TRIBHUVAN UNIVERSITY
INSTITUTE OF ENGINEERING
PULCHOWK CAMPUS
PULCHOWK, LALITPUR

Thesis No: M-108-MSESPM-2015/2019

Multiphase Flow Analysis between Sand and Water in Horizontal Pipe and Bends

by
Rama Sapkota

A FINAL REPORT
SUBMITTED TO THE DEPARTMENT OF MECHANICAL ENGINEERING
IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE
DEGREE OF MASTER OF SCIENCE IN
ENERGY SYSTEM PLANNING AND MANAGEMENT

DEPARTMENT OF MECHANICAL ENGINEERING
LALITPUR, NEPAL

NOVEMBER, 2019

TRIBHUVAN UNIVERSITY

INSTITUTE OF ENGINEERING

PULCHOWK CAMPUS

DEPARTMENT OF MECHANICAL ENGINEERING

The undersigned certify that they have read, and recommended to the Institute of Engineering for acceptance, a thesis entitled **"Multiphase Flow Analysis between Sand and Water inHorizontal Pipe and Bends"** submitted by RamaSapkota in partial fulfilment of the requirements for the degree of Master of Science in Energy System Planning and Management.

…………………………………………………

Supervisor, Dr. Laxman Poudel

Professor

Department of Mechanical Engineering

…………………………………………………

Supervisor, Hari Bahadur Dura

Assistant Professor

Department of Mechanical Engineering

…………………………………………………

External Examiner, Dr. Hari Prasad Neopane

Professor, Department of Mechanical Engineering

Kathmandu University

…………………………………………………

Committee Chairperson, Dr.Nawaraj Bhattarai

Head

Department of Mechanical Engineering

………………………………………..

Date

ABSTRACT

The deposition of impurities in pipelines is very common in various pipeline system existing in different industries, hydropower plants. The study is an approach on developing a model that predicts the sand particle behaviors in flowing water in closed pipelines. This study involves Computational Fluid Dynamics (CFD)analysis and predicted flow behavior in different conditions in a closed pipeline for sand (solid particles) and water(liquid) using Eulerian-Eulerian multiphase flow model. The study was performed for pipe of diameter 0.3m, variable length, volume fraction (15%-35%), particles (0.02mm-0.06mm), velocity (0.5m/s-2.5m/s).The results obtained were validated with the experimental data. Results showed that maximum particle concentration increases with decrease in velocity, increase in particle size and increase in volume fractions. The maximum concentration for inlet mixture velocity of 0.5m/s and 2.5m/s velocity at outlet was found to be 0.626 and 0.515 respectively, and at midplane was found to be 0.629 and 0.498 respectively.The maximum particle concentration for 15%, 25%, and 35% of initial mixture volume fraction was 0.586, 0.612 and 0.628 respectively at outlet and 0.536, 0.587and 0.629 at midplane respectively.Maximum particle concentration for particle size of 0.02mm,0.04mm and 0.06mm are 0.307, 0.612 and 0.629 respectively at outlet and 0.349, 0.629, 0.629 respectively at midplane. Maximum particle concentration for 90° sharp bend, curved bend of mean fillet radius 180mm and curved bend of mean fillet radius 210mm at bottom are 0.576, 0.476, and 0.466 respectively. Change in length of pipe didn't have any effect on concentration at certain point of pipe.

ACKNOWLEDGEMENT

I would like to express my deep and sincere gratitude to my research supervisors,Dr.Laxman Poudel,Professor, Department of Mechanical Engineering, Pulchowk Campus, Tribhuwan University andHari Dura ,AssistantProfessor, Department of Mechanical Engineering, Pulchowk Campus, Tribhuwan University forgivingmetheopportunitytodoresearchandproviding invaluableguidancethroughoutthisresearch.Theirdynamism,vision,sincerityand motivation have deeply inspired me. They have taught me the methodology to carry out the research and to present the research works as clearly as possible. It was a great privilege and honor to work and study under their guidance. Furthermore, I am grateful to program coordinator, Dr. Shree Raj Shakya, Assistant Professor, Department of Mechanical Engineering, Pulchowk Campus and all the faculties of Department of Mechanical Engineering, Pulchowk Campus for providing their immense support whenever needed.

I am extremely grateful to my parents for their love, prayers, caring and sacrifices for educating and preparing me for my future. I am very much thankful tofamilyfortheirlove,understanding,prayersandcontinuing support to complete this research work. Also, I express my thanks to my sisters, brother,sisterinlawandbrotherinlawsfortheirsupportandvaluableprayers.My Special thanks goes to my friend Er. Nipesh Regmifor the keen interest shown to complete this thesissuccessfully.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

D_p	:	Particle Diameter
L	:	Length of Pipe
V	:	Velocity
α_q	:	Volume fraction of phase q
V_q	:	Volume of Phase q
SG	:	Specific Gravity
$\emptyset$	:	Diameter

LIST OF ABBREVIATIONS

ANSYS	:	ANalysis SYStems
CFD	:	Computational Fluid Dynamics
CATIA	:	Computer Aided Three-Dimensional Interactive Application

CHAPTERONE: INTRODUCTION

1.1. Background

In hydropower plants, various industries, phase flow characterization has an essential importance due to its presence during the flow of fluid mixed with particles in various pipelines. Solid particle movement with liquid stream is found in many processes such as lubrication, sedimentation, mixed flow. In these flows, the laden particle affects the flow structure because it indicates the independent motion from the carrier flow. The solid particles and liquid phases are distributed in the pipe in a variety of flow configuration, called flow patterns. The flow pattern prediction is a major problem in two-phase flow analysis. Indeed, main variables like: pressure drop, behaviour of solid particles laden with liquid in pipe, liquid holdup, are strongly dependent on the existing flow pattern. These variables have to be predicted in order to reduce the erosion problems that such parameters could cause. The flow structure becomes complicated due to the interaction between particles and fluid. Understanding the mechanism of sand transport in multiphase flow lines has direct impact on estimation, design and detailed analysis. For instance the increasing amount of sand in horizontal pipelines produces a stationary sand deposit which creates a pressure drop and affects the rate of production (Goharzadeh & Rodgers, 2009). Therefore, basic understandings of particle motion and turbulence modulation in horizontal and vertical pipes are required.

Many experimental and theoretical studies have been carried out for study of particle movement and behavior in fluid. In recent years implementation of various new CFD techniques has allowed successful simulation in studying particle fluid behavior. However liquid and solid particle flow simulation has always been a challenge due to gravity induced particle accumulation on the bottom, re-suspended by the liquid flow like sedimentation(Patro & Patro, 2013).

1.2. Problem Statement

The deposition of impurities in pipelines is very common in various pipeline system existing in different industries, hydrpower plants. Deposition of excessive silt in pipelines damages different structures of industrial plant , hydropower stations causing cooling system, lubrication system failure. The impurities can be of different shape, size density however proposed thesis aims to narrow down the study for the volumetric fraction of spherical solid particles in fluid (range between 10%-35%) and study the particle fluid interaction for various particle size (0.02mm-0.06mm) in closed pipeline where particles properties is used as per sand properties and water is used as liquid. Designin Nepal are generally based on theoretical calculation and the closure insight regarding the behavior interaction between particle and fluid is required for the effective design of various pipeline related systems. Development of real time particle concentration and flow monitoring system for liquid flow is also a part of research in Nepal for sediment handling (Hydrolab, 2005).

1.3. Objectives

1.3.1. Main Objective

The main object of this work is to study and analyse the behaviour of sand in sand-water multiphase flow in closed channel.

1.3.2. Specific Objectives

The main objectives will be accomplished with the following auxiliary objectives:

i. To develop a numerical framework, set up physics for multiphase flow in closed channel and implement the model in ANSYS CFD
ii. To determine particle concentration patterns for different inlet conditions of velocity, particle size, volume fraction in straight and bend pipe

1.4. Assumptions and Limitations

i. The inter collision force between particles are neglected
ii. Sand and water particles properties only be assumed for modelling
iii. Constant temperature throughout the analysis
iv. The diameter of pipe is constant for all cases
v. The shape of particle is assumed to be spherical for all the studies.
vi. The study is confined to closed channel only
vii. CFD analysis will only be used for the analysis of the flow

1.5. Scope of Works

The scope of the study can be stated as follow

i. This method can be used for analysis of solid particle behavior in fluid flow in various pipelines, slurry flows, drainage system designs, pipelines of oil and petroleum and various pipeline system of industries and hydropower plants.It can be also used to quantify and predict the particle-fluid behavior and trends.
ii. This method can be used for determination of optimum velocityof flow in pipe for which the effect of sand particles at the wall shall be minimum for certain volume fraction conditions of flow.

CHAPTERTWO: LITERATURE REVIEW

Sand grains are generally broken rock particles that have been formed by physical weathering, or they are the resistant components of rocks broken down by chemical weathering. Sand grains generally have a rotund shape. According to British Soil Classification System, soils are classified into named Basic Soil Type groups according to size, and the groups further divided into coarse, medium and fine sub-groups(Atkinson, 2000):

Table 1 Sand Categorisation based on British Soil Classification

Very coarse soils	BOULDERS		>200mm
	COBBLES		60-200mm
Coarse soils	G GRAVEL	Coarse	20-60mm
		Medium	6-20mm
		Fine	2-6mm
	S SAND	Coarse	0.6-2.0mm
		Medium	0.2-0.6mm
		Fine	0.06-0.2mm
Fine soils	M SILT	Coarse	0.02-0.06mm
		Medium	0.006-0.02mm
		Fine	0.002-0.006mm
	C CLAY		<0.002

For this study soil particles diameter of 0.02mm-0.06mm is used whereas particles diameter of 0.27mm is used for experimental validation.

A multiphase flow is the flow of a mixture of phases such as gases (bubbles) in a liquid, or liquid (droplets) in gases, and so on. Liquid–solid flows consist of flows in which solid particles are carried by the liquid, and are referred to as slurry flows. Slurry flows cover a wide spectrum of applications that range from the transport of coals and ores to the flow of mud, sedimentations, drainage systems. These flows can also be classified as dispersed phase flows and are the focus of considerable interest in engineering research(Crowe, 2006).

The Eulerian-Eulerian approach models both phases (dispersed and continuous) as separate inter penetrating and interacting fluids in the shared computational domain using modified Navier-Stokes equations. The interaction forces between the phases are simulated as source terms in the equations describing each separate phase. The advantage of this approach is that full-scale process simulations with high solid loadings can be performed, since the computational effort is lower and two-way coupling is relatively easy to implement(Yang Ho Song & Lee, 2018).

2.1. Computational fluid dynamics

Computational fluid dynamics "CFD," is regarded as one of the main tools for the in-depth investigation and understanding of scientific or research problems related to fluid flows. It gives an insight into flow patterns that are difficult, expensive, or impossible to study using traditional (experimental) techniques. Compared to experiments, this technique can provide insights into anything related to fluid flow phenomena, for all desired quantities (eg, pressure, temperature, species concentrations), with high resolution both in time and space, values for any governing variable of the actual fluid domain, and conduct simulations and derive results for virtually any problem and realistic operating/boundary conditions. The main underlying numerical equations, representing any fluid phenomena, is the Navier–Stokes equations (G. Itskos & Grammelis, 2016).Many engineering problems depend on the numerical examination of fluid flow, which typically comprises of more than one phase. Numerous examples of two or more phases flowing simultaneously in a pipeline are encountered in industry. Because of the large differences in the physical properties of the two or more phases involved, such as differences in density and viscosity, one phase will normally flow more quickly in the pipe than the other. Thus when particles are transported in a horizontal pipeline the flow velocity is frequently inadequate to prevent non-uniform vertical solid concentration profile from being formed (Heywood & Cheng, 2002).

2.2. Governing equations for multiphase flow

There are two approaches for simulating these kinds of flows: Eulerian–Lagrangian (E–L) and Eulerian–Eulerian (E–E, or two-fluid). In the E–L approach, numerous discrete particles are tracked, and inter-particle collisions are simulated using either a hard- or soft-sphere model. In the E–E approach, the particles are simulated using a pseudo-fluid model. The advantage of the E–E approach is its ability to simulate large-scale engineering processes with acceptable computation requirements (Zhou L & Zeng, 2013). The most rigorous multiphase flow modeling approach is the two fluid modeling (separation approach).This approach is considered as a mechanistic model which in general, is the most accurate because they introduce models based on the physics of each of the different flow patterns (H.Shi & G.Oddie, 2005).With the Eulerian multiphase model, the number of secondary phases is limited only by memory requirements and convergence behavior. Any number of secondary phases can be modeled, provided that sufficient memory is available.

Many studies used the eulerian-eulerian model for the particle transportation and this study uses an eulerian-eulerian model. The two-phases are handled and analyzed as continuathat are preserved, and the volume occupied by one cannot be occupied by the other. The concept of volume fractions is hence introduced. These volume fractions are assumed as continuous functions of space and time and their sum is always constant. The governing equations for all phases can be obtained using the conservation equation for each (Yang Ho Song & Lee, 2018).The Eulerian-Eulerian model is best suited for high volumefractions of the dispersed phase which is averaged over eachcontrol volume. Each phase is governed by similar conservationequations and modelling is needed for interactionbetween the phases, turbulent dispersion of particles, andcollision of particle with walls.(Ofei & Ismail, 2016)

2.3. Equations in General Form

2.3.1. Volume Fraction Equation

The description of multiphase flow as interpenetrating continua incorporates the concept of phasic volume fractions, denoted here by α_q Volume fractions represent the space occupied by each phase,and the laws of conservation of mass and momentum are satisfied by each phase individually.

The volume of phase q, V_q is defined by

$$V_q = \int_V \alpha_q \, dV$$

Where

$$\sum_{q=1}^{n} \alpha_q = 1$$

The effective density of phase is

$$\hat{\rho}_q = \alpha_q \rho_q$$

where ρ_q is the physical density of phase q.

2.3.2. Conservation of Mass

The continuity equation for phase q is

$$\frac{\partial}{\partial t}\left(\alpha_q \rho_q\right) + \nabla.\left(\alpha_q \rho_q \vec{v}_q\right) = \sum_{p=1}^{n}(\dot{m}_{pq} - \dot{m}_{qp}) + S_q$$

where $\vec{v}_q$ is the velocity of phase q and $\dot{m}_{pq}$ characterizes the mass transfer from the p^{th} to q^{th} phase, and $\dot{m}_{pq}$ characterizes the mass transfer from phase q to phase p .

2.3.3. Conservation of Momentum

The momentum balance for phase q

$$\frac{\partial}{\partial t}\left(\alpha_q \rho_q \overrightarrow{v_q}\right) + \nabla.\left(\alpha_q \rho_q \overrightarrow{v_q}\overrightarrow{v_q}\right)$$

$$= -\alpha_q \nabla_p + \nabla.\bar{\bar{\tau}}_q + \alpha_q \rho_q \vec{g} + \sum_{P=1}^{n}(\vec{R}_{pq} + \dot{m}_{pq}\vec{v}_{pq}$$

$$- \dot{m}_{pq}\vec{v}_{qp}) + (\vec{F}_q + \vec{F}_{lift,q} + \vec{F}_{wl,q} + \vec{F}_{vm,q} + \vec{F}_{td,q}$$

where $\bar{\bar{\tau}}_q$ is the q^{th} phase stress-strain tensor and μ_q and λ_q viscosity of are the shear and bulk phase q, $\vec{F}_q$ is an external body force, $\vec{F}_{lift,q}$ is a lift force, $\vec{F}_{wl,q}$ is a wall lubrication force $\vec{F}_{vm,q}$ is a virtual mass force, and $\vec{F}_{td,q}$,is a turbulent dispersion force (in the case of turbulent flows only). R_{pq} , is an interaction force between phases, and p is the pressure shared by all phases.

2.4. Related Works

Oshinowo & Bakker (2002)studied the distribution of solids in stirred tanks under a range of solids loadings (0.5 to 50 % by volume) .Results were predicted using CFD and validated against experimental data obtained from the literature. The multiphase flow was modeled using the Eulerian Granular Multiphase model. This paper also reviewed the established design parameter in the context of scale-up and compare it to the quality of solids dispersion as a means of assessing correct scale-up in suspension tank design. The results of this study described a straightforward procedure to obtaining comprehensive information about reactor behavior with complex CFD models.

A. Ekambara & H.Masliyah (2009)used CFD simulation to investigatethe effect of in situ solidsvolume concentration, particlesize, mixture velocity, and particle diameter on local timeaveragedsolids concentration profiles, particle and liquidvelocity profiles, and frictional pressure loss.. The study revealed thatParticleswere asymmetrically distributed in the vertical planewith the degree of asymmetryincreasing with increasingparticle size.

G. Micale & Godfrey (2000)simulated the particle concentration distribution in two-phase stirred tanks on the basis of information on the three-dimensional flow field as obtained by numerical solution of the flow equations (CFD) using the well-known k-ϵ turbulence model. Two modelling approaches (k-ϵ turbulence and k-ω turbulence model)were attempted. The comparison of experimental data with simulation results was satisfactory with both simulation approaches. Differences between the two approaches concerning their accuracy and computational effort were discussed. The need to make a suitable estimate of the particle drag coeffcients in turbulent fluid media was emphasized.

Ajay K Yerrumshetty & James (2009) reported the results of applying a two-fluid model developed for dilute turbulent gas-solid flow to the case of dense liquid-solid flow of relatively coarse particles in a horizontal channel. Many features of the two-fluid model developed for gas-particle flows were relevant to liquid solid flows. He recommended on further development in the model for the solids volume fraction, which must be capable of handling flow regions where the particle concentrations are sufficiently high to strongly inhibit the fluid motion and significantly suppress the fluid turbulence. The dramatic increase in mean solids concentration with the depth was observed.

Patro & Patro (2013)simulated fully developed gas–solid flow in a horizontal pipe using the two fluid model .The solid phase stresses were modeled using kinetic theory of granular flow (KTGF). The computed results for velocity profiles and pressure drop were compared with the experimental data. The particle-wall collision and lift was considered in modeling .Paper also presented the effect of flow parameters like gas velocity, particle properties and particle loading on pressure drop prediction in different pipe diameters. Pressure drop increases with gas velocity and particle loading .With respect to particle diameter, pressure drop first increased, reaches a peak and then decreased.

Tamer Nabil & El-Nahhas (2013)developed a generalized slurry flow model using the computational fluid dynamics simulation technique (CFD) to have better insight about the complexity of slurry flow in pipelines. The model was utilized to predict concentration profile, velocity profile and their effect on pressure drop taking the effect of particle size into consideration. The two-fluid model based on the Eulerian-Eulerian approach along with a standard k-ε turbulence model with mixture properties was used, whereby both the liquid and solid phases were considered as continua. The computational model was mapped on to a commercial (CFD) solver FLUENT 6.3 and compared with experimental data.

Boris V Balakin (2010)focused on computational study of the process of sedimentation of spherical particles in suspensions with high particle concentrations with the two-fluid Eulerian approach. Convectional flow patterns were found and studied during the simulations. The presence of these patterns, which are also observed experimentally, makes the sedimentation process dependent on the rheological behavior of the suspension. The results of the simulation were validated with experimental results. The present paper showed that Eulerian-Eulerian simulations can account for some of the detailed processes taking place in a settling suspension of particles.

M.A. Delele & Mellmann (2016) analyzed the solid and fluid flow behaviors inside a rotary drum using computational fluid dynamics (CFD). The CFD model developed was based on an Eulerian–Eulerian multiphase flow approach. The capability of the multiphase CFD model to predict the transverse and axial solid flow patterns, the fluid flow profile, and particle residence time was assessed. The experiments were conducted on pilot scale rotary drums. As could be proved by measured and simulated results, the particle flow near the bed surface exerted a strong entrainment effect on the transverse air flow in the proximity of the bed surface. The study demonstrated the capability of the multiphase CFD model to predict the particle and fluid flow behaviors simultaneously. The results confirmed the capability of the multiphase CFD model to study solid and fluid flow characteristics in rotary drum systems. The paper also recommended improvement in the accuracy of the model is needed, particularly for predicting the residence time of the particles.

T.N & A.Y (2016) studied a computational fluid dynamics (CFD) simulation which adopted the inhomogeneous Eulerian-Eulerian two-fluid model in ANSYS CFX-15 to examine the influence of particle size (90 μm to 270 μm) and in situ particle volume fraction (10% to 40%) on the radial distribution of particle concentration and velocity and frictional pressure loss. For a constant particle volume fraction, the radial distribution of particle concentration increased with increasing particle size, where high concentration of particles occurred at the bottom of the pipe. Particles of size 90 μm were nearly buoyant especially for high particle volume fraction of 40%. The CFD study shows that knowledge of the variation of these parameters with pipe position is very crucial if the understanding of pipeline wear, particle attrition, or agglomeration is to be advanced.

Messa & Malavasi (2015)proposed a new two-fluid modelfor the simulation of fully suspended liquid-solid slurry flowsin horizontal pipes. The model is claimed toaddress wallboundary conditions for solid phase, viscosity of the slurrymixture which incorporates particle shape, and a solutionalgorithmwhich reducescomputational burden.The authors have emphasised that the new model increased the accuracyof the pressure gradient predictions without affecting themodel’s capability in reproducing other engineering featuressuch as solid volume fraction and velocity distributions.

L. Ma & Xie (2015)conducted a CFD study for calculating erodentparticle trajectories in slurry flow. It involves the capturingof the movement of erodent particles using the discretephase method (DPM) and calculating the interfaces betweenfluid phase and gas phase using the volume of fluid (VOF)method. The authors revealed that their model results werein reasonable agreement with experimental observations interms of normal impact velocity on the specimen surface.

S. A.Miedema (2016)proposed a framework for predicting head loss and limit deposit velocity in slurry flow. The framework is based on constant spatial volumetric concentration curves and uniform sand or gravels for five flow regimes in Newtonian fluid, namely, the stationary or fixed bed regime, the sliding bed regime, the heterogeneous regime, the homogeneous regime, and the sliding flow regime. The author concluded that the new framework explained the behaviourof very small particles in terms of the mobilisation of thelubrication effect of the particle poor viscous sublayer.

CHAPTERTHREE: RESEARCH METHODOLOGY

This chapter presents an insight methodology based on which research was carried out.

3.1. Research Framework

In order to meet the objectives of the study, research framework as shown in Figure 1 was implemented.The whole methodology been divided into four major stages; literature review, CFD analysis, result tabulation and analysis, and validations of result.

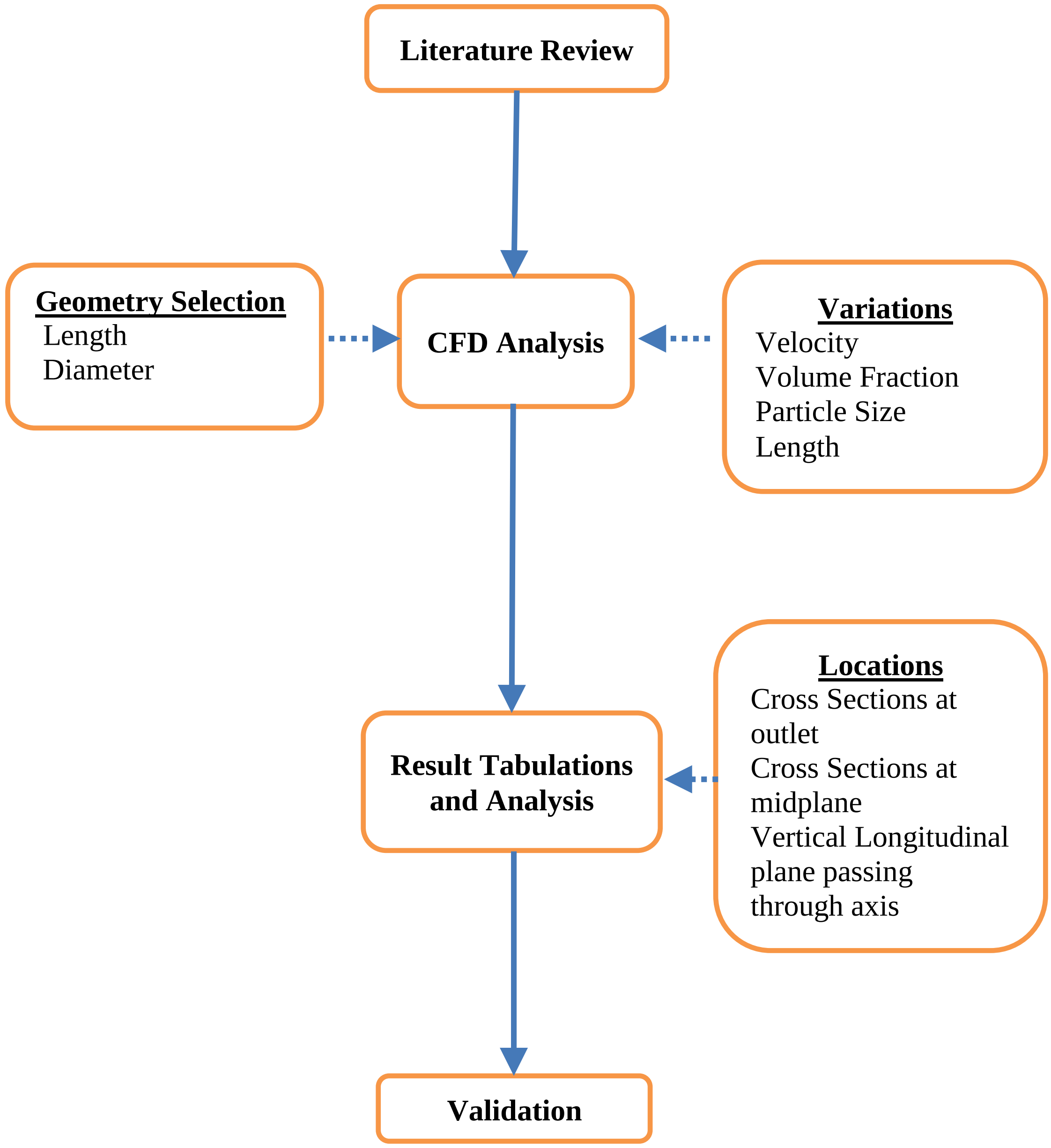


Figure 1 Research Methodology Chart

3.1.1. Literature Review

The research was started with the literature review of various related papers and documents. The various case studies related to multiphase flow were studies and the various experimental papers were studied followed by the geometry set up and case studies. The study was focused for various velocities, volume fractions, particle size and length of straight pipe and results were focused at outlet regions, midplane regions and longitudinal planes of the geometry.

3.1.2. CFD Analysis

Computational fluid dynamics is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process

- Analysis begins with a mathematical model of a physical problem.
- Conservation of mass, momentum and energy must be satisfied throughout the region of interest.
- Fluid properties are modelled empirically.
- Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, inviscid, two-dimensional).
- Provide appropriate initial and boundary conditions for the problem.
- CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanics in the fluid region of interest.
- The solution is post-processed to extract quantities of interest (e.g. lift, drag, torque, heat transfer, separation, pressure loss, etc.).
- For the entire CFD analysis between water and sand(silt) particle in closed pipe horizontal and perpendicular pipe of different dimensions is modelled and simulations were performed.
- Horizontal and perpendicular pipe of 0.3m diameter was used for the study for different velocity, volume fraction, length and particle size. The studies were carried out for velocities of 0.5m/s, 1m/s, 1.5m/s, 2m/s, 2.5m/s; Volume fraction of 15%, 25%, 35%; Particle size of 0.02mm, 0.04mm, 0.06mm and

length of 6m, 8m, 10m, 12m for horizontal pipes. For perpendicular pipes 90° sharp bends, curved bends with mean fillet radius of 180mm and 210mm were used.

3.1.3. Result Tabulation and Analysis

CFD analysis is followed by analysis of the results with different methods like contour plots, vector plot, streamlines, data curve etc. CFD POST 18.1 was used for contour Plots, streamlines, and exporting the data to Microsoft Excel for appropriate graphical representations and data analysis and report generation. The result obtained from CFD analysis of horizontal and perpendicular pipes were first studied in CFD POST 18.1. The analysis was done at bends for perpendicular pipes whereas at midplane, outlet and transverse vertical region of horizontal pipes. The variation in particle concentration after the simulation for different input condition, pattern of solid particles deposition in the bottom of pipes were the focus of the study. The obtained results for different input parameters and conditions were arranged in the form of data and contour plots for the further analysis.

3.1.4. Validation

Prior to every study based on CFD analysis validation of model is necessary for the determining accuracy and reliability of the model. Here the model was validated against the experimental data obtained from (Randall G. Gillies & Xu, 2004)which is shown in Figure 8. The computed result from model showed good agreement with the experimental study conducted. Details of the study is explained in 3.2.5 section of this report.

3.2. CFD Model Analysis

CFD analysis were performed for various cases of straight pipes, perpendicular pipes of different bend conditions for various cases of velocity, volume fraction, size of particles and length of pipe. After modelling the geometryand meshing of pipe ,different boundary conditions were specified.Water was taken as primary phase and sand particles as secondary phase where properties of water and sand particles (densities, size of particles) were defined. Eulerian- Eulerian method with RNG $k-\epsilon$ model was used. The methodology adopted for the CFD analysis of 3d pipe and bends is shown in Figure 2.

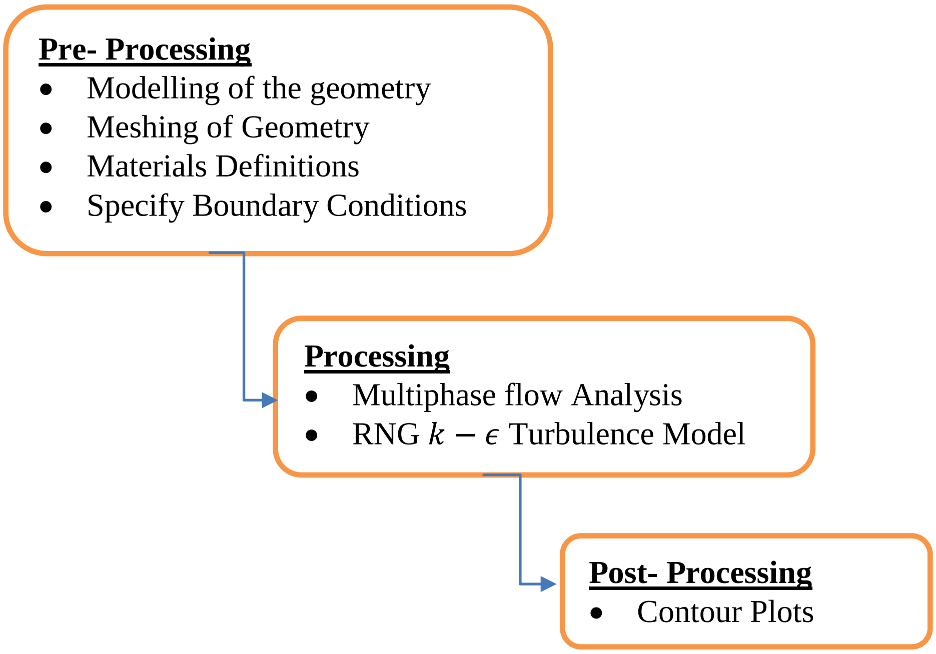


Figure 2 Methodology for CFD Analysis of Particle Behaviour

The analysis was done in three steps.

A. Pre-Processing

- Horizontal and perpendicular pipes of 0.3m diameter was modelled using ANSYS WORKBENCH 18.1 and CATIA V5R20.
- Geometry Modelling was flollowed by meshing and defining the boundary of inlet, walls and outlet in the pipe.
- Water and sand particles were defined for Primary and Secondary Phases.
- Various boundary conditions were specified.

B. Processing

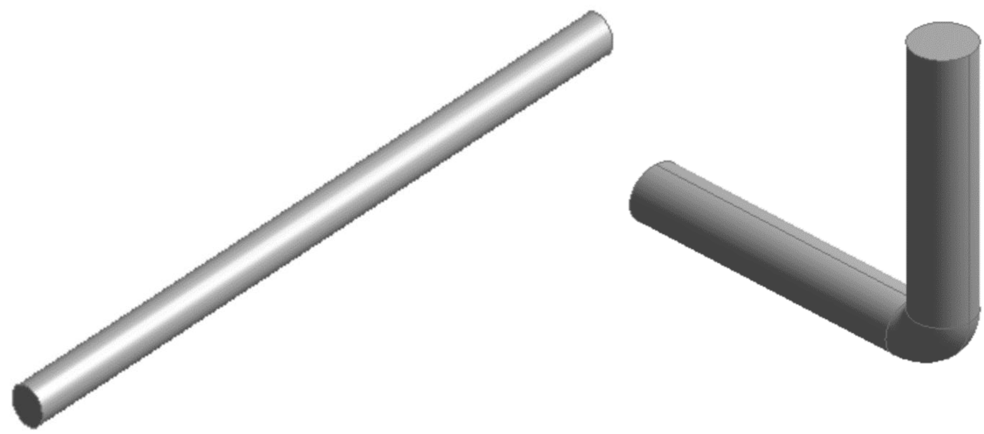
- Eulerian Method with RNG $k - \epsilon$ turbulence model was used .

C. Post-Processing

- Contour Plots obtained from all the simulation cases were used for study of particle deposition pattern.

3.2.1. Modelling Geometry

The three-dimensional (3D) horizontal pipe $(\emptyset 0.3m\,, L = 6m)$ geometry was modelled for analysis for variable velocity, inlet mixture particle volume fraction, and variable particle diameter whereas horizontal pipe$(\emptyset 0.3m\,, L_1 = 6m, L_2 = 8m, L_3 = 10m, L_4 = 12m)$ geometry was modelled for analysis of particle concentration at 4m distance from inlet for various length pipe using ANSYS 18.1 Workbench Design Modeller.The three-dimensional (3D) perpendicular pipe of $(\emptyset 0.3m\,, L_1 = 1.5, L_2 = 1.5)$ with sharp and curved bend structure was modelled using CATIA V5R20.Figure3presents the meshed 3D geometry of both pipe.

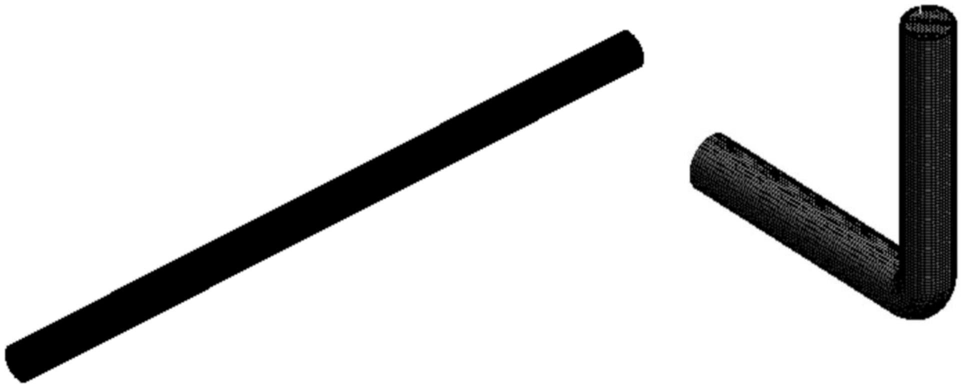


a) Straight Pipe b) Bend Pipe

Figure3 Isometric view of pipe

3.2.2. Mesh Generation

Both horizontal and perpendicular 3D pipe of different geometry was meshed into small grid cell using using unstructured assembly cutcell mesh type with 10 inflation layers created at the boundary of the pipe to accurately capture the floweffect in that region. The number of elements were optimised after mesh independence analysisstudyuntil the results were no affected. From the mesh independence analysis study the optimised number of elements used was 7,10,362.Figure 4presents the meshed 3D geometry for horizontal and perpendicular pipe.



a)Straight Pipe b) Bend Pipe

Figure 4 Meshing of Pipes

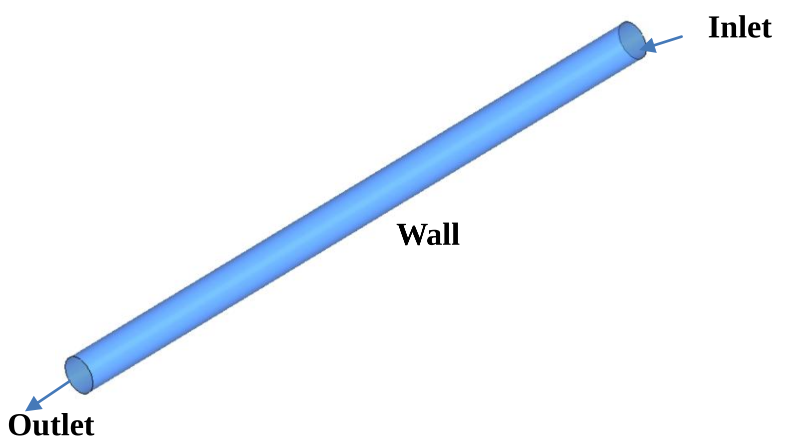
The details for meshing of geometry is mentioned in Table 2.

Table 2 Meshing Details of Pipe

Sizing	
Size function	Curvature
Relevance center	Fine
Curvature Normal Angle	Default (18.0°)
Min Size	8.e-005m
Max Tet Size	1.024e-002m
Growth Rate	1.15
Minimim Edge Length	0.942480m
Quality	
Smoothing	Medium
Mesh Metric	Orthogonal Quality
Min	0.78436
Max	1.
Average	0.99241
Standard Deviation	2.55e-002
Inflation	
Inflation Options	Smooth transition
Transition Ratio	0.272
Maximum Layers	10
Growth Rate	1.2

3.2.3. Physics Setup and Boundary Conditions

At the inlet of the horizontal pipe, mixture velocity and volume fraction of both liquid and particles phases were specified. Mixture of different volume fractions and different velocities were imosed . At the outlet, static pressure was specified. At the wall, no-slip condition was imposed. To initiate the numerical solution, average volume fractions andinitial mixture velocitywere specified as initial conditions the initial velocity for both liquid and solid particles was specied as same. The detailed



boundry condition specified for the study in horizontal is shown in theFigure 5.

Figure 5 Boundary for Horizontal Pipe

The boundary conditions applied for the particle flow in water for the horizontal pipe is mentioned in Table 3.

Table 3 Boundary Conditions for Horizontal Pipe

Locations	Boundary Conditions	Remarks
Inlet	Speed	Variable velocity $\alpha_{sand} = Variable\ (between\ 10\% - 35\%)$
Wall	No Slip	Smooth
Outlet	Pressure outlet	Average Static Pressure with Relative Pressure of 0

The boundry condition specified for the study of perpendicular pipe is shown in Figure 6.At the inlet of the perpendicular pipe, mixture velocity of 0.5m/s and volume fraction of 25% solid particles were specified.At the outlet, static pressure was specified. At the wall, no-slip condition was imposed.

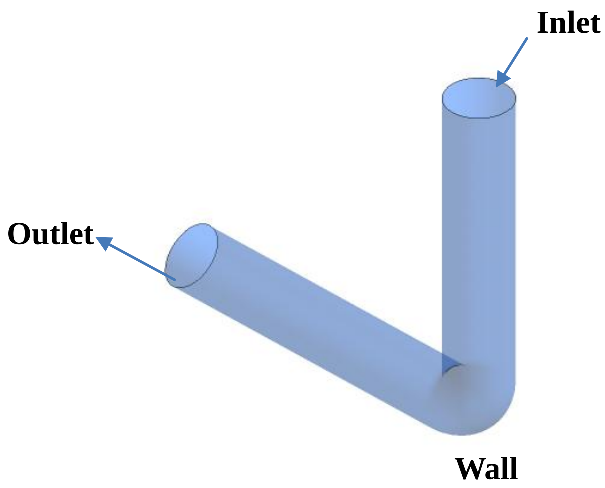


Figure 6Boundary Conditions for Curved Bend Pipe

The boundary condition in detail for perpendiculat pipe is mentioned in Table 4.

Table 4 Boundary Condition for Pipe Bend

Locations	Boundary Conditions	Remarks
Inlet	Speed	0.5m/s
Wall	No Slip	Smooth
Outlet	Pressure outlet	Average Static Pressure with Relative Pressure of 0

The overall simulation parameters used for the entire study is tabulated in Table 5.

Table 5 Simulation Data

Simulation Parameters	**Values taken for the study**
Pipe Diameter	300mm
Pipe Length	6m, 8m, 10m, 12m
Size of Particle	0.02mm, 0.04mm, 0.06mm
Length of pipes	6m, 8m, 10m, 12m
Particle volume fraction	15%, 25%, 35%
Density of sand	2650 kg/m^3
Density of water	998 kg/m^3
Velocity of the mixture	0.5 m/s, 1m/s, 1.5m/s, 2m/s, 2.5m/s
Turbulence equation	$k - \epsilon$ model

3.2.4. Solver

In this transient simulation particle water slurry study, the Navier-Stokes governing equations together with their closure terms were solved using ANSYS FLUENT 18.1 solver. The mass and momentum equations were discretised using the control volume technique. The firstorder implicit method was adopted for time discretisation, whereas the second-order implicit method was also adopted for space in solving the conservation law equations. The SIMPLE algorithm was utilised to solve the pressure-velocity coupling in the momentum equations,. The solution was assumed to be converged when the root mean square (RMS) of the normalised residual error reached 10−4 for all simulations. The detail solution method is presented in table.

Table 6 Solution Method

Pressure -Velocity Coupling	
Scheme	Phase Coupled SIMPLE
Spatial Discretization	
Gradient	Least Squares Cell Based
Momentum	Second Order Upwind
Volume Fraction	First Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Turbulent Dissipation Rate	Second Order Upwind

3.2.5. Model Validation

The experimental data was obtained from the research paper based on experimental results obtained by(Randall G. Gillies & Xu, 2004). Experiments were conducted in closed loop of pipe and velocity of the mixture was maintained to 5.4m/s using a pump. The particles propertiesemployed were as follows: size,dp = 270 μm; specific gravity, SG = 2.65; and in situparticle volume fraction of 0.1, 0.2, 0.3, 0.36 and 0.45. The mixture velocityused is 5.4m/s. Gamma ray gauze was used to determine the total in-situ particle concentrations Simulation were carried out by modelling the geometry as per experimental data to investigate the accuracy of models inpredicting the experimental particleconcentration profiledata. However, for the validation of model data of the volume fraction 30% and 45% was used and simulations were carried out for same geometry, inlet velocity of 5.4m/s, particle size of 0.27mm as used in experimental set up and volume fractions of 30% and 45% respectively.

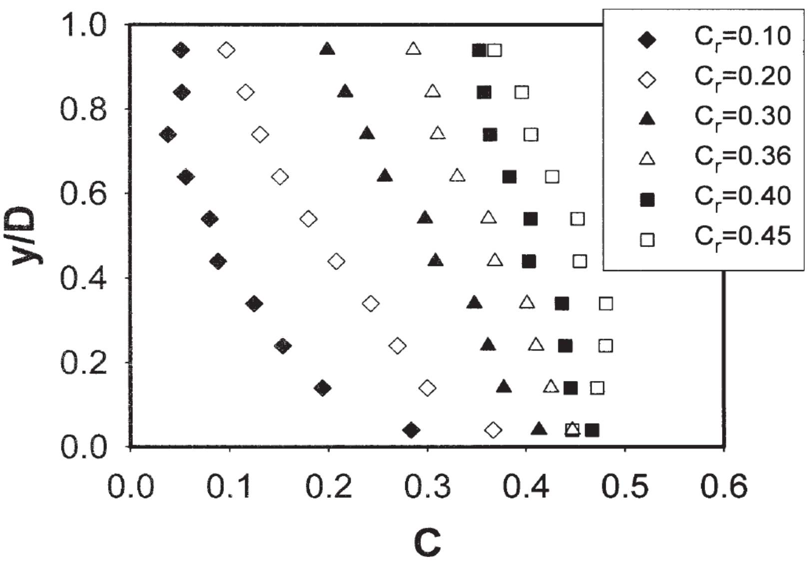


Figure 7 Experimental Data(Randall G. Gillies & Xu, 2004)

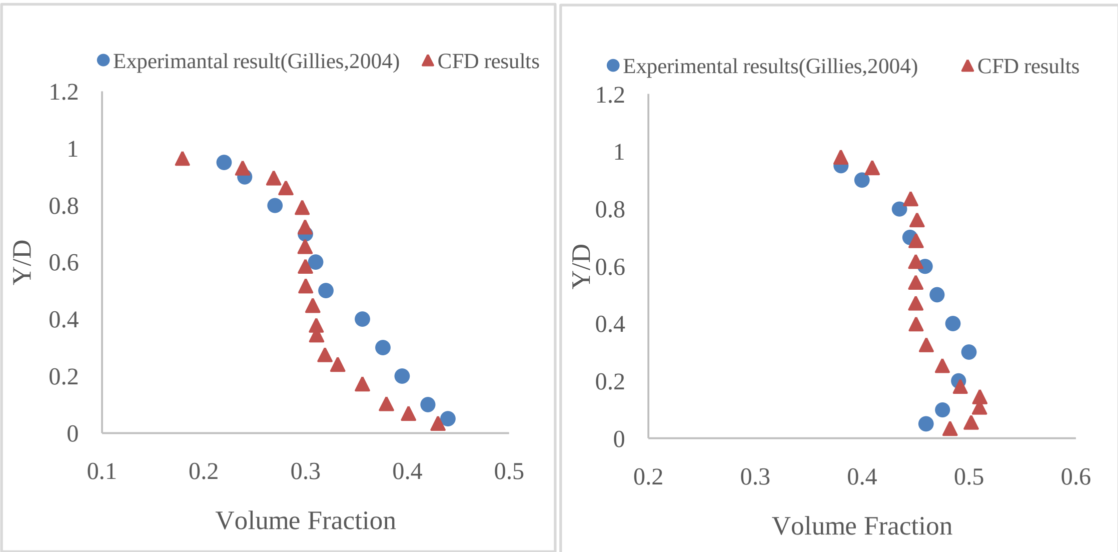


Figure 8 Comparison between Experimental Results and CFD Analysis Results for Volume fraction 0.3 and 0.45

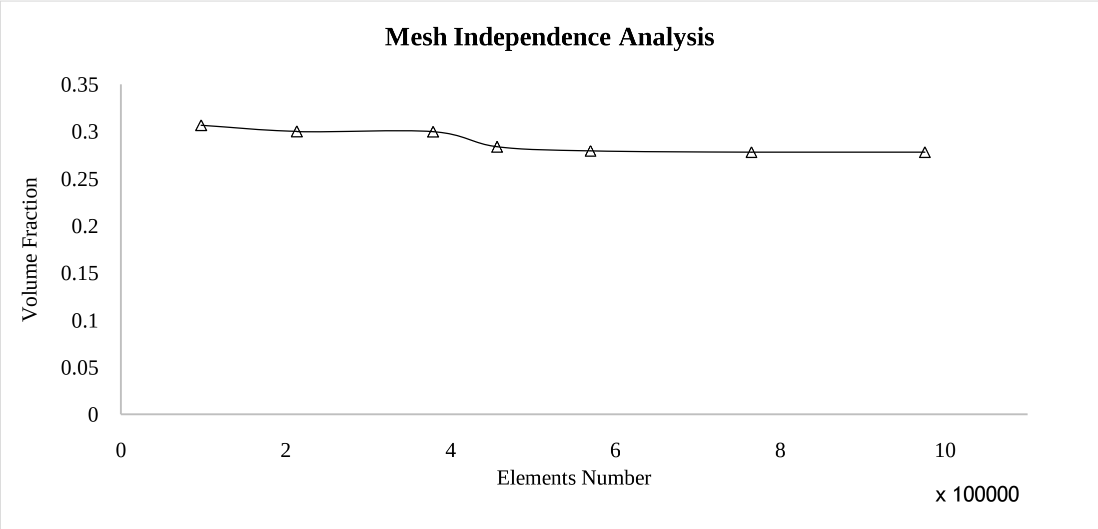
The results obtained after simulations were compared to the experimental data for the volume fraction of 0.3 and 0.45 respectively which is shown in Figure 8. Despite errors in results the trend of the experimental data was achieved in the CFD results. Surprisingly, the slight decrease in particle concentration at bottom region for initial volume fraction of 0.45 was also predicted by the model which is not seen in case of volume fraction 0.3. Hence it can be concluded that the results obtained from the model developed can be used for liquid-solid particle flow in a pipeline.

3.2.6. Mesh Independence Study

To establish the accuracy of the CFD solution, the pipe flow was analysed using the standard k-ε model, and RNG model, at uniform V_{in} = 0.5m/s, and volume fraction 25%. The grid convergence study was performed by developing three different meshes: with a coarse, medium, and fine grid for all seven different meshes of the pipe to predict the flow on normalised mesh cells to determine how the mesh quality affects CFD simulation results. The number of elements and the simulation time for the seven cases simulated using the RNG k-ϵ model are highlighted in Table 2, summarise the key characteristics of the meshes, and it is very clear that CFD simulation results for particle concentration at point is highly dependent on the number of mesh element considered.

Meshes of different sizes with the number of elements varying from 98552 to 871725 at convergence criteria of 10^{-4}were used for study. The graph was plotted between number of elements and volume fraction at point(0,-0.13,3) of pipe as shown in Figure 9.

Figure 9 Mesh Independence Analysis



As per results from above seven cases shown in Figure 9, change in volume fraction of sand particles between 600000 - 971724 was found negligible. Hence total 710362 elements was used for all subsequent computations.

3.3. Case Studies

Horizontal pipe of dimension (Dia=0.3m, L=6m) was used for multiphase flow analysis involving variation in particle size, volume fractions and velocity whereas for the case study involving impact of change in length at particle concentration the length was varied. Three different geometry of perpendicular pipe with different fillet radius was used as case studies for the analysis and prediction of the flow behaviour in bends, impact of bend surface over the settlement and flow.

3.3.1. Effect of Length

Volume fraction along vertical axis at distance 4 m from inlet was calculated for Horizontal pipe of different length ($L_{1=}$6m, $L_{2=}$8m, L_3=10m, L_4=12m) with constant diameter of 0.3m. The main purpose of this calculation was to analyse the effect of length of pipe on volumetric fractions at certain distance from inlet. The velocity of inlet mixture was taken 0.75m/s and 0.04mm diameter of sand particle was taken for the calculations. The volume fractions of sand particles in mixture for the study was taken as 0.25.

3.3.2. Effect of Velocity

Various simulations were carried out at inlet velocity 0.5m/s, 0.75m/s, 1m/s, 1.5m/s, 2m/s, 2.5m/s. All the simulation were carried out for the particle size of diameter 0.04mm and volume fraction of 0.25 for variable velocity. The geometry of pipe for the study was (Length, L=6m and Diameter,Dia=0.3m). The purpose this calculation was to study effect of variable velocity in sand particle distribution and behaviour in flow for different velocity.

3.3.3. Effect of Particle Size

Various simulations were carried out for particle diameter $d_1 = 0.02mm, d_2 = 0.04mm, d_3 = 0.06mm$at inlet velocity 0.5m/s. Initial volume fraction for all the simulation cases were 25%.The diameter of pipe taken for the study was 0.3m and length L=6m for all three cases. The main purpose of this calculation was to analyse the settlement of particle along the pipe and impact of different particle size on volumetricconcentration along the pipe.

3.3.4. Effect of Volume fraction

The calculation were also performed at volumetric fractions of 15%, 25%, 35%. The main purpose of this calculation was to analyse the settlement of particle along the pipe and impact of volume fraction on volumetric deposition of sand particle in bottom along the pipe for different initial volume fractions. All the simulations were carried out at constant particle diameter of $d_1 = 0.04mm$ and velocity of 0.5m/s with length of the pipe $l = 06m$ and $d_{pipe} = 0.3m$.

3.3.5. Effect of bends

The calculation was also performed at perpendicular bends, and curved bends for 180mm and 210mm mean fillet radius. The geometry for bends was created in CATIA V5R20. Pipe with perpendicular bend was modelled in ANSYS Design Module. The main purpose for this study was to analyse the impact of curved surface of pipe over corner surfaced pipe in settlement of sand particles in bottom of horizontal pipe surfaces. Three different simulations were carried out for the pipe $d_{pipe} = 0.3m$horizontal and vertical length 1.5m with. All the simulations were performed at $d_p = 0.04mm$,velocity 0.5m/s and Volume fraction 30%.

4.1. Results

This section includes results obtained by CFD analysis for various case study as mentioned in 3.4 Case Studies section. The results obtained are first studied and analysed in CFD POST 18.1 through contour plots of volume fractions in all cases. Various results obtained from CFD analysis are discussed below.

4.1.1. For different length

The contour plots at (l=4m from inlet) and vertical plane for all four cases are shown below in Figure 10and Figure 11.

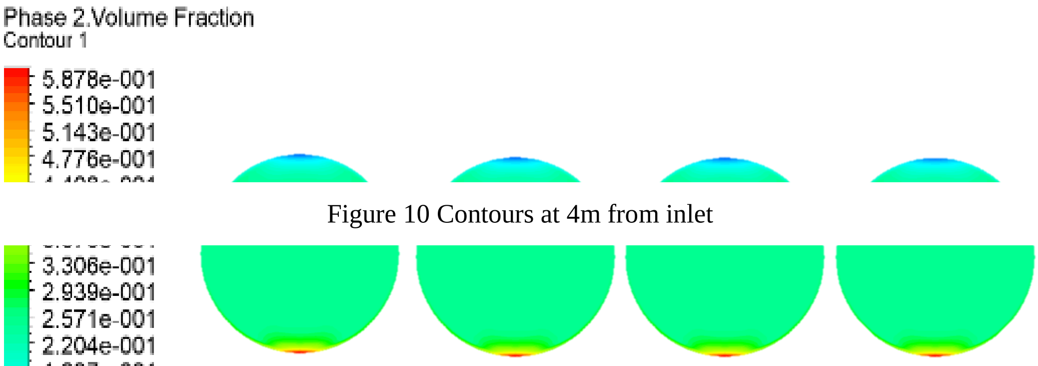


Figure 10 Contours at 4m from inlet

The contours at 4m cross section from inlet was identical for all cases. The patterns of particle concentration for all four cases are also identical at all the sections. Hence the results suggestthat the particle concentration at the certain point from the inlet is unaffected for different length pipe provided the same condition of velocity, particle size, initial volume fraction.

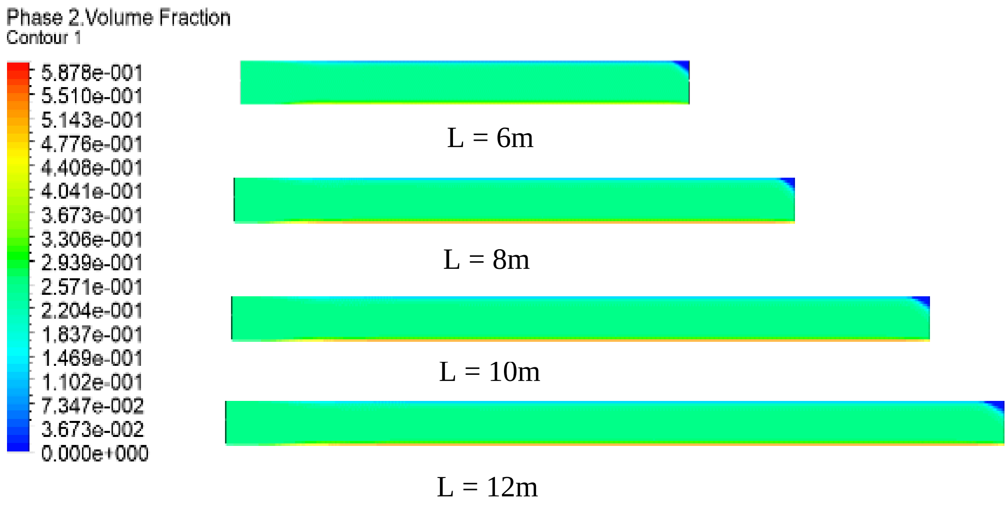


Figure 11 Effect of length at vertical plane

No changes in contours were seen at the distance of 4 m from inlet for the different length of pipe. Outlet pattern and volumetric distribution of particles along the pipe was same and length of pipe had no effect in particle concentration at certain distance from inlet.Figure 11 is the computed result abtained after simulations.

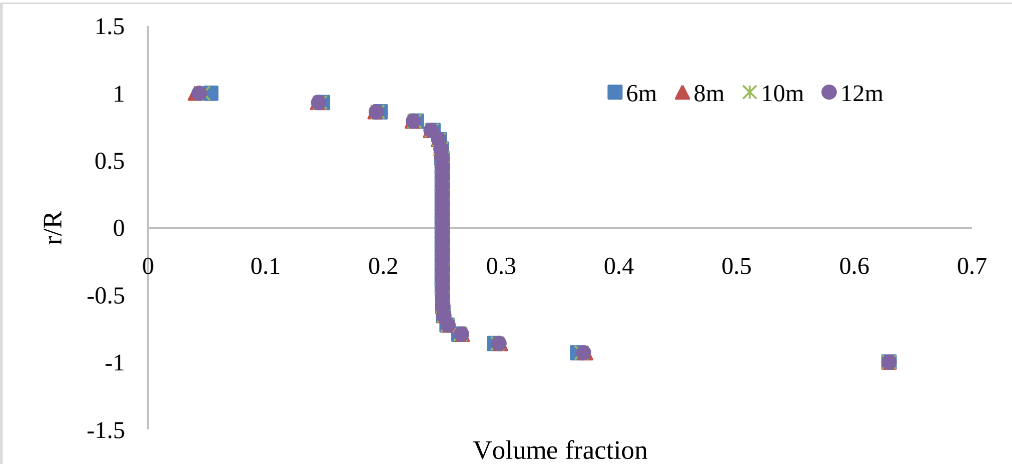


Figure 12 Effect of Length

No significant change was seen in the particle concentration when the length of pipe was altered. The particle concentration within vertical axis at the length 4m from inlet was same for all pipe at constant velocity and particle size. Maximum particle concentration in all the four cases was 0.629 approx at the bottom of the pipe.

4.1.2. Effect of Velocity

The contours at outlet and vertical plane are shown below in Figure 12 and Figure 13for various homogenoeus mixture inlet velocities.

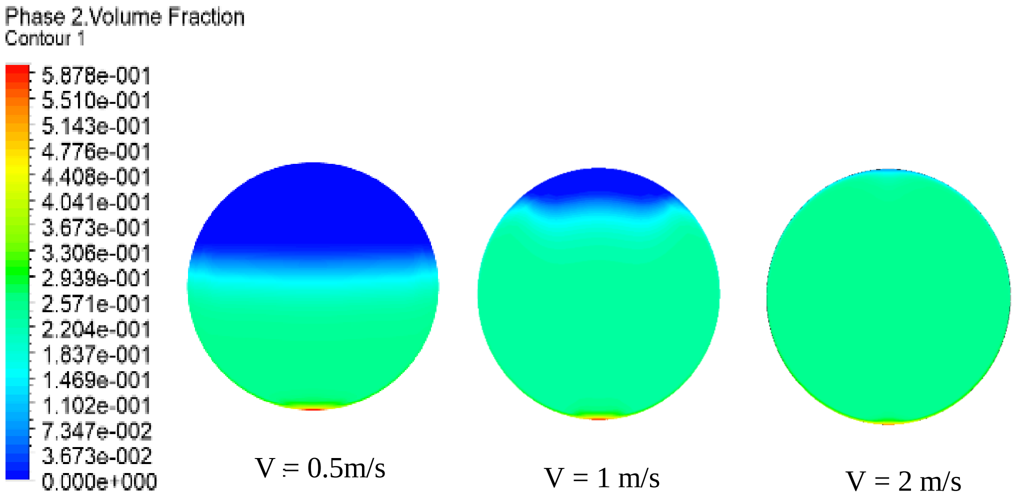


Figure 12 Outlet Contours at different velocity

The mixture distribution at outlet contour ismore homogenous with increase in velocity of the flow and the settlement of particle is increased with decreasein velocity. The particle distribution at bottom region is denser for low velocities. As the velocity of mixture increases the density of settling particles decreases.

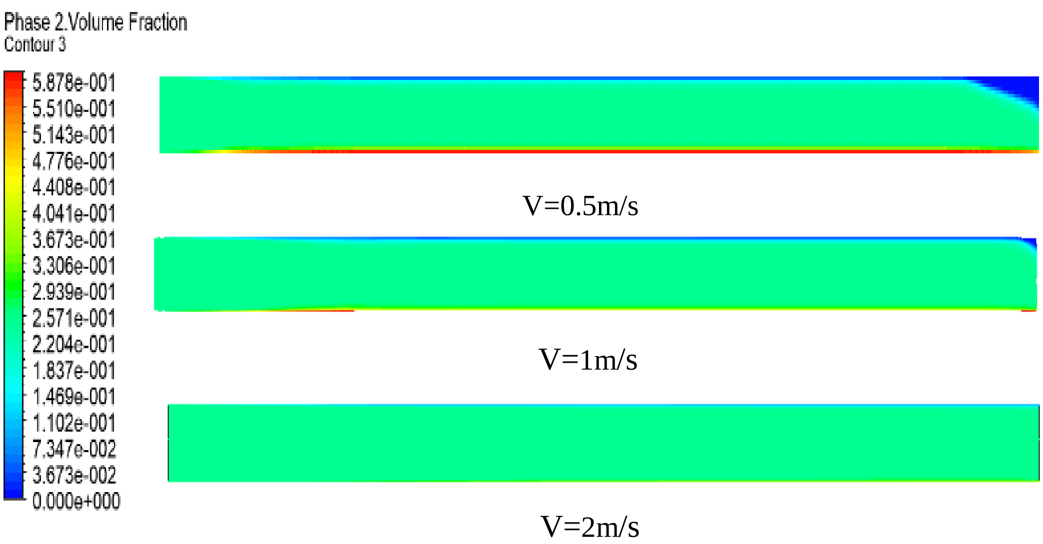


Figure 13 Contours at transverse vertical axis

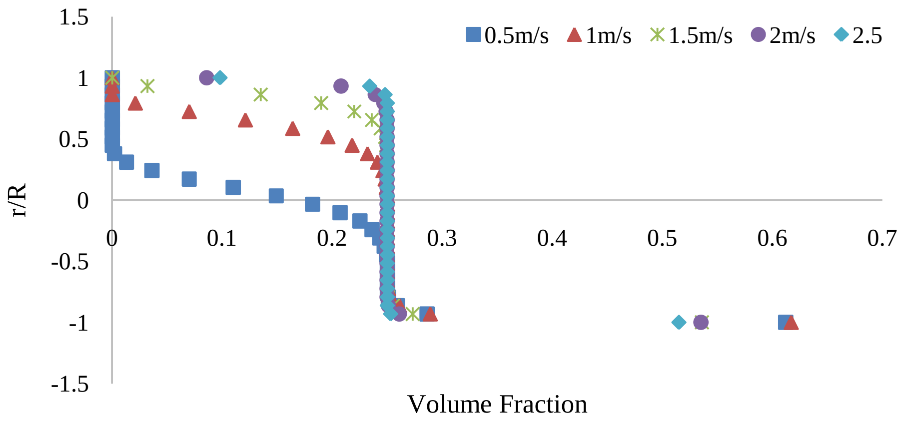


Figure 14 Effect of Velocity at Outlet

Figure 14 shows that the deposition of sand particle is maximum at the bottom region of the pipe and concentration of particle at bottom region is maximum at lower velocities. With the increase in velocities, the settlement of particles is low. The particle concentration is more homogeneous at the middle region. As the velocity is increased rate of settlement of particles at upper half region of pipe is also decreased whereas rapid settletement is seen in upper region at low velocities. The maximum concentration for 0.5m/s , 1m/s, 1.5m/s 2m/s, 2.5m/s velocity at outlet was found to be 0.626 ,0.617,0.536,0.535,0.515 respectively.

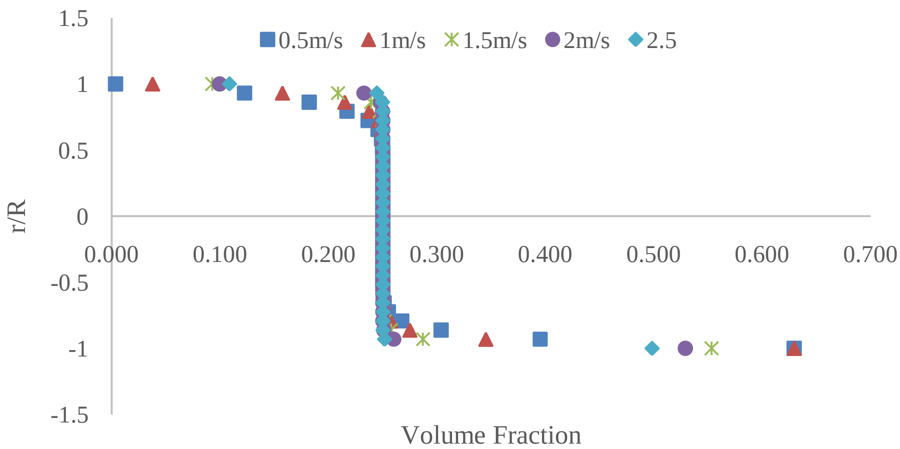


Figure 15 Particle concentration at midplane of pipe

Pattern of particle concentration distribution is seen more uniform at central plane for various inlet velocity of mixture however the of deposition of particle is low at high velocity and high at low velocity.The maximum particle concentration for 0.5m/s , 1m/s, 1.5m/s 2m/s, 2.5m/s velocity at midplane was found to be 0.629, 0.629, 0.553, 0.529, 0.498 respectively.

4.1.3. Effect of Volume Fraction

The following Figure 16and Figure 17depicts the contours of sand particles obtained after the simulations at outlet and mid transverse plane for the volume fraction of sand at 15%, 25%, 35% for at same condition of inlet mixture velocity, particle size and length of pipe.

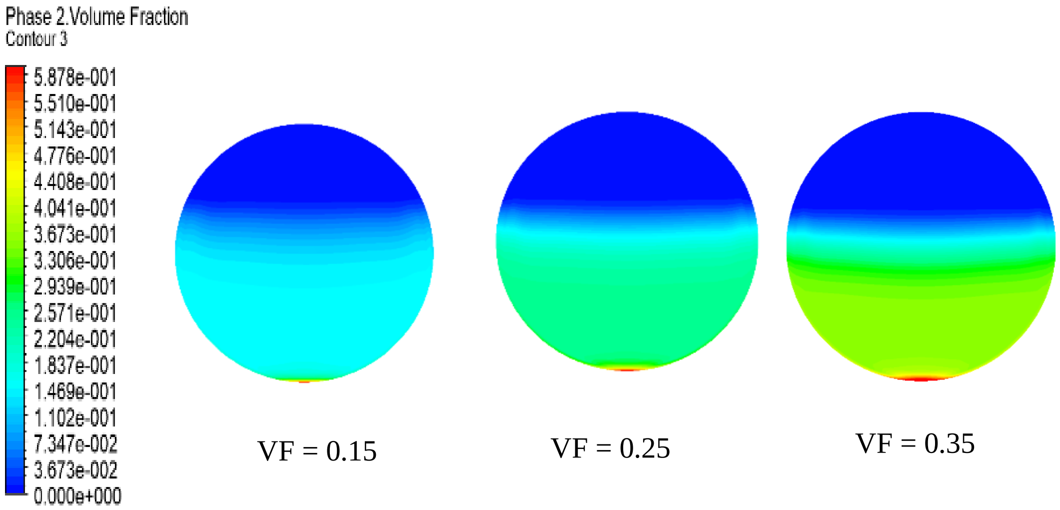


Figure 16 Outlet Contours

The sand particle concentration at outlet contour bottom region increaes with increase in initial volume fraction of mixture also the settlement of particle is increased with increase in initial volume fraction of mixture. The particle distribution at bottom region is denser. The deposition of particles at lower initial volume fraction is seen very less. This result can be due the impact of low overall density of sand particles at initial volume fractions ensuring high water impacts on particles.

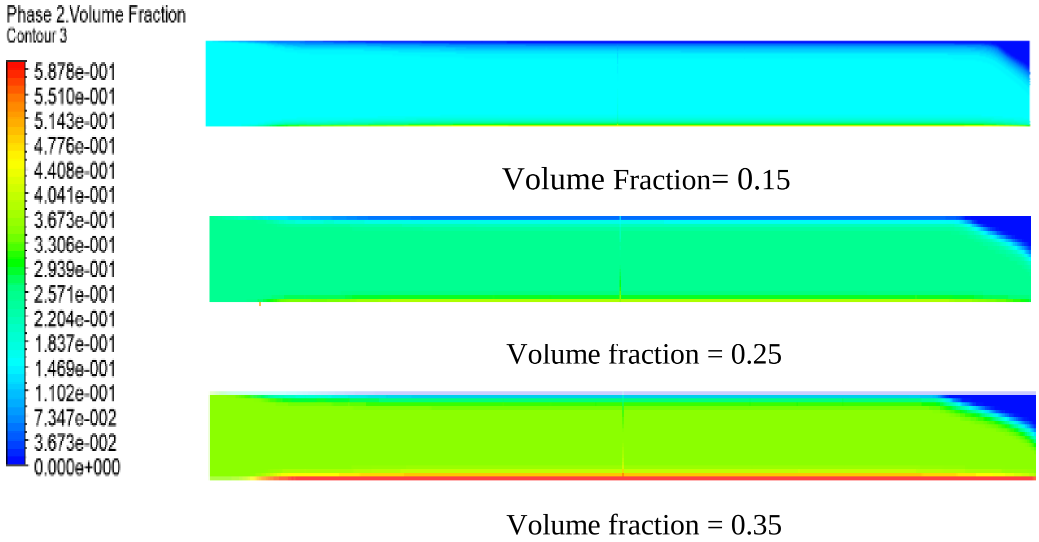


Figure 17Contours of transverse vertical plane

The contours along the length is shown above. Pattern of particle deposition along the length of pipe is similar in all three cases. The mixture remains homogeneous at certain distance from inlet and starts depositing in bottom. In the middle region of pipe, the flow is more homogenous, changes is evident in outlet region. The deposition of sand particle increases with increase in initial volume fraction of mixture. However, at lower initial volume fraction of the mixture the deposition of particle in bottom is least. Computed results of sand particle concentration at the outlet region and midplane after the simulation is shown in Figure 19 and Figure

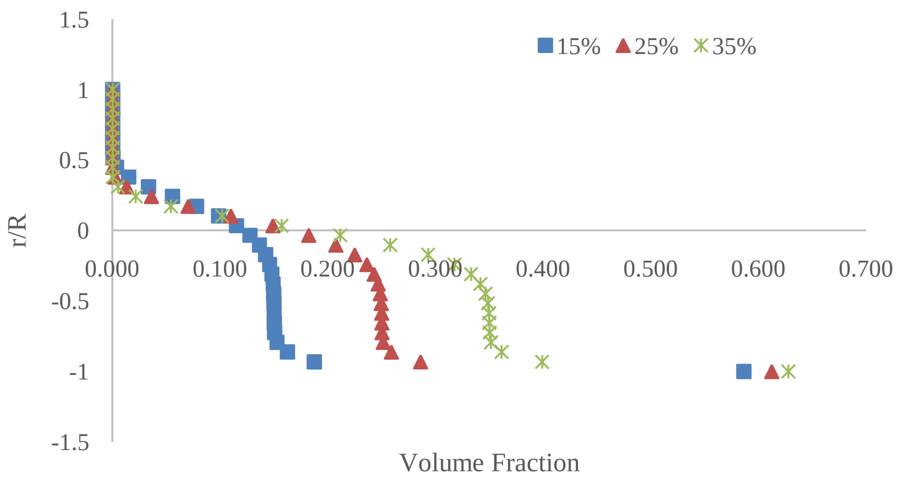
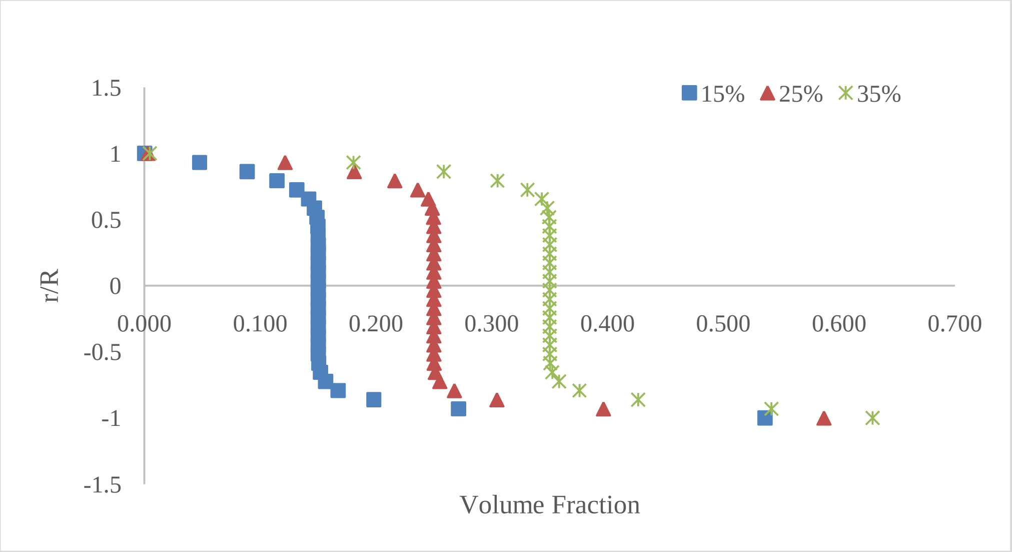


Figure 18Particle concentration at outlet

20respectively.

The particle concentration in bottom region is increasing significantly with the increase in sand particle initial volume fraction. The upper half region at outlet seems to have almost same pattern of particle distribution for all the cases whereas more variation is seen in lower region.The maximum particle concentration for 15%, 25%, and 35% volume fraction was 0.586, 0.612 and 0.628 respectively.

Figure 19 Effect of Volume Fraction at Central Plane

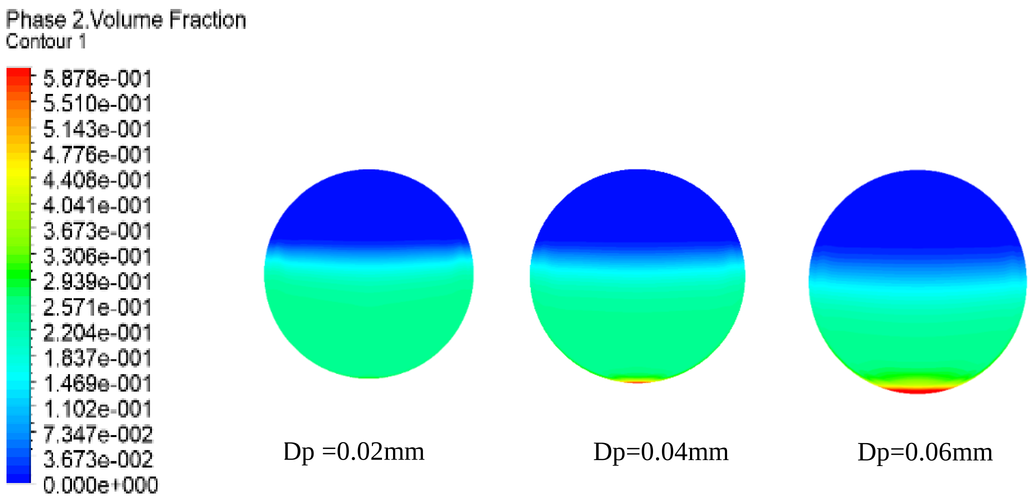


Particle concentration at the middle section of pipe was found increasing as the mixture volume fraction of sand was increased from 15% to 35%. The maximum particle concentration for 15%, 25%, and 35% volume fraction was 0.536, 0.587 and 0.629 respectively.

4.1.4. Effect of Particle Size

Figure 21 and Figure 21below shows the contours at outlet and vertical plane obtained after simulation for case studies involving different particle sizei.e., d_p=0.02mm, 0.04mm, 0.06mm. All three simulations were carried out at 0.5m/s initial velocity and 25% of initial volume fraction with pipe length of 6m and diameter of 0.3m.

Figure 20 Outlet Contours



The sand particle concentration at bottom region ofoutlet contourincreases with increase in particle size. The particle distribution at bottom region is denser. The deposition of sand particles for small size is seen very less. The result suggest that the inertial force of water is unable to carry away large particle sand in comparison to small particles.

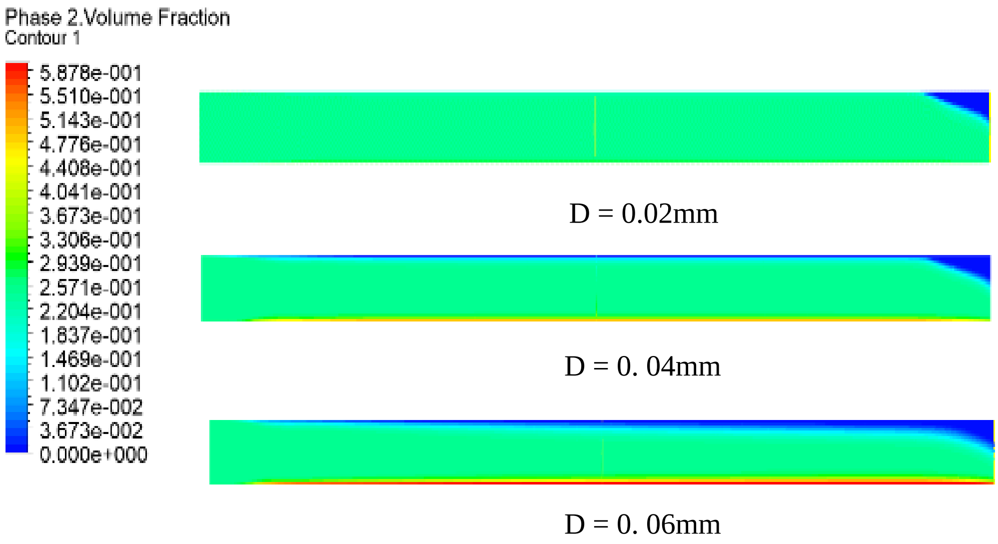
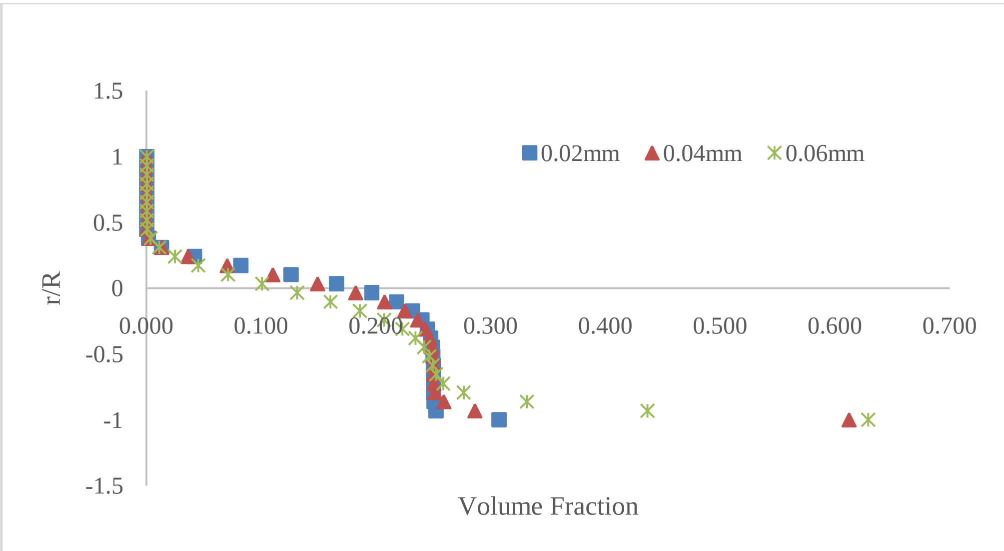


Figure 21 Transverse Vertical plane contours

With increase in particle diameter, settlement of the particles were increased. Also the particle concentration at bottom is increased i.e., settlement is more concentrated in bottom region for large particle size than small particle size. The result obtained from this case study was computed as shown in Figure 23 and Figure 23 .

Figure 22 Particle Concentration at Outlet



Particle concentration at the top region of pipe at outlet was seen negligible in all three cases whereas was found increasing as the particle size of sandwas increased from 0.02mm to 0.06mm at the bottom of pipe at outlet. The increase is maximum

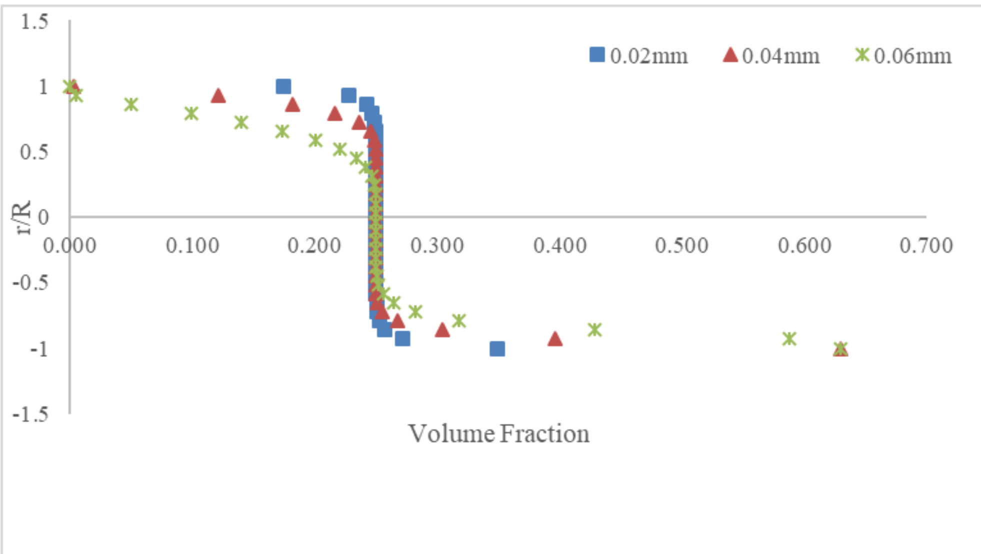


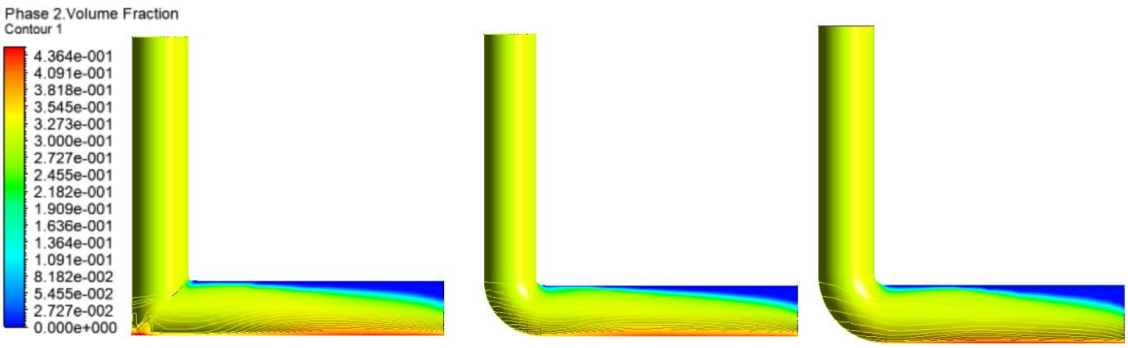
Figure 23 Particle Concentration at Midplane

concentration of particle for particle size 0.02mm to 0.04mm is much more than increase in 0.04mm to 0.06mm i.e maximum particle concentration for particle size 0.02mm at outlet is 0.307 whereas for particle size of 0.04mm and 0.06mm is 0.612 and 0.629 which depicts huge consecutive differences.

Sand particles are densely concentrated in between -1 to -1.5 radial distance of pipe and the density is more in cases of high volume fraction. Maximum particle concentration for initial particle size of 0.02mm in midplane is 0.349 whereas for initial volume fraction of0.04mm and 0.06mm is 0.629 and 0.629 respectively.

4.1.5. Effect of Bends

Two different kind of bends was used for the study. Curved and sharp bend were used to study the impact of bend surface.



Perpendicular bend | Mean fillet radius 180mm | Mean fillet radius 210mm

Figure 24 Contours at wall of different pipes

Figure 24andFigure 25 shows the wall contours and midplane contours respectively of the sand particles for all three cases. The obtained result shows that deposition of sand is higher in 90° bend than curved pipes. The impact of smooth surfaces in curved bends enables the smooth flow of solid sand particles decreasing the deposition in comparison to the 90° bend. Also, deposition of particle at midplane and wall region is seen random at the 90° bend pipe than in comparison to the curved pipe. The particle settlement at bottom region of 90° bend was found more than other two curved bend pipe. The flow was homogenous in vertical region of pipe in all the cases which might be due to the impact of gravity acting on the direction of inlet flowwhereas change in concentration patterns is seen at the horizontal pipe.

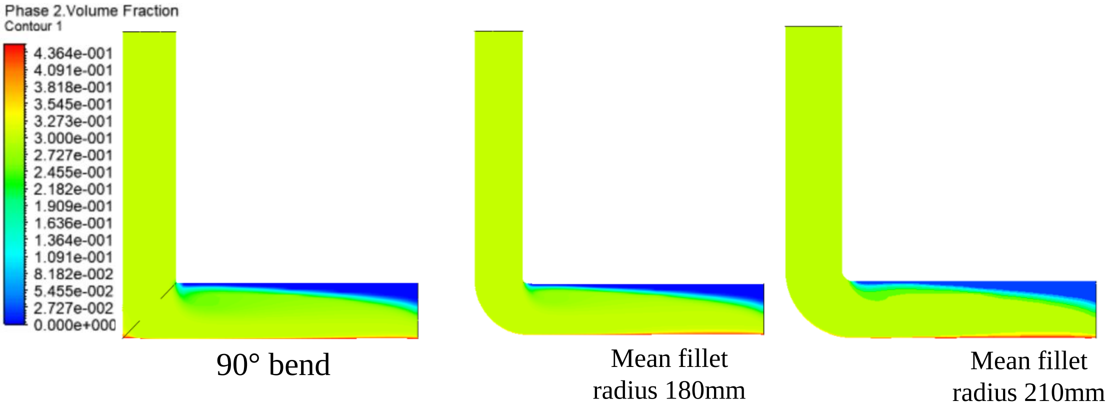


Figure 25Contours at Transverse midplaneof different pipes

From the results obtained from simulation, maximum particle concentration for 90° sharp bend, curved bend of mean fillet radius 180mm and curved bend of mean fillet radius 210mm at bottom are 0.576, 0.476, and 0.466 respectively. i.e. as the degree of curvature is increased the maximum particle concentration decreases.

CHAPTER FIVE: CONCLUSIONAND RECOMMENDATIONS

5.1. Conclusions

Hence the model developed was suitable for predicting two phase flow. However various error and variations were seen.

The major findings of thesis work can be summarized as below

- CFD analysis was carried out using multiphase approach between liquid water and solid particles sand in 3D horizontal pipe, bend pipe of 90° sharp bend and curved bend and the experimental data obtained from research paper was used for model validation.
- The maximum concentration for inlet mixture velocity of 0.5m/s , 1m/s, 1.5m/s 2m/s, 2.5m/s velocity at outlet was found to be 0.626 ,0.617,0.536,0.535,0.515 respectively, and at midplane was found to be 0.629, 0.629, 0.553, 0.529, 0.498 respectively.
- The maximum particle concentration for 15%, 25%, and 35% of initial mixture volume fraction was 0.586, 0.612 and 0.628 respectively at outlet and 0.536, 0.587 and 0.629 at midplane respectively.
- Maximum particle concentration for particle size of 0.02mm ,0.04mm and 0.06mm are 0.307, 0.612 and 0.629 respectively at outlet and 0.349, 0.629, 0.629 respectively at midplane.
- Impact of bend surface in solid particle flow with liquid particles were studied and deposition of particles for different bend conditions were analysed for give inlet conditions. From the results maximum particle concentration for 90° sharp bend, curved bend of mean fillet radius 180mm and curved bend of mean fillet radius 210mm at bottom are 0.576, 0.476, and 0.466 respectively.

5.2. Recommendations

- The bends results are predictions based on the model developed after the experimental validations of results of the horizonal pipes hence the experimental analysis of bends is upcoming interests.
- More closure in bend conditions of various bend angle is the further interest of the study for both curved and non- curved cases.

REFERENCES

A. Ekambara, R. S. S. K. N. & H.Masliyah, J., 2009. Hydrodynamic simulation of horizontal slurry pipeline flow using ANSYS-CFX. *Industrial Engineering and Chemistry Research*, Volume 48, p. 17.

Ajay K Yerrumshetty, D. J. B. & Bugg, J. D., 2009. A TWO-FLUID MODEL OF TURBULENT LIQUID-SOLID FLOW IN A HORIZONTAL CHANNEL. *Sixth International Symposium on Turbulence and Shear Flow Phenomena*, pp. 537-540.

Atkinson, J., 2000. *Soil Classification.* [Online] Available at: https://environment.uwe.ac.uk/geocal/SoilMech/classification/default.htm [Accessed 10 10 2018].

Boris V Balakin, A. C. H. P. K. &. L. D. R., 2010. Eulerian-Eulerian CFD Model for the Sedimentation of Spherical Particles in Suspension with High Particle Concentrations. *Engineering Applications of Computational Fluid Mechanics.*

Crowe, C. T., 2006. *Multiphase Flow Handbook.* s.l.:CRC Press Taylor and Francis Group.

G. Itskos, N. N.-S. K. K. V. D. & Grammelis, P., 2016. Environment and Development. In: *Energy and the Environment.* s.l.:elsevier, pp. 363-452.

G. MICALE, G. M. F. G. A. B. & GODFREY, J., 2000. CFD SIMULATION OF PARTICLE DISTRIBUTION IN STIRRED VESSELS. *Trans IChemE, Vol 78,* p. 1.

Goharzadeh, A. & Rodgers, P., 2009. Experimental characterization of solid particle transport by slug flow using Particle Image Velocimetry. *Journal of Physics: Conference Series,* pp. 1-2.

H.Shi, J. H. L. D. K. A. L. D. B. A. & G.Oddie, 2005. Drift flux modelling of two phase flow in wellbores. *SPE Journal,* p. 2.

Heywood, N. I. & Cheng, D. C.-H., 2002. FLOW IN PIPES. *Physics in Technology,* Volume 15, pp. 1-2.

Hydrolab, 2005. *::HYDRO LAB::.* [Online] Available at: http://www.hydrolab.org/resndev.htm [Accessed 2 3 2017].

L. Ma, C. H. & Xie, Y., 2015. Modeling of erodent particle trajectories in slurry flow. *Wear,* Volume 334, pp. 49-50.

M.A. Delele, F. W. G. F. & Mellmann, J., 2016. Studying the solids and fluid flow behavior in rotary drums based on a multiphase CFD model. *Powder Technology,* pp. 260-271.

Messa, G. V. & Malavasi, S., 2015. Improvements in the numerical prediction of fully-suspended slurry flow in horizontal pipes. *Powder Technology,* pp. 358-367.

Ofei, T. N. & Ismail, A. Y., 2016. Eulerian-Eulerian Simulation of Particle-Liquid Slurry Flow in Horizontal Pipe. *Journal of Petroleum Engineering,* Volume 2016, p. 2.

Oshinowo, L. M. & Bakker, A., 2002. Cfd Modelling of Solid Suspended in Stirred Tanks. *Symposium on Computational Modeling of Metals, Minerals and Materials,* p. 2.

Patro, P. & Patro, B., 2013. Kinetic Theory Based CFD Modeling of Particulate Flows in Horizontal Pipes. *International Journal of Mathematical, Computational, Physical, Electrical and Computer Engineering,* pp. 1-7.

Randall G. Gillies, C. A. S. & Xu, J., 2004. Modelling Heterogeneous Slurry Flows at High Velocities. *Saskatchewan Research Council Pipe Flow Technology Centre, 15 Innovation Boulevard, Saskatoon, SK, Canada,* p. 4.

S. A.Miedema, 2016. The heterogeneous to homogeneous transition for slurry flow in pipes. *Ocean Engineering,* Volume 123, pp. 422-430.

Tamer Nabil, I. E.-S. & El-Nahhas, K., 2013. Computational Fluid Dynamics Simulation of the Solid-Liquid Slurry Flow in a Pipeline. *Seventeenth International Water Conference,* pp. 2-3.

Yang Ho Song, R. Y. ,. E. H. L. & Lee, J. H., 2018. Predicting Sedimentation in Urban Sewer Conduits. *water,* pp. 3,4.

Zhou L, Y. Y. C. F. & Zeng, Z., 2013. Two-phase turbulence models for simulating dense gas–particle flows. *Particuology,* pp. 6,7.

ANNEXES

Table 7 Mesh Independence Analysis

S.N.	Elements Number	Volume Fraction
1	97,552	0.30647
2	213,576	0.299845
3	378,952	0.299783
4	456,382	0.283609
5	569,664	0.279267
6	765,038	0.277994
7	975,521	0.277954

Table 8 Computational Results for Different Length

	Length of Pipe			
Radial Distance(r/R)	6m	8m	10m	12m
1.0000	0.0532	0.0401	0.0467	0.0434
0.9310	0.1483	0.1439	0.1461	0.1450
0.8621	0.1974	0.1927	0.1950	0.1939
0.7931	0.2281	0.2244	0.2262	0.2253
0.7241	0.2421	0.2399	0.2410	0.2404
0.6552	0.2475	0.2466	0.2471	0.2468
0.5862	0.2491	0.2488	0.2490	0.2489
0.5172	0.2497	0.2497	0.2497	0.2497
0.4483	0.2499	0.2499	0.2499	0.2499
0.3793	0.2500	0.2500	0.2500	0.2500
0.3103	0.2500	0.2500	0.2500	0.2500
0.2414	0.2500	0.2500	0.2500	0.2500
0.1724	0.2500	0.2500	0.2500	0.2500
0.1034	0.2500	0.2500	0.2500	0.2500
0.0345	0.2500	0.2500	0.2500	0.2500
-0.0345	0.2500	0.2500	0.2500	0.2500
-0.1034	0.2500	0.2500	0.2500	0.2500
-0.1724	0.2500	0.2500	0.2500	0.2500
-0.2414	0.2500	0.2500	0.2500	0.2500
-0.3103	0.2500	0.2500	0.2500	0.2500
-0.3793	0.2500	0.2500	0.2500	0.2500
-0.4483	0.2500	0.2500	0.2500	0.2500
-0.5172	0.2501	0.2501	0.2501	0.2501
-0.5862	0.2503	0.2504	0.2504	0.2504
-0.6552	0.2511	0.2514	0.2512	0.2513
-0.7241	0.2541	0.2552	0.2546	0.2549
-0.7931	0.2641	0.2669	0.2655	0.2662

-0.8621	0.2943	0.2994	0.2968	0.2981
-0.9310	0.3649	0.3717	0.3683	0.3700
-1.0000	0.6295	0.6299	0.6297	0.6298

Table 9 Computational Results for different velocities at midplane of pipe

Radial Distance	**Initial Mixture Velocity**				
r/R	**0.5m/s**	**1m/s**	**1.5m/s**	**2m/s**	**2.5m/s**
1.000	0.00345	0.03752	0.09242	0.09938	0.10841
0.931	0.12245	0.15728	0.20852	0.23270	0.24435
0.862	0.18206	0.21480	0.23899	0.24789	0.24969
0.793	0.21689	0.23754	0.24689	0.24971	0.25000
0.724	0.23647	0.24605	0.24921	0.24996	0.24999
0.655	0.24542	0.24886	0.24983	0.24998	0.24998
0.586	0.24871	0.24971	0.24997	0.24999	0.25000
0.517	0.24971	0.24992	0.25000	0.25000	0.25001
0.448	0.24995	0.24998	0.25000	0.25000	0.25000
0.379	0.25000	0.24999	0.25000	0.25000	0.25000
0.310	0.25000	0.25000	0.25000	0.25000	0.25000
0.241	0.25000	0.25000	0.25000	0.25000	0.25000
0.172	0.25000	0.25000	0.25000	0.25000	0.25000
0.103	0.25000	0.25000	0.25000	0.25000	0.25000
0.034	0.25000	0.25000	0.25000	0.25000	0.25000
-0.034	0.25000	0.25000	0.25000	0.25000	0.25000
-0.103	0.25000	0.25000	0.25000	0.25000	0.25000
-0.172	0.25000	0.25000	0.25000	0.25000	0.25000
-0.241	0.25000	0.25000	0.25000	0.25000	0.25000
-0.310	0.25000	0.24999	0.25000	0.25000	0.25000
-0.379	0.25000	0.24999	0.25000	0.25000	0.25000
-0.448	0.25001	0.24999	0.25000	0.25000	0.25000
-0.517	0.25004	0.25000	0.25000	0.25000	0.25001
-0.586	0.25023	0.25009	0.25001	0.25000	0.25000
-0.655	0.25121	0.25046	0.25005	0.24999	0.24998
-0.724	0.25500	0.25187	0.25033	0.25000	0.24999
-0.793	0.26742	0.25695	0.25168	0.25010	0.25003
-0.862	0.30373	0.27488	0.25744	0.25091	0.25010
-0.931	0.39515	0.34499	0.28694	0.26017	0.25152
-1.000	0.62950	0.62950	0.55329	0.52917	0.49833

Table 10 Computational Results for different velocities at outlet of pipe

Radial Distance	**Initial Mixture Velocity**				
r/R	**0.5m/s**	**1m/s**	**1.5m/s**	**2m/s**	**2.5m/s**
1.000	0	0	0	0.086	0.098
0.931	0	0	0.032	0.208	0.234
0.862	0	0	0.135	0.239	0.248
0.793	0	0.021	0.19	0.247	0.25
0.724	0	0.07	0.22	0.249	0.25
0.655	0	0.121	0.236	0.25	0.25
0.586	0	0.164	0.244	0.25	0.25
0.517	0	0.196	0.248	0.25	0.25
0.448	0	0.218	0.249	0.25	0.25
0.379	0.002	0.232	0.25	0.25	0.25
0.310	0.013	0.241	0.25	0.25	0.25
0.241	0.036	0.246	0.25	0.25	0.25
0.172	0.07	0.248	0.25	0.25	0.25
0.103	0.11	0.249	0.25	0.25	0.25
0.034	0.149	0.25	0.25	0.25	0.25
-0.034	0.182	0.25	0.25	0.25	0.25
-0.103	0.207	0.25	0.25	0.25	0.25
-0.172	0.225	0.25	0.25	0.25	0.25
-0.241	0.236	0.25	0.25	0.25	0.25
-0.310	0.243	0.25	0.25	0.25	0.25
-0.379	0.247	0.25	0.25	0.25	0.25
-0.448	0.249	0.25	0.25	0.25	0.25
-0.517	0.25	0.25	0.25	0.25	0.25
-0.586	0.25	0.25	0.25	0.25	0.25
-0.655	0.25	0.25	0.25	0.25	0.25
-0.724	0.25	0.251	0.25	0.25	0.25
-0.793	0.251	0.253	0.251	0.25	0.25
-0.862	0.259	0.261	0.255	0.251	0.25
-0.931	0.286	0.289	0.273	0.261	0.253
-1.000	0.612	0.617	0.536	0.535	0.515

Table 11 Computational Results for different Initial Volume fraction at outlet of pipe

Radial Distance	**Initial Volume Fraction**		
r/R	**15%**	**25%**	**35%**
1	0.000	0.000	0.000
0.931	0.000	0.000	0.000
0.862	0.000	0.000	0.000
0.793	0.000	0.000	0.000
0.724	0.000	0.000	0.000
0.655	0.000	0.000	0.000
0.586	0.000	0.000	0.000
0.517	0.000	0.000	0.000
0.448	0.004	0.000	0.000

0.379	0.015	0.002	0.000
0.31	0.033	0.013	0.005
0.241	0.056	0.036	0.022
0.172	0.078	0.070	0.054
0.103	0.098	0.110	0.102
0.034	0.115	0.149	0.157
-0.034	0.128	0.182	0.211
-0.103	0.136	0.207	0.258
-0.172	0.142	0.225	0.293
-0.241	0.146	0.236	0.318
-0.31	0.148	0.243	0.333
-0.379	0.149	0.247	0.342
-0.448	0.150	0.249	0.346
-0.517	0.150	0.250	0.349
-0.586	0.150	0.250	0.350
-0.655	0.150	0.250	0.350
-0.724	0.150	0.250	0.350
-0.793	0.153	0.251	0.352
-0.862	0.162	0.259	0.361
-0.931	0.187	0.286	0.399
-1	0.586	0.612	0.628

Table 12 Computational Results for different Initial Volume fraction at midplane of pipe

Radial Distance	**Initial Volume Fraction**		
r/R	**15%**	**25%**	**35%**
1	0.000	0.003	0.005
0.931	0.048	0.121	0.180
0.862	0.089	0.181	0.258
0.793	0.114	0.216	0.305
0.724	0.132	0.236	0.331
0.655	0.142	0.245	0.343
0.586	0.147	0.249	0.348
0.517	0.149	0.250	0.349
0.448	0.150	0.250	0.350
0.379	0.150	0.250	0.350
0.31	0.150	0.250	0.350
0.241	0.150	0.250	0.350
0.172	0.150	0.250	0.350
0.103	0.150	0.250	0.350
0.034	0.150	0.250	0.350
-0.034	0.150	0.250	0.350
-0.103	0.150	0.250	0.350
-0.172	0.150	0.250	0.350
-0.241	0.150	0.250	0.350
-0.31	0.150	0.250	0.350

-0.379	0.150	0.250	0.350
-0.448	0.150	0.250	0.350
-0.517	0.150	0.250	0.350
-0.586	0.151	0.250	0.350
-0.655	0.152	0.251	0.352
-0.724	0.156	0.255	0.358
-0.793	0.167	0.268	0.376
-0.862	0.198	0.304	0.427
-0.931	0.271	0.396	0.542
-1	0.536	0.587	0.629

Table 13 Computational Results for different sand particles at outlet of pipe

Radial Distance	**Particle size**		
r/R	**0.02mm**	**0.04mm**	**0.06mm**
1	0.000	0.000	0.000
0.931	0.000	0.000	0.000
0.862	0.000	0.000	0.000
0.793	0.000	0.000	0.000
0.724	0.000	0.000	0.000
0.655	0.000	0.000	0.000
0.586	0.000	0.000	0.000
0.517	0.000	0.000	0.000
0.448	0.000	0.000	0.001
0.379	0.002	0.002	0.003
0.31	0.013	0.013	0.011
0.241	0.042	0.036	0.025
0.172	0.082	0.070	0.045
0.103	0.126	0.110	0.071
0.034	0.165	0.149	0.101
-0.034	0.196	0.182	0.131
-0.103	0.218	0.207	0.160
-0.172	0.232	0.225	0.186
-0.241	0.240	0.236	0.207
-0.31	0.245	0.243	0.223
-0.379	0.247	0.247	0.234
-0.448	0.249	0.249	0.242
-0.517	0.249	0.250	0.246
-0.586	0.250	0.250	0.249
-0.655	0.250	0.250	0.252
-0.724	0.250	0.250	0.258
-0.793	0.250	0.251	0.276
-0.862	0.250	0.259	0.331
-0.931	0.252	0.286	0.436
-1	0.307	0.612	0.629

Table 14 Computational Results for different sand particles at midplane of pipe

Radial Distance	**Paricle size**		
r/R	**0.02mm**	**0.04mm**	**0.06mm**
1	0.174	0.003	0.000
0.931	0.228	0.121	0.005
0.862	0.242	0.181	0.050
0.793	0.247	0.216	0.099
0.724	0.249	0.236	0.140
0.655	0.250	0.245	0.173
0.586	0.250	0.249	0.200
0.517	0.250	0.250	0.220
0.448	0.250	0.250	0.234
0.379	0.250	0.250	0.242
0.31	0.250	0.250	0.246
0.241	0.250	0.250	0.249
0.172	0.250	0.250	0.250
0.103	0.250	0.250	0.250
0.034	0.250	0.250	0.250
-0.034	0.250	0.250	0.250
-0.103	0.250	0.250	0.250
-0.172	0.250	0.250	0.250
-0.241	0.250	0.250	0.250
-0.31	0.250	0.250	0.250
-0.379	0.250	0.250	0.250
-0.448	0.250	0.250	0.251
-0.517	0.250	0.250	0.252
-0.586	0.250	0.250	0.256
-0.655	0.250	0.251	0.265
-0.724	0.251	0.255	0.282
-0.793	0.253	0.268	0.318
-0.862	0.257	0.304	0.429
-0.931	0.272	0.396	0.588
-1	0.349	0.629	0.629