Analysis of Effect of Particle Size on Pressure Drop and Phase Distribution in a Slurry pipeline

Using Ansys Fluent Software

Sridhar Sahoo
M.tech Scholar
Dept. of Mechanical Engineering (Thermal)
NIT, RAIPUR

Mr. Satish Dewangan
Assistant Professor
Dept. of Mechanical Engineering
NIT, RAIPUR

Dr. Shobhalata Sinha
Professor
Dept. of Mechanical Engineering
NIT, RAIPUR

Abstract **- The hydraulic transport of solid particle is very much demanding area in industry as well as academic research area. As Slurry comes under Multiphase flow, its analysis is a bit difficult and hectic work if done experimentally, till now lots of research paper on the slurry flow in pipeline has been published. Experimental work is done in some paper to reduce the specific power consumption required for transportation of slurry. In this paper, it has been analyzed through simulation and checked the parameters of slurry at different location of the pipeline. It will help in determining the particular rheology of slurry to be transported and a way of optimization to it. In order to investigate the parameters , a slurry of glass bead and water is made where the glass-beads of three different sizes 50µm,75µm and 100µm respectively were used at a particular concentration of .25(i.e. 25%) . The whole problem is solved using a simulation software Ansys v12.0 and in fluent solver using eulerian-eulerian approach to solve the problem. The simulation analysis shows that the effect of higher size particle is somewhat different from the lower size particle .After attaining the fully developed condition the concentrations of solid particles present in slurry is plotted at different planes for three different size particles, and the relative pressure drop is compared for all three particle sizes. It shows a better accuracy with the experimental results of other researchers in this field.**

Keywords: Ansys_Fluent , Multiphase flow, Slurry Flow, Fully Developed Flow.

I. INTRODUCTION

Hydraulic transportation of solid materials in the form of slurry through pipelines has been one of the widely used methods for conveying bulk quantity of materials. Slurry is a mixture of a solid particles and fluid held in suspension where water is the most commonly used fluid. For laminar to a turbulent flow a single-phase liquid of low absolute viscosity can be allowed to flow at low speeds. The mixture resists the flow in highly viscous mixturesbecause of excessively low shear rate in the pipeline. Due to the relatively low operation and maintenance costs, slurry pipelines are typically used in chemical and mining industries for long distance transport of bulk materials such as oil sand ore, coal, copper, iron and phosphate concentrates, among others, to processing plants. Reduction in energy requirement for hydraulic transportation of solids has been the main concern of researchers. Effective design and control of hydraulic transport system requires the successful prediction of slurry flow behavior in the pipeline, slurries are showing distinct two phase behavior which is affected by mutual particle-particle and particle- liquid interaction. The power required for the pump is determined from the pressure losses in the pipelines, to minimize the size of motor, pump, gearbox and other item, therefore an addition to optimize pressure loss. El-Nahnas et al.(1997) and Kaushal et al.(2005) proposed the effect of particle size on the flow behavior and the effective power consumption .

In this paper, three-dimensional concentration distributions and pressure drops are modeled using eulerian model in 54.9 mm diameter and 3 meter long horizontal pipeline, the slurry of water and glass-bead has been analyzed at a particular concentration for different diameter glass beads varying as 50µm,75µm and 100µm and a specific gravity of 2.47, for a velocity of 2.5m/s and concentration of 25%, using the computational approach in Ansys v12.0 ,Fluent solver. The problem can be further worked by taking consideration to higher size of particle and for different concentration and velocity.

II. MATHEMATICAL MODEL

Eulerian model has been chosen, as it allows for multiple separate phases. The phases can be liquids, gases or solids in any combination. An eulerian approach was used for each phase. In this model granular flow is taken in consideration where glass-beads have been designated as granular phase, A single pressure is shared by all phases. Momentum and continuity equations are solved for each phase; k-ε turbulence model is applied to all phases or to mixtures.

2.1. Governing equations:

2.1.1. Continuity equation

$$\nabla \cdot (\alpha_p \rho_p \vec{v}_p) = 0$$

Where p either solid or fluid

2.1.2. Momentum equations

For fluid phase:

$$\nabla \cdot (\alpha_f \rho_f \vec{v}_f \vec{v}_f) = -\alpha_f \nabla P + \nabla \cdot \bar{\bar{\tau}}_f + \alpha_f \rho_f \vec{g} + K_{sf}(\vec{v}_s - \vec{v}_f) + F_L + F_{VM}$$

For solid phase:

$$\nabla \cdot (\alpha_s \rho_s \vec{v}_s \vec{v}_s) = -\alpha_s \nabla P - \nabla P_s + \nabla \cdot \bar{\bar{\tau}}_s + \alpha_s \rho_f \vec{g} + K_{fs}(\vec{v}_s - \vec{v}_f) + F_L + F_{VM}$$

f , s,α, ρ, ν, g,$F_L$and F_{VM} represents fluid phase, solid phase, concentration of particle, density , velocity , acceleration due to gravity , lift force and virtual mass force respectively.

2.1.3. Energy equation:

$$\frac{\partial}{\partial t}(\alpha_q \rho_q h_q) + \nabla \cdot (\alpha_q \rho_q \vec{u}_q h_q) = \alpha_q \frac{\partial P_q}{\partial t} + \bar{\bar{\tau}}_q : \nabla \vec{u}_q - \nabla \cdot \vec{q}_q + S_q + \sum_{p=1}^{n} (Q_{pq} + \dot{m}_{pq} h_{pq} - \dot{m}_{qp} h_{qp})$$

q, h_q , $\vec{q}_q$, S_q , Q_{pq} ,h_{qp} represents phase ,specific enthalpy of q phase, heat flux, source term , heat exchange between p and q phase and interphase enthalpy respectively.

III. Methodology

3.1. Geometry

The geometry of the pipe was modeled using default AnsysDM v12.0. The pipe was of 3m length and 54.9 mm diameter straight pipeline. The length of the pipe was sufficient for confirmation of fully developed flow. The pressure drop was computed between last 0.2m from outlet, and concentration distribution was observed at three different point of 2.8m, 2.9m and 3m from inlet respectively.

3.2. Mesh

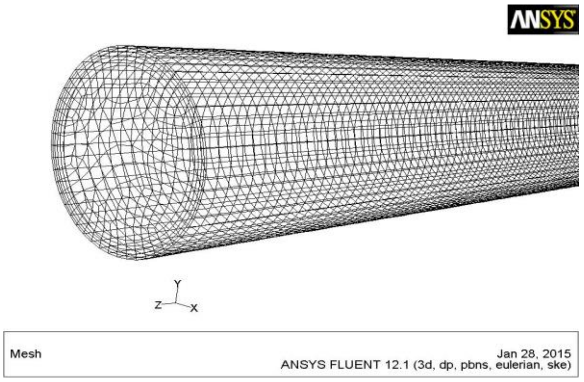

Fig 1. Three dimensional meshing of slurry pipeline

A mesh of hexahedral elements was used to discretize the computation domain using AnsysMeshing, as shown in fig 1. The grid distribution on the circumference of pipeline was uniform. Mesh having 3, 67,562nodes and 1 cell zone and 4 face zone. For accuracy in solving, the mesh size is kept very fine near wall so that the phase and velocity variation near the wall was found out.

3.3. Solver

All the governing equation such as continuity, momentum and energy, for all the boundary conditions was solved using CFD solver FLUENT v12.0

3.3.1. Model

This is a pressure based absolute velocity formulated steady state approach for the problem. Multiphase Eulerian, implicit linear equation scheme was used to solve the resultant scalar system of equations; k-ε turbulence model with standard wall function treatment near the wall was used. The granular viscosity and bulk viscosity was kept as syamlal-obrien andlun-et-al. respectively.

3.3.2. Boundary conditions

The velocity inlet condition was used for the inlet of wall with a flow velocity of 2.5 m/s at a 25% concentration, similarly outflow condition was used for outlet wall. For the wall there is no slip condition and stationary wall condition prevails.

3.3.3. Solution Controls

The phase coupled simple scheme was selected for pressure velocity compounding; first order upwind scheme for volume fraction, turbulence kinetic energy, turbulence dissipation rate was used whereas second order upwind scheme was used for momentum.To avoid divergence.the under relaxation factors for pressure was .3 , 1 for density and body force , 0.7 for momentum and volume fraction.

IV. Results And Discussion

4.1.Concentration distribution of solid particle

The results for distribution of solid particle have been plotted in fig. 2-4 for different plane (at 2.8m, 2.9m and 3m from inlet respectively). It is observed that the concentration of solid deposit increases at lower portion of the pipe with increase in distance from inlet, due to the effect of gravity.

The concentration profiles have been plotted at different location taking area weighted average along a vertical line drawn on the plane, for observing the difference in distribution of solid particle. as shown in fig. 5-7.

4.2. Pressure loss

Comparison of pressure drops is presented in fig 8. It is observed that as size of the particle increases the pressure drop also increases due to the frictional loss, loss due to shear etc.

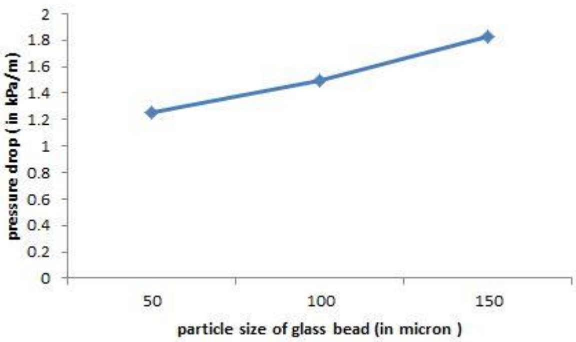

Fig. 8. Pressure drop at different particle size

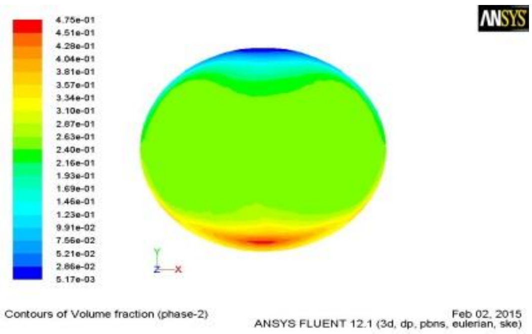

Fig. 2.1.At z=2.8m from inlet

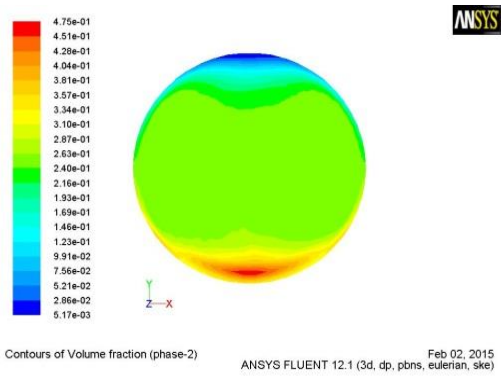

Fig. 2.2.At z=2.9m from inlet

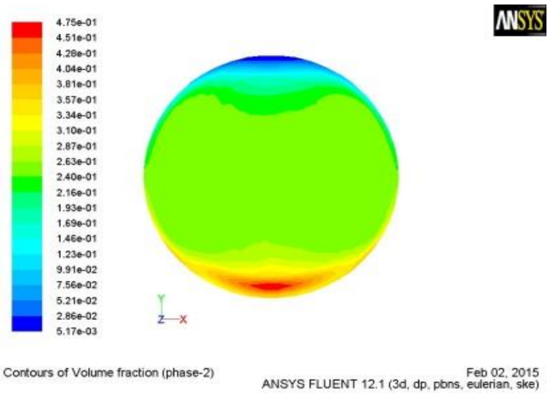

Fig. 2.3.At z=3m from inlet

(Fig. 2. Concentration distribution for 50 micron particle)

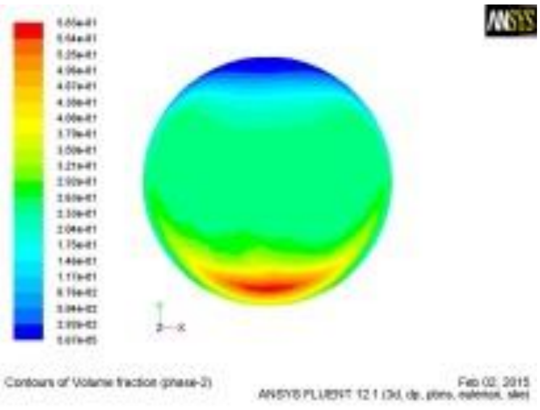

Fig. Fig 3.1. At z=2.8m from inlet

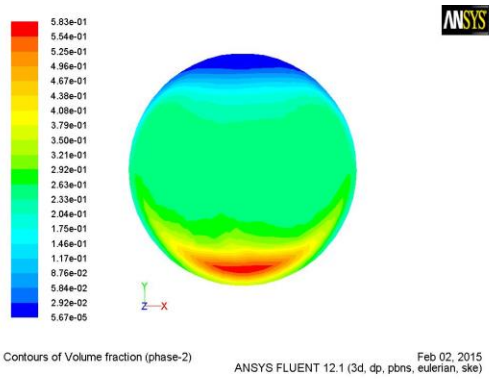

Fig. 3.2. At z=2.9m from inlet

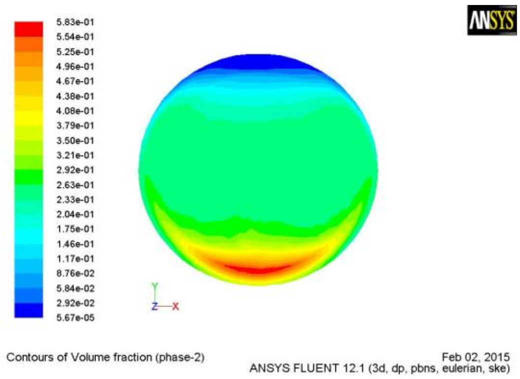

Fig. 3.3. At z= 3m from inlet

(Fig. 3. Concentration distribution for 75 micron particle)

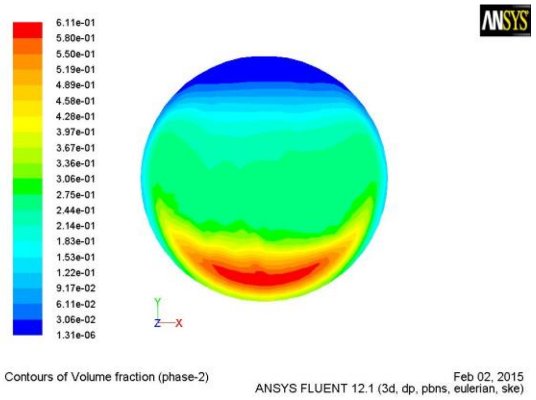

Fig. 4.1. At z=2.8m from inlet

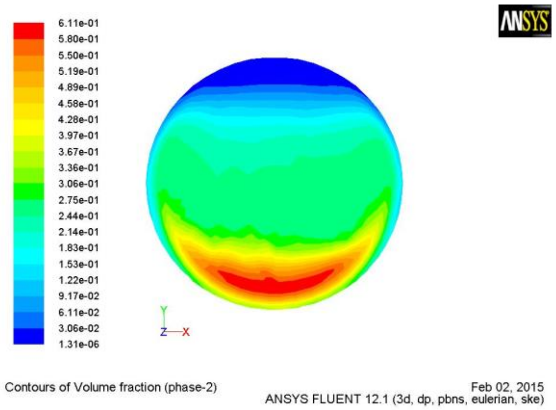

Fig. 4.2. At z=2.9m from inlet

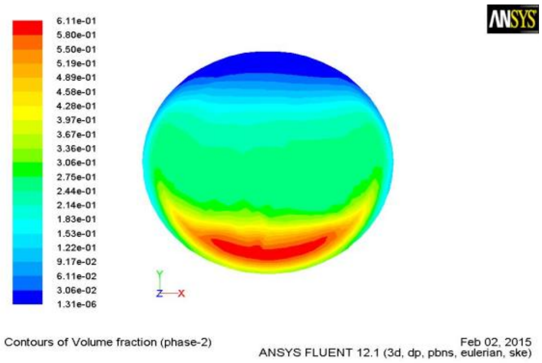

Fig. 4.3. At z= 3m from inlet

(Fig. 4. Concentration distribution for 100 micron particle)

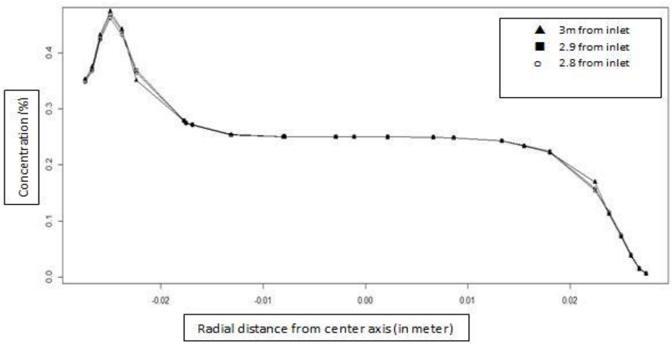

Fig. 5. Concentration profile variation for50 micron particle

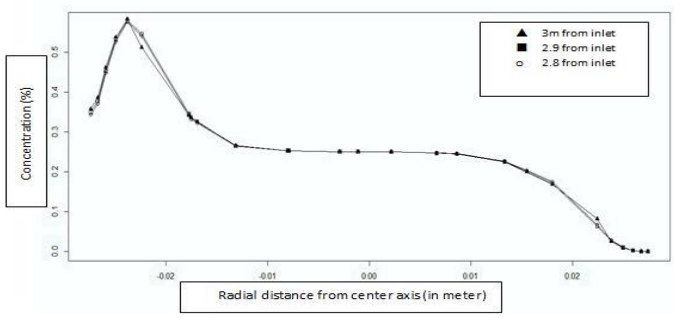

Fig. 6. Concentration profile variation for 75 micron particle

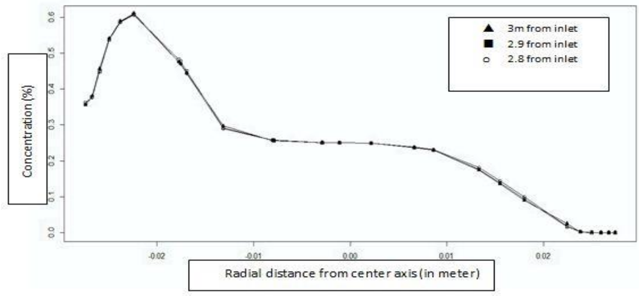

Fig.7.Concentration profile variation for 100 micron particle

It is seen that trend of the simulations result matches accurately with the experimental results of [5]. And further research can be done for higher size particles as well as for type of slurry taking into consideration.

V. CONCLUSION

The deposition of solid particle increases in the bottom portion of the pipeline as distance is increased from inlet , due to decrease in velocity and effect of gravity .

Concentration of solid particle is much less, can be assumed zero for upper portion of the pipeline.

Pressure drop increases with increase in particle size for a particular concentration, due to increase in frictional force and wall shear stress.

REFERENCES

[1]. J. Ling, P.V. Skudarnov, C.X. Lin, M.A. Ebadian"Numerical Investigations Of Liquid-Solid Slurry Flows In A Fully Developed Turbulent Flow Region" International Journal of heat and Fluid Flow 24(2003) 389-398.

[2]. Kamal El-Nahhas , Nageh Gad El-Hak, MagdyAbouRayan and Imam El- sawaf "Effect Of Particle Size Distribution On The Hydraulic Transport Of Settling Slurries "Thirteenth International Water Technology Conference,IWTC13 2009, Hurghada, Egypt.

[3]. Tamer Nabil 1, Imam El-Sawaf 2, and Kamal El-Nahhas "Computational Fluid Dynamics Simulation Of The Solidliquid Slurry Flow In A Pipeline"Seventeenth International Water Technology Conference, I WTC 17, Istanbul, 5-7, November 2013.

[4]. D.R. Kaushal, Kimihiko Sato, Takeshi Toyota ,KatsuyaFunatsu , Yuji Tomita "Effect Of Particle Size Distribution On Pressure Drop And Concentration Profile In Pipeline Flow Of Highly Concentrated Slurry" International Journal of Multiphase Flow 31 (2005) 809–823.

[5]. D.R.Kaushal, T.Thinglas, Yuji Tomita, Shigeru Kuchii, Hiroshi Tsukamoto "Cfd Modeling For Pipeline Flow Of Fine Particles At High Concentration" International journal of Multiphase Flow 43(2012) 85-100

[6]. Fluent Inc. ,FLUENT 12.0, User's Guide, Theory Guide