COB-2021-1254
COMPARISON BETWEEN DPM AND DDPM METHODS APPLIED TO SLURRY FILTRATION PROCESS

Matheus Dutra Baptista Oliveira
Cristiana Brasil Maia
Mechanical engineering department – Pontifícia Universidade Católica de Minas Gerais - Belo Horizonte, Minas Gerias, Brazil.
matheus.mdbo@gmail.com
cristiana@pucminas.br

Abstract. *The objective of computational analysis is to virtually represent a real physical behavior on a computer screen. Through computational simulation, it is possible to induce geometrical and numerical modifications reaching the most efficient operational parameters of equipment before its fabrication. A coupling of computational fluid dynamics and discrete element methods (CFD-DEM) could be performed in one-way and two-way analysis, with the objective to join two distinct methods of calculation in a single solver evaluating the iterations between fluid and particles. Discrete particle method (DPM) method uses one-way analysis calculating the influence of fluid in massless particles and dense discrete phase method (DDPM) method considers fluid influence in particles and particles influence in fluid, a two-way analysis. This paper uses both methods (DPM and DDPM) for a CFD-DEM coupled analysis studying a filtration equipment that transports slurry (fluid + particles) to different exits. A rotational valve restrains the main exit, creating a rising pressure location, which forces the slurry through a porosity region, with the objective to reduce the number of particles in the exiting water. The obtained results indicates that the precision needed to reach must be pointed to decide the most suitable simulation method, while physical results of fluid and particle flow and computational processing consumption varies.*

Keywords: Slurry filtration, CFD-DEM, DPM, DDPM

1. INTRODUCTION

Filtration is a process that is often applied with the objective of drying or extract humidity of a product. The drying process continues going through constant innovation and research, principally because of its enormous area of application. One of the justifications for the constant research applied for this process is the amount of energy consumed, especially when dealing with its application in large industrial companies. (HALL, 1988) explains in his text that the efficiency of drying should be measured to satisfy the product application, that in some cases, the excessive drying is prejudicial for transportation and storage.

Filtration is a multiphase process where solid particles move according to the fluid flow until a wall restricts the particles movement and allow the fluid phase to pass through. Computational analysis could be adopted to interpret and preview the behavior of multiphase fluids, however, as (ARIYARATNE et al., 2018) describes, in the field of multiphase flow modeling, system containing one or more particles are the most complex and challenging, while it is necessary to choose a numerical method capable to predict particle-particle, particle-fluid and particle-wall interactions.

Computational fluid dynamics (CFD) is a method with equations that could calculate the fluid flow and Discrete element (DEM) method uses equations for calculate particles movement. Recent papers adopt these methods coupling, as (MA et al., 2015) that used CFD and DEM through DPM (Discrete particle method) to visualize the particles motion in accordance with the fluid flow and (FATAHI; FARZANEGAN, 2018) uses the CFD and DEM through DDPM (Dense discrete particle method) for a Knelson separator analysis, and the results were validated with experimental data. The DPM method do not consider the particle-particle neither particle-wall iteration, is a one-way method, in other side, DDPM is a four-way analysis, that consider the iteration of particle-particle, fluid-particle and particle-fluid, providing more accurate results, because the particles motion also interferes in fluid motion.

Focusing the same objective of this article, other papers also compare results adopting different simulation methods applied to fluid-particles interaction. (MANGADODDY et al., 2020) presents a review of different kinds of methods to simulate fluid-particles interaction, as DPM (one way) with CFD-DEM coupling (two way) and to empirical results applied to thumbling mills, also compares ASM methods with DPM method. Applying the comparison for fluidized beds, (WU et al., 2021) compares the DDPM and TFM (tow fluid model), that applies Eulerian-Eulerian approach through the kinetic theory of granular flow (KTGF) for particles behavior calculation, different from DDPM, that uses Eulerian-Lagrangian approach through Newton's laws of motion. (OSTERMEIER et al., 2019) also applies the model's comparison to fluidized bed, but they compares three methods, TFM, DDPM and CFD-DEM changing the time steps values and number of elements to verify the impact in computational processing time.

The present article considers a comparison of two methods (DPM and DDPM) in a simulation of a drying through filtering equipment which, by the pressure of slurry column pulls the material against a porosity layer, responsible for retaining a percentage of particles, resulting in separating the solid from liquid phase.

2. METHODOLOGY

2.1 Computational fluid dynamics (CFD)

According to (VERSTEEG; MALALASEKERA, 2007) the laws of mass and momentum conservation are described by equations (2) and (3).

$$\frac{\partial \rho}{\partial t} + \rho.\nabla u = 0 \tag{1}$$

$$\rho\left(\frac{\partial u}{\partial t} + \rho.\nabla u\right) = -\nabla P + \mu_{eq}\nabla^2 u + S \tag{2}$$

where t is the time, u the flow velocity, p the pressure, μ_{eq} the dynamic equivalent viscosity and S the source term.

As (JANUÁRIO; MAIA, 2019) describes, to represent the turbulence behavior, many models could be used, but for this work was selected the k-ε, represented by the equations (4), (5) and (6).

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \tag{3}$$

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

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

where μ_t is the turbulent viscosity, $C_\mu, C_{1\epsilon}, C_{2\epsilon}, \sigma_k \; and \; \sigma_\epsilon$ the model constants and E_{ij} the rate of deformation components.

2.2 Discrete particle method (DPM) and dense discrete particle method (DDPM)

According to (POURARIA et al., 2020) the governing equation for particles motion is written with:

$$m_s\frac{d\vec{u_s}}{\partial t} = m_s\frac{3C_d}{4}\vec{u_s} = \frac{|\vec{u_s} - \vec{v_s}|}{d_s}(\vec{v_f} - \vec{u_s}) + m_s\frac{\vec{g}(\rho_s - \rho_f)}{\rho_s} + m_s\frac{\rho_f}{\rho_s}\vec{v_f}\nabla\vec{v_f} + \overrightarrow{F_{KTGF}} \tag{6}$$

where u and v are the velocities vectors, m is the mass, C_d the drag coefficient, ρ the density, d the diameter, g the gravitational acceleration. The f subscription defines the fluid phase and s the solid phase.

The first three terms on the right side of the equation represents the drag force, buoyancy force and pressure gradient force respectively. What differs the DPM and DDPM equations is the last term, that is used for the DDPM method, and represents the particle-particle collision and translations through the Kinect theory of granular flows (KTGF).

$$\overrightarrow{F_{KTGF}} = -m_s\frac{1}{\rho_s}\nabla\overline{\overline{\tau}}_s \tag{7}$$

$$\overline{\overline{\tau}}_s = -p_s\overline{\overline{I}} + a_s\mu_s\left(\nabla\vec{u_s} + \nabla\vec{u_s}^T\right) + a_s\left(\lambda_s - \frac{2}{3}\mu_s\right)\nabla.\vec{v_s}\overline{\overline{I}} \tag{8}$$

where p_s indicates solid pressure, $\overline{\overline{I}}$ is the unit stress tensor, μ_s and λ_s shear and bulk viscosities arising from particle momentum exchange due to translation and collision.

The solid shear viscosity is the sum of Kinect, collision and frictional viscosities.

$$\mu_s = \mu_{s,col} + \mu_{s,kin} + \mu_{s,fr} \tag{9}$$

$$\mu_{s,col} = \frac{4}{5}a_s\rho_s d_s g_0(1 + e_{ss}).\frac{\theta_s}{\pi}\frac{1}{2}a_s \tag{10}$$

$$\mu_{s,kin} = \frac{a_s\rho_s d_s\sqrt{\theta_s\pi}}{6(3-e_{ss})}\left[1 + \frac{2}{5}(1 + e_{ss})(3e_{ss} - 1)a_s g_0\right] \tag{11}$$

where e_{ss} is the coefficient of restitution for particle collisions, g_0 is the radial distribution function and θ_s is the granular temperature.

Bulk viscosity is represented by:

$$\lambda_s = \frac{4}{3} a_s{}^2 \rho_s d_s g_0 (1 + e_{ss}) a_s{}^2 g_0 \theta_s \qquad (12)$$

And solids pressure by:

$$p_s = a_s \rho_s \theta_s + 2\rho_s(1 + e_{ss}) a_s{}^2 g_0 \theta_s \qquad (13)$$

3. MATERIALS AND METHOD

Simulations were created with the objective of predict the particles behavior inside a fluid while going through equipment composed by a porous media region. In the physical model (Figure 1), the porous media is responsible to retain a percentage of particles, while the other percentage goes through it. With the complexity of create a geometry of porous media, because of the porous scale relative to the equipment, a pressure loss is considered in porous media region, representing the physical behavior of fluid.

Figure 1. Physical model

The mesh (Figure 2) was created in a surface with only 2D elements. The total number of elements was 91993 while 98,5% were quadratic and the others triangular. Mesh was divided in 4 different zones, slurry zone, purified water zone and porosity zone, according to figure 3. All the mentioned zones were water domain, while the porosity is a water domain with a specified porosity value, aiming to retain or deviate particles. The boundary conditions described in Figure 3 were associated with the related elements in mesh and the numerical parameters were considered according to Table 1.
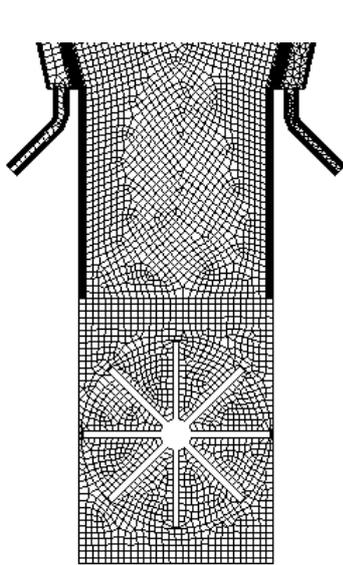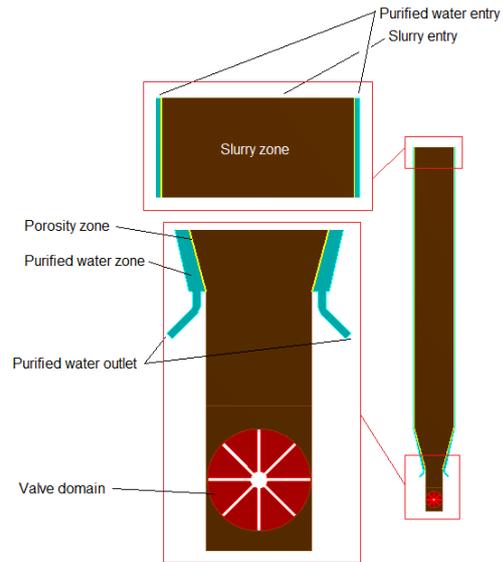
Figure 2. Mesh elements

Figure 3. Domain zones

Table 1. Boundary conditions

Parameter	Region	Value
Velocity inlet	Slurry entry	0.1 m/s
Pressure inlet	Purified water entry	1 atm
Pressure outlet	Purified water outlet	1 atm
Pressure outlet	Slurry outlet	1 atm
Rotational velocity	Valve domain	1 rad/s
Wall rough	External wall	0.5
Fluid porosity	Porous region	1
Porosity – Viscous resistance	Porous region	21111000

A computer with and Intel Core i7 10750H CPU @ 2.60GHz with 8 cores, 16Gb of Ram memory a Geforce RTX 2060 GPU and a SSD M.2 was used to performe both simulations, DPM and DDPM, where the conditions for each of them were described in the following topics.

3.1 Discrete particle method (DPM)

The k-ε turbulence method was used in the analysis.

CFD-DEM coupling was performed using the standard formulation of the Lagrangian multiphase discrete phase mode (DPM), that according to (FATAHI; FARZANEGAN, 2018) do not calculate the particle-particle interaction as for example particle's collision. This formulation is most used for a low fraction of particles, what in consequence consumes a lower processing time. Particles were considered massless with a virtual mass physical model, what also do not consider them diameter. A one-way coupling was performed, because as (STONE et al, 2019) describes, a two-way coupling do not affect the deposition of the particles, so it has no effect in an analysis of a one direction flow, as performed in this article.

The simulation was performed in a transient regime with total time of 20 seconds, defined in function of the 200 time steps with a 0,1 time step. As considered in (FATAHI; FARZANEGAN, 2018), a first-order upwind scheme was performed in the volume finite method (VFM), which considers (ANSYS, no date) the cells center values for element's faces. A SIMPLE algorithm was set to this simulation to, according to (ANSYS, no date) maintain a more conservative under-relaxion value, while a turbulence model is being performed.

3.2 Dense discrete particle method (DDPM)

DDPM simulation was performed using k-ε turbulence method, the Eulerian multiphase method with 1 discrete phase, the injection was performed with 1m/s normal to the entrance surface, and with particles of 1e-6 meter of diameter with a flow rate of 1e-10 kg/s and particles with density of 8030 kg/m³.

Following the DPM simulation, DDPM was performed with 20 seconds, defined in function of the 200 time steps with a 0,1 time step. A first-order upwind scheme was performed in the VFM, SIMPLE algorithm.

4. RESULTS

4.1 Discrete particle method (DPM)

Simulation time and results were extracted and compared with DDPM simulation.
DPM simulation took a total of 1 hour and 3 minutes of processing time in a computer with the configurations described in previews topics.

Results could be seemed in Figures 4 and 5, where fluid streamlines velocities are represented in function of the time and in Figures 6 ant 7, where particles velocities are represented in function of the time.
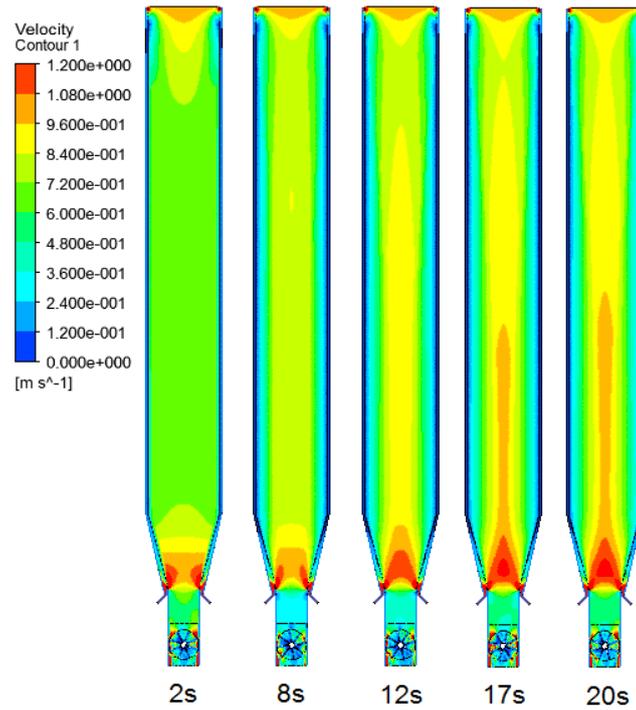

Figure 4. Fluid's velocity in DPM simulation
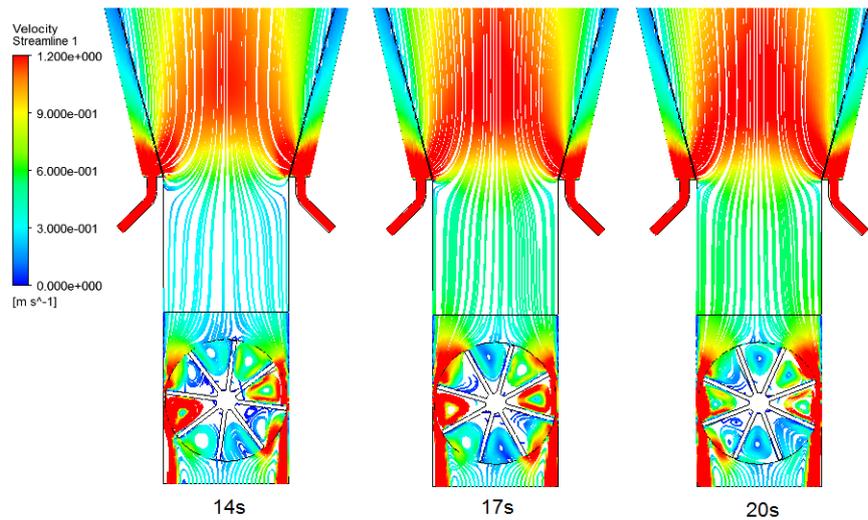
Figure 5. Fluid velocity in exit area in DPM simulation
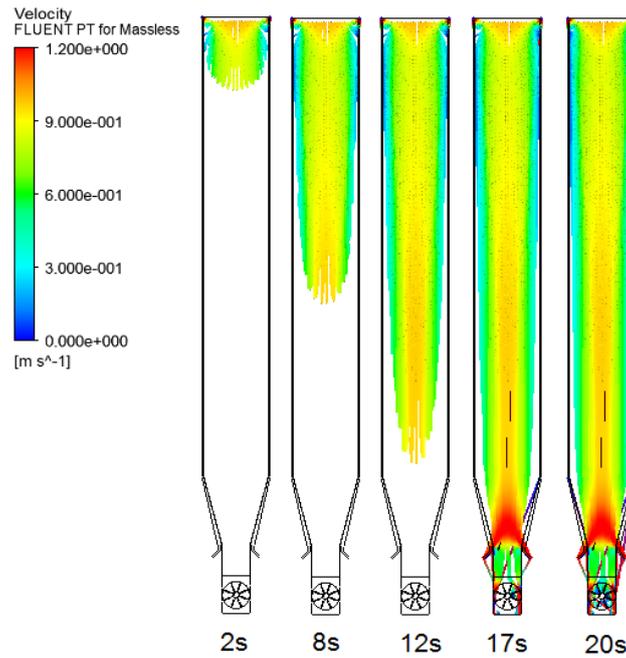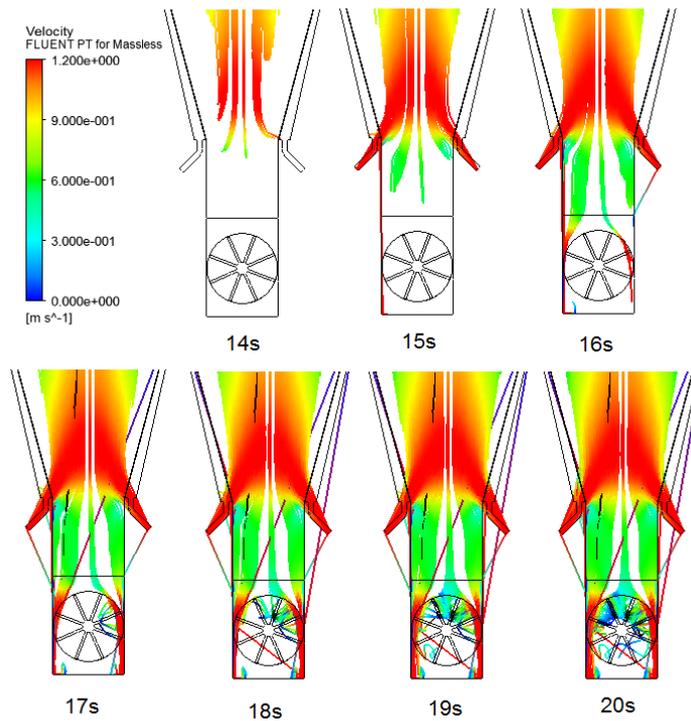
Figure 6. Particle's velocity in DPM simulation

Figure 7. Particle's velocity in the exit area in DPM simulation

It is possible to notice the similarity of fluid's and particle's flow behavior, also the linear flow during all the domain.

4.2 Dense discrete particle method (DDPM)

The simulation was performed in a total of 9 hours and 22 minutes in same number of time steps with same processing configurations of DPM simulation.
Results of fluid's flow could be seemed in Figures 8 and 9 and results from particles flow in Figures 10 and 11.
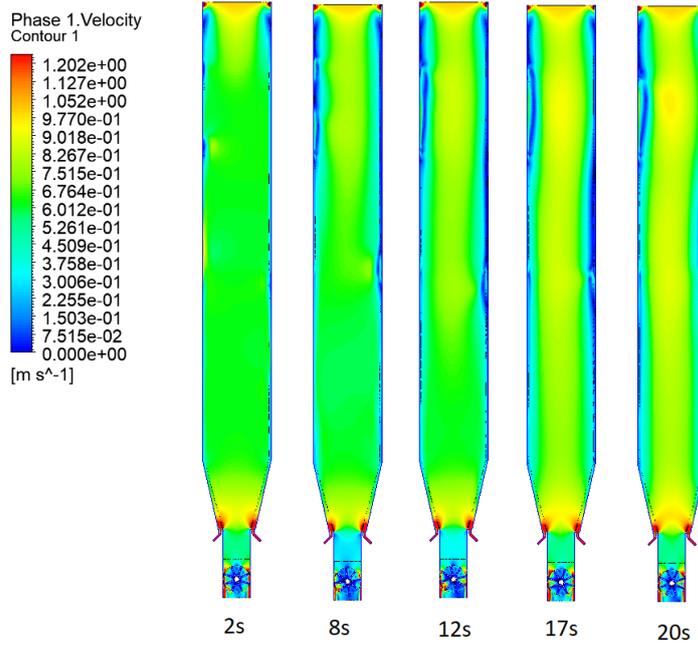
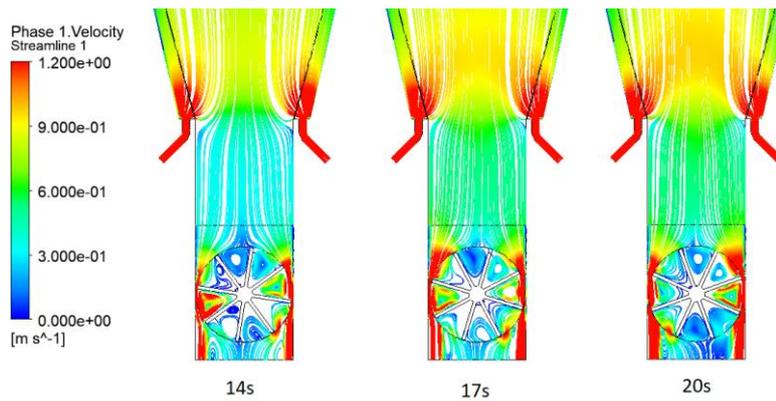Figure 8. Fluid's velocity in DDPM simulation

Figure 9. Fluid's velocity in exit area in DDPM simulation
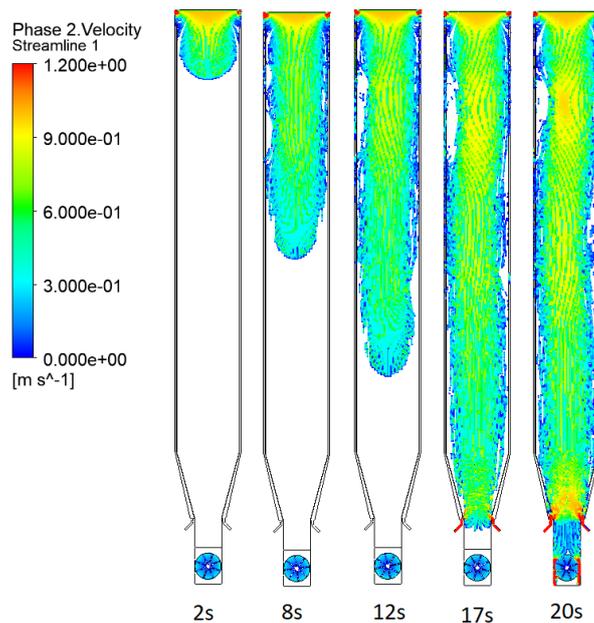

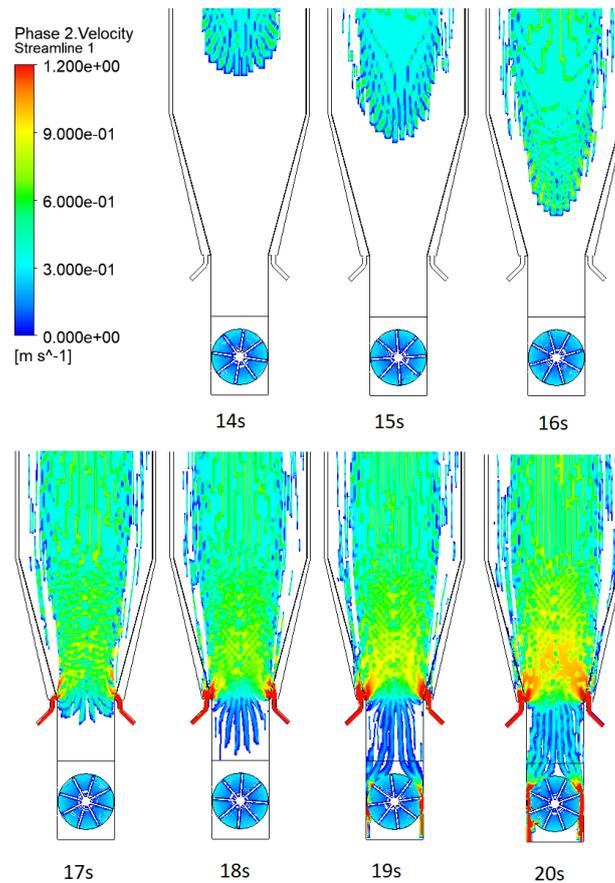Figure 10. Particle's velocity in DDPM simulation

Figure 11. Particle's velocity in the exit area in DDPM simulation

As DPM simulation, it's possible to notice the similarity of fluids and particle's flow, however the flow had some points of direction changes, different from DPM simulation. Also, comparing same simulation time, in DDPM simulation the particles got in the same position after particles of DPM simulation.

5. CONCLUSION

In both simulations, DPM and DDPM, the restriction caused by the rotative valve forces the fluid and particles to go through the porosity regions, making possible to create a particle filtering and expel a lower percentage particle in clean water. The regions of preference particles retention could be identified, indicating a cleaning improvement in that region while making the physical experiment.

By DPM simulation, comparing the fluid and particles behavior, was clear to notice the influence that the fluid phase causes in the particles in a one-way simulation. Also, it is possible to notice that was a low time processing simulation.

By the DDPM simulation could be seem some points of change of direction because of the influence of particles in fluid's phase and fluid in particle's phase. This conclusion was reached because this behavior was not observed in DPM. Another point to be registered is the difference of position of particles when compared to DPM, behavior caused because of the change of direction and, also, the drag force and particles rotation, parameters not presented in DPM simulation. Processing time was approximately 9 times greater than DPM simulation.

With the results of these two methods of perform a multiphase simulation with particles, it's possible to conclude that the level of results precision must be pointed before the choice of which method to perform. The DDPM method gives more precise results, but for this filter application, for example, a DPM simulation already gives reasonable results with less processing requirements.

An experimental test must be performed to validate the obtained results.

6. ACKNOWLEDGEMENTS

7. REFERENCES

ANSYS Fluent theory guide.

Ariyaratne W.K. Hiromi, Manjula E.V.P.J., Ratnayake Chandana, Melaaen Morten C. CFD Approaches for Modeling Gas-Solids Multiphase Flows – A Review. *Proceedings of the 9th EUROSIM & the 57th SIMS 680 September 12th-16th,* Oulu, Finland, 2016.

Fatahi M.R., A. Farzanegan. An analysis of multiphase flow and solids separation inside Knelson Concentrator based on four-way coupling of CFD and DEM simulation. *Minerals Engineering* vol. 126 p. 130–144, 2018

Fernandes C., Semyonov D., Ferrás L. L, Miguel Nóbrega J.. Validation of the CFD-DPM solver DPMFoam in OpenFOAM through analytical, numerical and experimental comparisons. *Granular Matter,* 20:64, 2018.

Fuping Qian, Naijin Huang, Jinli Lu, Yunlong Han. CFD–DEM simulation of the filtration performance for fibrous mediabased on the mimic structure. *Computers and Chemical Engineering* vol.71 p. 478–488, 2014

Hall, C. W. Handbook of Industrial Drying. *Drying Technology*, v. 6, n. 3, p. 571–573, 1988.

Hassan a Pouraria, Farzin Darihaki b, Ki Heum Park c, Siamack A. Shirazi b, Yutaek Seo. CFD modelling of the influence of particle loading on erosion using dense discrete particle model. *Wear*, 460–461, 2020.

Januário, J. R., Maia, C. B. Modeling and numerical simulation of slurry flow. *Brazilian Journal of Development*, v. 5, n. 10, p. 20462–20477, 2019.

Luke Stone, David Hastie, Stefan Zigan. Using a coupled CFD – DPM approach to predict particle settling in a horizontal air stream. *Advanced Powder Technology* vol. 30 p. 869–878. 2019

Ma, L. et al. Modeling of erodent particle trajectories in slurry flow. *Wear*, v. 334–335, p. 49–55, 2015.

MANGADODDY, N. et al. Computational modelling of particle-fluid dynamics in comminution and classification: a reviewMineral *Processing and Extractive Metallurgy: Transactions of the Institute of Mining and Metallurgy,* Taylor and Francis Ltd., , 2 abr. 2020.

Mannheimer, R. J. Laminar and turbulent flow of cement slurries in large diameter pipe: A comparison with laboratory viscometers. *Journal of Rheology*, 35(1), 113–133. 1991.

Mohammad A. Elyyan, Mai Doan, Yeong-Yan Perng. Fluid-structure interaction modeling of subseajumper pipe, *Proceedings of the ASME 2014 33rd International Conference on Ocean, Offshore and Arctic Engineering*, sdf

OSTERMEIER, P. et al. Comprehensive investigation and comparison of TFM, DenseDPM and CFD-DEM for dense fluidized beds. *Chemical Engineering Science*, v. 196, p. 291–309, 16 mar. 2019.

POURARIA, H. et al. CFD modelling of the influence of particle loading on erosion using dense discrete particle model. *Wear*, v. 460–461, 15 nov. 2020.

VERSTEEG, Henk Kaarle; MALALASEKERA, Weeratunge. An introduction to computational fluid dynamics: the finite volume method. Pearson education, 2007.

WU, Y. et al. Comparative study of two fluid model and dense discrete phase model for simulations of gas–solid hydrodynamics in circulating fluidized beds. *Particuology*, v. 55, p. 108–117, 1 abr. 2021.

YUE, C.; ZHANG, Q.; ZHAI, Z. Numerical simulation of the filtration process in fibrous filters using CFD-DEM method. *Journal of Aerosol Science*, v. 101, p. 174–187, 2016.

8. RESPONSIBILITY NOTICE

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