Proceedings of IOE Graduate Conference, 2019-Winter
Peer Reviewed
Year: 2019 Month: December Volume: 7
ISSN: 2350-8914 (Online), 2350-8906 (Print)

Multiphase Flow Analysis between Sand and Water in Horizontal Pipe

Rama Sapkota [a], Hari Bahadur Dura [b], Laxman Poudel [c]

[a, b, c] *Department of Mechanical Engineering, Pulchowk Campus, Institute of Engineering, Tribhuvan University, Nepal*
Corresponding Email: [a] sapkota.rama@pcampus.edu.np, [b] duraharis@pcampus.edu.np, [c] p12_laxman@yahoo.com

Abstract
Multiphase flow is very common phenomenon in industrial, hydro power, drainage systems, petroleum pipelines system etc. This 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 for prediction of 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 0.3 m, variable length, volume fraction (15%-35%), particles (0.02 mm - 0.06 mm), velocity (0.5 m/s - 2.5 m/s). 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 0.5 m/s and 2.5 m/s velocity at outlet was found to be 0.626 and 0.515 respectively, and at mid-plane 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, 0.587 and 0.629 at mid-plane respectively.Maximum particle concentration for particle size of 0.02 mm ,0.04 mm and 0.06 mm are 0.307, 0.612 and 0.629 respectively at outlet and 0.349, 0.629, 0.629 respectively at mid-plane. Change in length of pipe didn't have any effect on concentration at certain point of pipe.

1. Introduction

In hydro-power plants, various industries, phase flow characterization has an essential importance due to its presence in the flow of fluid mixed with particles in various pipelines system. 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, behavior 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 [1]. 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 [2].

1.1 Problem Statement

The proposed research aims to narrow down the study for the volumetric fraction of solid particles in fluid (range between 10%-30%) and study the particle fluid interaction for various particle size. Moreover design in 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 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 [3].

1.2 Related Works

Patro & Patro (2013) simulated fully developed gas–solid flow in a horizontal pipe using the two fluid model which 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, reached a peak and then decreased[2].

Boris V Balakin (2010) focused on the computational study of the process of sedimentation of spherical particles in suspensions with high particle concentrations with the two-fluid Eulerian approach. 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[4].

A. Ekambara & H.Masliyah (2009) used CFD simulation to investigate the effect of in situ solids volume concentration, particle size, mixture velocity, and particle diameter on local time averaged solids concentration profiles, particle and liquid velocity profiles, and frictional pressure loss. The study revealed that particles were asymmetrically distributed in the vertical plane with the degree of asymmetry increasing with increasing particle size in horizontal pipe[5] .

G. Micale & Godfrey (2000) simulated The particle concentration distribution in two-phase stirred tanks w on the basis of information on the three-dimensional flow field as obtained by numerical solution of the flow equations using the well-known k-epsilon turbulence model. Two modeling approaches (k-epsilon turbulence and k-omega 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 coefficients in turbulent fluid media was emphasized[6].

The solid and fluid flow behaviors inside a rotary drum was analyzed using CFD by M.A. Delele & Mellmann (2016) . 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 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 [7].

T.N & A.Y (2016) adopted the CFD simulation in-homogeneous Eulerian-Eulerian two-fluid model to examine the influence of particle size and particle volume fraction on the radial distribution of particle concentration and velocity and frictional pressure loss.The study showed that knowledge of the variation of these parameters with pipe position is very crucial in the understanding of pipeline wear, particle attrition, or agglomeration is to be advanced[8].

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[9].

2. Methodology

The study was based on CFD analysis of flow followed by geometry set up, mesh independence analysis, and boundary conditions set up described in following sections:

2.1 CFD Analysis

2.1.1 Governing Equations

Volume Fractions Equations
The volume fraction equation is given by

$$V_q = \int_V \alpha_q dV \tag{1}$$

where

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

α_q = Volume fraction of phase q

Conservation of Mass Equations

The continuity equation for phase q, is given by

$$\frac{\partial}{\partial t}(\alpha_q \rho_q) + \nabla.(\alpha_q \rho_q \overrightarrow{v_q}) = \sum_{p=1}^{n} \dot{m}_{pq} - \dot{m}_{qp} + S_q \quad (2)$$

Equations for Conservation of Momentum

The momentum balance equation for phase q, is given by

$$\frac{\partial}{\partial t}(\alpha_q \rho_q \overrightarrow{v_q}) + \nabla.(\alpha_q \rho_q \overrightarrow{v_q}\overrightarrow{v_q}) = -\alpha_q \nabla_p + \nabla.\overline{\tau_q} + \alpha_q \rho_q \overrightarrow{g} + \sum_{p=1}^{n} (\overline{R}_{pq} + \dot{m}_{pq}\overrightarrow{v_{pq}} - \dot{m}_{qp}\overrightarrow{v_{qp}}) + (\overrightarrow{F_q} + \overrightarrow{F}_{lift,q} + \overrightarrow{F}_{wl,q} + \overrightarrow{F}_{vm,q} + \overrightarrow{F}_{td,q}) \quad (3)$$

Where,
V_q = Volume of phase q,
$\overline{\tau}_q$ = qth phase stress-strain tensor,
$\overrightarrow{F}_q$ = an external body force,
$\overrightarrow{F}_{lift,q}$ = lift force,
$\overrightarrow{F}_{wl,q}$ = wall lubrication force,
$\overrightarrow{F}_{vm,q}$ =virtual mass force,
$\overrightarrow{F}_{td,q}$ =turbulent dispersion force,
$\overline{R}_{pq}$ =interaction forces between phases respectively.

2.1.2 Geometry & Meshing of Pipe

The geometry of pipe for the study is as shown in Figure 1.

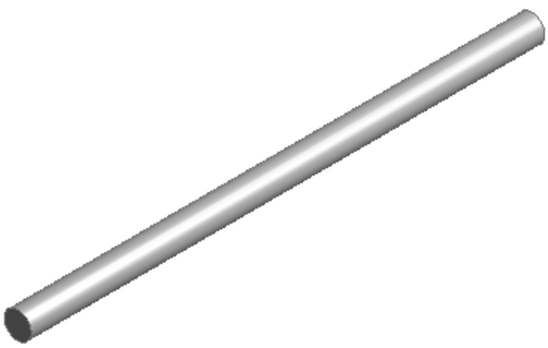

Figure 1: Isometric view of pipe

Pipe diameter of 0.3m at different length (6m, 8m, 10m and 12m) was modeled using ANSYS 18.1 Workbench Design Modeller

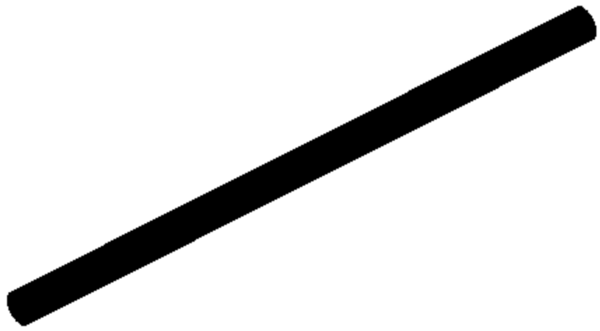

Figure 2: Meshing of pipe

Unstructured cut-cell mesh with 10 inflation layers at the boundary region of the pipe was created to accurately capture the flow effect in that region.710362 number of elements after mesh independence analysis were used meshing of pipe.

2.1.3 Physics Setup

Studies were carried out with boundary conditions as shown in Figure 3.

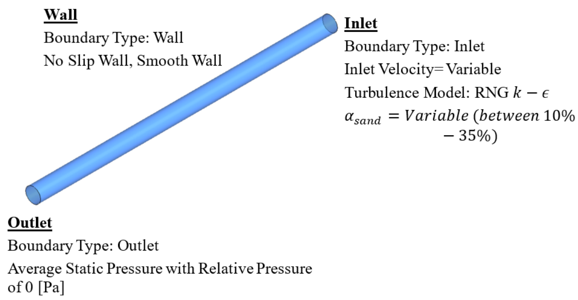

Figure 3: Boundary conditions for pipe

At the inlet of the pipe, velocity of homogeneous mixture of sand and water was defined.Volume fraction of sand particles in water (liquid), particles size (diameter of sand particles) were specified. At the outlet, static pressure was specified. At the wall, no-slip condition was imposed. To initiate the numerical solution, average volume fractions and mixture velocity were specified as initial conditions. Density of sand 2650 kg/m^3 and water 998.8 kg/m^3 was used for calculation. Pipe diameter was taken constant for all cases whereas parameters inlet velocity, particle size, initial mixture concentrations and lengths were varied for different cases.

2.1.4 Solver

In this simulation of sand-water flow study, the Navier-Stokes governing equations together with their closure terms were solved using ANSYS Fluent 18.1 solver using Eulerian-Eulerian Multifluid VOF model . The mass and momentum equations were discretised using the control volume technique. The first order 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 utilized 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 normalized residual error reached 10^{-4} for all simulation.

2.2 Mesh Independence Analysis

Mesh independence study was performed by taking the RNG k-ε model with function with the inlet velocity of 0.5 m/s. Seven meshes of different sizes with the number of elements varying from 98552 to 871725 were used for mesh independence study. Volume Fraction of sand particle at a point was observed as a key parameter of interest. In the present work the convergence criteria of 10^{-4} was used. The graph was plotted between number of elements and volume fraction at point(0,-0.13,3) of pipe as shown in Figure

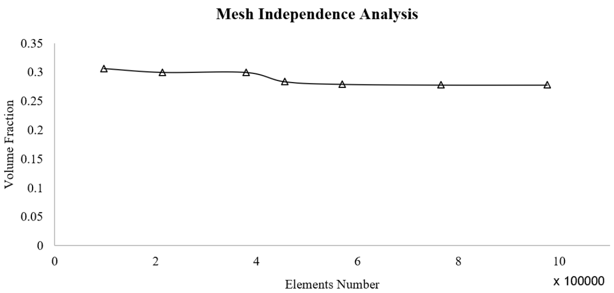

Figure 4: Mesh independence Analysis

As per results from above seven cases change in volume fraction of sand particles between 600000 - 871724 was found negligible. Hence total 710362 elements was used for all subsequent computations.

2.3 Case Studies

2.3.1 Effect of Pipe Length

Volume fraction along vertical axis at distance 4 m from inlet was calculated for Horizontal pipe of different length i.e (6 m, 8 m, 10 m and 12m). The main purpose of this calculation was to analyse the effect of length of pipe on volumetric fractions at certain distance from inlet.

2.3.2 Effect of Particle Size

Various simulations were carried out for particle diameters $d_1 = 0.02mm$, $d_2 = 0.04mm$ and $d_3 = 0.06mm$ at inlet velocity 0.5 m/s. The main purpose of this calculation was to analyze the settlement of particle along the pipe and impact of particle size on volumetric concentration along the pipe for different particle size.

2.3.3 Effect of Initial Volume Fraction

The calculation were also performed at volumetric fractions of 15%, 25% and 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 particle size.

2.3.4 Effect of Particle Velocity

Various simulations were carried out at inlet velocity (0.5 m/s, 1 m/s, 1.5 m/s, 2 m/s and 2.5 m/s). The purpose of this calculation was to study effect of velocity in sand particle distribution in flow for different velocity and analyse the gravity dominance of sand particles at low velocities.

3. Results and Discussions

Various simulations were performed for different cases of inlet mixture velocity,particle size, initial volume fractions and different pipe length for CFD analysis of sand water flow in horizontal pipe. Results obtained for various initial conditions were obtained as follows.

3.1 Results at Variable Length of Pipe

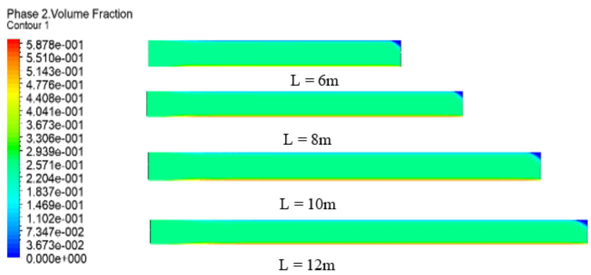

Figure 5: Contours at 4m from inlet

Contours along the length of pipe for different length is shown.The patterns of particle concentrations were identical for all cases at bottom region.

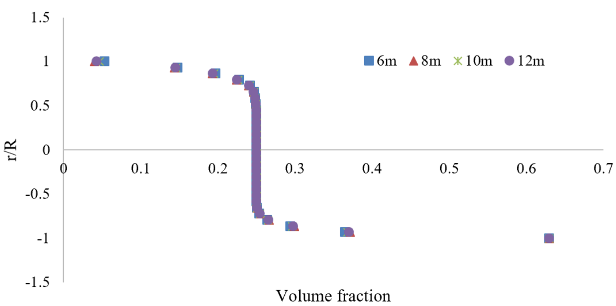

Figure 6: Particle concentrations at 4m inlet

Effect of length was negligible in deposition of particles at certain distance from inlet for different pipe lengths.

Maximum particle concentration in all the four cases was 0.629 approx at the bottom of the pipe.

3.2 Results at Variable Particle Size

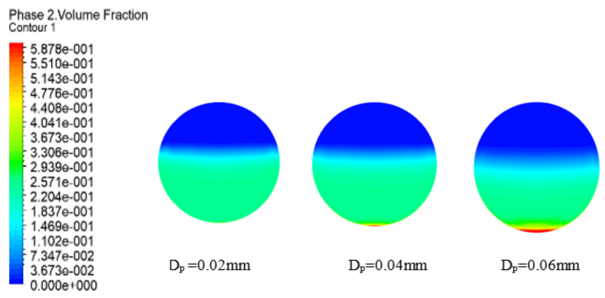

Figure 7: Outlet contours at different particle size

With the increase in particle size, deposition of sand particles at bottom is increased. the pattern gets denser at lower regions and the density of particle at lower regions increases with increase in particle size.

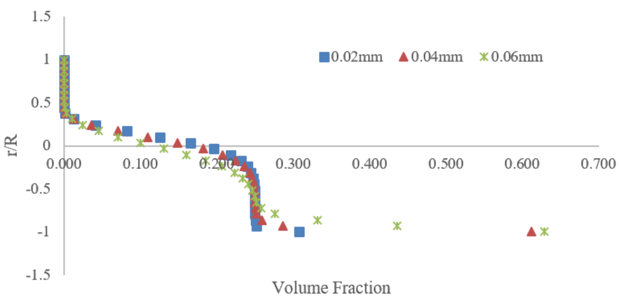

Figure 8: Particle concentrations at outlet

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 sand was increased from 0.02mm to 0.06mm at the bottom of pipe at outlet. The increase is maximum 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.

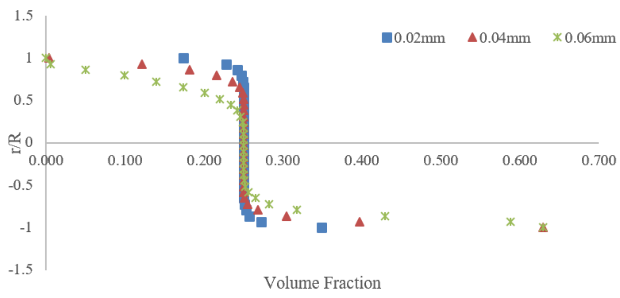

Figure 9: Particle concentrations at mid-plane

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 of 0.04mm and 0.06mm is 0.629 and 0.629 respectively.

3.3 Results at Variable Initial Fraction of Mixture

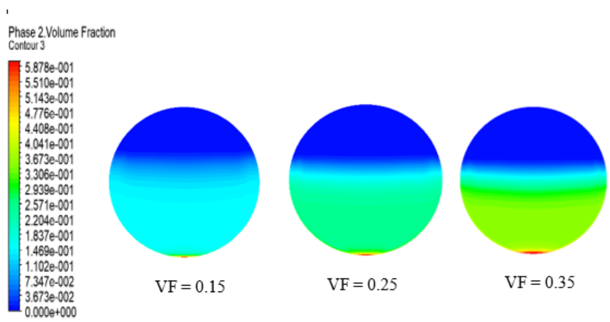

Figure 10: Outlet Contours at different initial volume fraction

The sand particle concentration at outlet contour bottom region increases 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.

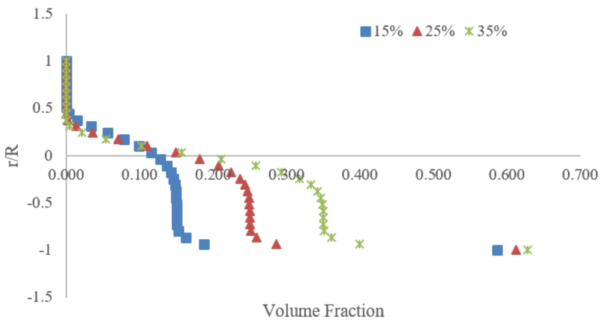

Figure 11: Particle concentrations at outlet

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.

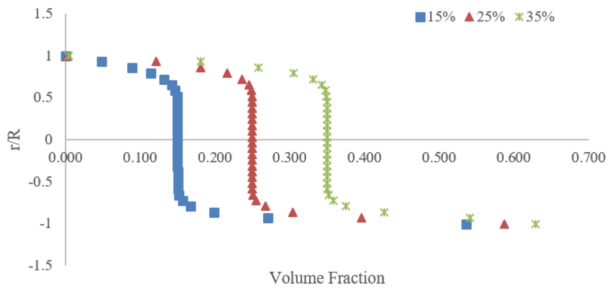

Figure 12: Particle concentrations at mid-plane

Particle concentration at the middle section of pipe was found increasing as the mixture volume fraction of sand was increased from 15

3.4 Results at Variable Initial Velocity

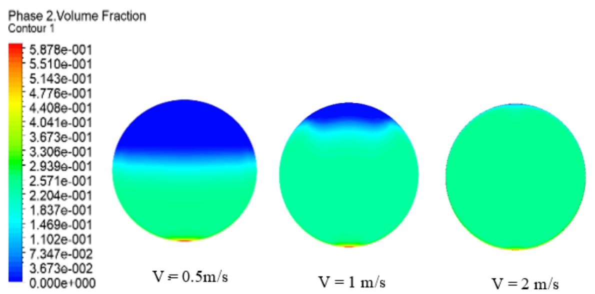

Figure 13: Contours at outlet for different velocity

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

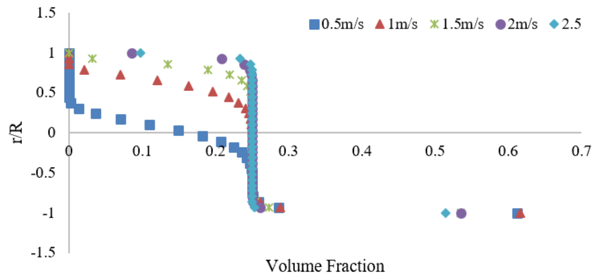

Figure 14: Particle concentrations at outlet

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 settlements 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.

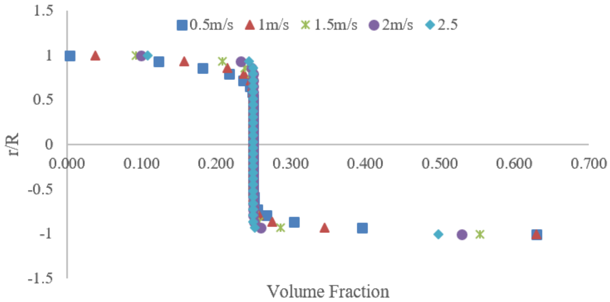

Figure 15: Particle concentrations at mid-plane

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. Conclusions

Deposition of sand particles increases with the decreasing velocity and decreases with the increase in velocity hence the model can also be used for determining the optimum velocity for minimum deposition of solid particles in pipe for different geometry.

For the constant velocity deposition of particles increased with the increase in initial particle volume fraction and particle size. However the settlement of particles were uniform in mid-planes and random in outlet section all cases. This might be due to outlet sections more exposure to atmospheric conditions.

The maximum concentration for inlet mixture velocity of 0.5m/s , 1m/s, 1.5m/s 2m/s and 2.5m/s velocity at outlet was found to be 0.626, 0.617, 0.536, 0.535 and 0.515 respectively, and at mid-plane was found to be 0.629, 0.629, 0.553, 0.529 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, 0.587 and 0.629 at mid-plane 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 and 0.629 respectively at mid-plane.

Change in pipe length had no impact in particle concentration at certain distance from inlet.

5. Recommendations

More closure in bend conditions of various bend angle is the further interest of the study for both curved and sharp bend cases.

Acknowledgments

The authors would like to acknowledge and extend heartfelt gratitude to Mr. Sudip Bhattrai for his continuous help and suggestions to keep the study in track. The authors are grateful to Institute of Engineering [IOE], Department of Mechanical Engineering, Pulchowk Campus for providing access to computer workstations.

References

[1] A. Goharzadeh and P. Rodgers. Experimental characterization of solid particle transport by slug flow using particle image velocimetry. *Journal of Physics: Conference Series*, 2009.

[2] Patro Pandaba and Patro Brundaban. Kinetic theory based cfd modeling of particulate flows in horizontal pipes. *International Journal of Mathematical, Computational, Physical, Electrical and Computer Engineering*, pages 1–7, 2013.

[3] http://www.hydrolab.org/resndev.htm. 2017.

[4] Boris V Balakin, Alex C Hoffmann, Pawel Kosinski, and Lee D Rhyne. *Eulerian-Eulerian CFD Model for the Sedimentation of Spherical Particles in Suspension with High Particle Concentrations*, volume 1. 2010.

[5] A. Ekambara, R. S. Sanders, K. Nandakumar, and H.Masliyah. Hydrodynamic simulation of horizontal slurry pipeline flow using ansys-cfx. *Industrial Engineering and Chemistry Research*, 48:17, 2009.

[6] G. Micale, G. Montante, F. Grisafi, A. Brucato, and J. Godfrey. Cfd simulation of particle distribution in stirred vessels. *Trans IChemE*, 78:1, 2000.

[7] M.A. Delele, F. Weigler, G. Franke, and J.Mellmann. Studying the solids and fluid flow behavior in rotary drums based on a multiphase cfd model. *Powder Technology*, pages 260–271, 2016.

[8] T.N Olfei and A.Y Ismail. Eulerian-eulerian simulation of particle-liquid slurry flow in horizontal pipe. *Journal of Petroleum Engineering*, 2016.

[9] Tamer Nabil, El-Nahhas, and I. E.-S. Computational fluid dynamics simulation of the solid-liquid slurry flow in a pipeline. *Seventeenth International Water Conference*, pages 2–3, 2013.