EULER-LAGRANGE APPROACH OF DUST EMISSION AND DISPERSION FROM STOCKPILES OF GRANULAR MATERIALS

Bruno Furieri, bruno.furieri@ifes.edu.br
Edilson Luiz do Nascimento, edilsonluiz@gmail.com
Maria Clara S. F. Caliman, mcaliman@ifes.edu.br
Enilene R. Lovatte, enilene.lovatte@ifes.edu.br
Instituto Federal do Espírito Santo, Av. Vitória, 1729, Jucutuquara, Vitória, 29046-020, Brazil

Jane Meri Santos, jmerisantos@yahoo.com.br
Neyval Costa Reis Junior, neyval@gmail.com
Universidade Federal do Espírito Santo, Av. Fernando Ferrari, 514, Goiabeiras, Vitória, 29075-910, Brazil

Abstract. *The present work studies the dispersion of silt particles using an Euler-Lagrange approach. The main results are focused on instantaneous three-dimensional fields of particle location and concentration over the domain and on the ground surface surrounding the source. We analyse more finely the turbulence structure on and surrounding the stockpile to easily understand the take-off and dispersion process. The present work focuses on the investigation of particles behaviour emitted by aeolian erosion of stockpiles of granular materials. The Eulerian approach simulates the fluid phase (wind flow) by means of Large Eddy Simulations (LES) to incorporate the effects of atmospheric turbulence. The Lagrangian approach simulates the unsteady particle tracking for the discrete phase (granular materials). The numerical results well reproduced the mean velocity profile, while the turbulence intensity accounted for differences of 2% or less compared to the experimental results. The Euler-Lagrange approach presented very interesting results about particles behavior downstream the pile. Some profiles helped to explain the zones of higher and lower concentration downstream the pile. High concentration values of granular material particles were found parallel to the highest friction zone on the ground near the pile. It was found that concentrations were in the range* 1.10^{-4} *g/cm*^{3} *to* 6.10^{-4} *g/cm*^{3} *throughout the study area, consistent with the range of the experimental order of magnitude in the bed of particles found in literature.*

Keywords*: Euler-Lagrangian Approach, Turbulent Dispersion, Large Eddy Simulation, Non-erodible particles, Stockpile.*

1. INTRODUCTION

Airborne particles affect population wellbeing and health and may create adverse conditions for social activities due to nuisance caused by dust. Particles emissions to atmosphere may occur due to wind erosion events. Wind erosion is a natural process that disturbs a given surface causing entrainment of particles into atmosphere. The erosion occurs if wind flows on a surface containing agglomerates of particles and exceeds a certain threshold friction velocity. Aeolian erosion involves three main forces: (i) aerodynamic forces which tends to remove particles from the surface where they are settled and gravitational and (ii) cohesive forces that tend to limit particles removal (Shao, 2008). Some authors studied the collision of airborne particles with the surface, re-emissions and particle trajectories (Shao, 2008; Alfaro et al, 2004; Kok & Renno, 2009; Ren & Huang, 2010; Carneiro et al, 2013). Other studies have reported the effects of air flow structure around a source or obstacle on wind erosion (Wiggs et al. (1996) and McKenna et al. (1997)).

There may be effects of turbulent flow impinging on stockpiles on the exchange of mass, momentum and energy between the phases, called continuous (flow) and discrete (particle). Therefore, one must assess the turbulent vortices behavior and their interactions with the eroded surface. The increasing of computational processing capacity, the improvement of Lagrangian particle transport technique, besides the development of methodologies for boundary conditions for the LES technique allowed for the numerical simulation of dispersion of particles emitted from stockpiles. At the same time, the advances in the techniques of the atmospheric boundary layer representation in wind tunnels have also allowed for a better understanding of the problem.

The aim of this work is to investigate the dispersion of silt particles using the Euler-Lagrange approach. In the present paper instantaneous three-dimensional fields of particle location and concentration over the entire domain and on the ground surface are shown. In addition, resuspension has been quantified. In the present paper we are analyzing more finely the turbulence structure on and surrounding the stockpile to easily understand the take-off process.

2. NUMERICAL SIMULATION METHODOLOGY: EULER-LAGRANGE APPROACH AND LARGE EDDY SIMULATION

2.1 Boundary and Initial conditions

The initial conditions to determine velocity field and dispersion and turbulence parameter to simulate LES, the model were κ-ω SST. The model κ-ω SST uses the original formulation κ-ω model near the wall and $\kappa - \varepsilon$ model far from wall.

At the inlet, longitudinal velocity, turbulent kinetic energy and specific dissipation data were obtained from a calculation in a simple three-dimensional channel. The symmetry condition was applied to the top wall of the computational domain. The sides and wall were considered fixed walls. A mass flow rate value to obtain a 5.5 m/s was used to input condition. This velocity was chosen as experiments in wind tunnels so that the velocity at 0.4 m from the bottom wall of the numeric field is equal to the velocity at the wind tunnel center (Turpin, 2010), i.e., 6.5 m/s. To satisfy the conditions imposed by this approach, the mesh wall was refined so that the first mesh point is within $z^+ \leq 1$. Figure 1 represents the simulation domain, in dimensionless units as a function of stockpile height and Figure 2 shows the mesh surrounding the stockpile surface. It is observed the ground around the stockpile (Figure 2(a)) around the computational domain and a description of the mesh used in the stockpile wall (Figure 2(b)). The mesh was produced by extrusion of triangular cells. The computational mesh has around 10 million cells enough to allow LES investigation. For modeling dispersion by Lagrangian approach is necessary first to know the rate of emission of particles inserted in the fluid flow. For this, we used a method in which particles are injected into the domain from the pile surface. The strategy used to inject the particles was adapted from Furieri (2012), Badr and Harion (2007) and Turpin and Harion (2009).

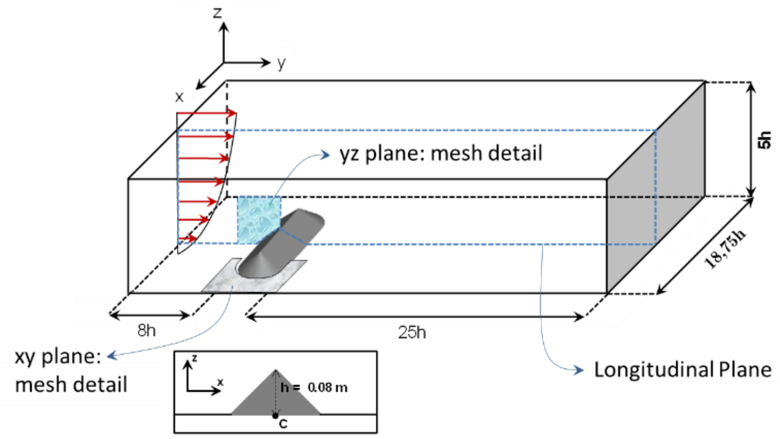

Figure 1 – Simulation domain with cut in plan and surface to show mesh.

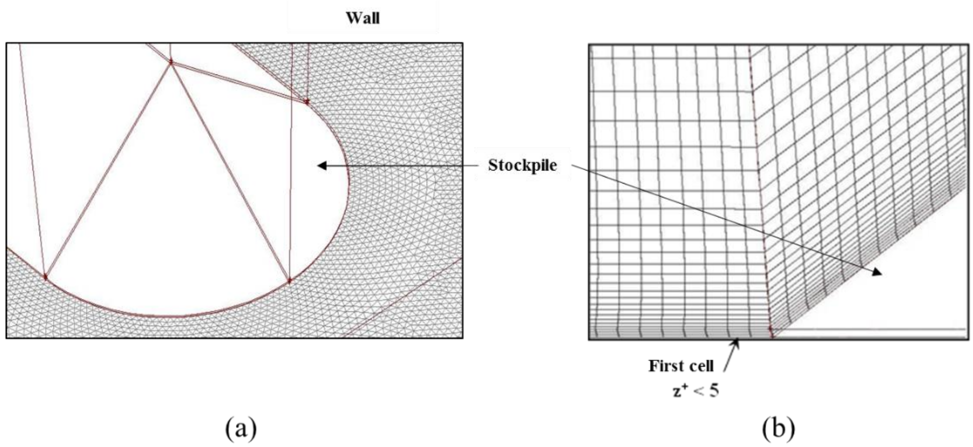

Figure 2 – Computational mesh: (a) on the ground around the stockpile (z=0); (b) vertical and longitudinal plane representing the middle of the stockpile (x=0).

2.1 Euler-Lagrange approach

The Euler-Lagrange approach is applied in two steps: transport equations solution for the continuous phase (Eulerian) and simulation of a second discrete phase, called particle (Lagrangian). The Euler-Lagrange approach requires knowledge of particle-domain interactions. As particles may leave the domain or impinge the ground or stockpile surface, or in some areas the particles may disappear (passing through the outlet boundary) or collide and reflect on a wall.

The quantity of particles emitted is accounted based on the USEPA model modified to consider the influence of non-erodible particles in agglomerates stockpiles containing a range of particles diameters. In this case, the emitted mass flux decreases with time as noticed in the experimental work of Furieri et al. (2013). The positions of particles modify with time, as in a transient regime. Each particle moves over a certain number of time steps, not necessarily reaching a final destination, before the solution update. The calculations of discrete and continuous phases occur simultaneously (although the time step of the two phases may be different). The Euler-Lagrange approach and mathematical modeling inherent in this approach simulates the already widespread studies of the particle phase and the forces acting on the particles together with the size distribution of the particles.

The Euler- Lagrange approach and the mathematical model included in this work are part of studies widely used in the particle phase and the forces that act along the particles of different sizes.

2.2 Large Eddy Simulation (LES)

LES consists of directly solving the large scales of the turbulent flow and model the small scales using a sub-grid model. A filter is applied within the conservation equations and the computational solution is obtained transiently. From the point of view of the energy spectrum, LES start from a value within the range of scales between inertial and large, while the RANS methodology models the entire spectrum. Based on this argument, it is justified to carry out LES (Large Eddy Simulation), where the filtered solution of the equations solves directly the turbulent structures transporting mass and momentum, while models smaller structures. There is a wide range of sub-grid models among them we will use the WALE. The main advantage of the WALE model is the ability to reproduce the transition from laminar to turbulent flow, and does not require a second filtering, as the dynamic Smagorinsky-Lilly model.

The use of conservation equations can be simplified, or changed, with the aim to facilitate both the theoretical analysis and numerical computation. Among these simplifications, we have: (i) conditions of neutral stability; (ii) atmospheric flow limited to microscale; (iii) incompressible fluid and constant viscosity; (iv) Lagrangian approach for calculating the concentration of pollutants; (v) spherical particles.

The governing flow equations are solved numerically employing the commercial software FLUENT 15.0 (Ansys Fluent, 2015), employing the numerical method called pressure-based solver, based on finite volume (Versteeg and Malalasekera, 2007). The velocity field is obtained from the momentum equation, while the pressure field is obtained from the pressure correction equation, by manipulating the equations of continuity and momentum. This technique is based on volume control, consisting of the following steps: (i) domain division in discrete control volumes using a computational grid, (ii) integration of the governing equations in the individual control volumes, with the goal of building the algebraic equations of unknown variables (velocity and pressure) and (iii) linearization of the discretized equations and solving the linear system of equations to produce the update of the dependent variables.

The numerical method used for solving the discretized equation with implementation for structured and unstructured meshes based on Gauss theorem and the weight functions of the finite element method. One of the biggest challenges for the simulation of atmospheric flows is the representation of the incident flow. According to Tabor and Baba-Ahmadi (2010), one of the techniques usually employed to represent the incident flow is the artificial generation of the turbulent profiles based on the fluctuation around the average values, technique employed in this work. This paper proposed using turbulence generation method related to the velocity at the entrance proposed by Smirnov and Celik Shi (2001), called Spectral Synthesizer.

Thus, the input conditions are entered by an average profile and velocity fluctuations by means of Spectral Synthesizer method using the filtered conservation equations characteristic of turbulence model of the large scales. The solid surface is handled automatically by subgrid modelling, so that there is no need for damping function. The coupling rate-pressure method was used SIMPLEC (Patankar, 1980), forward in time was used Implicit method and 2nd Order Interpolation method for Upwind pressure and 2nd Order Momentum the central differences. The simplifying assumptions were applied as follows: (i) incompressible fluid; (ii) constant laminar viscosity; (iii) stable neutral condition; (iv) solution of the atmospheric flow limited to microscale.

An important approach to the study of stockpiles when there is incident flow is the analysis of the influence of the obstacle in the flow. Several studies analyzing flow into stockpiles of granular materials (Alfaro et al. (2004), Carneiro et al. (2013), McKenna et al. (1997)). A technique used to simulate the boundary condition in the domain entry for SLE is the mathematical synthesis. In this technique for the generation of fluctuations is based on the use of mathematical processes not directly related to the turbulence to generate a random field in the entry of the domain with properties similar to turbulence. The method used in this study is called RFG (Random Flow Generation) (Kok et al. (2004)).

The tracking of the particles was simulated in Ansys Fluent package; it is possible due to a model that takes into account a force balance in each particle that is injected into the area of the stockpile surface. The flow of fluid and particles are treated following Euler-Lagrange methodology (as previously explained). The fluid is treated as eulerian, using the Navier-Stokes equations, while the dispersed phase is resolved by tracing a large number of particles which are subjected to the fluid flow.

3. RESULTS AND DISCUSSIONS

3.1. Validation of turbulence and particle dispersion modelling: bed of particles

In order to verify the precision/accuracy of the mathematical model proposed for the representation of atmospheric flow and the emission and dispersion of particles of different particle sizes emitted into the atmosphere due to wind erosion, numerical simulation experiment was carried out carried out wind tunnel (Zhang et al. (2007)) which is simulated in a particle bed. Once validated the model, numerical simulation of an oblong stockpile was made with particles of different grain sizes on the surface of another experiment in wind tunnel (Carneiro et al. (2013), McKenna et al. (1997). The simulations were carried out using Ansys Fluent.

Figure 3(a) shows the inlet vertical profiles of mean velocity. The numerical results show reasonable agreement with the experimental profile obtained by Zhang et al. (2007) and predict well the thickness of the boundary layer, which means that velocity no longer, suffers the influence of the surface presence. Figure 3(b) presents the vertical profile of particle concentration in $g.cm^{-2}$ at the bed of particles. Very near the wall, there are some discrepancies between these results. However, from the height of 3 mm, the numerical results are in good agreement with the experimental data.

It is important to observe two relevant concerns presented by Zhang et al. (2007) about their experiments: (i) concentration near the wall: concentrations from 3 mm are in the range of 5.10^{-4} $g.cm^{-2}$ and 9.10^{-6} $g.cm^{-2}$ to 20 mm and (ii) fast concentration decay with height: reproduction of the experimental data is very difficult, especially in the region very close to the wall.

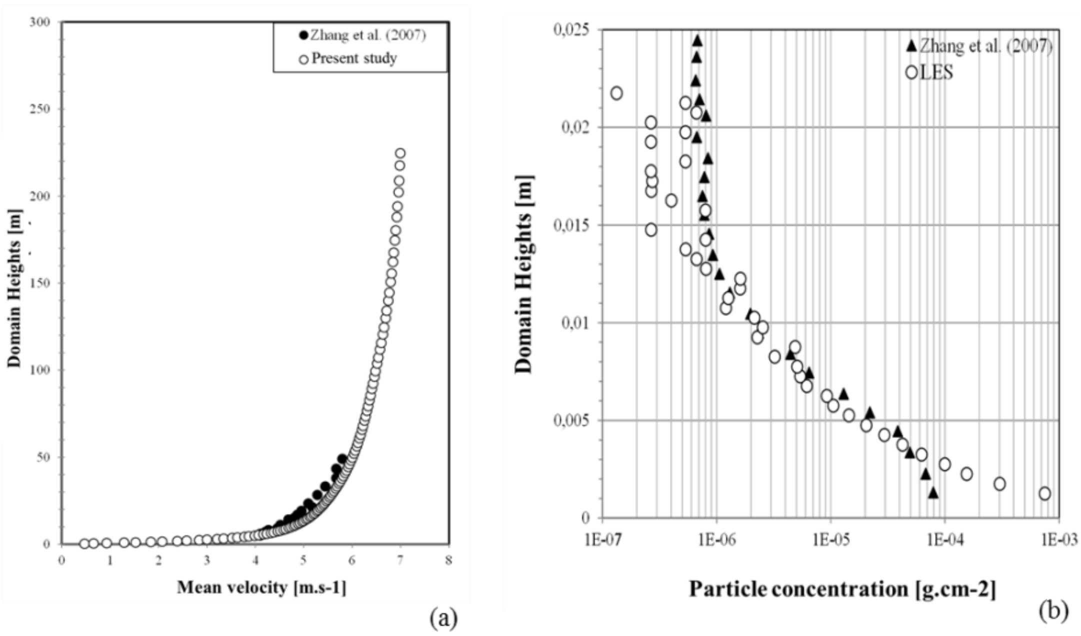

Figure 3 – Vertical profiles of (a) inlet mean velocity and (b) particle concentration at the bed of particles

3.2. Dispersion of particles emitted from an isolated stockpile

Figure 4(a) represents a pile found in areas of actual handling of ore or coal in steel industries, for instance.

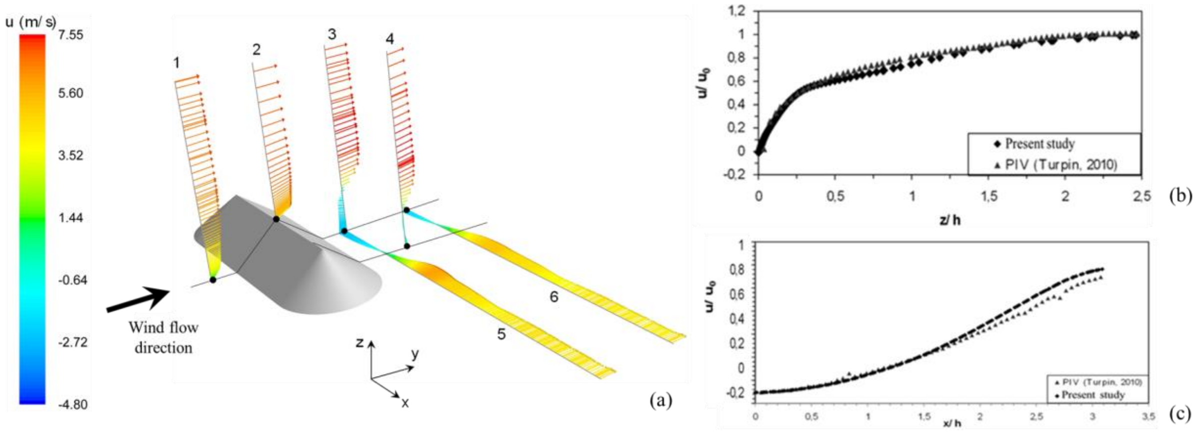

Figure 4 – (a) Numerical domain and velocity profiles location, (b) mean streamwise velocity plotted in Profile 1 and (c) mean streamwise velocity plotted in Profile 5.

Wind tunnel experimental work using PIV were used to validate the numerical simulations (Turpin (2010) and Furieri et al. (2013)). The numerical vertical profile 1 (Figure 4(b)), located at the entrance of the domain, is superposed on the experimental profile, showing that the numerical conditions on the inlet predicts well the wind tunnel. The error encountered between the experimental and numerical velocity profile was approximately 3.7%. Profile 5 (Figure 4(c)) is the mean horizontal velocity, located 0,005 m from the surface and 0,154 m after the center of the pile shows reasonable agreement in reproducing the experimental curve. A reasonable agreement was found in profile 5 over the horizontal distance up to 2 times the stockpile height, with a slight increase in velocity after that distance. The relative error was about 4.7%.

Figure 5 presents vertical profiles of particle concentration. Concentration values ranged between 4.10^{-5} and 1.10^{-7} g.cm^{-3}. Maximum values of concentration were found near the ground and the lowest concentrations slightly above the pile height. This profile can be explained by the proximity of the recirculation zone, in which case, there are local negative instantaneous velocities in comparison with the accelerating fluid at the top of the pile, as observed in Figure 2. Thus, the region analyzed have moving particles hop closer to the surface, rather than particles that are eroded from the side of the stockpile (in larger amounts) and the top (in smaller quantities) since these regions present the major instantaneous velocities of the fluid. The profile at 3.5h is located into the recirculation zone. The slightly larger height of the plume of contaminants is due to the inclination of the pile. This causes a slightly larger suspension height and a somewhat larger concentration zone. The profile at 6h coincides with the end of the recirculation zone and the continuity of turbulent wake. Once there, in this region, there is a smaller influence of the pile in line with the turbulent wake. Particles of smaller size appear in the heights of the plume, while larger ones tend to start heels and deposition on the surface

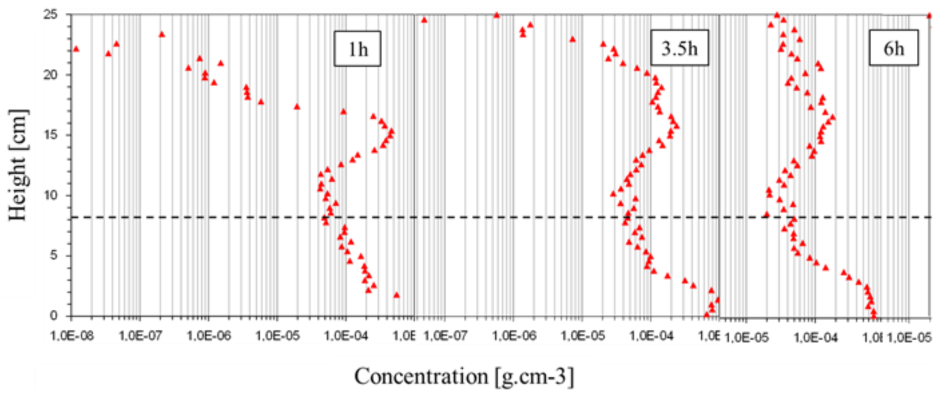

Figure 5 – Vertical profiles of particle concentration at each 0.2 m downstream from the pile.

Figure 6 represents the horizontal contours of settled particles, taken on the entire surface area. For this, the area was divided into squares of 1 cm (length) x 1 cm (width), totalizing 42.750 points where the concentration is calculated. One can observe areas where there are more particles. This is due to areas where there is an intense movement of particles at the sides of the pile. The highest concentrations are about 2.10^{-6} g.cm^{-3} at the end of the domain. The lowest concentrations are in the order of 1.10^{-7} g.cm^{-3}.

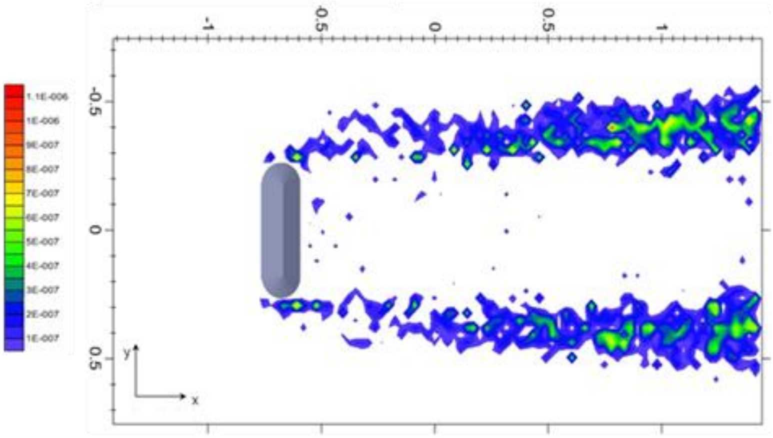

Figure 6 – Settled particles concentration [µg/m³] downstream the pile. The wind flows from left to right.

Figure 7 presents the particles distribution over the domain colored by their diameters. One can notice that the minimum and maximum diameters tested are in the range 100-194 µm. The blue particles (the smallest ones) remain easily inside the recirculation zone. Red particles (the coarsest ones) are deposited on the ground surrounding.

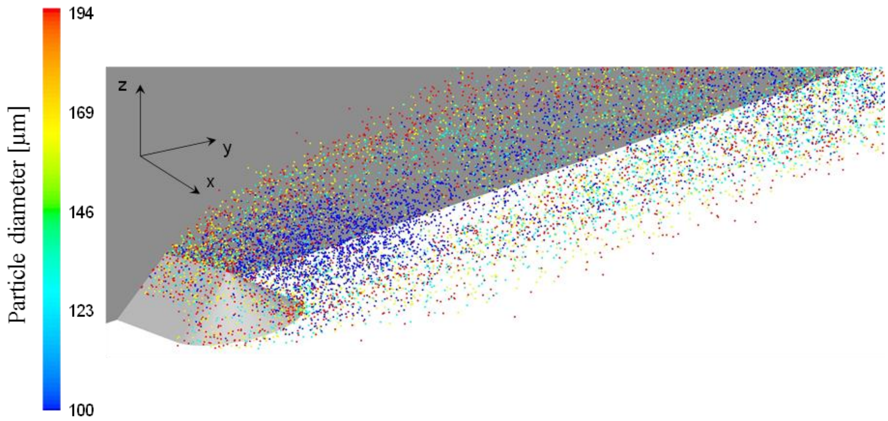

Figure 7 – Distribution of the particles colored by their diameters.

Figure 8 shows quantitative analyses of the resuspension of fine particles seen over the computational domain. The plots below show two variables that help to investigate the resuspension: resuspension factor (RF) and resuspension rate (RR). The term resuspension has been used to indicate the re-entrainment in the atmosphere of a granular material that was previously deposited. The term suspension or entrainment is used when the material was not previously deposited by atmospheric flow. The resuspension has been expressed as a factor or rate of resuspension. The resuspension factor is defined as RF = C/S where C is the airborne particles concentration and S is the concentration on the surface. The resuspension rate is defined the fraction of the surface species removed per unit time, RR = R/S. One can observe the high Resuspension Factor: the particles are emitted by the continuous emission from the stockpile in the time range shown in Figure 6, with a free surface velocity of 6.5 m/s, generating considerable recirculation zone and turbulent wake. The resuspension rate increases as fast as more particles are inserted into the domain.

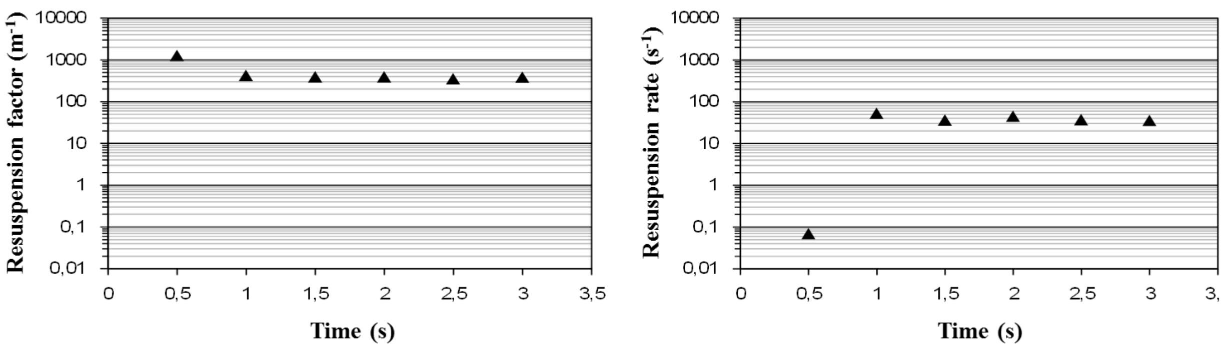

Figure 8 – Temporal evolution of (a) Resuspension Factor and (b) Resuspension Rate.

4. CONCLUSIONS

The suspension and deposition of MP of different sizes emitted from piles by aeolian erosion under neutral stability were investigated by applying a Eulerian-Lagrangian approach and LES turbulence model. Experimental results in wind tunnel (Zhang et al., 2007 and Turpin, 2010) were used to validate the numerical simulations. The present studied analysed the dispersion of particles emitted from an isolated pile subject to the incident turbulent flow.

The numerical results well reproduced the mean velocity profile, while the turbulent intensity accounted for differences of 2% or less compared to the experimental results. The Euler-Lagrange approach presented very interesting results about particles behavior downstream the pile. Some profiles helped to explain the zones of higher and lower concentration found downstream the pile over distances larger than 10 times the pile length. The highest concentration values on the ground near the pile were found on a zone parallel to the highest friction (on the pile surface – its lateral walls).

5. ACKNOWLEDGEMENTS

FAPES, CAPES and CNPq financially supported this work.

6 REFERENCES

Alfaro, S.; Rajot, J.; Nickling, W. Estimation of PM_{20} emissions by wind erosion: main sources of uncertainties. Geomorphology, vol. 59, pp. 63–74, 2004.

Carneiro, M.; Araújo, N.; Pahtz, T.; Herrmann, H. Mid-air collisions enhance saltation. Physical Review Letters, v. 111, p. 058001, 2013.

Furieri, B.; Russeil, S.; Santos, J.M.; Harion, J.L. Effects of non-erodible particles on aeolian erosion: Wind-tunnel simulations of a sand oblong storage pile. Atmospheric Environment, vol. 79, p. 672-680, 2013.

Kok, J. F.; Renno, N. O. A comprehensive numerical model of steady state saltation (COMSALT). Journal of Geophysical Research, Vol. 114, D17204, 2009.

McKenna, C. N.; Lancaster, T. N.; Nickling, W.G. Relations between dune morphology, air flow, and sediment fluxon reversing dunes, Silver Peak, Nevada. Sedimentology. v. 44, pp. 1103-1113, 1997.

Ren, S.; Huang, N. A numerical model of the evolution of sand saltation with consideration of two feedback mechanisms. The European Physical Journal E, v. 33, n. 4, p. 351–358, 2010.

Shao, Y. Physics and modelling of wind erosion. Atmospheric and Oceanographic Sciences Library, Vol. 37, 2a. Ed., 2008;

Turpin, C. Amélioration des modèles de quantification des émissions particulaires diffuses liées à l'érosion éolienne de tas de stockage de matières granulaires sur sites industriels. PhD thesis, University of Valenciennes, França, 2010.

Wiggs, G. F.S.; Livingstone, I. Warren, A. The role of streamline curvature in sand dune dynamics: evidence from field and wind tunnel measurements. Geomorphology. v. 17, pp. 29-46, 1996.

Zhang, W.; Wang, Y.; Lee, S. J. Two-phase measurements of wind and saltating in an atmospheric boundary layer. Geomorphology. v. 88(1-2), p. 109-119, 2007.

7. RESPONSIBILITY NOTICE

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