Abstract. Among the many geological processes of interest to oil industry, understanding turbidity currents can help explain where and how organic matter was deposited and perhaps transformed by other processes of geological scale into hydrocarbons. The intent of the work, is to assess local, small scale parameters and their upscaling. The hybrid model is based on a Lagrangian-Eulerian approach under a class of the named Unresolved Discrete Particle Method (UDPM). In this approach, a Lagrangian description is used for the particle system employing the Discrete Element Method (DEM) while a fixed Eulerian mesh is used for the fluid phase modeled by finite element method (FEM). Fluid motion equations are solved by an appropriate FEM implementation. Closure equations are used to compute drag and lift forces over the particles in the DEM framework. Volume averaged momentum sink terms are included in the fluid equations. The resulting coupled DEM-FEM model is integrated in time with a subcycling scheme. The aforementioned scheme was applied in the simulation of a seabed current to analyze which mechanisms lead to the emergence of bedload transport and sediment suspension, and also quantify the effective viscosity of the seabed in comparison with the ideal no-slip wall condition. To compare the behavior of the particles falling in a fluid medium, a simulation of a salt plume in free fall was performed, and the main characteristics of the system are discussed in comparison with a qualitative experiment.

Keywords: DEM, FEM, bedload, UDPM
1 INTRODUCTION

Among the many geological processes of interest to oil industry, understanding turbidity currents (typically a gravity or density current) can help explain where and how organic matter was deposited and perhaps transformed by other processes of geological scale into hydrocarbons. In this work we describe a contribution to the study of turbidity transport in scales smaller than TFM (two-fluid models). The intent of the work, part of a large scale simulation project, is to assess local, small scale parameters and their upscaling. The hybrid model is based on a Lagrangian-Eulerian approach under a class of the named Unresolved Discrete Particle Method (UDPM) according to the classification presented in Hoef (2008). In this approach, a Lagrangian description is used for the particle system employing the Discrete Element Method (DEM) while a fixed Eulerian mesh is used for the fluid phase modeled by finite element method (FEM). This technique has been successfully applied to the study of fluidized bed in catalytic reactors (Hoomans, 1996).

Fluid motion for the incompressible and viscous fluid is governed by Navier-Stokes equations which are solved by an appropriate FEM implementation (Elias, 2005). Closure equations are used to compute drag and lift forces over the particles in the DEM framework (Cho, 2005). Volume averaged momentum sink terms are included in the fluid equations. The resulting coupled DEM-FEM model is integrated in time with a subcycling scheme.

2 MOTION EQUATIONS

In this paper a three-dimensional Lagrangean approach is used with the Discrete Element Method (DEM) to describe the particles, and a three-dimensional Eulerian approach with the Finite Element Method (FEM) is used to describe the fluid behavior.

2.1 Particle motion

According to Hoef (2008), there are two main classifications to the simulation of particle’s interaction: the hard-sphere model and the soft-sphere model. In the hard-sphere model the particles are modeled as rigid bodies and interact through instantaneous collisions. However, in the soft-sphere model, the equations of motion of each particle are solved numerically, requiring a contact force model. The spring-damper model is the most widely used, showing a good compromise between accuracy and efficiency.

Due to the large number of possible simultaneous contacts, directly related to the high concentration of particles, the soft-sphere model is the most suitable and, therefore, the implemented. The equation of motion of each particle can be written as:

\[ m \cdot \ddot{x} = f_G + f_B + f_C + f_A \]  

(5)

where \( \ddot{x} \) is the particles’ accelerations, \( m \) the particles’ mass, \( f_G \) is the gravity force, \( f_B \) is the buoyancy, \( f_C \) is the contact force and \( f_A \) is the drag force. Other possible ways of interaction between particles (e.g. adhesion, aggregation, disaggregation) can be naturally treated within the framework of DEM technique.

Once the particles have been mapped in the Eulerian domain discretized by a tetrahedra mesh, it is possible to determine by interpolation the fluid velocity at the point occupied by the
particle. The drag forces acting in each particle can be processed through empirical laws involving the relative velocity between the particle and the fluid.

The contact force comes from the soft-sphere model, consisting of linear elements of stiffness and damping (O’Sullivan, 2011) in the normal and tangential directions of the sphere contact. Additionally, for the tangential direction it is considered the Coulomb friction law (Figure 1).

\[ F_c = K_c \delta_c + C_c \dot{\delta}_c \quad (n, t) \]  

\[ F_t = -\min(\mu_{\text{coulomb}} F_n, F_t(\delta_t, \dot{\delta}_t)) \frac{\dot{\delta}_t}{|\dot{\delta}_t|} \]  

where \( F_n \) is the normal, \( K_n \) the stiffness coefficient, \( C_n \) the damping coefficient, \( \delta_n \) the penetration at the sphere, \( \dot{\delta}_n \) the temporal penetration rate, \( F_t \) the tangential force, \( \mu_{\text{coulomb}} \) the friction coefficient and \( F_t(\delta_t, \dot{\delta}_t) \) the tangential force before slipping, been identical to the equation of normal force with the respective coefficients.

\[ \text{Figure 1: Contact model between two spheres: (a) normal direction, (b) tangential direction.} \]

The gravity and buoyancy forces are calculated trivially. According to Hoomans (2000), the drag force acting on a particle can be modeled as:

\[ f_d = \frac{1}{8} \pi d_p^2 C_d \rho_f |v| \cdot v \]  

where \( d_p \) is the particle’s diameter, \( \rho_f \) is the fluid’s specific mass and \( v \) is the fluid’s superficial relative velocity, defined as:

\[ |v| \cdot v = \varepsilon^2 |u - v_p| (u - v_p) \]  

being \( \varepsilon \) the porosity, \( u \) the fluid’s velocity and \( v_p \) the particle’s velocity. The effective drag coefficient \( C'_d \) consists of a correction to the drag coefficient \( C_d \) due to the porosity. According to Wen and Yu (1966), this correction can be applied following the equations 10, 11 and 12.

\[ C'_d = \varepsilon^{-4.7} \cdot C_d \]
The drag coefficient of an isolated particle is modeled by Rowe and Henwood (1961) as:

\[
C_d = \begin{cases} 
\frac{24}{Re_p} \cdot (1 + 0.15(Re_p)^{0.687}), & Re_p \leq 1000 \\
0.44 & Re_p \geq 1000 
\end{cases} 
\] (11)

Where the Reynolds number is defined as:

\[
Re_p = \frac{\varepsilon \cdot \rho_f \cdot |u - v_p| \cdot d_p}{\mu_f} 
\] (12)

2.2 Fluid motion

While the particle motion is described through discrete elements, the fluid is described as a continuum medium, thus the fluid domain is divided in tetrahedral cells, cells sufficiently small to represent the fluid motion, and big enough to accommodate inside several particles, homogenizing locally the porosity.

In this work, it was supposed that the variation of porosity in time and space is negligible for the fluid, so that the set of equations of Navier-Stokes for the incompressible Newtonian fluid flow does not need to be corrected by porosity. Thus, according to Hoomans et al. (1996), the resulting equations are listed in equations 13, 14 and 15.

\[
\frac{\partial u_i}{\partial x_j} = 0 
\] (13)

\[
\rho_f \frac{\partial u_i}{\partial t} + \rho_f \frac{\partial (u_j \cdot u_i)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \mu_f \frac{\partial^2 u_i}{\partial x_j \partial x_j} - F_{fp} 
\] (14)

\[
F_{fp} = \beta \cdot (u - \bar{v}_p) 
\] (15)

where \(\beta\) is the coefficient of moment transfer, \(\rho_f\) the fluid’s specific mass, \(p\) the pressure, \(\mu_f\) dynamic viscosity and \(\bar{v}_p\) the mean velocity of the particles contained in an element. The last term, \(F_{fp}\), represents the moment transfer from the particle to the fluid, acting as a source or a sink.

According to Kuipers (1992), for porosities below 0.80, \(\beta\) is defined by the Ergun equation in the following form.

\[
\beta = 150 \left(1 - \varepsilon\right)^2 \frac{\mu_f}{d_p} + 1.75(1 - \varepsilon) \rho_f \left|u - \bar{v}_p\right| 
\] (16)

while for porosities above 0.80, the following correlation was presented by Wen and Yu (1966):

\[
\beta = \frac{3}{4} C_d \frac{\varepsilon \cdot (1 - \varepsilon)}{d_p} \rho_f \left|u - \bar{v}_p\right| \cdot \varepsilon^{-2.65} 
\] (17)
2.3 General aspects of the coupled implementation

The overall diagram of the coupling is presented in Fig. 6. After the solution of the set of fluid’s equation (FEM/CFD cycle), the drag forces for each particle are calculated, which allows the progress of the solution for particle motion (DEM cycle). Typically, particle dynamics requires small time steps compared to fluid flow, demanding a subcycle approach, i.e., the computation of various DEM cycles for each FEM/CFD cycle. After the finalization of DEM cycle, the source/sink terms $F_p$ are evaluated for each finite element in the mesh and proceeding to a new step of the Navier-Stokes equations integration for the FEM/CFD cycle.

3 STUDIED CASES

The operationality of the code is verified in a qualitative way, using two cases for which two-way coupling between DEM and FEM is intrinsically important.

- Channel flow;
- Salt plume.

In the following sections, it is presented the description and results of both cases.

3.1 Channel flow

The aim of this case is to compare the velocity profile of the fluid near the bottom of the channel using the no-slip boundary condition at the bottom wall and using a bed of particles to represent the soil (Figure 3).
The simulation addresses the channel flow configuration shown in Figure 5. Representing the soil, a polydisperse distribution comprising 4800 particles with diameter ranging from 0.2mm to 0.3mm were settled in a channel of rectangular cross section with 12mm height, 1.8mm width and 12mm length. The entire channel was discretized with 6000 tetrahedra as depicted in Figure 4.

The employed boundary conditions are the velocity of 30mm/s at the left wall for the nodes above the seabed and 0mm/s for the nodes bellow seabed (if applied), no-slip at the bottom wall, free slip at the upper wall and no penetration at side walls (Figure 5). At time $t = 0s$ the particles are frozen and the sink term is nulled to the fluid develop a steady state profile. This methodology was applied to facilitate the convergence. After 0.05s the particles were released and the sink term computed.
The solution was achieved after the simulation reaches the steady state. The Figure 6 illustrates the obtained velocity profiles. Each marker represents a measure realized at the respective node of the mesh.

It can be observed from the graph that the slopes of the profiles are slight different. This difference is due to the interaction of the fluid and the particle that has a different effect than the no-slip wall condition. To assess this difference, both slopes were calculated resulting in a
1.86° for the no-slip cond. seabed and a 2.82° for the particulate seabed. The ratio between both slopes is 152%, therefore the effective viscosity of the fluid in the particulate seabed is 50% greater than the effective viscosity of the fluid in the no-slip condition seabed.

Table 1: Physical properties attributed to the simulation of channel flow.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid time step</td>
<td>0.005</td>
<td>s</td>
</tr>
<tr>
<td>Simulation duration</td>
<td>5</td>
<td>s</td>
</tr>
<tr>
<td>Dynamic viscosity</td>
<td>0.001</td>
<td>Pa.s</td>
</tr>
<tr>
<td>Fluid’s specific mass</td>
<td>1000</td>
<td>Kg/m³</td>
</tr>
<tr>
<td>Fluid domain size</td>
<td>12 x 12 x 1.8</td>
<td>mm</td>
</tr>
<tr>
<td>Mesh</td>
<td>Structured, 6000 tetrahedra with 0.6 mm side</td>
<td>-</td>
</tr>
<tr>
<td>DEM time step</td>
<td>0.00001 (500 subcycles)</td>
<td>s</td>
</tr>
<tr>
<td>Number of particles</td>
<td>4800</td>
<td>Unit</td>
</tr>
<tr>
<td>Particle specific mass</td>
<td>2170</td>
<td>Kg/m³</td>
</tr>
<tr>
<td>Contact rigidity</td>
<td>1.0E+5</td>
<td>N/m</td>
</tr>
<tr>
<td>Particle diameter</td>
<td>&lt; 0.2 , 0.3 &gt;</td>
<td>mm</td>
</tr>
</tbody>
</table>

3.2 Salt plume in free fall

Due to the lack of physical experiments involving particles with controlled parameters, this work presents a comparison between a numerical simulated and a qualitative experiment of a salt plume. This experiment consists in a batch of salt released in a recipient filled with water. The main objective of this case is to compare the numerical and experimental features of the salt plume.

A minimal characterization of the salt batch was performed over a sample of 300 salt grains, measuring grain sizes using digital imaging and a caliper as a reference scale. The results were interpolated assuming a Gaussian distribution of the equivalent diameter of the salt set. The probability density function and the numerical measurements are presented in Figure 7. The additional properties utilized in the numerical simulation are listed in Table 2.

The salt plume was represented numerically by a set of 28224 spherical particles organized in a volume 20x20x40mm in a fluid domain with 100x100x250mm. The salt plume is centered in the X-Y plane and is distant 200mm from the bottom. The diameter of the particles was generated randomly according to the adjusted Gaussian distribution. The boundary condition for the fluid is no-slip in every wall.
Figure 7: Salt grain size distribution.

Time evolution of the numerical experiment are presented in Figure 8 and Figure 9. The formation of a dome shape is the main feature observed. The dome shape formation mechanism is due to the recirculation of the fluid. Initially, the fluid and particles are in repose. After the particles start to move downward driven by the gravity force, a drag force is generated at the particle and, in reaction, the same force is exercised over the fluid trough the source moment. This source moment impels the fluid in center of domain downward that, because of incompressibility, impels the surrounding fluid upward, causing the circulation of the fluid. The circulation creates an axial component of the fluid velocity that transports the particles axially, creating the dome profile.

Figure 8: Evolution of the numerical “salt plume”.

![Salt grain size distribution](image1)

![Evolution of the numerical “salt plume”](image2)
Figure 9: Evolution of fluid velocity field of the numerical “salt plume”.

The experimental result is illustrated at Figure 10. First, a similar dome shape is observed. Despite the salt plume is not centered, the set of images shows an initial small conglomerate of salt grains that gradually transforms in a dome shape with a tail of grains, the same features that appeared in the numerical simulation. Secondly, the numerical and experimental results have inherent differences because of several points, as cubic shape of salt grains, chemical interaction between salt and water, and initial conditions of salt grains.

Figure 10: Evolution of the experimental “salt plume”.

Finally, it is important to highlight that the feature presented in the salt plume case can be only observed in a two-way simulation, since the particles motion due to gravity generates the fluid flow that causes the dome shape of the particles. Although quantitative data could not be recovery for comparison purposes, the numerical simulation provided a deeper knowledge of the phenomenon.
Table 2: Properties of the salt plume numerical experiment

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid time step</td>
<td>0.001</td>
<td>s</td>
</tr>
<tr>
<td>Simulation duration</td>
<td>1.0</td>
<td>s</td>
</tr>
<tr>
<td>Dynamic viscosity</td>
<td>0.001</td>
<td>Pa.s</td>
</tr>
<tr>
<td>Fluid’s specific mass</td>
<td>1000</td>
<td>Kg/m³</td>
</tr>
<tr>
<td>Fluid domain size</td>
<td>0.1 x 0.1 x 0.25</td>
<td>m</td>
</tr>
<tr>
<td>Mesh</td>
<td>Structured, 12500 tetrahedra with 0.01 m side</td>
<td>-</td>
</tr>
<tr>
<td>DEM time step</td>
<td>0.00001</td>
<td>Unit</td>
</tr>
<tr>
<td>Number of particles</td>
<td>28224</td>
<td></td>
</tr>
<tr>
<td>Particle specific mass</td>
<td>2170</td>
<td>Kg/m³</td>
</tr>
<tr>
<td>Contact rigidity</td>
<td>1.0E+2</td>
<td>N/m</td>
</tr>
<tr>
<td>Particle diameter</td>
<td>Gaussian Distribution ($\mu = 309; \sigma = 114$)</td>
<td>µm</td>
</tr>
</tbody>
</table>

FINAL REMARKS

This work presented the main aspects of a successful implementation of a DEM-FEM hybrid approach to a problem of bedload transport.

The case studies showed excellent qualitative results, representing the main physical features expected. For future work we intend to extend the verification of the implementation with other cases and possible validation by comparing the experimental results.

ACKNOWLEDGMENTS

The authors recognize and acknowledge the invaluable support given to this work by Petrobras S.A., CNPq and ANP, through the Programa de Formação de Recursos Humanos, PRH-02.

REFERÊNCIAS


Hybrid FEM-DEM approach applied to bedload transport


