X CONGRESSO NACIONAL DE ENGENHARIA MECÂNICA 2018
20 A 24 DE MAIO DE 2018 SALVADOR – BA – BRASIL

Applying variable viscosity to numerical simulation of slurry flow

João Rodolfo Januário, joaorjanuario@gmail.com[1]
Cristiana Brasil Maia, cristiana@pucminas.br[1]
[1] Pontifícia Universidade Católica de Minas Gerais, Av: Dom José Gaspar, 500 Coração Eucarístico - Belo Horizonte - MG 30535-901.

Abstract: *This work has the objective of modeling different non-Newtonian fluid laws from heterogeneous mixtures between water and minerals, showing as advantages and disadvantages found in the method employed of viscosity variation, to increase the transport efficiency of the same. The study method shows numerical methodologies of CFD, using the opensource software openfoam, together with the validation of experimental results obtained in the literature, as well as of fluid characterization methods, for the determination of the productions by this type of Material in transport piping. With the results produced by paths to studies of this type can be shortened, always seeking the most efficient method for each situation. Variable viscosity had results not to rely on further experimental data for value interpolation.*

Keyword: *Mineral pulps, Critical velocity, Heterogeneous mixtures, variable viscosity.*

1. INTRODUCTION

One of the first published works on pulp transport is the work of Durand (1952), where the velocities of deposition of distinct particles in water were measured. The numerical studies allow the velocity profiles to be found in each cross section of the duct, making possible this same calculation with the minimum velocity found, causing errors related to the average velocity to be reduced.

Turian et al. (1987) concluded that the ratio of the critical transport velocity to the pipe diameter is approximately equal to the square root of this diameter, and that for larger particle compound fluids, the critical condition is independent of particle size. Similarly, Turian et al. (1998) studied the friction losses of a pulp in a venturi tube. The authors verified the coefficients of flow resistance for heterogeneous non-Newtonian fluids, and. it was found that the coefficients of resistance have a higher sensitivity when approaching the transition regime, which precedes the turbulent, but the approximations used by the authors in the laminar regime are adequate for use. The non-Newtonian fluid used by the authors did not allow a deeper study of their rheology, because the particles deposited too quickly. In addition, numerical studies have an inherent uncertainty in the process.

Bijjan et al. (2012) verified that in pseudoplastic fluids, the drag coefficient increases with the Reynolds number, while in Newtonian fluids and dilatory fluids show a decline of the same coefficient. The same behavior was observed in relation to the Strouhal number. Wahba (2013) found a behavior for dilatory fluids, where the transient pressure variation when acting on the system had a faster attenuation in relation to the Newtonian fluid. In contrast, pseudoplastic fluids presented a much longer time for stabilization, indicating a lower cushioning.

For studies using numerical methods to be validated, that is, to accurately represent the physical phenomenon, global variables must be compared. For the present study, the validation was done by Souza Pinto (2014), based on analytical modeling and experimental validation of fluids containing different concentrations of apatite, hematite and quartz, flowing through 25.4mm and 50mm tubes; evaluating the loss of load and the critical velocity of each case. The test made by the author introduced the pulp in the tubing through a centrifugal pump, and then a zone of interest was determined, where the flow was completely developed, and then measured flow (to obtain average velocity) and pressure, thus being able to relate velocity with loss of pressure. In addition, the transparent tubes allowed to observe the moment of deposition of particles in the tube, determining the point where the velocity is critical, and also the point where the minimum loss value of pressure should be measured. One of the first results obtained by the author is that to avoid the deposition of particles in the tube of 25.4mm were required velocities smaller than in the tube of 50 mm. The critical velocity results led to the proposition of a new analytical model for critical velocity calculation, based on Wasp and Slatter (2004), with deviations of at most 10% from the experimentally obtained curves.

Blais et al. (2016), using CFD-DEM modellig, did not have results corresponding initially to the experimental results, and the error was attributed to the absence of viscous dissipation in the particle scale. For this, a dissipation system was developed in the finite volume mesh to compensate for this problem, decreasing this difference. Gopaliya et al. (2016) showed that at all conditions of velocity and particle size, the growth of the pressure gradient with the particle concentration growth was observed. Another important result was the decline in turbulent viscosity with increasing particle concentration. The observed turbulence dominates in the region of lower solids concentration, and its intensity decreases with increasing particle concentration.

2. PHYSICAL AND NUMERICAL MODELS

Veersteeg and Malalasekera (2007) showed the laws governing fluid behavior, both in space and in time, are the laws of mass conservation (Eq. 1) and the law of conservation of momentum (Eq. 2).

$$\frac{\partial \rho}{\partial t} + \rho . \nabla u = 0 \qquad (1)$$

$$\rho\left(\frac{\partial u}{\partial t} + u . \nabla u\right) = -\nabla p + \mu \nabla^2 u + S \qquad (2)$$

Where ρ – fluid specific mass (kg/m³); t – time (s); u – flow velocity (m/s); p – pressure (Pa); μ – dynamic viscosity (Pa.s); S – Source term.

To solve turbulent problems, the most well-known methodology was proposed by Reynolds, where the properties of the fluid are given as the sum of an average value and a floating part, thus indicating the property fluctuations in each direction. Warsi (2006) studied that for each type of solution chosen, excluding DNS, there are still several ramifications of turbulence models. Some of the RANS models use as a parameter the so-called turbulent kinetic energy represented by the letter k, and forms of dissipation of this energy, such as the Wilcox model, k-ω, where ω is the frequency of dissipation of this energy and the model of interest of this study, k-ε, where ε represents this energy dissipation. The k-ε model is represented by Equations 3, 4 and 5.

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \qquad (3)$$

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho k u_i}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + 2\mu_t E_{ij} E_{ij} - \rho\epsilon \qquad (4)$$

$$\frac{\partial \rho \epsilon}{\partial t} + \frac{\partial \rho \epsilon u_i}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\epsilon}\frac{\partial \epsilon}{\partial x_j}\right) + C_{1\epsilon}\frac{\epsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\epsilon}\rho\frac{\epsilon^2}{k} \qquad (5)$$

Where μ_t – turbulent viscosity (Pa.s); C_μ, $C_{2\epsilon}$, $C_{1\epsilon}$, σ_k and σ_ϵ – constants obtained by iterations of data fitting in experiments with turbulent flows; E_{ij} represents rate of deformation component.

Fox, Pritchard and McDonald (2010) showed that the relation between shear stress and strain rate is not linear (Eq.6).

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

Where k - consistency index (Pa.s); n – behavior index; y - height of the fluid column (m); τ – Shear stress.

In turbulent flow, an equivalent viscosity can represent this behavior. This means that the equivalent viscosity is a function of these two parameters and can be applied as a constant for each flow value. Wilson et al (2006) cited the definition (Eq.7).

$$\mu_{eq} = \frac{\rho\sqrt{\frac{\tau}{\rho}}D}{exp\left(\frac{u}{2{,}5\sqrt{\frac{\tau}{\rho}}}\right)} \qquad (7)$$

3. COMPUTATIONAL METHOD

The domain of solution, that are the proposal of this work, has dimensions of 1.6 meters of height and 2.2 meters of length in the total. The tube has a nominal diameter of 2 inches, which means that it has an internal diameter of 50.1mm; according to the commercial tables. Four hexahedral meshes were made (Table 1).

Table 1 – Produced mesh

Mesh	Number of elements
1	250560
2	443700
3	794952
4	1841856

Figure 1 shows the mesh spatially from the start of the tube to the outlet to the atmosphere. Figure 2 shows the differences between the coarser and more refined meshes.

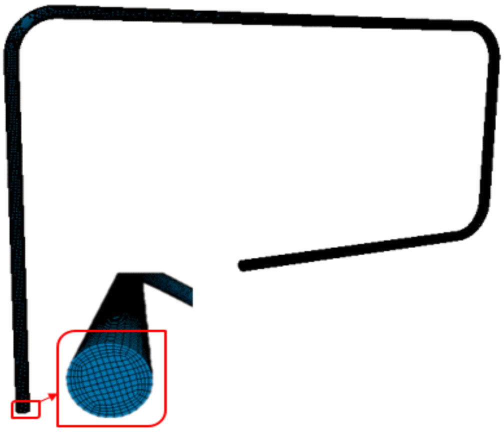

Figure 1: Mesh 1

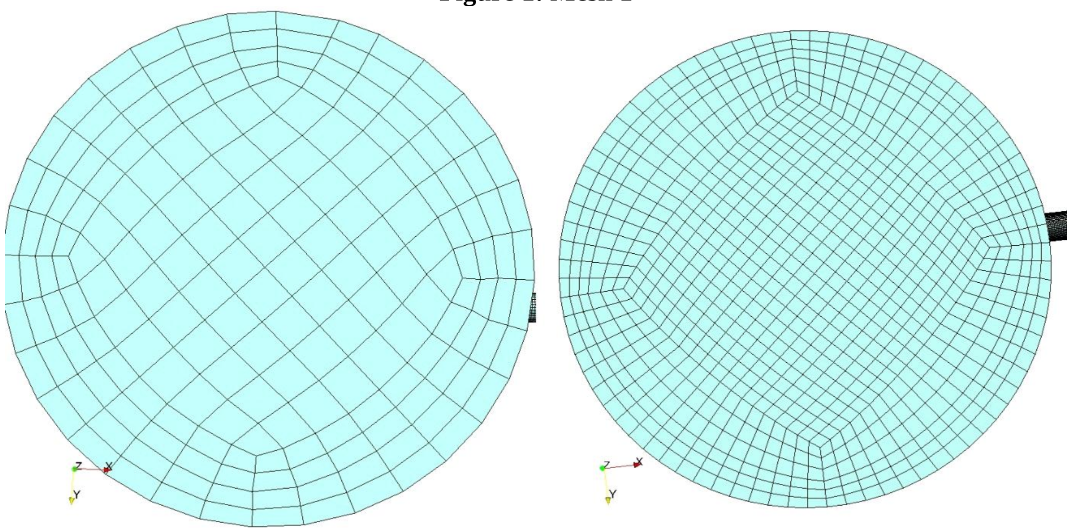

Figure 2: Comparation between mesh 1 (left) and mesh 4 (right)

The fluid used in this simulation is a mixture of water and apatite, 12% in volume, with particles of 297 to 249 micrometers of granulometry.

We used the same results of pressure loss to determine k (consistency index) and n (behavior index). In this case, it was necessary to use the Laun (1983) methodology, because for non-Newtonian fluids, the shear stress on the wall is not proportional to strain rate using the equivalent viscosity. Then, the author suggests using the shear stress at a distance of 83% of the radius value, and the ratio between this value and the equivalent viscosity is the real strain rate. With this value, we can interpolate a curve and find k and n.

Figure 3 shows the curve interpolated. The values found is 0,504 for k and 0,363 for n, wih R^2 about 0,97.

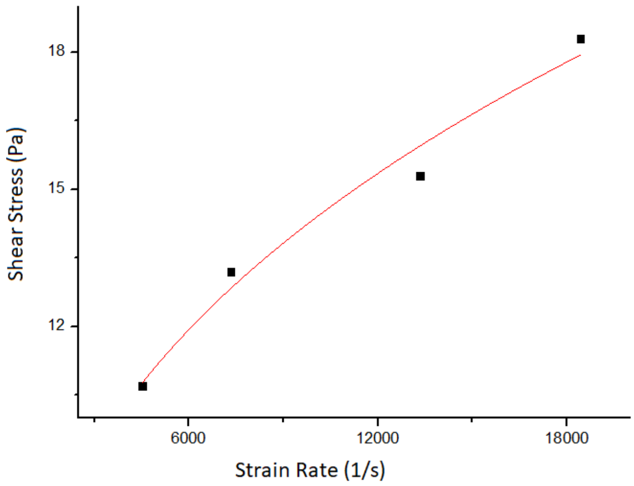

Figure 3: Adjusted curve

Boundary conditions applied in this model was the same conditions used for Souza Pinto (2014), making possible a comparative analysis with numerical and experimental results. The average velocity applied in the tube inlet was according to table 2. The 1,3m/s case was only used in third method, because we did not see correspondence in results below critical speed in only CFD methods. Tube outlet has a atmospheric pressure condition. Walls of the tube has no slip condition. The k-ε was the turbulence model utilized.

Table 2 – Average velocities applied

Average velocity (m/s)
1,8
2,1
2,4
2,7

The software utilized in for these simulations was openfoam, an opensource alternative to CFD cases, and the solver applied was pimplefoam.

4. RESULTS AND DISCUSSION

Applying the variable viscosity using the most refined mesh (mesh 4), the results reached differences of up to 9.47% (Table 3).

Table 3 – Numerical pressure losses x Experimental pressure losses

Average velocity (m/s)	Numerical pressure loss (Pa/m)	Experimental pressure loss (Pa/m)	Diference (%)	Average velocity (m/s)
1,8	931,6	1029	9,47	1,8
2,1	1199,6	1274	5,84	2,1
2,4	1499,4	1476	1,59	2,4
2,7	1825,9	1769	3,22	2,7

It can be observed that the calculated values of the pressure loss are lower than the experimental values at lower flows and are higher at higher flows, which indicates that there is a divergence of the results in a range outside these values, that is, it is possible to interpolate flow values that are among the experimental values used, but the extrapolation can may increase the differences between them.

Figure 4 shows, on the right, the pressure curve along the pipeline interest section for the 1.8m / s case and displays in a color scale the pressure from the pump inlet at the lower left end to the outlet to the atmosphere. The images corresponding to the other flows are like this one.

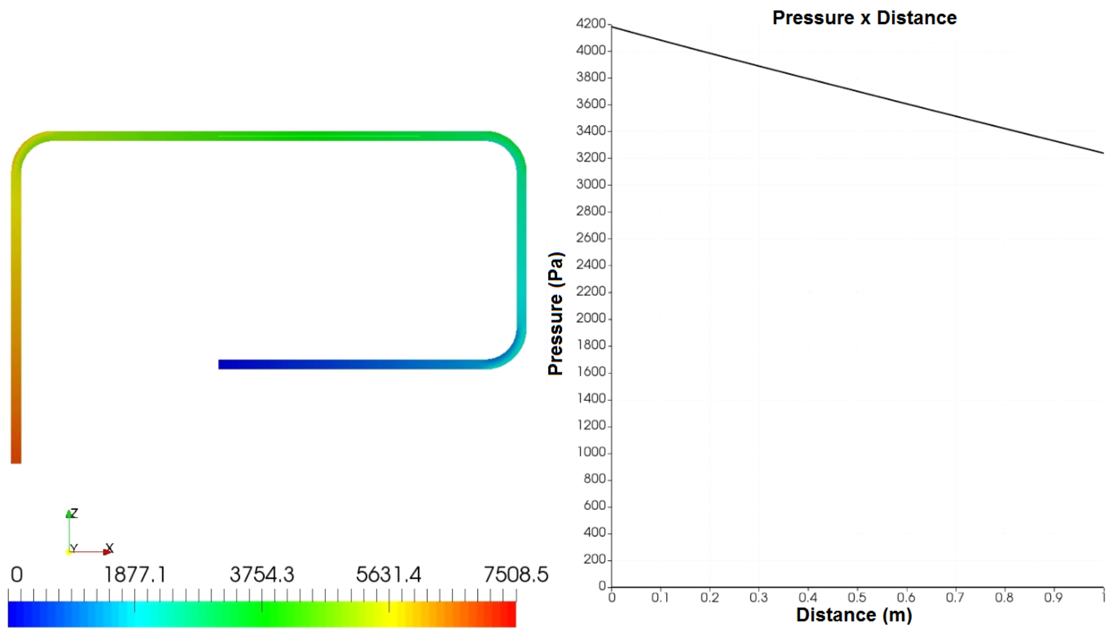

Figure 4: Numerical pressure results with average velocity about 1,8m/s

To calculate the uncertainty, the meshes 2,3 and 4 of Table 1 were used, the best results being those produced with the mesh 4 (Table 4). For this calculation, the equation was used for the fine mesh about the coarse mesh.

Table 4 – Numerical uncertainty

Average velocity (m/s)	Numerical pressure loss (Pa/m)	Uncertainty (Pa/m)	Uncertainty (%)
1,8	931,6	0,4	0,04
2,1	1199,6	1,7	0,14
2,4	1499,4	3,7	0,25
2,7	1825,9	0,01	0,0003

5. CONCLUSION

Variable viscosity modeling is independent of experimental results, once the fluid is characterized, it is possible to replicate it to different geometries and different flow rates. The interpolation curve of the characteristic values of the fluid proved to be effective for interpolated values but indicates divergence for extrapolated values. This type of modeling requires very refined meshes due to the high viscosity gradient between the elements of the mesh, reaching results up to 10% difference from the experimental data.

6. ACKNOWLEDGMENTS

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

7. REFERENCES

Bijjam, Sudheer., Dhiman, Amit Kumar., 2012. CFD analysis of two-dimensional non-Newtonian power-law flow across a circular cylinder confined in a channel. Chemical Engineering Communications 199, 767-785.

Blais, Bruno. Lassaine, M., Goniva, C., Fradette, L., Bertrand, F., 2016. Development of an unresolved CFD–DEM model for the flow of viscous suspensions and its application to solid–liquid mixing. Journal of Computational Physics 318, 201-221.

Durand, R., 1952. The hydraulic transportation of coal and other materials in pipes. In: Conf. of National Coal Board, London.

Fox, Robert; McDonald, Alan. 2010. Introduction to Flud Mechanics. John Wiley & Sons, Inc.

Gopaliya, Manoj Kumar. Kaushal, D. R., 2016. Modeling of sand-water slurry flow through horizontal pipe using CFD. Journal of Hydrology and Hydromechanics 64, 261-272.

Laun, Hans Martin. 1983. Polymer melt rheology with a slit die. Rheologica Acta 22, 171-185.

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

Turian, R. M., Hsu, F. L. G., Ma, T.W., 1987. Estimation of the critical velocity in pipeline flow of slurries. Powder Technology 51, 35-47.

Turian, R. M., Hsu, F. L., Ma, F. L. G., Sung, M. D., Plackman, G. W. , 1998. Flow of concentrated non-Newtonian slurries: 2. Friction losses in bends, fittings, valves and Venturi meters. International Journal of Multiphase Flow. 24, 243-270.

Versteeg, Henk Kaarle. Malalasekera, Weeratunge. 2007. An introduction to computational fluid dynamics: the finite volume method. Pearson Education.

Warsi, Zahir UA. 2005. Fluid dynamics: theoretical and computational approaches. CRC press.

Wasp, E.J., Slatter, P.T., 2004. Deposition velocities for small particles in large pipes. In: 12th International Conference on Transport & Sedimentation of Solid Particles, Prague, Czech Republic, 20–24.

Wilson, Kenneth C. Addie, G. R., Sellgren, R. C. 2006. Slurry transport using centrifugal pumps. Springer Science & Business Media.

8. AUTHORAL RESPONSIBILITY

The authors are the only one responsible for the content of this work