



25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-1897

Transport And Deposition Modeling Of Rock Cuttings In The Ocean Floor

Paulo Costa Porto de Figueiredo Barbosa

João Marcelo Vedovoto

Aristeu da Silveira Neto

UFU - Federal University of Uberlândia, Av João Naves de Ávila, 2121, Campus Santa Mônica, Uberlândia, MG CEP 38400-902, Brazil

paulo.barbosa@ufu.br; vedovoto@ufu.br; aristeus@ufu.br

Abstract. *The process of drilling oil wells generates waste that needs to be monitored and measured. In this sense, the present work proposes the modeling that involves the transport, dispersion, and deposition of rock cuttings on the ocean floor with an emphasis on coarse solids. It is intended to determine the thickness of the deposited cuttings layer as well as the spatial distribution area of the discards. For this will be used a Computational Fluid Dynamics model coupled in 2 ways of a discrete parcel model. Good results were obtained, which showed that the presented model was able to qualitatively exemplify the occurrence of the spatial distribution of the rubble.*

Keywords: *particles transport, CFD-DPM, 2-ways*

1. INTRODUCTION

Due to the environmental impact caused by the increasing exploration and production of oil and gas, it is necessary to monitor the dispersion of residues resulting from such activity. In this context, drilling of oil wells has become the target of environmental licensing processes by the responsible for control the use of natural resources - IBAMA.

According to the data ELPN / IBAMA N^o 039/05, some models are required to the trajectory and dispersion studies of each type of effluent that intend to throw to the sea (drilling fluid, rubble, water from production, effluents from sealing test, etc.). For this purpose, computerized tools, operational data, oceanographic, and others may be used.

In this sense, the present work proposes a transport, dispersion, and deposition modeling of rock cuttings on the ocean floor with an emphasis on coarse solids. It is intended to determine the thickness of the deposited rubble layer as well as the spatial distribution area of the discards. A computational model will be created for the transport of particles, considered as Lagrangian, in a continuous medium, numerically treated eulerian.

The numerical-computational simulation problem was performed in MFSim, that is a computational platform in development by the Fluid Mechanics Laboratory (MFlab) at the Federal University of Uberlândia. Nowadays, to obtain better results, the MFSim can insert adaptive dynamical meshes for complex problems, since the resolution mesh is by the scale of the phenomena.

The dynamic behavior of the continuous phase (seawater and drilling mud), will be approximated using the Eulerian framework, where the flow is described by the Navier-Stokes equations. The behavior of the particles is described by a Lagrangian approach through the Discrete Particle Method (DPM). In this methodology, the movement of each particle follows Newton's second law.

According to Norouzi *et al.* (2016) the coupling between the phases may occur in terms of the momentum, energy, and mass. The modeling of the coupling is based on the contribution of each phase in the preponderant phenomena of the flow. The computational cost increases according to the effects associated with time and length scales. The interaction between phases can be characterized in three different ways: 1-way coupling where the continuous phase affects the transport of particles due to drag, but the particles do not influence the fluid flow; 2-way coupling where both phases are influenced by each other, but the dispersed phase does not interact with your self; and 4-way coupling where the continuous phase affects the transport of particles, the particles also influence the flow of the fluid, and collisions between particles must be considered.

In this work, a 2-way model will be evaluated, the results obtained with this formulation will be essential for a more realistic approach to be presented later.

2. MATHEMATICAL MODELING

For the considered flow (seawater and drilling mud), the dynamics of the continuous phase was modeled in the Eulerian referential, where the fluid dynamics is modeled through the Navier-Stokes equations. For the formulation of the behavior

of the particles present in the flow, a Lagrangian approach was used through the Discrete Particle Method (DPM). In this methodology, the movement of each particle obeys Newton's second law.

A two-way strategy is retained for the present work. In this case, both phases are influenced by each other, but the dispersed phase does not interact with yourself (e.g. tailings collision and particle-particle interaction). The dynamics of the continuous phase is expressed by the Eq. (1) and Eq. (2) Eq. 2. These equations were initially shown by (Anderson and Jackson, 1967).

$$\frac{\partial}{\partial t} (\rho_F) + \frac{\partial}{\partial x_i} (u_{F_i}) = 0, \quad (1)$$

$$\frac{\partial \rho_F u_{F_i}}{\partial t} + \frac{\partial \rho_F u_{F_i} u_{F_j}}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\mu_F \mu_t) \left(\frac{\partial u_{F_i}}{\partial x_j} + \frac{\partial u_{F_j}}{\partial x_i} \right) \right] + \rho_F g_i + F_i, \quad (2)$$

where ρ_F is the specific mass of the fluid, μ_F is the dynamic viscosity, μ_t is the eddy viscosity, p is the pressure field, u_F is the velocity vector of the fluid, g is the gravitational field acceleration, the indexes i, j and k are associated with the coordinate axes, and f_{F_p} is the vector of external forces.

It is emphasized that the term f_{F_p} allows the coupling of the Navier-Stokes equations to the equations of motion of the particles (rubble). It was described by de Souza *et al.* (2014) and is given by:

$$f_{F_{p_i}} = -n \left\langle m_p \left(\frac{du_{p_i}}{dt} \right) - \left(1 - \frac{\rho_F}{\rho_p} g_i \right) \right\rangle. \quad (3)$$

DPM is capable to solve the fluid-particle interaction and calculates the displacement of particles within the flow. In this methodology (Schwarzkopf *et al.*, 2011) the motion of each particle obeys Newton's second law, mathematically expressed by Eq. (4) and Eq. (5):

$$m_p \frac{du_{p_i}}{dt} = f_{D_i} - f_{BW_i} + f_{S_i} + f_{M_i}, \quad (4)$$

$$\frac{dx_{p_i}}{dt} = u_{p_i}, \quad (5)$$

where f_D , f_{BW} , f_S e f_M are drag force, combination of the forces weight and buoyancy, lift force due to the non-uniform flow (Saffman) and lift force due to rotation (Magnus), respectively.

The drag force is given as:

$$F_{D_i} = \frac{\pi \rho_F d_p^2}{8 m_p} C_d \|u_f - u_p\| (u_{f_i} - u_{p_i}) \quad (6)$$

The drag coefficient C_d is showed in de Souza *et al.* (2014).

$$\begin{aligned} \frac{24}{Re} & \quad Re \geq 1000 \\ 0.44 & \quad Re < 1000 \end{aligned} \quad (7)$$

were:

$$Re = \frac{\rho_F d_p |u_{F_i} - u_{p_i}|}{\mu_F} \quad (8)$$

Combined buoyancy and weight are described by:

$$F_{Bw_i} = g_i \left(1 - \frac{\rho_F}{\rho_p} \right) \quad (9)$$

The lift force due to the non-uniform flow (Saffman) can be expressed as:

$$F_{S_i} = \frac{\pi \rho_F d_p^3}{8 m_p} [(u_{F_j} - u_{p_j}) \times \omega_k] \quad (10)$$

were:

$$\omega_k = \varepsilon_{ijk} \frac{\partial u_{F_i}}{\partial x_j} \quad (11)$$

An, the lift force due to the rotation (Magnus) can be expressed like:

$$F_{M_i} = \frac{\pi \rho_F d_p^2}{8 m_p} \|u_f - u_p\| \frac{\Omega_j \times (u_{f_i} - u_{p_i})}{\|\Omega_j\|} \quad (12)$$

were Ω is the angular velocity of the particle.

3. Numerical Model

In this work, it was analyzed the emission of particles with 10 mm diameter through a 5 m long nozzle and 0.5 m radius located at $x = 15$ m, $y = 0$ m, and $z = 15$ m. It was also considered the action of ocean flow in the x -direction, which acts in a way to influence the dispersion of the particles. For the simulation, a rectangular domain of $[40 \times 40 \times 30]$ m in the directions x , y , and z respectively was considered (See Fig. 1), The smaller characteristic mesh size was discretized in volumes whose lateral has a size of 0.21 m, the intermediate mesh has volumes whose side has a size of 0.42 m and the thicker mesh volumes whose side has a size of 0.84 m. The boundary conditions are shown in Tab. 1.

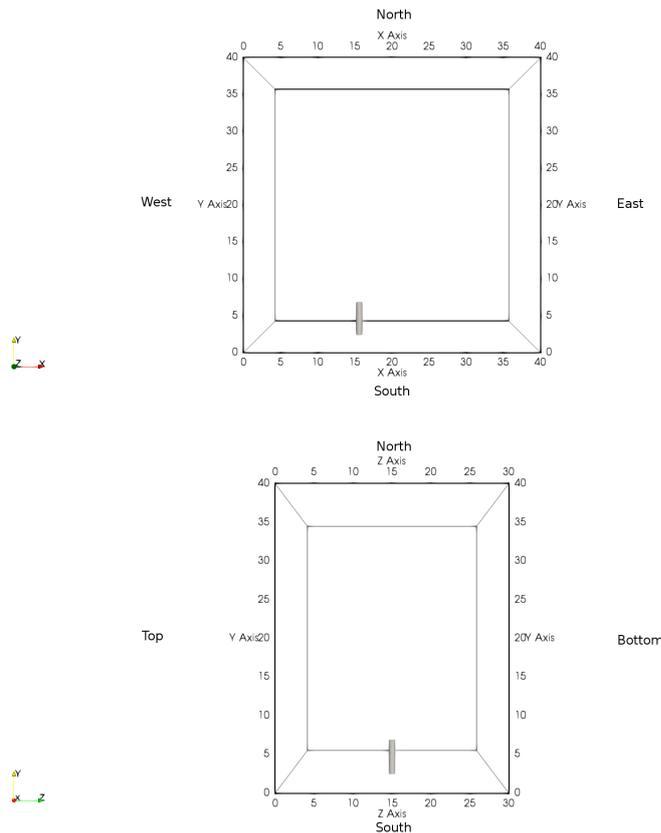


Figure 1. Computational domain

Table 1. Boundary conditions used in the numerical model.

Boundary condition location	Velocities (m/s) and gradients (1/s)		
	u	v	w
$x = 0$ m (west)	0.1 m/s for depth of 1000 m 0.22 m/s for depth of 2000 m	0	0
$x = 40$ m (east)	wave equation ⁽¹⁾	wave equation ⁽¹⁾	wave equation ⁽¹⁾
$y = 0$ m (south)	0	0	0
$y = 40$ m (north)	$\frac{\partial u}{\partial y} = 0$	$\frac{\partial v}{\partial y} = 0$	$\frac{\partial w}{\partial y} = 0$
$z = 0$ m (top)	$\frac{\partial u}{\partial z} = 0$	$\frac{\partial v}{\partial z} = 0$	$\frac{\partial w}{\partial z} = 0$
$z = 40$ m (bottom)	$\frac{\partial u}{\partial z} = 0$	$\frac{\partial v}{\partial z} = 0$	$\frac{\partial w}{\partial z} = 0$

⁽¹⁾ Orlandi (1976)

Data and properties used for drilling mud and the particles leaving the nozzle are shown in Tab. 2. It should be noted that in the simulations presented in this report it is considered that the fluid flowing in the domain, and the fluid exiting the nozzle (drilling mud) have the same properties.

Although this work presents the problem through simplified modeling, the results were able to exemplify the occurrence of the spatial distribution of the rock cuttings qualitatively. Three cases were simulated:

The first one considered the emission of particles through a 5 m length nozzle located on the ocean floor at a 2000 m

of depth, being influenced by the action of an ocean current in the x-direction with a velocity of 0.2 m/s that acts on the dispersion of the particles. The disposal time was 15 s and the coefficient of restitution (Cr) is equal to 1;

$$Cr = \frac{V_f}{V_i} \tag{13}$$

were V_f and V_i are the velocity of the particle after the collision to a wall and the velocity before the collision, respectively.

In the second case, the same condition was considered but now the coefficient of restitution (Cr) is equal to 0;

The third one was simulated by using the same parameters of the previous case, added with the presence of a second immersed boundary simulating the effects generated by the drilling column with angular velocities equal to 100 rpm;

Table 2. Data and properties used for drilling mud and the particles leaving the nozzle.

Proprieties	Rock cuttings particles	Drilling mud
Nozzle flow rate	0.004 m ³ /s	2.09 m ³ /s
Density (ρ)	2500 kg/m ³	1024 kg/m ³
Velocity	0.0056 m/s	2.66 m/s
Dynamic Viscosity (μ)	0.0147 Pa.s	0.00098 Pa.s

4. Results

Note, as expected, in Fig. 2, Fig. 3 and, Fig. 4 the flow imposed in the x-direction provides a dispersion of cuttings along the preferential direction. Even that, the occurrence of vortices due to the presence of the nozzle and the cross-flow influenced the trajectory taken by the particles, creating more significant concentrations in the areas of proximity to the nozzle.

The coefficient of restitution does not change substantially the extent of disposal (see Fig. 2 and, Fig. 3). It is also noted that the presence of the perforation column in Fig. 4, provides a more concentrated discard at the wellhead since it can induce stronger downside rotating vortices. The Fig. 2, Fig. 3 and, Fig. 4 are colored by the number of particles (np) in each finite volume.

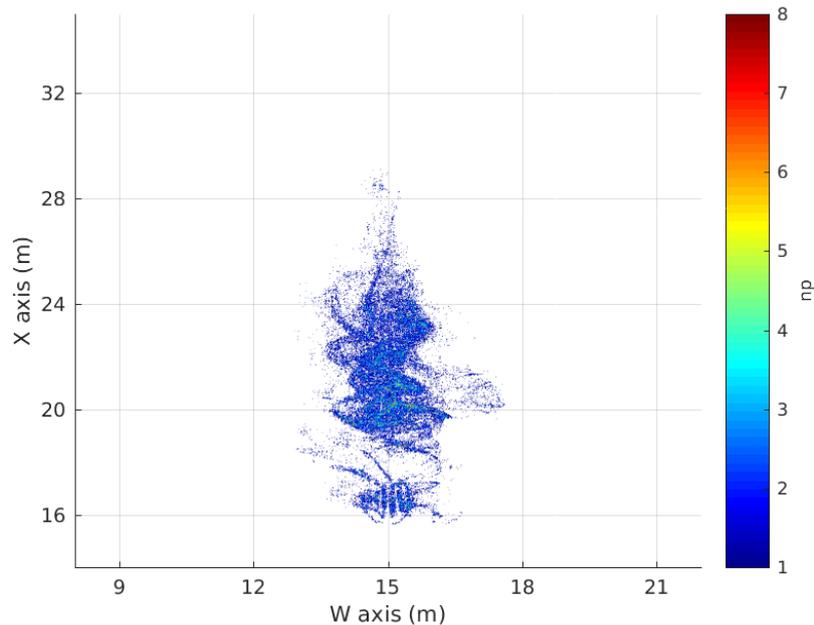


Figure 2. Scatter plot for 15 s simulation of gravel dispersion with $V_e = 0.2 \text{ m/s}$ and $CR = 1$

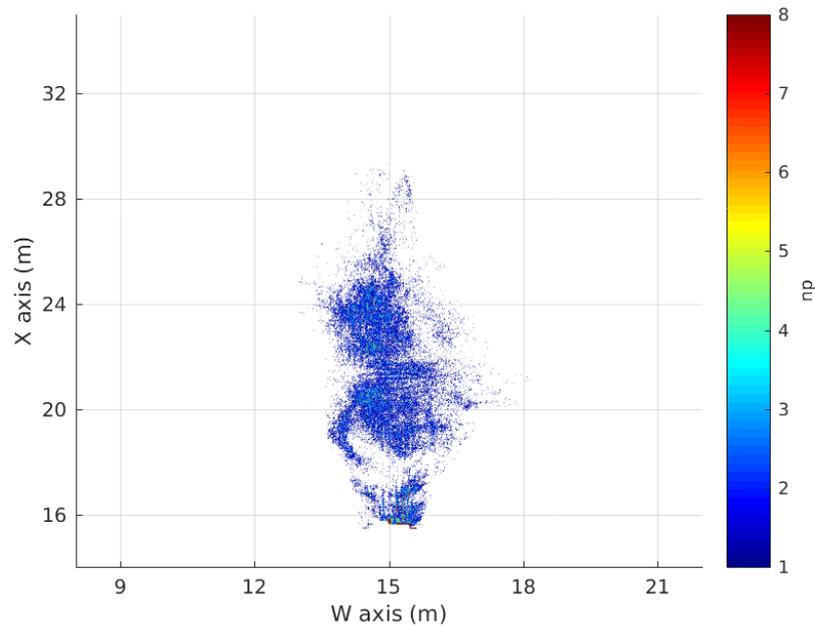


Figure 3. Scatter plot for 15 s simulation of gravel dispersion with $Ve = 0.2$ m/s and $CR = 0$

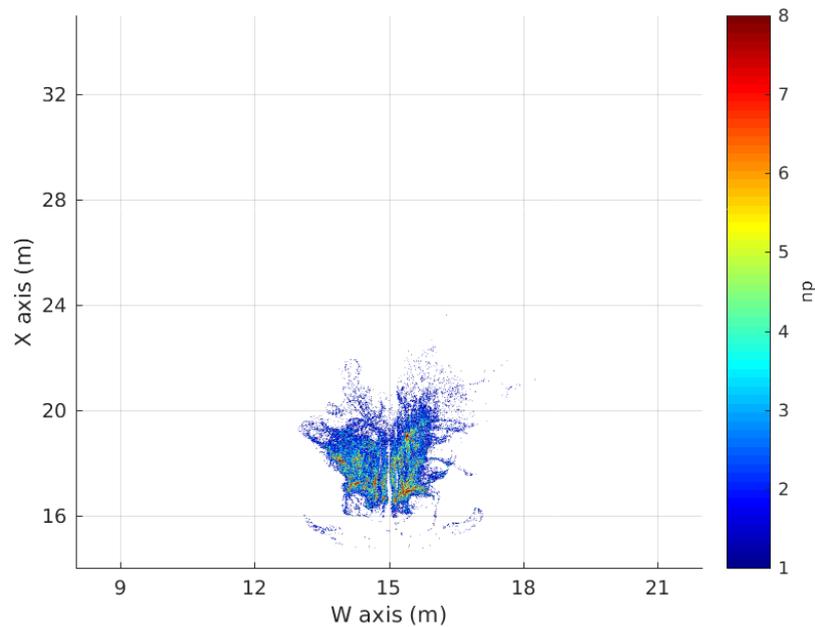


Figure 4. Scatter plot for simulation of 15 s gravel dispersion with $Ve = 0.2$ m/s, $CR = 0$ and with the drill column rotating at 100 rpm

5. Conclusions and Perspectives

The model was able to exemplify the occurrence of the spatial distribution of the rock cuttings due to certain preponderant parameters in each studied case in spite of the assumed simplifications. For a more realistic approach, the flexible sphere model (DEM) should be implemented. It can solve the fluid-particle interaction and calculate the displacement of the particles within the flow, considering the interaction between the particles employing a contact force. It is also necessary to include models for evaluation particle breakage and resuspension due to external forces. In this work, it was observed that the region of interest has a large volumetric fraction of particles. Therefore formulations for a dense continuous phase flow must be implemented.

6. ACKNOWLEDGEMENTS

The authors are thankful for the financial support provided to the present research effort by Petrobras and CAPES.

7. REFERENCES

- Anderson, T.B. and Jackson, R., 1967. "Fluid mechanical description of fluidized beds. equations of motion". *Industrial & Engineering Chemistry Fundamentals*, Vol. 6, No. 4, pp. 527–539.
- de Souza, F.J., Silva, A.L. and Utzig, J., 2014. "Four-way coupled simulations of the gas–particle flow in a diffuser". *Powder technology*, Vol. 253, pp. 496–508.
- Norouzi, H.R., Zarghami, R., Sotudeh-Gharebagh, R. and Mostoufi, N., 2016. *Coupled CFD-DEM modeling: formulation, implementation and application to multiphase flows*. John Wiley & Sons.
- Orlanski, I., 1976. "A simple boundary condition for unbounded hyperbolic flows". *Journal of computational physics*, Vol. 21, No. 3, pp. 251–269.
- Schwarzkopf, J.D., Sommerfeld, M., Crowe, C.T. and Tsuji, Y., 2011. *Multiphase flows with droplets and particles*. CRC press.

8. RESPONSIBILITY NOTICE

The following text, properly adapted to the number of authors, must be included in the last section of the paper:
The author(s) is (are) the only responsible for the printed material included in this paper.