MODELLING AND NUMERICAL SIMULATION OF SLURRY FLOW

JANUÁRIO, João Rodolfo, joaorjanuario@gmail.com
MAIA, Cristiana Brasil, cristiana@pucminas.br
Pontifícia Universidade Católica de Minas Gerais , Av. Dom José Gaspar, 500, Coração Eucarístico, Belo Horizonte, MG, Brasil, 30535-901

Abstract. *In countries with large territorial extension, the transport of production is always a crucial factor in the competitiveness of enterprises or even in their maintenance. The transport of raw material can be done in several ways. A low cost solution would be mixing the particulate with water, producing a slurry , which can be transported along pipes for hydraulic pumps. Particle transport through pipes is an important operation in many industries, and detailed information of the process is required. CFD studies in solid-liquid slurry flows have been developed by many authors, but still have not been widely performed. In this paper, an open-source software was used to simulate the slurry flow in a pipeline. An equivalent viscosity was used to represent the viscosity of the slurry. Numerical results were compared to experimental data from the literature, and a maximum difference of 2.2% was found for the pressure drop.Results showed the regions with higher deposition of particles, and indicated that the mass flow rate should be increased to prevent this problem.*

Keywords*: Rheology, Computational Fluid Dynamics, Non-Newtonian fluid, Slurry*

1. INTRODUCTION

Transportation of slurries through pipeline is common in many industries including foods, pharmaceuticals, chemicals and mining industries. It is important to develop accurate models to predict pressure drop, velocity profile, and concentration distribution of the slurry (Nabil et al., 2014). In many of these applications the fluid is highly viscous and may have a Newtonian or non-Newtonian rheology and flow is usually turbulent, which makes the analysis more complex (Lahiri, Ghanta, 2010).

Currently, the study of non-Newtonian fluids is becoming important in industrial development. For the simulation of a fluid particle (in a homogeneous or heterogeneous mixture), the knowledge of the location of the suspended particle and of the flow deposition rate is necessary to avoid obstructions in the path. An important parameter in the design of the ducts is the critical flow velocity; where the pulp has a Newtonian fluid behavior, and the generated turbulence prevents the particles to accumulate in the ducts, especially where there are greater pressure losses (PINTO, 2014).

Pinto (2014) developed an analytical model to determine the critical velocity of fluids containing different concentrations of apatite, hematite and quartz, flowing through a PVC pipe. The results were validated by comparison to experimental data. The results of critical velocity showed maximum deviations in 10% from the curves obtained experimentally.

Analytical and numerical studies can be performed in the design of pipes with slurry flows. Analytical studies give an overall view of the flow. Numerical studies can predict specific problems, such as regions with deposition of particles.

One of the first papers on the slurry transport was the work of Durand (1952), who measured pressure losses distributed on the system, and determined the deposition rate. Recently, Sutalo et al. (2006) developed a numerical model of the flow of mineral pulp on inclined surfaces in industrial equipment. According to the authors, the knowledge of this behavior would lead to increased efficiency in the devices. The authors used the Herschel-Bulkley model to determine the viscosity of the fluid. The numerical simulation was performed using the commercial software ANSYS-CFX software, and the results were validated by comparison to experimental data. The results indicated that the analytical model is suitable to describe the flow behavior in the first inclined plate. Although the error greatly increased in subsequent plates, the CFD model was able to calculate the difference of the height in the fluid's free surface at the subsequent plates, by considering the film height's variation in time due to transient effects, ensuring higher accuracy of the results.

Balakin et al. (2011) made a comparison between an experimental model and a numerical model to calculate the formation, size, quantity and the influence on the hydrates of water flow, similar compounds with ice, that modify the fluid properties (viscosity for example) and can increase the velocity and pressure losses. For the author, combining water and Freon R11 led to the formation of hydrates. The author also observed that using a homogeneous flow-water mixture hydrates, the numerical model presented satisfactory results. The experiment with the formation of a fixed bed of solid particles showed that the diameter of formed particles is inversely proportional to the velocity flow, reducing the losses, as seen in the experiment. The viscosity presented values ten times higher near to the particles bed.

One must design the ducts to achieve optimal operating parameters, such as critical velocity, and still use the minimum amount of material in order to decrease the safety factors. Numerical methods show punctual defects distributed along the path, enabling specific design in various pipeline sections, from the inlet, where there is introduction of slurry, until the exit, at atmospheric pressure near the ports.

In this paper, the flow of water and slurry (water mixed with 12% apatite) were predicted using OpenFoam software. The viscosity of the non-Newtonian fluid was determined using the equivalent viscosity model. Experimental results from the literature (Pinto, 2014) were used to validate the model.

2. MATHEMATICAL MODEL

The governing equations are the conservation equations of mass, momentum and energy, given by (Versteeg, Malalasekera, 2007):

$$\frac{\delta\rho}{\delta t}+\frac{\delta\rho}{\delta x_j}(\rho u_j)=0 \tag{1}$$

$$\frac{\delta}{\delta t}(\rho u_i)+\frac{\delta}{\delta x_j}(\rho u_j u_i)=-\frac{\delta P}{\delta x_j}+\frac{\delta}{\delta x_j}\left(\mu\frac{\delta T}{\delta x_j}\right)+S_u \tag{2}$$

$$\frac{\delta}{\delta t}(\rho T)+\frac{\delta}{\delta x_j}(\rho u_j T)=\frac{\delta}{\delta x_j}\left(\frac{k}{C_p}\frac{\delta T}{\delta x_j}\right)+S_T \tag{3}$$

It was used the k-ε turbulence model to evaluate the turbulence inside the system. The transport equations for the turbulent kinetic energy k and the dissipation of the turbulent kinetic energy ε are:

$$\frac{\delta}{\delta t}(\rho k)+\frac{\delta}{\delta x_i}(\rho U_i k)=2\mu_t E_{ij}E_{ij}-\rho\varepsilon+\frac{\delta}{\delta x_j}(\frac{\mu_t}{\sigma_\varepsilon}\frac{\delta k}{\delta x_j}) \tag{4}$$

$$\frac{\delta}{\delta t}(\rho\varepsilon)+\frac{\delta}{\delta x_i}(\rho U_i\varepsilon)=C_{1e}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij}-C_{2e}\rho\frac{\varepsilon^2}{k}+\frac{\delta}{\delta x_j}(\frac{\mu_t}{\sigma_\varepsilon}\frac{\delta\varepsilon}{\delta x_j}) \tag{5}$$

In the previous equations, it is assumed a Newtonian fluid. In Newtonian fluids, the shear stress is directly proportional to the deformation rate. In non-Newtonian fluids, this relationship is not observed. They may be adequately represented by the power-law model, which for one-dimensional flow becomes (Fox, Pritchard, McDonald, 2012)

$$\tau=k\left(\frac{du}{dy}\right)^n \tag{6}$$

Where n is called the flow behavior index and k is the consistency index. This equation is rewritten in the form

$$\tau=\eta\frac{du}{dy} \tag{7}$$

η is the apparent viscosity.

In this work, the fluid is water with 12% of apatite, which is a non-Newtonian fluid. Literature (Garcia, 2014) indicates that the problem can be modelled using the Navier-Stokes equations, replacing the absolute viscosity by the apparent viscosity η.

The apparent viscosity is determined using the model described by Wilson et al. (2006). Using experimental data of the pressure drop from the literature, the shear stress is determined according to

$$\tau=\frac{\Delta P}{L}\times\frac{D}{4} \tag{8}$$

D and L represent the tube diameter and length, respectively.

With the shear stress, the apparent viscosity, referred as equivalent viscosityμ_{eq}, is determined according to Wilson et al. (2006). The same author indicates a way to calculate the equivalent viscosity, based on the mean velocity and shear velocity, the shear stress divided by specific mass square root (Eq. 9).

$$\mu_{eq}=\frac{\rho\times\sqrt{\frac{\tau}{\rho}}\times D}{\exp\left(\frac{Vm}{2{,}5\times\sqrt{\frac{\tau}{\rho}}}\right)} \tag{9}$$

Where ρ represents the density of the fluid and Vm represents the average velocity inside the tube.

3. METHODOLOGY

In this paper, it is predicted the behavior of the flow of water mixed with 12% apatite in a tube system. Experimental data of geometry, properties and mass flow rate from the literature (Pinto, 2012) was used an input data. Numerical results of pressure drop were compared to experimental data.

3.1 Experimental setup

Experimental data from the literature (Pinto, 2014) was used as input to the mathematical model. The experimental pulp pumping plant is shown in Figure 1. The nominal diameter of the tubes is 2". Five velocities were evaluated by the author, and the pressure drop between points A and B was measured in each case, as indicated in Tab. 1. The fluid used was water with 12% of apatite, with granulometry of 249-297 μm.

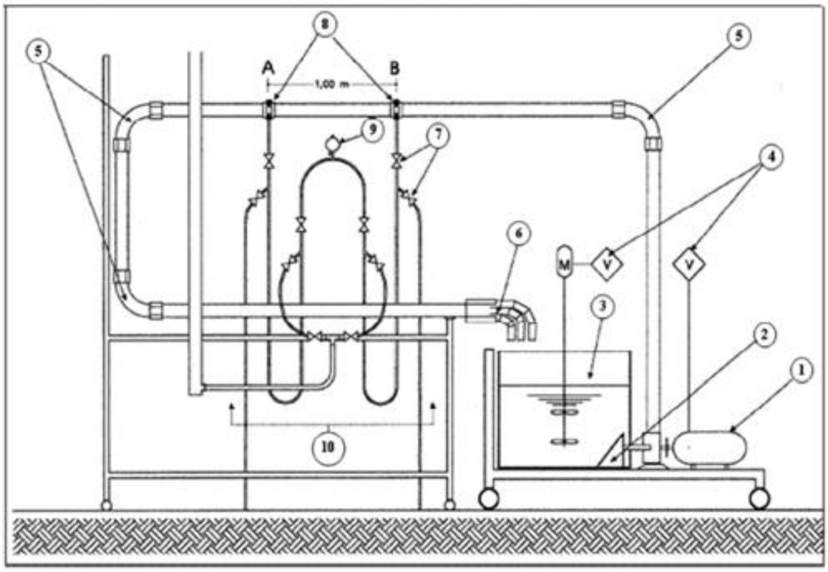

Figure 1. Experimental setup (Pinto, 2014)

Table 1. Experimental mean velocity and pressure loss data of slurry

Mean Velocity(m/s)	Pressure drop (Pa/m)
1.3	1176
1.8	1029
2.1	1274
2.4	1469
2.7	1764

Data from Tab. 1 was used in Eq. (6) to determine the equivalent viscosity to be used in Navier-Stokes equations. Also, the geometric parameters were used to build the numerical model. The internal diameter of the tube is 50.1 mm (corresponding to a nominal diameter of 2", the height of the system is 1620 mm, and the total length is 5130 mm (Pinto, 2014). The material of the tubes is PVC.

Pinto (2014) also run experiments using pure water (density of 1000 kg/m^3), and the results of pressure drop between points A and B are shown in Tab. 2.

Table 2. Experimental mean velocity and pressure loss data of water

Mean Velocity(m/s)	Pressure Loss(Pa/m)
1.3	300
1.9	620
2.5	1013
3.0	1490

These results were used to validate the model used for a Newtonian fluid.

3.2 Numerical model

The software OpenFOAM and the solver buoyantPimpleFoam were used in the numerical simulation. The main assumptions are: steady state flow, constant properties, and heat transfer neglected. According to experimental data, it was assumed a density of 1257 kg/m³.

The boundary conditions adopted were velocity prescribed at the inlet (according to Tab. 1), and in the outlet, atmospheric pressure. In the walls, no slip condition was adopted. Four cases were simulated, corresponding to the velocities given in Tab. 1.

A mesh test was performed, as indicated in Tab. 3. To evaluate the independence of the results relative to the mesh, three conditions were tested. The conditions were the maximum velocity of water (3 m/s), the minimum velocity of slurry (1.3 m/s) and the maximum velocity of slurry (2.7 m/s). The velocity and the pressure at a specified point were compared for each mesh.

For water at 3 m/s, the difference between the velocity data for meshes 1 and 2 was 1.79% and between meshes 2 and 3, 1.04%. For the pressure, the difference between meshes 1 and 2 was 0.12% and between meshes 2 and 3, 1.89%. For slurry at 1.3 m/s, the differences of velocity were 0.33% and 0.67%, and the differences of pressure were 0.10% and 0.58%, respectively, for meshes 1 and 2 and 2 and 3. For slurry at 2.7 m/s, the differences of velocity were 2.28% and 0.52%, and the differences of pressure were 0.144 and 1.43%, respectively, for meshes 1 and 2 and 2 and 3.

Since results did not significantly vary between the evaluated meshes, and considering the increase on the computational time required to run the simulations, mesh 1 was selected. The mesh used is shown in Fig. 2.

Table 3. Mesh test

Mesh	Number of elements
1	250560
2	443700
3	794952

Table 4. Mesh test for water at mean velocity 3.0m/s

Mesh	Pressure point 1	Pressure point 2	Velocity point 1	Velocity point 2
Mesh 1	92170	90673	3.2	3.3
Mesh 2	92159	90683	3.2	3.3
Mesh 3	91983	90512	3.3	3.3

Table 5. Mesh test for slurry at mean velocity 1.8m/s (left) and 2.7m/s (right)

Mesh	Pressure point 1	Pressure point 2	Velocity point 1	Velocity point 2	Pressure point 1	Pressure point 2	Velocity point 1	Velocity point 2
Mesh 1	85928	84869	2.0	2. 2	89353	87587	3.0	3.1
Mesh 2	85912	84867	2.0	2.2	89313	87526	3.0	3.2
Mesh 3	85822	84779	2.0	2.2	89138	87346	3.0	3.2

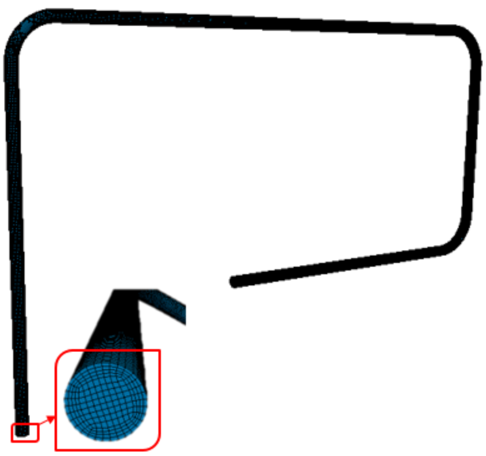

Figure 2. Mesh

The problem was solved using a computer with 32Gb of RAM and a i7 processor with 3.4GHz. The solver ran until a convergence of 10^{-6} was obtained for all variables. The numerical uncertainties were evaluated according to the methodology described by Eça et al, 2014.

4. RESULTS AND DISCUSSION

The first analysis was the comparison between numerical and experimental data, using water at two velocities, 1.3 m/s and 3.0 m/s. Table 6 shows the results for the pressure drop between points A and B. It can be seen that the maximum difference, 0.82%, was obtained for the higher velocity.

Table 6. Comparison of experimental and numerical data about water

Mean Velocity (m/s)	Numerical pressure drop (Pa)	Experimental pressure drop (Pa)	Error (%)
1.3	301.4	300	0.47
3.0	1477.8	1490	0.82

Figure 3 presents the pressure field inside the system (left side) and the pressure in the centerline of the tube between points A and B (right side). It can be seen that, between points A and B, the pressure decreases linearly, as expected.

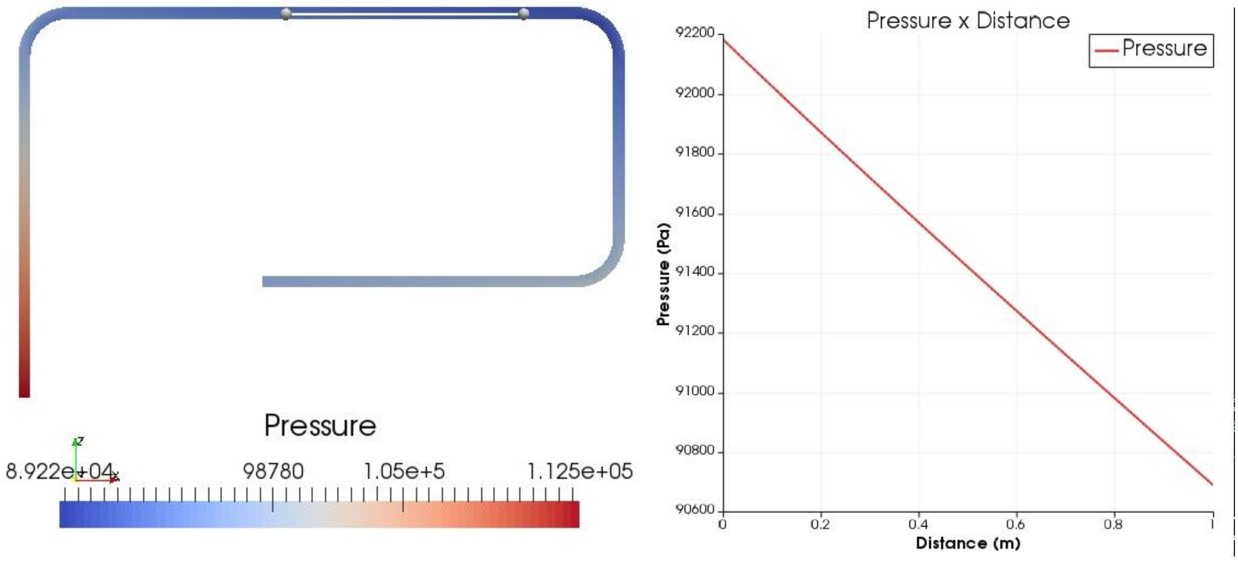

Figure 3. Results using water and mean velocity 3.0m/s

The second analysis was performed using water mixed with apatite. In order to run the simulations, it was necessary first to determine the equivalent viscosity, using experimental data of pressure drop and average velocity. The values are described in Tab. 7. The equivalent viscosity decreases with increasing velocity.

Table 7. Equivalent viscosity

Mean Velocity (m/s)	Shear Stress (Pa)	Equivalent Viscosity (Pa.s)
1.8	13.1	0.0056
2.1	16.2	0.0044
2.4	18.7	0.0029
2.7	22.4	0.0026

According to Pinto (2014), the determination of the equivalent viscosity can only be performed using this analysis for velocities higher than the critical velocity, 1.8 m/s in this case. Therefore, the numerical simulations were only done for velocities higher than 1.8 m/s. Figure 4 shows the pressure field and the pressure drop between points A and B, for the average velocity of slurry of 2.7 m/s. The general behavior is similar to the behavior of water, but with higher pressure drops. For the other velocities, similar results were obtained, and will not be shown.

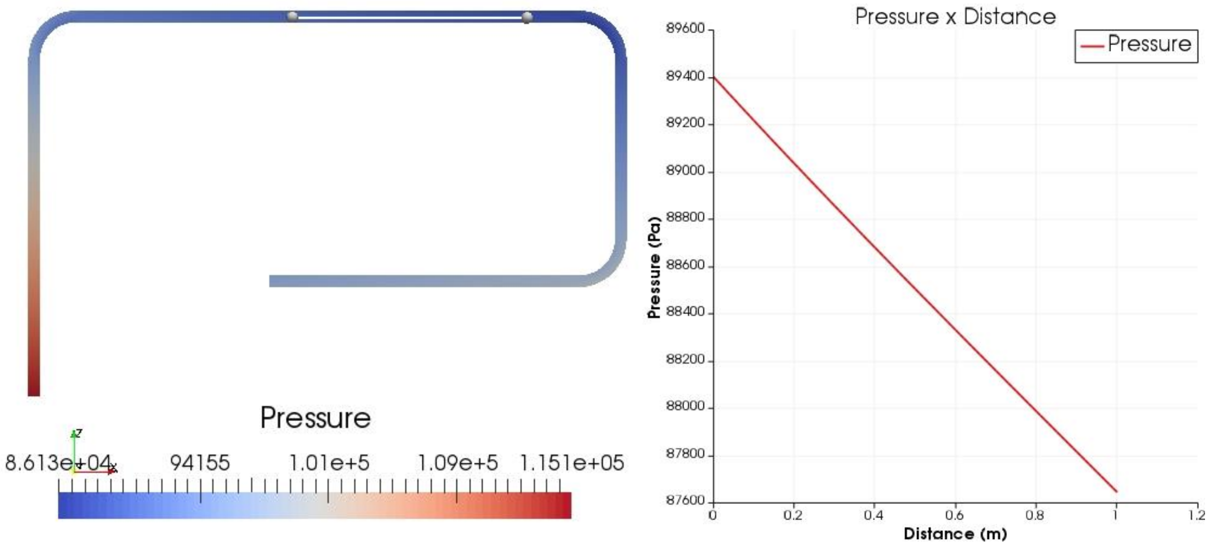

Figure 4. Results using slurry and mean velocity 2.7m/s

Table 8 presents the comparison between experimental and numerical values of the pressure drop between points A and B, for the evaluated velocities. The maximum difference was 2.18%, corresponding to the lower velocity. The higher difference in this velocity may be attributed to the critical velocity, which is the limit to use the equation for the equivalent viscosity. It can be seen also that the pressure drop increases with the velocity, as already obtained by experimental data.

Table 8. Comparison of experimental and numerical data about slurry

Mean Velocity (m/s)	Numerical pressure drop (Pa)	Experimental pressure drop (Pa)	Error (%)
1.8	1006.6	1029	2.18
2.1	1256.2	1274	1.40
2.4	1459.9	1469	0.62
2.7	1782.3	1764	1.04

Figure 5. Velocity streamlines for slurry at 2.7m/s

Figure 5 shows the streamlines for the slurry with average velocity of 2.7 m/s. It can be seen that the maximum velocity was about 3.3m/s and no zones of recirculation are found.

5. CONCLUSIONS

Copyright © 2012 by ABCM

In this work, a numerical study of the slurry flow was performed. Numerical results of slurry and water were compared to experimental data, and low differences were observed. A pressure loss increase of about 31% was observed with the slurry, for similar velocities.

It was assumed that the non-Newtonian fluid problem could be solved using Navier-Stokes equations, using an equivalent viscosity. The low difference between numerical and experimental results evidence that this methodology is suitable to this problem.

6. ACKNOWLEDGEMENTS

The authors generously acknowledge the support of PUC Minas, FAPEMIG and CNPq.

7. REFERENCES

Balakin, B.V., A.C. Hoffmann. P. Kosinski. Experimental study and computational fluid dynamics modeling of deposition of hydrate particles in a pipeline with turbulent water flow. Chemical engineering science. 2011. P755-765.

Durand, R. The hydraulic transportation of coal and other materials in pipes. Conf. Of national coal board, londres. 1952.

Eça, Luis; Hoekstra, Martin. "A procedure for the estimation of the numerical uncertainty of cfd calculations based on grid refinement studies".journal of computational physics, v. 262, p. 104-130, 2014.

Fox, R.. W., Mcdonald, A. T., Pritchard, P. J., & introdução, à. (2012). Mecânica dos fluidos. Rio de janeiro, 8.ed.

Garcia, Luís Paulo. Obtenção de parâmetros reológicos de polpas minerais contendo partículas grossas a partir de ensaios de bombeamento. Unisanta. 2014.

Nabil, Tamer; El-Sawaf, Imam; El-Nahhas, Kamal. "Sand-water slurry flow modelling in a horizontal pipeline by computational fluid dynamics technique". International water technology journal. Vol. 4- n.1, march 2014.

Pinto, Tc Souza. D. Moraes Junior, P.T. Slatter, L.S. Leal Filho. "Modelling the critical velocity for heterogeneous flow of mineral slurries". International journal of multiphase flow, v. 65, p. 31-37, 2014.

Pinto, Thiago César De Souza. Modelagem da velocidade crítica de transporte de polpas minerais contendo partículas grossas. Thesis. University of são paulo. 2012.

Sandip Kumar Lahiri, K.C. Ghanta. "Slurry flow modelling by cfd". Chemical industry & chemical engineering quarterly 16 (4) 295−308 (2010)

Šutalo, I. D.; Bui, Anh; Rudman, Murray. "The flow of non-newtonian fluids down inclines". Journal of non-newtonian fluid mechanics, v. 136, n. 1, p. 64-75, 2006.

Veersteg, H. K.; Malalasekera, W. An introduction to computational fluid dynamics.essex, england: pearson education limited, 2007.

Wilson, Kenneth C. et al. Slurry transport using centrifugal pumps. Springer science & business media, 2006.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.