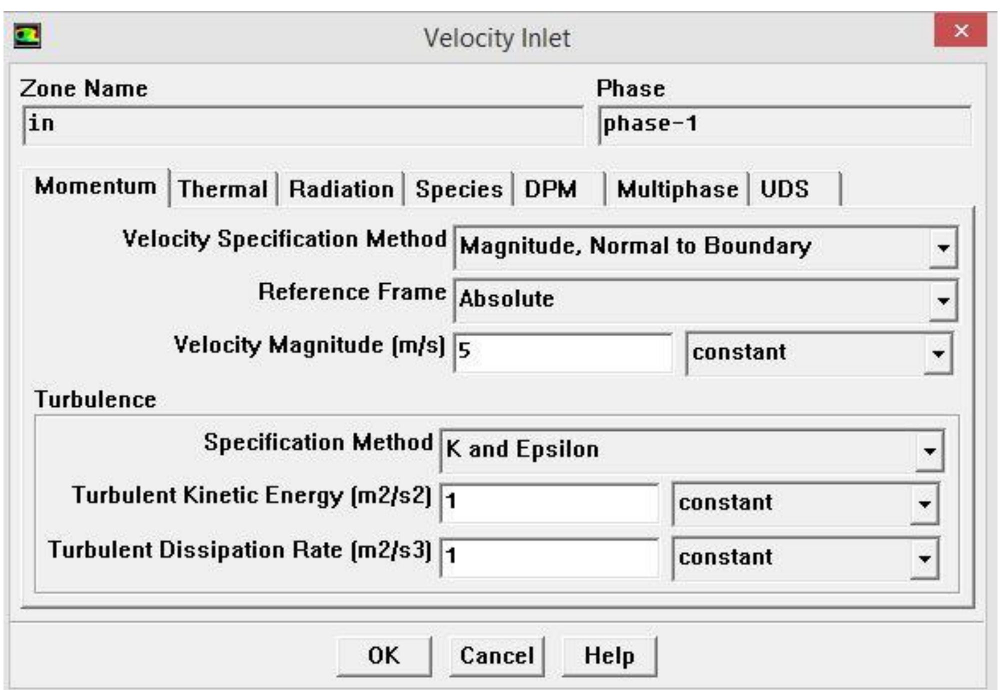

Figure 6.6: Fluent Boundary Condition Setting

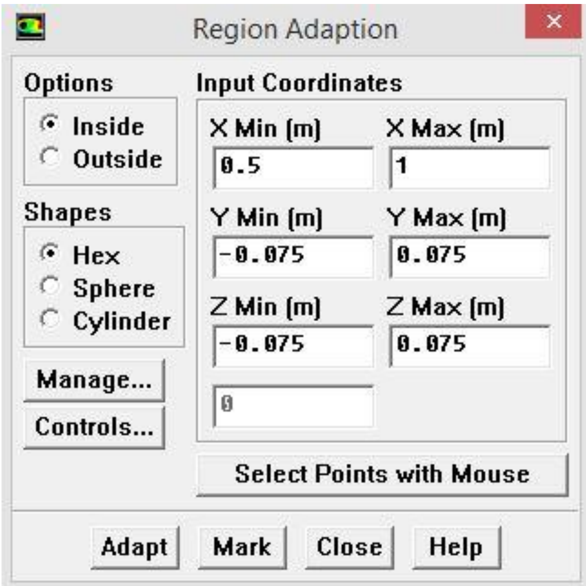

Figure 6.7: Fluent Region Adaption

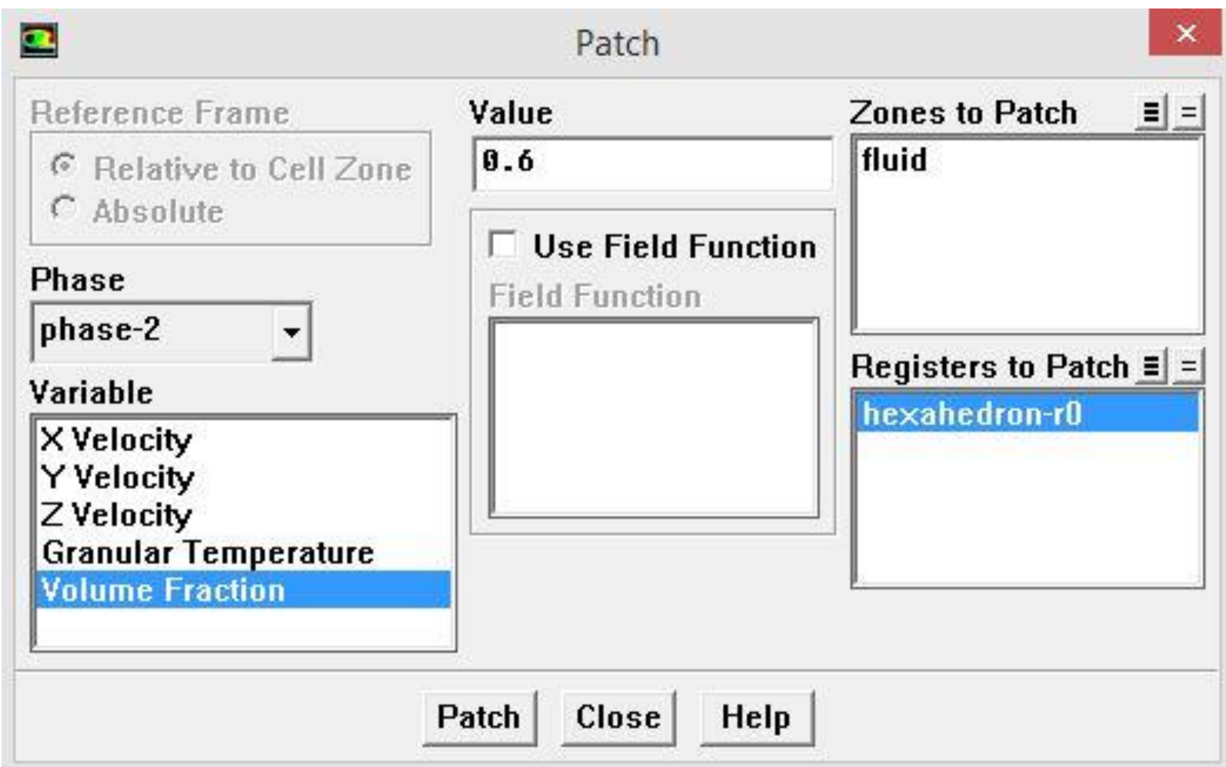

Figure 6.8: Fluent Plug Patching

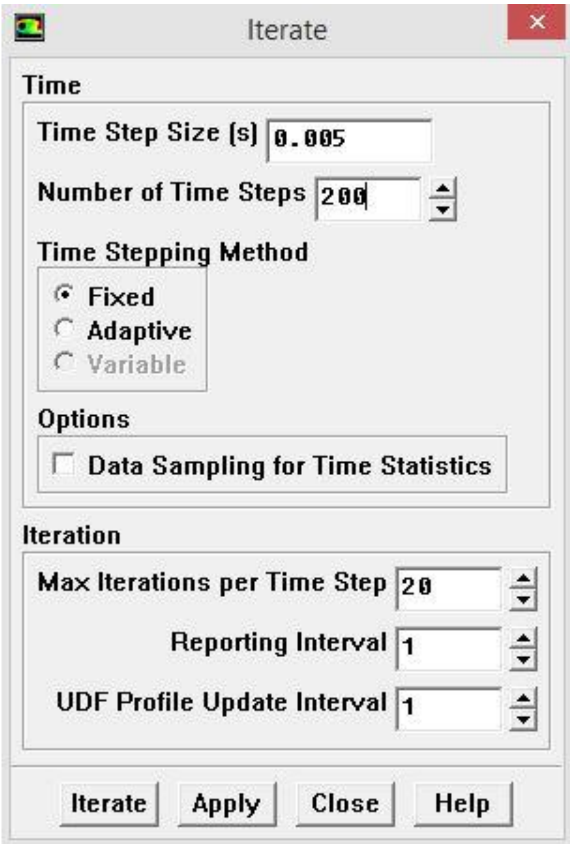

Figure 6.9: Fluent Iteration Setting