MOGASE: A GAUSSIAN MODEL FOR THE DIFFUSION OF ATMOSPHERIC POLLUTANTS

October 27th 2015

Model overview

MOGASE is a Gaussian plume model for the simulation of pollutants dispersion in the atmosphere. The pollutant surface concentration is computed by solving the following equation:

$$\chi(x, y, 0) = \frac{Q}{2\pi V\ \sigma_y \sigma_z} \exp\left(-\frac{1}{2}\left(\frac{y^2}{\sigma_y^2}\right)\right) \exp\left(-\frac{1}{2}\left(\frac{H^2}{\sigma_z^2}\right)\right) \qquad [gm^{-3}]$$

where

(x, y, 0) [m] are the coordinates of a point at surface in a reference system centered in the point emission source and oriented (X-axis) along the wind direction.

V [ms^{-1}] is the wind speed Q

[gs^{-1}] is the flow rate

σ_y, σ_z, [m] are the sigma of lateral and vertical dispersion of the plume

H [m] is the effective height of the plume

Dispersion coefficients

The computation of σ_y, σ_z and H is performed in MOGASE by choosing between different parameterization schemes. For the σ computation, BRIGGS OPEN COUNTRY (O) and BRIGGS URBAN (U) are available for rough terrains and urban terrains respectively. Tables 1 and 2 report the formulas used as a function of class of stability (this one selected by the user through the GUI during the input setup phase).

Stability class	Σ_y	Σ_z
A	$0.22x(1 + 0.0001\ \ )^{-1/2}$	$0.20x$
B	$0.16x(1+0.0001x)^{-1/2}$	$0.12x$
C	$0.11x(1+0.0001x)^{-1/2}$	$0.08x(1+0.0002x)^{-1/2}$
D	$0.08x(1+0.0001x)^{-1/2}$	$0.06x(1+0.0015x)^{-1/2}$
E	$0.06x(1+0.0001x)^{-1/2}$	$0.03x(1+0.0003x)^{-1}$
F	$0.04x(1+0.0001x)^{-1/2}$	$0.016x(1+0.0003x)^{-1}$

Table 1: Briggs Open Country

Stability class	σ_y	Σ_z
A-B	$0.32x(1+0.0004x)^{-1/2}$	$0.024x(1+0.001x)^{-1/2}$
C	$0.22x(1+0.0004x)^{-1/2}$	$0.020x$
D	$0.16x(1+0.0004x)^{-1/2}$	$0.014x(1+0.0003x)^{-1/2}$
E-F	$0.11x(1+0.0004x)^{-1/2}$	$0.08x(1+0.0015x)^{-1/2}$

Table 2: Briggs Urban

Effective height

Before introducing the concept of effective height H, two parameters need to be mentioned: the buoyancy coefficient F and the stability parameter s:

$$F = \frac{T_g - T_a}{T_g} gwr^2$$

$$s = \frac{g}{T_a} \left(\frac{\delta T}{\delta z} + 0.0098\right)$$

where

T_g [K] is the plume temperature

T_a [K] is the air temperature

g [ms^{-2}] is the acceleration of gravity

ω [ms^{-1}] is the plume wind speed

r [m] is the stack radius

$\delta T/\delta z$ [Km^{-1}] is the vertical temperature gradient

The effective height H is defined as the sum of the geometric height of the source h_s and the plume rise Δh:

$H = h_s + \Delta h$

Two different behaviors for Δh can be considered:

1) A transition phase, close to the emission source, where Δh varies
2) A leveling phase where Δh is constant

The step from the transition phase to the leveling one happens at distance x_f from the point source, and it is computed by using the parameterizations reported in Table 3.

$s \leq 0, u \geq 1$	$x_f = 6.48 F^{2/5} h^{3/5}$
$s > 0, u > 1$	$x_f = 2us^{-1/2}$

Table 3: Step from the transition to the leveling phase.

For the computation of the plume rise Δh, MOGASE makes use of the Briggs formulas for main power plants (GROSSI IMPIANTI “G”) e minor emissions (PICCOLE EMISSIONI “P”).

Main power plants (G)

In this case, Δh assumes the value reported in Table 4.

	Transition phase	Leveling phase
$s \leq 0, u \geq 1$	$\Delta h = 1.6F^{1/3}x^{2/3}u^{-1}$	$\Delta h = 1.6F \quad x_f^{2/3}u^{-1}$
$s > 0, u \geq 1$		$\Delta h = 2.6\,[F/(u\,s)]^{1/3}$
$s > 0, u < 1$	$\Delta h = 1.6F^{1/3}x^{2/3}u^{-1}$	$\Delta h = 5.1\,F^{1/4}\,S^{-3/8}$

Table 4: Main power plants

where

x [m] is the downwind distance

T_g [K] is the plume temperature

T_a [K] is the air temperature

g [ms^{-2}] is the acceleration of gravity

ω [ms^{-1}] is the plume wind speed

u [ms^{-1}] is the wind speed

r [m] is the stack radius

$\delta T/\delta z$ [Km^{-1}] is the vertical temperature gradient

Minor emissions

In this case, Δh is computed as:

$$\Delta h = 6r(\frac{w}{u} - 1)$$

where

w [ms^{-1}] is the plume wind

speed u [ms^{-1}] is the wind speed

r [m] is the stack radius

Input-Output

The model requires the following input dataset:

1. Information about the domain size and resolution
2. Meteorological input: stability class, wind speed and direction, average values of air temperature
3. Information about the emission sources:
    - ✓ Location and geometric features of the stacks
    - ✓ Speed and temperature of the plume, flow rate
    - ✓ Size distribution and chemical speciation of particulate matter (if emitted)
    - ✓ Features of the plume rise and vertical dispersion

The model output is provided both in graphic and text format.

Use of the model MOGASE

The MOGASE software works in Matlab. Once Matlab has been opened, the current directory has to be changed by selecting the folder where the software is located. For launching the code, the command "mogase" needs to be digited in the Matlab command window, in order to open the GUI interface.

The interface is logically divided in 4 parts, related to the

1. input of meteorological conditions
2. input of stacks features
3. input of city features (for cities located inside the domain)
4. buttons for the simulation run

Regarding the meteorological input dataset, the following variables must be specified:

- ✓ wind speed [ms^{-1}]
- ✓ wind direction (taking into account that 0 represents a wind flowing towards South, 90 towards West, 180 towards North and 270 towards East)
- ✓ Air temperature [K]
- ✓ Stability class (from A (strongly unstable) to F (strongly stable))
- ✓ Land use (O=OPEN COUNTRY for a rural domain, U=URBAN for a urban area)

For the input of stack features, the software can take into account up to 3 stack emission sources simultaneously. The main information required can be summarized as follow:

- ✓ Point source coordinates (comma separated). The domain is a 20x20 km^2 resolution grid with coordinates (0, 0) in the low left corner.
- ✓ Stack height [m]
- ✓ Stack radius [m]
- ✓ Power plant dimension (the emission type can be jet (P) or buoyant (G). This choice has influence on the computation of the effective height)
- ✓ Plume wind speed [ms^{-1}]
- ✓ Flow rate [gs^{-1}]

Regarding the input of cities features, the coordinates of each city need to be specified.
The buttons of the GUI interface are used for:

- ✓ Running the simulation and creating 2D maps of surface concentration.

- ✓ Clearing all fields

- ✓ Interacting with the maps by using the button “conc puntuali”. A point selection on the map provides coordinates and concentration value of that point on the right side of the interface.

- ✓ Running the simulation and creating concentration vertical profiles