SURFACE VELOCITY CONTOUR ANALYSIS IN THE AIRBORNE DUST GENERATION DUE TO OPEN STORAGE PILES

Javier Toraño†, Rafael Rodríguez† and Isidro Diego†

† GIMOC, Mining Engineering and Civil Works Research Group
Oviedo School of Mines
C\ Independencia 13, 33005, Oviedo, Spain
e-mail: jta@uniovi.es
Web page: http://www.uniovi.es

Key words: Computational fluid dynamics, storage piles, airborne dust, dust emission

Abstract. In the framework of the Research Project CTM2005-00187/TECNO, “Prediction models and prevention systems in the particle atmospheric contamination in an industrial environment” of the Spanish National R+D Plan of the Ministry of Education and Science, 2004-2007 period, there have been developed several CFD models to simulate particulated material emission from mineral stockpiles. US EPA regulation determines the influence of the wind in the pile surface through tables and figures obtained from several tests done in atmospheric wind tunnels, depending on two typical shapes of stockpiles: a cone and a flat top pile. In order to create a computer based system that obtains the particulated material emission factor, CFD was selected as the way to simulate the effect of wind gusts in the pile surface. Several models were developed using the commercial code Ansys CFX 10.0, starting from several 3-D meshes of different resolutions generated using ICEM CFD 10.0. There were selected medium complexity turbulence models in order to obtain affordable resolution times in single processor machines, as well as following advices contained in related bibliography. These models were: k-epsilon (with and without surface roughness) and k-w based Shear-Stress-Transport (SST), combined with different logarithmic and plain wind profiles. Results were compared against the experimental data included within EPA and the best fit was obtained with a roughness k-epsilon model using a logarithmic wind profile.
1 INTRODUCTION

In the framework of the Research Project CTM2005-00187/TECNO, “Prediction models and prevention systems in the particle atmospheric contamination in an industrial environment” of the Spanish National R+D Plan of the Ministry of Education and Science, 2004-2007 period, there have been developed several CFD models to simulate particulated material emission from mineral stockpiles.

PM10 or PM50 material can be easily thrown in suspension from the stockpiles, creating an environmental problem not only in the vicinity of the mineral stockyards but also at long distances from the particle source. One of the ways to establish the amount of the particle emission is the use of US EPA regulations¹, which relates the stock material characteristics, the atmospheric parameters, the stacking and reclaiming operations and the shape of the wind flow around the stockpiles to create a particle emission factor.

US EPA determines the influence of the wind in the pile surface through tables and figures obtained from several tests done in atmospheric wind tunnels², depending on two typical shapes of stockpiles: a cone and a flat top pile. In order to create a computer based system that obtains the particulated material emission factor, CFD was selected as the way to simulate the effect of wind gusts in the pile surface.

Several models were developed using the commercial code Ansys CFX 10.0, starting from several 3-D meshes of different resolutions generated using ICEM CFD 10.0. There were selected medium complexity turbulence models in order to obtain affordable resolution times in single processor machines, as well as following advices contained in related bibliography⁴,⁶. These models were: k-epsilon (with and without surface roughness) and k-w based Shear-Stress-Transport (SST), combined with different logarithmic and semi-logarithmic wind profiles. Results were compared against the experimental data included within EPA and the best fit was obtained through a roughness k-epsilon model using a logarithmic wind profile.

After the successful velocity field simulation, currently we are developing the study of the evolution of the particulated material inside this air flow, as well as the subsequent comparison with the experimental data obtained by means of optical particle samplers, already used by our research team in previous particle movement model validation experiments³,⁵. User Fortran routines are being developed to simulate the emission of particles according to flow shape.
2 THE NEED FOR CFD CALCULATIONS

Open areas are the common practice for storing huge aggregate bulky cargoes, therefore dust lift must be considered as a significant environmental and operational problem. Several inputs influence over the quantity of dust to be generated:

- Wind characteristics like direction, speed, gusts, etc.
- Wind Accessibility.
- Elevation of the surface exposed to the wind.
- Particle parameters: diameter, density, shape, moisture, etc.
- Presence of non erodible elements and crust formation.
- Disturbance of the surface.
- Etc.

As it is foreseen, the study of this phenomenon is highly complicate due to this great number of variables to be considered so any theoretical – numerical simulation approach would probably result on a forced assumption of several simplifications but on the other hand an empirical valuation would only cover a limited range of working conditions, even more, for a given location it is known that conditions are really fluctuant.

When trying to establish a level of airborne dust generated from an open pile, one of the most extended methodologies is the USEPA (1998a), which covers not only that but from unpaved roads up to explosives detonation giving emission factors and procedures to estimate total emissions, control methods, etc.

For piles, USEPA gives several parameters and inputs for cone and flat top oval configurations. The study includes only this two pile shapes, shapes that unquestionably plays a role in the dust emission phenomena. When evaluating other pile shapes or multiple piles configuration Boolean operations adding the results of calculations done to this two simplified pile shapes must be done, not accounting for possible effects of obstacles or barriers in the wind path.

The problem of estimating the dust emission from a stockyard can be solved specifically for each stockyard disposition using CFD commercial codes. A 3D model of the mineral stockyard can be adequately meshed and wind field calculated in this spatial distribution.

But first there must be determined the emission rate for a known pile area, as well as check the CFD methods accuracy in defining the wind around a pile.

3. US EPA

The emissions from the storage pile activities may be estimated using the information from sections 13.2.4 and 13.2.5 of AP-42 Compilation of Air Pollutant Emission Factors.

The methodology applied by this section 13.2.5 Industrial Wind Erosion requires first to establish wind parameters being the fastest mile used to convert the values obtained from reference anemometers to equivalent friction velocity:

\[ u^* = 0.053 \, u_{10}^+ \]  

(1)
where
- \( u^* \) is the equivalent friction velocity
- \( u_{10}^* \) is the fastest mile of wind at 10 m height

This equation only applies to flat piles or those with little penetration into surface wind layer.

The emission factor for surface airborne dust subject to disturbances may be obtained in units of grams per square meter (g/m²) per year as follows:

\[
EmissionFactor = K \sum_{i=1}^{N} P_i
\]

where
- \( K \) is particle size multiplier
- \( N \) is number of disturbances per year
- \( P_i \) is the erosion potential corresponding to the observed fastest mile of wind for the \( i^{\text{th}} \) period between disturbances, calculated by equation [3].

\[
P = 58(u^* - u_i^*)^2 + 25(u^* - u_i^{*})
\]

\[
P = 0 \quad \text{for} \quad u^* \leq u_i^*
\]

where
- \( u^* \) is the friction velocity (m/s)
- \( u_i^* \) is the threshold friction velocity (m/s)

Therefore the erosion is directly affected by particle size distribution as well as by the frequency of disturbance\(^1\); because of the nonlinear form of the erosion, each disturbance must be treated separately.

But equation [1] assumes a typical roughness height of 0.5 cm and height to base ratio not exceeding 0.2; if the pile exceeds this value significantly, it would be necessary to divide the pile area into subareas of different degrees of exposure to wind. When talking about higher height ratios, it would be necessary to use:

\[
u^* = 0.10 \ u_S^*
\]

\[
u_S^* = \frac{u_s}{u_r} \ u_{10}^*
\]

EPA Section 13.2.5 gives for representative cone and oval top flat pile shapes the ratios of surface wind speed (\( u_s \)) to approach wind (\( u_r \)) derived from wind tunnel studies, so the total surface of the pile is subdivided into areas of constant \( u^* \) where equations [3], [4] & [5] can be used. Areas are defined as constant \( u_s/u_r \) values as well as its position over the pile surface.

4 CFD SIMULATION

Ansys CFX was selected to develop a numerical simulation of the piles, obtaining the
areas of constant $u^*$ where equation [3] could be used and doing the validation of the procedure by comparing its results with USEPA as the experimental reference study, when applied for cone and flat top oval piles.

CFD codes require a high number of simulation decisions to be taken. These are mainly relative to the boundary conditions and the turbulence model selected. Apart from common boundary conditions in the limits of the simulation domain the one to be focused on is the wind entrance. Initially we try a plain inlet, we mean a constant wind independent from height, as we were using models quite close to ground. As per related bibliography\(^1\) we immediately change to try using logarithmic wind profiles.

Regarding turbulence models, CFX 10.0 offers a wide range of them. The selection of one of these models depends in the physics of the problem we try to simulate, but also in the quality and density of the mesh that we are using to represent the simulated domain and the capacity of the computer(s) that are going to solve the calculation. Highly complex models, as DES or LES could get probably an accurate solution, but will take dozens of hours to generate an adequate fine mesh and hours of multiple parallel processing to get to the solution.

Our approach is to start obtaining a solution with medium complexity models, easy and quickly applicable to solve engineering problems over extensive physical domains, with no need for huge meshes that require multiprocessing capabilities of the calculation code and its cluster-type hardware associated.

There were selected medium complexity turbulence models following advices contained in related bibliography\(^4,6\), among others. These models were: k-epsilon (with and without surface roughness) and k-w based Shear-Stress-Transport (SST), combined with different logarithmic and semi-logarithmic wind profiles.

For all models an average of 100 to 200 iterations were needed to obtain results,
established at a RMS value of convergence rate of 1E-05. Figure 1 shows, left side, one of the outputs of the CFX-Solver, where clean convergence of the RMS residuals reaches 1E-05 level, and also shows in the right side the visualization of the maximum values of the residuals of the W component of the Momentum and Mass Value. In case of the flat top pile we can see colored in blue the location of the areas where the maximum residual is 1E-04, undoubtedly related with the rotation of the air flow behind this obstacle.

4.1 Cone

The first simulation was made for a cone pile according to the sketch shown in Figure 2. In order to simplify numerical calculations, a 3D model by Solidworks simulating just one half longitudinal side of the case was used and using symmetry process on CFX, the whole range of values was obtained.

Applied onto all models developed, the close-to-wall area (boundary layer) above ground and pile was meshed using prisms while the rest of the surrounding air was divided into tetrahedral cells. Modeling with hybrid tetrahedral grids with near surface prism layers result in smaller analysis models, better convergence of the solution and better analysis results. Mesh is shown in figure 3.

The model was meshed by approximate 2x105,000 cells and a revolution control surface 25 cm over the cone was defined in order to measure the ratio \( u_s / u_r \), named Variable 1, following the equivalent procedure than Stunder and Arya².

Although this model was simulated as a symmetric domain, subsequent simulations using meshes of 210,000 cells without symmetry simulating the same geometry showed similar results. Seems like convergence is not affected in this type of problems when using symmetry
conditions, an observation that will be used in future complex simulations where trying to save cell number is a must.

Figure 3: Mesh used in case of pile model

This mesh was imported to CFX which execute calculations according to boundary conditions and resolution factors referred above. In case of the cone 6 groups of calculations were done:
- Turbulence k-ε. Roughness 0. Logarithmic wind profile.
- Turbulence k-ε. Roughness present. Logarithmic wind profile.
- Turbulence SST. Roughness 0. Logarithmic wind profile.
- Turbulence k-ε. Roughness present. Plain wind profile.
- Turbulence k-ε. Roughness 0. Plain wind profile.
- Turbulence SST. Roughness 0. Plain wind profile.

All solutions converged adequately. The speed ratio defined in EPA is measured in a surface parallel to the cone surface and by means of contour area calculations we obtain the results shown in Figure 4. EPA values are shown in red color, being the simulation objective to reach.

Results are not perfect, but are quite close particularly in the medium and high speed ratio values, which is important as this high speed areas are the zones where dust emission will occur following EPA procedure. There can be seen the relevance of using a wind speed profile as an inlet boundary condition. The first three simulations using the logarithmic profile obtain better results than the last three simulations that just use a plain wind input, except in
the area “0.2a”, windward low speed area. Strangely SST turbulence model gets worst results than the k-ε one, even being a model much more elaborated that need 25% extra calculation time.

\[
RMSE = \sqrt{\frac{1}{N} \sum_{i=1}^{N} (y_i - \hat{y}_i)^2}
\]

where
N is the number of measurements
\( y_i \) is the obtained value
\( \hat{y}_i \) is the average value

Results are shown in Table 1. There can be seen how the best fit is obtained using the k-ε model, applying roughness and using a logarithmic wind profile. Table 2 shows the detailed results of this model compared to the reference included in EPA.

<table>
<thead>
<tr>
<th></th>
<th>k-ε Roughness logarithmic wind profile</th>
<th>k-ε NO roughness logarithmic wind profile</th>
<th>SST NO roughness logarithmic wind profile</th>
<th>k-ε Roughness plain wind profile</th>
<th>k-ε NO roughness plain wind profile</th>
<th>SST NO roughness plain wind profile</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>5,74%</td>
<td>6,70%</td>
<td>7,34%</td>
<td>12,82%</td>
<td>15,28%</td>
<td>16,33%</td>
</tr>
</tbody>
</table>

Table 1: RMSE values in different simulations (Cone)
The graphical result of this model is shown in figure 5 (wind coming from the left). Notice the existence of negative values, which indicates that certain areas have flow direction opposite to main stream upwind, which has been obviated in case of the numerical comparison referred above. Future simulations including dust movement will take advantage of this numerical modelling capability.

Therefore it could be considered that computational approach by CFX has the same level of accuracy than USEPA standard when determining values of friction velocities.

### 4.2 Flat top oval

The second model that EPA takes into consideration a flat top pile with different wind
directions blowing, at 0° (Pile B1), 20° (Pile B2) and 40° (Pile B3), as this model is not symmetrical as the cone was. The geometry as is modelled using Solidworks is shown in Figure 5.

![Figure 5. Flat top oval pile model](image)

On this model approximately 569,000 cells were required to obtain a smooth convergence with residuals on $10^{-5}$ level. Again the air volume next to the ground and the pile surface was modelled using prisms.

![Figure 6: Section of the mesh used in case of the flat top oval pile](image)

This mesh is given to CFX and calculated in several cases in the same way as the cone model was. Figure 7 shows results in case of wind blowing at 0° bearing, this is, case B1 of EPA. Calculations again show a better fit using the k-ε turbulence model using roughness in ground surfaces and a logarithmic wind profile. Again the use of a logarithmic wind improves
dramatically results over a flat profile

Nevertheless results are not as good as in the case of the cone pile. A strong area of high speed wind appears, EPA region 1.1, that is 0 in EPA values but ranges from 8,11% to 27% in case of simulations presented in this paper. In any case the method with the best fit $k-\varepsilon$ and logarithmic, is the lowest value.

![Figure 6: Calculation results in case of flat top oval pile](image)

Results of the RMS calculation for the different models are shown in Table 3. There we can see how the best fit is obtained using the $k-\varepsilon$ model, applying roughness and using a logarithmic wind profile. Table 4 shows the detailed results of this model compared to the reference included in EPA.

<table>
<thead>
<tr>
<th>Model</th>
<th>Area Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k-\varepsilon$ Roughness</td>
<td>5.77%</td>
</tr>
<tr>
<td>$k-\varepsilon$ NO roughness</td>
<td>6.84%</td>
</tr>
<tr>
<td>SST NO roughness logarithmic wind profile</td>
<td>7.98%</td>
</tr>
<tr>
<td>$k-\varepsilon$ Roughness</td>
<td>10.18%</td>
</tr>
<tr>
<td>$k-\varepsilon$ NO roughness</td>
<td>11.90%</td>
</tr>
<tr>
<td>SST NO roughness plain wind profile</td>
<td>12.87%</td>
</tr>
</tbody>
</table>

Table 3: RMSE values in different simulations (Flat top oval pile, 0° wind bearing)
Table 4: Comparison of subareas by USEPA and CFD - Flat top oval pile, 0° wind bearing

<table>
<thead>
<tr>
<th></th>
<th>EPA Pile A</th>
<th>CFX</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2a</td>
<td>5%</td>
<td>9.31%</td>
</tr>
<tr>
<td>0.2b</td>
<td>35%</td>
<td>0.00%</td>
</tr>
<tr>
<td>0.2c</td>
<td>0%</td>
<td>19.01%</td>
</tr>
<tr>
<td>0.6a</td>
<td>48%</td>
<td>30.14%</td>
</tr>
<tr>
<td>0.6b</td>
<td>0%</td>
<td>18.80%</td>
</tr>
<tr>
<td>0.9</td>
<td>12%</td>
<td>14.63%</td>
</tr>
<tr>
<td>1.1</td>
<td>0%</td>
<td>8.11%</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>100.00%</strong></td>
<td><strong>100.00%</strong></td>
</tr>
</tbody>
</table>

Figure 7. Wind blowing from left with bearing 0° (up left), 20° (up right) and 40° (down).
The same calculations can be done in case of wind blowing in bearings 20º and 40º against the pile. In the same way as above the best fit is in case of k-ε turbulence and logarithmic wind profile.

Figure 7 shows all speed ratio profiles obtained using the simulation. Tables 5 and 6 show results for the percentage of areas in case of this two new wind bearings. It is noticeable that now the high speed areas are correctly simulated by the calculations, as an opposite to the 0º bearing case.

<table>
<thead>
<tr>
<th>EPA Pile B2</th>
<th>CFX</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2a</td>
<td>3%</td>
</tr>
<tr>
<td>0.2b</td>
<td>28%</td>
</tr>
<tr>
<td>0.2c</td>
<td>0%</td>
</tr>
<tr>
<td>0.6a</td>
<td>29%</td>
</tr>
<tr>
<td>0.6b</td>
<td>22%</td>
</tr>
<tr>
<td>0.9</td>
<td>15%</td>
</tr>
<tr>
<td>1.1</td>
<td>3%</td>
</tr>
<tr>
<td>Total</td>
<td>100.00%</td>
</tr>
</tbody>
</table>

Table 5: Comparison of subareas by USEPA and CFD - Flat top oval pile, 20º wind bearing

<table>
<thead>
<tr>
<th>EPA Pile B3</th>
<th>CFX</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2a</td>
<td>3%</td>
</tr>
<tr>
<td>0.2b</td>
<td>25%</td>
</tr>
<tr>
<td>0.2c</td>
<td>0%</td>
</tr>
<tr>
<td>0.6a</td>
<td>28%</td>
</tr>
<tr>
<td>0.6b</td>
<td>26%</td>
</tr>
<tr>
<td>0.9</td>
<td>14%</td>
</tr>
<tr>
<td>1.1</td>
<td>4%</td>
</tr>
<tr>
<td>Total</td>
<td>100.00%</td>
</tr>
</tbody>
</table>

Table 6: Comparison of subareas by USEPA and CFD - Flat top oval pile, 40º wind bearing

The accuracy of the model is proven by the low values of RSME in all cases, which are 5.77% for 0º (see Table 3), 5.21% for 20º and 4.75% for 40º.

Once again it could be considered that computational approach by CFX has the same level of accuracy than USEPA standard when determining values of friction velocities.

5. PARTICLE INJECTION

Then the studies referred above guide us to use terrain roughness fixed at 0.5 cm and wind defined by a logarithmic profile.

From all available models, turbulence choice is the k-ε model. Calculation convergence is considered acceptable when RMS residuals get values under $10^{-5}$ (CFX-10 support documents\textsuperscript{13}). According to CFX-10.0 Solver Manual values of residuals at this level means very tight convergence, required for geometrically sensitive problems.
After the successful velocity field simulation, currently we are developing the study of the evolution of the particulated material inside this air flow, as well as the subsequent comparison with the experimental data obtained by means of optical particle samplers, already used by our research group in previous particle movement model validation experiments\textsuperscript{3,5}.

User FORTRAN routines are being developed to simulate the emission of particles according, among other factors, to flow values.

Figures 8 and 9 show outputs of test models where concrete particle injection is carried out with a high vertical injection, completely unrealistic, just for method validation purposes. The particles are injected in the surface of the pile at an amount that is adequately related to the speed of the wind at a certain distance from the pile. As can be seen each one of the trajectories, only a small number of them are shown, is colored depending on the particle mass flow rate, that is related to the color in the pile surface, which is defined by the air speed. E.g. red colors in the surface contours means high wind speeds (as can also be seen in figure 4 in case of cone and figure 7 in case of flat top pile), and trajectories starting from this area are also colored in red, which means high levels of particle emission.

Results shown here are obtained using CFX Particle Tracking method, a Lagrangian particle trajectory method. Eulerian-Eulerian multiphase methods have also been tried but up to now seem that particle tracking is the more promising choice.

![Figure 8: dust emission according to wind speed (cone)](image)
Another way to show this effect is to measure the concentration at a certain height over the pile and compare it with the air flow speed. Figure 10 shows this comparison in case of the flat top pile simulation. Again we can see how particle concentration is related with the speed profile over the pile. Please notice that wind is blowing from right to left.

Figure 10: Dust concentration (µg/m³, left) and air speed (m/s, right)

Another studies under development involves the influence of the shape of the pile in the dust emission. In order to compare other possible pile configurations there has been...
developed a semicircular pile, a shape widely extended in the stockyards as it can be easily created using rotating conveyor belts. Figure 11 shows analogue results as above.

Figure 11: Semicircular pile meshed (left), dust emission (right)

6. ACKNOWLEDGEMENTS


We also want to acknowledge the help and support from the Ansys CFX Technical Support Team in the development of these studies.

REFERENCES


