PAPER 2008-355

Oilsands Hydrotransport Process Control Using Computational Fluid Dynamics

O. GARCIA, C. DIAZ-GOANO, F. FORBES
University of Alberta, Chemical and Materials Engineering Department

Abstract

Hydrotransport is a method of transporting a slurry from the mine to a bitumen extraction facility. The process is a hybrid system, with discrete event loading of ore into a continuous slurrying system. Control of hydrotransport has been limited to flow rate, density, temperature, and flow rate of additives for variable feedstock. The process must be operated conservatively to prevent plugging of pipelines, at the expense of downstream separation performance. The characteristics of the slurries expose the pumps to abrasive conditions, which are magnified during inefficient pump operation. Therefore, the objective of this project is to develop a mathematical model to simulate and control the hydrotransport process.

The model consists of three modules. The first module simulates, using Computational Fluid Dynamics (CFD), the turbulent slurry flow. A second module compares the output generated by the CFD simulator against the process constraints. The third module computes the new input parameters for the CFD module to overcome the imposed constraints. An optimization scheme is developed such that it traces and corrects the operational parameters to meet the requirements of minimal velocity of the fluid to assure an efficient operation of the pumps. The complete interaction of the different modules represents the foundation towards more complex multiphase modeling of the hydrotransport process. A parametric study is performed for two different fluids that represent two different ore grades with estimated density and viscosity. For each case, there is a minimum operational velocity, corresponding to a discharge pressure of the pumps and the pumps' performance.

Introduction

Hydrotransport is one of the key processes of the Oil sands industry. Major improvements in terms of flexibility and energy efficiency were achieved when it was incorporated to the Oil sands industry in the decade of the 90's. Efficient pipelines to transport the ore to the extraction facility replaced costly conveyor belts. Now, hydrotransport is where the extraction process begins. Hot water, caustic and air are added to the crushed lumps on gigantic cyclofeeders where the slurry is prepared and then pumped. From this moment on, digestion starts kilometres away from the extraction plant.

The new characteristics of the process made possible a conversion from a high temperature extraction process, the traditional Clark's process [1], to a ground-breaking low energy extraction process.

The slurry is a highly abrasive mixture of water, bitumen, sand, clays, rocks and air. Each component plays an important role in the digestion and effective aeration and liberation of the bitumen from the sand grains. Temperature of the slurry, length of the pipe, characteristics of the ore, amount of air and amount of chemicals added to the slurry are among the multiple factors that affect the performance of the process.

All these parameter need to be wisely controlled to ensure a high quality product and avoid or minimize two major effects from the operational point of view:

1. Plugging or sanding of the pipelines. This is a direct consequence of operating the system below the minimum deposition velocity, which normally ranges between 3 m/s to 5 m/s.
2. Wear of pumps, pipelines and measurement devices. The slurry is an abrasive mixture. This effect is magnified when the operational velocities are too high and also when the pumps operate out of specification. As a result, maintenance costs and the life of the equipments and pipes are dramatically affected by the operation parameters. Rotation of pipes is performed on a regular basis to counterbalance the effects of abrasion.

Hence, reaching optimal operation is a key factor to ultimately minimize additional costs related to maintenance and replacement of expensive equipment.

As hydrotransport is not a fully observable process, research is required into new measurement and control methods for both existing and next-generation hydrotransport and extraction processes. The progress and availability of powerful computers have made possible to develop models that could mimic physical phenomena. Computational Fluid Dynamics (CFD) is a growing discipline that allows researchers to give a rapid and inexpensive interpretation and evaluation to complicated real processes.

Numerical Experiments and Results

For this project, Fluent, a commercial finite element solver, was used as the Computational Fluid Dynamics (CFD) simulation software package [2]. The domain grids were generated with Gambit, Fluent's geometry and mesh generation software [2].

Single – phase simulations

Mesh Validation

One of the first steps for the numerical simulation of fluid flows is the transformation of the underlying differential equation into a finite dimensional space. This discretization of the domain is done by generating a mesh. The quality of the mesh places an important role in determining the accuracy of the solution. For this reason, special care was given to the mesh generation process.

Some simplifications and assumptions were made to set up the system of study:

- Based on the existence of the "maximum maximorum" [3], that is, the velocity above which all the solids are in suspension, our first assumption was that the velocity conditions were always above this critical value of velocity. This condition made possible to threat the slurry as a homogeneous fluid.
- Fines were assumed not to form flocs. This allows treating the fluid as Newtonian defining and averaged and constant value for viscosity. Density was considered to be constant too.

The generated mesh should represent an accurate approximation of the domain to effectively reproduce the physics of the process when using fluids of different physical properties. To achieve this, two slurries of different properties were used. The main characteristics of the systems are given in table 1.

	Slurry A	Slurry B
Density [kg/m3]	1200	1700
Viscosity [Pa.s]	0.002006	0.004012

Table 1: Characteristics of the fluids used

For the validation of the mesh independence some criteria was established:

Reynolds numbers for any condition were over 300000 for this reason the system was modeled as a turbulent system, specifically using the K-epsilon model to solve the Navier – Stokes equations according to the following equations:

$$\rho\left(\frac{\partial v}{\partial t}+\vec{v}\cdot\nabla\vec{v}\right)=-\nabla p+\mu\nabla^2\vec{v}+f \qquad (1)$$

The dimensionless distance y+ was used to define how far from the wall the first grid node was going to be located. y+ relates physical and transport properties according to this expressions [2], [4]:

$$y^+=\frac{\rho\cdot U_\tau\cdot y_p}{\mu} \qquad (2)$$

$$U_\tau=\sqrt{\frac{\tau_w}{\rho}} \qquad (3)$$

$$\tau_w=\frac{\lambda\cdot\rho\cdot u^2}{8} \qquad (4)$$

$$\lambda=\frac{0.3164}{\mathrm{Re}^{0.25}} \qquad (5)$$

Where y_p is the geometric parameter of interest, representing the distance from the centroid of the first element to the wall.

Each fluid was evaluated for 3 m/s and 5/m, yielding 4 different scenarios. From each of those scenarios and element aspects ratios, a fine mesh was obtained based on an y+ of 35. As a result, 4 meshes were obtained and later used as controls to evaluate reproducibility when designing the final average mesh.

The following step was to take the mesh with the greatest distance from the wall as a reference to estimate new y+'s. All the estimated y+ observed were lower than 300, meeting the criteria of validity required by the software [2]. These estimations were later verified by running all 4 cases with this mesh.

In the case of y+'s, the estimated values were within good agreement with those calculated by the software.

The results for pressure drop and velocity profiles were effectively reproduced using this new average mesh.

First grid node from the wall [m]	0.00135
Element width [m]	0.00405
Boundary layer growth rate	1.1
Total number of elements [M]	1.5

Table 2: Characteristics of the average mesh.

Optimization modules

The optimization modules were developed for single-phase evaluations to predict values of operational velocities based on restrictions of velocities i.e. a range of operational velocities and also a maximum value of pressure drop, related to the optimal performance of the pumps. MATLAB was used to develop the modules and to create the interface to run Fluent in batch mode.

The optimization loop consisted of three basic modules:

CFD Execution module.

The CFD execution module consisted in different functions that perform 2 main tasks: create Fluent input files, one of which would dynamically change after each optimization iteration and generate a script file to call fluent in batch mode.

Different functions generated the input files. The run parameters were fed into Fluent in two separate input files:

1. A first file that corresponded to the constant part of Fluent input file (template file), such as the dimension of the problem and the desired path for the folder that would store the output files. Other information included in this file consisted of the turbulence models, the discretization scheme, tolerance parameters and physical properties of the material. These factors were not influenced by the results of each optimization iteration and therefore would not change throughout the optimization run.
2. A second input file that was required to run Fluent from MATLAB in batch mode. The inputs to this file were the mesh file to be used, and the boundary conditions (BC), inlet magnitude of the velocity and the outlet pressure. Additionally, all the parameters that were associated to the BC, for example the turbulence intensity calculated from equation 6 [2].

$$I = 0.16 \cdot \mathrm{Re}^{-1/8} \qquad (6)$$

The last step of the module was related to the execution function. This function used the script file the desired location of the output and also the number of processors to be used. The time of each run is also recorded.

Post processing module

The objective of this module was to prepare the output data for further analysis. The first step was to take the binary output file from Fluent and to create a file with all the data. Then the mesh was read and the result was a structure that contained a table with the nodes, another with the faces and one extra table with the cells. Each of the above fields consisted of two parts: the ID and the value fields. Having all this information stored by nodes, multiple analyses could be made. From velocity profiles at any position to the evaluation of the pressure drop which was defined as the primary objective for this specific case.

Comparison Module.

This module was responsible of identifying the velocity of the system for the pumps to generate the optimal head, specified as the pressure drop restriction, and to operate as close as possible to the best efficiency point. Velocity variation was possible due to the fact the variable speed pumps were used in this type of systems. If the velocity was within the range imposed, the loop would continue until the optimal velocity was obtained.

A limit case could arise when the velocity was below the minimum operational velocity, then once again an inefficient operation of the pumps would be expected because that lower limit represents the minimum deposition velocity. Below that point a stationary bed of deposits would appear at the bottom of the pipe. If the velocity required to reach the optimal head was beyond the point of the velocity range, the system would stop and suggest using the maximum value of velocity at the expense of lower efficient operation unless a higher operational velocity was allowed and abrasion could not represent a major inconvenience.

Figures 1 and 2 show the convergence results of the optimization program for a given set of velocity/pressure constraints. The figures show the relative error between the set and current velocity and pressure vs. the iteration number. It can be observed how after 6 iterations, the solution was reached.

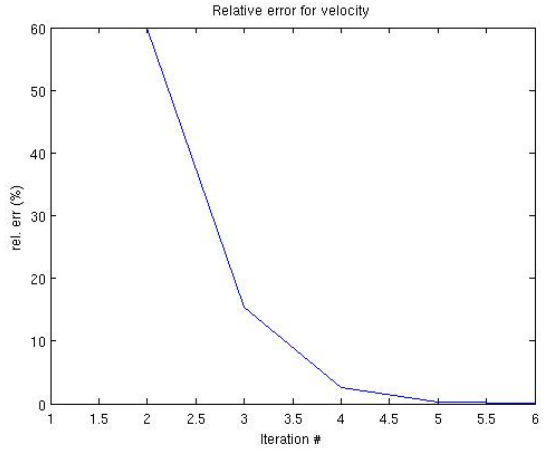

Figure1: Relative error for velocity with respect to the iteration number.

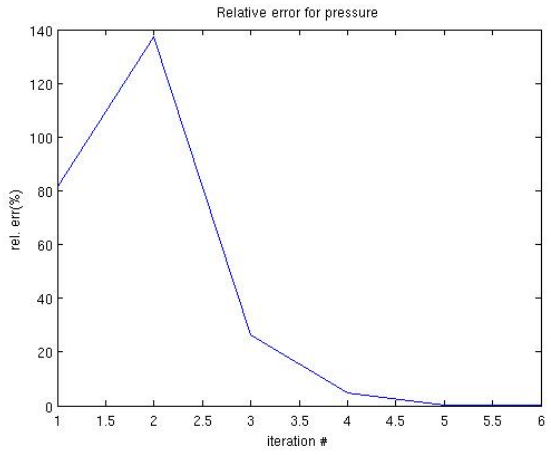

Figure 2: Relative error for pressure with respect to the iteration number.

A scheme of the optimization module is shown in figure 3

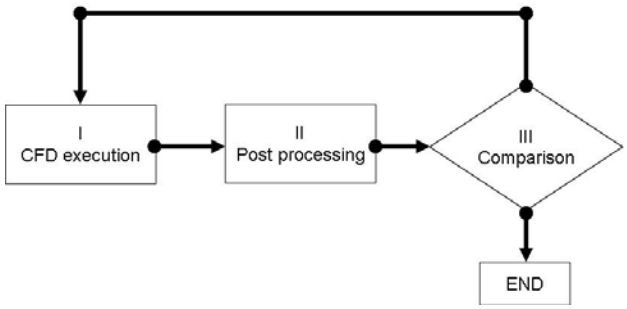

Figure 3. Optimization scheme.

Multiphase simulations.

After the optimization routines for steady state conditions were developed and validated, the code was extended to consider transient behavior of the system due to changes in the feed characteristics. Since transient studies involved much longer CPU times, scalabilities issues were also explored. The question whether a model represented by a smaller domain (a smaller mesh size implied shorter time of program execution) could be used to effectively capture the behavior of the system was also investigated.

From the literature review, only a few authors have considered the case of fluid displacement. Bannwart et al.[5] evaluated the displacement of oil with water after the shut – down of a core annular flow loop. In this case, a thin layer of water at the bottom of the pipe acted as a by – pass at the beginning of the displacement defining the transient behavior of the pressure drop.

Regner et al. [6] evaluated the flushing of industrial yogurt lines with water. Once again in this case when performing the simulation, they established an annular initial condition with water. Additionally to this, they based all their analysis of their results establishing the breakthrough time as t = 0.

In this study, the initial time was taken as the time when the displacing fluid just enters the domain and the pressure drop analysis is made from t = 0 until the steady state for the displacing fluid is reached. For this case some assumptions were implemented to perform the evaluation:

To minimize any interaction between the fluids, total immiscibility criterion was used. Each of the fluids was treated as a homogeneous fluid that in the case of a slurry it meant that the system was maintained at a velocity higher than the maximum critical deposition velocity [7].

A base case was chosen to evaluate this type of scenario. A kerosene – water system was studied with the physical an interfacial properties used by Chakrabarti [8] in his experiments. A summary of the data used is shown in table 3. The change was only in the characteristics of the feed while maintaining the velocity input the same.

Fluid	Density [kg/m^3]	Viscosity [Pa.s]
Kerosene	787	0.00120
Water	1000	0.00084

Table 2: Physical properties of Kerosnee and water

The interfacial tension values used was 0.045 N/m

CPU time savings.

Numerical experiments were used to evaluate different strategies towards the reduction of computational time. The first strategy involved reducing the problem from a 3D to a 2D axisymmetric case scenario and evaluating the consequences of this simplification. The second strategy involved reducing the length of the discretized domain. A 3D case (half pipe) was used as a benchmark to compare the performance of the other possible alternative.

The cases were run in a Dual Core AMD Opteron Processor 885 cluster, using 6 CPUs for each run. The cluster was 64 bit running Scientific Linux. The 3D model consisted on a mesh with 31,005 elements and 38,640 nodes. The other 2D models had 39,375 elements and 47,256 nodes and 3445 elements and 4140 nodes respectively.

3D model (half pipe)

Breakthrough time is defined as the tame it takes for the displacing fluid to reach the end of the domain and it is one of the registered parameters to be used as part of the analysis. Figure 4 shows a slide of the 3D domain showing a snap shot for the breakthrough time.

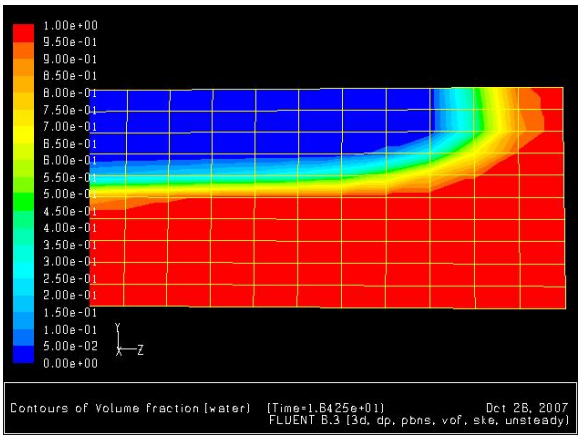

Figure 4. Breakthrough time snap shot

Another control parameter was the time it takes the displaced fluid to completely leave the domain. The author defined this time as the 'displacement time'. A scheme showing a snap shot for the displacement time is presented in figure 5.

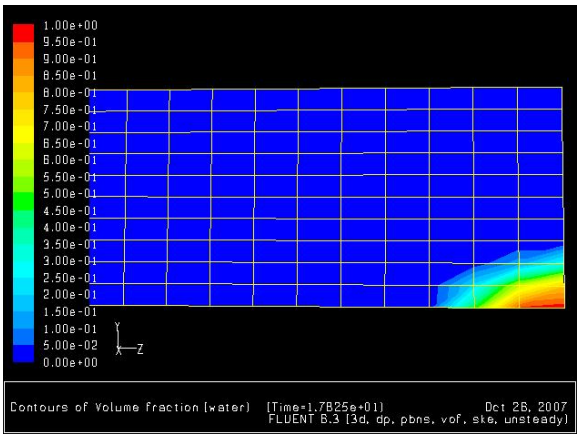

Figure 5. Displacement time.

As it can be observed, this is a gravity driven phenomenon. Density difference between the fluids defines the gap between breakthrough time and displacement time.

2D axisymmetric model.

The alternate mode to use was a 2D axisymmetric model. Together with this assumption comes the simplification of the gravity effect. It was expected that the breakthrough time and the displacement time coincide with the previous case. The mesh created for this case was equivalent to the one used in the 3D model. A summary of the observation is shown in figure 6.

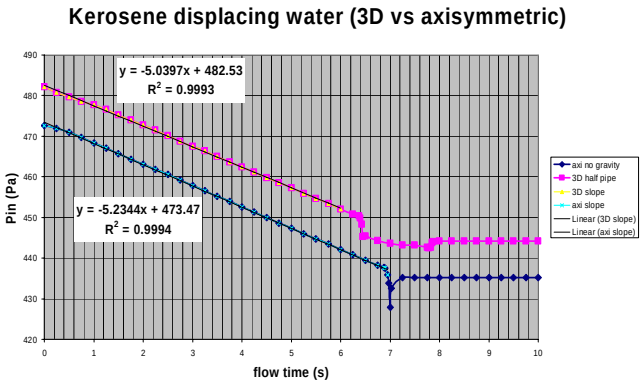

Figure 6. Pressure drop with respect to time. Comparison between a 3D model and a 2D axisymmetric model

The lower curve represents the axisymmetric evaluation. It can be seen that the breakthrough time was in between the values for breakthrough time and displacement time of the 3D model. In this particular study, the precise evaluation of the breakthrough time was not important. However, it should be noted that both results show the same linear behavior of the transient pressure drop with time. In this case a difference between the two models exists but it is a constant difference and the average relative error between the values of pressure drops is of approximately 3.3%. In terms of industrial applications, this value could be considered to be within the range of error of a device used to measure pressure at any point.

The scalability of this scenario was evaluated by changing the length of the domain from 3.5 m to 20 m. The results presented in figure 7, show that a smaller system could be used to effectively extrapolate the breakthrough time. Additionally, the relative error for the slope of the pressure drop decay was only of approximately 0.3% between the two cases. This confirms that when maintaining the same diameter, the model can be scalable down in the direction of the flow.

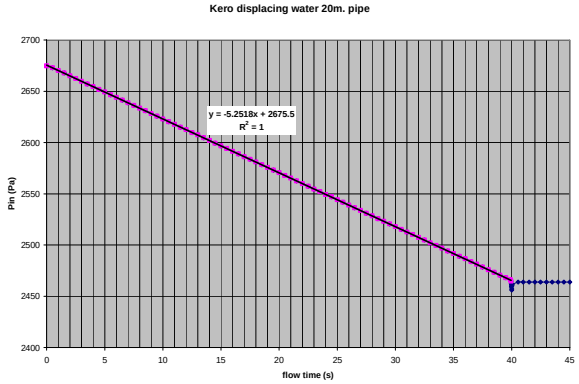

Figure 7. Pressure drop with time for a 20m domain.

As a consequence of this new approach, the execution time of the simulations was reduced from 3 hours of the 3D model to less than 40 minutes when the 2D axisymmetric model was implemented.

Conclusion

In this study, an optimization software that included CFD simulations was successfully developed. This software could be a very useful tool for industrial operations. The program not only can compute the optimal operating conditions but also provides a detailed fluid dynamic description of the flow that allows to further increase the understanding of the system.

The transient behavior of the pressure drop follows a linear trend when switching from one steady state to another. This observation is valid when the inlet velocity is maintained constant and the change comes in the form of a displacing fluid with different physical properties.

Most CFD packages require large amounts of CPU time to produce accurate results. For any simulation study, the mesh generation is a crucial step in the simulation process, as it strongly influences the accuracy, stability and computational cost of the model. In our case, a 2D axisymmetric model represents an economic alternative when compared to a full 3D model. The difference in pressure between the two models is of approximately 3%.

Acknowledgement

The authors would like to thank Yuri Shardt for his collaboration towards the development of the optimization modules. Srinath Madhavan and Dr. Joe Mmbaga for their support.

NOMENCLATURE

ρ	=	Density of the fluids [kg/m^3]
v	=	velocity [m/s]
t	=	time [s]
p	=	Pressure [Pa]
μ	=	Viscosity of the fluid [Pa.s]
f	=	external forces i.e. gravity
y^+	=	Solution dependent dimensionless distance
u_τ	=	Friction velocity [m^2/s]
y_p	=	Distance from the centroid of the first element to the wall [m]
τ_w	=	Shear stress at the wall [kg.m/s^2]
λ	=	Dimensionless coefficient of resistance
Re	=	Reynolds number
I	=	Turbulence intensity [%]
R^2	=	Geometric correlation parameter for line fitting

REFERENCES

1. K. A. Clark and S. D. Pasternak. *Ind. Eng. Chem.* 24 (1932), p. 1410

2. FLUENT 6.3 documentation, *Ansys2007.*

3. Thomas, A.D., Predicting the deposit velocity for horizontal turbulent pipe flow of slurries, *Int. J. Multiphase flow, Vol. 5, pp. 113 – 129. 1979.*

4. Schlichting, H Boundary Layer Theory, *8th revision, Springer 2000.*

5. Bannwart A.C, Barbosa A., and Ribeiro G.S., A model for interpretation of a re – start experiment of viscous oil – water annular flow, *Proc. Of the 18th IASTED Intl. conference, May – June, 2007.*

6. Regner. M., Henningsson M., Wilklund J., Ostergren K., and Tragardh C., Predicting the displacement of yoghurt by water in a pipe using CFD, *Chem. Eng. Technol. Vol. 30, No. 7, pp. 844 – 853. March 2007.*

7. Wilson, K.C., Deposition – limit nomograns for particles of various densities in pipeline flow, *6th Intl. Conference on the hydraulic transport of solids in pipes. Paper A1. September 1979.*

8. Chakrabarti, D.P., Das, G, and Ray S., Pressure drop in liquid – liquid two phase horizontal flow: experiment and prediction, *Chem. Eng. Technol. Vol. 28, No. 9 pp. 1003 – 1009. 2005.*