

Knowledgebase

Using Pollutant and Water Spray Sources in ENVI-met

Version 11/2021 Updated for V5

INTRODUCTION

The simulation the dispersion of passive and active pollutants is one of the most complex tasks when using ENVI-met. The calculation of the transport and deposition of pollutants requires everything a full microclimate simulation needs to be run successfully plus additional information concerning the pollutant emissions.

Moreover, the simulation of Water Spray features also requires a special setup of pollutant sources we will discuss in this document.

This Knowledgebase should help you to find your way in this process.

STEP 1: CREATE AN EMISSION SOURCE



DB Manager

To include pollutant dispersion into your simulation there are several steps which have to be followed. First of all, the Source has to be created and defined in the *DatabaseManager*.

1. Open the DBManager and create a new user-defined pollutant source ①.
Remember, in your project you might use the global User Database or you can decide to use an individual project database (see the base tutorials on our website).
2. Give the Source a unique ID ②
3. Set the Source Geometry ③.
ENVI-met distinguishes between point, line or area sources. Which type you select depends on the nature of your objects. For example, chimneys are typical point sources while street lanes are line sources. Area sources are more seldom but can be imagined e.g. as type of landcover that emits pollutants.

Water Spray: Water Spray sources can be of two types:

Fountain type (ID= 1): Emitting water droplets from level z=0 to the source height

Spray type (ID=2): Emitting water droplets only in the source height.

You must use the **Special ID field (A)** to define as which type of water spray your source should be interpreted.

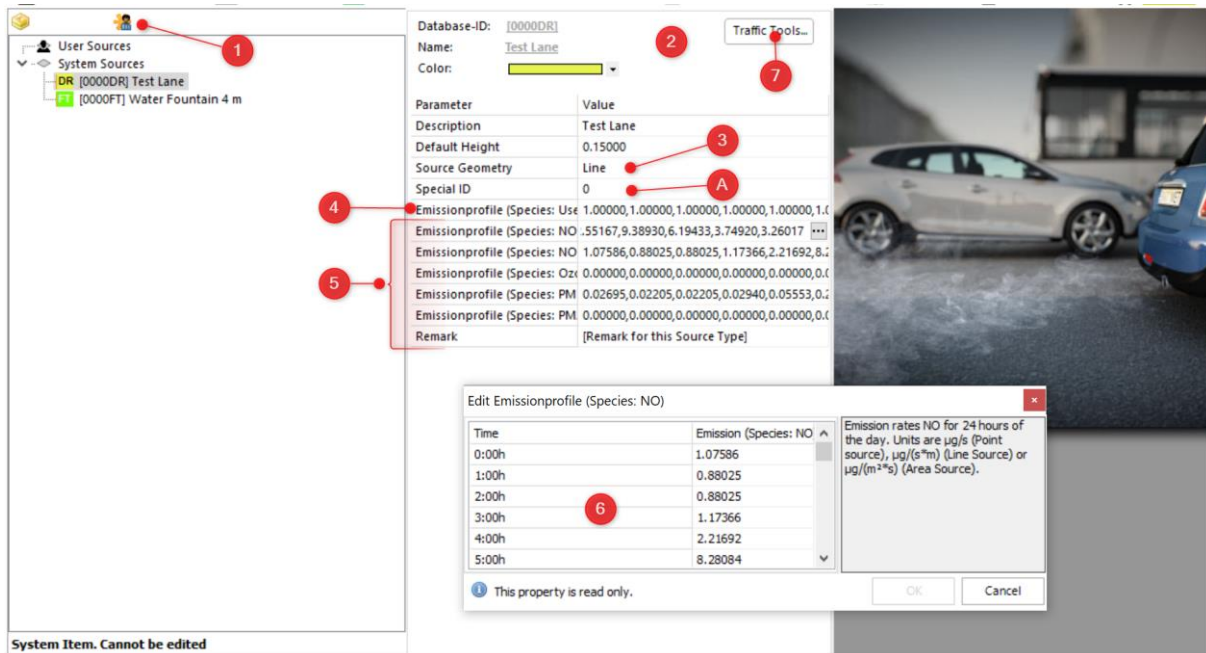
4. Define emission the rates for the selected species (④, ⑤):
Important: If you decide to use the **Single Pollutant Mode** in your simulation, you need to insert the hourly emission data in the **Species: User** database.
Even if your species is one of the defined species in the rows below ⑤, e.g. PM10 you still need to enter your data in the Species: User row.
The physical or chemical properties of the User pollutant will be set in the .SIMX configuration file.
The rows for NO, NO2, Ozone, PM10 and PM2.5 ⑤ are only used in Multi Pollutant Mode.

Click on the edit button to open the emission editor ⑥

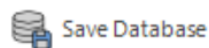
If you work with traffic emissions, you might also want to use the Traffic Toolbox accessible through the "Traffic Tools" button ⑦.

- If you want to use **Multi Pollutant Mode** you can enter the hourly emission data for the predefined 5 pollutants in the assigned rows (5).
You still can use a 6th User-Defined pollutant in the simulation.

Note, that Multi Pollutant Mode is only available in the full versions of ENVI-met.



Save your edits in the used database (User Database or Project Database).



STEP2: LOCATING YOUR SOURCE IN SPACES

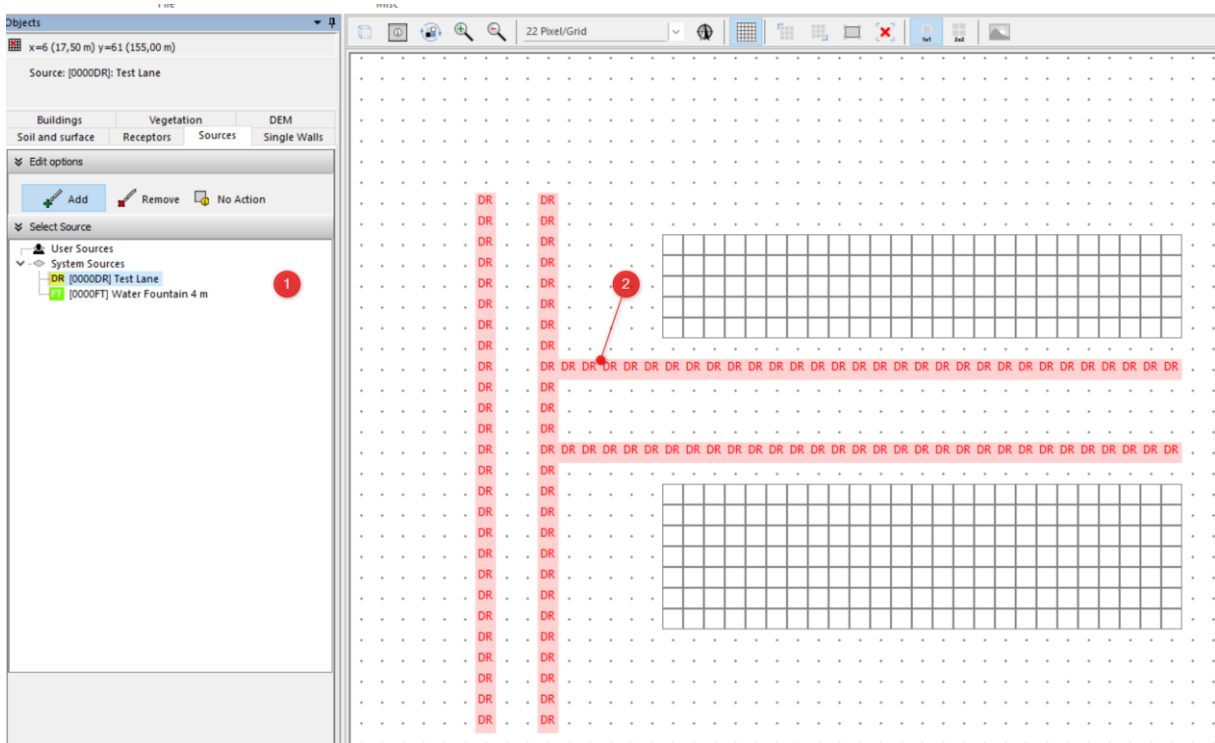


After adding one or more pollutant sources to your system, they are available for selection in SPACES (1).

Make sure that you have selected the right project in SPACES in order to have the right sources available. If you have changed or added new sources, just re-select the project in SPACES to reload your recent data.

The pollutants are located under the topic Sources and the chosen database (1).

Now you can select your source and digitize it at those grid cells, where the pollutants shall be emitted (2) by Selecting "Add" and mark the grids.



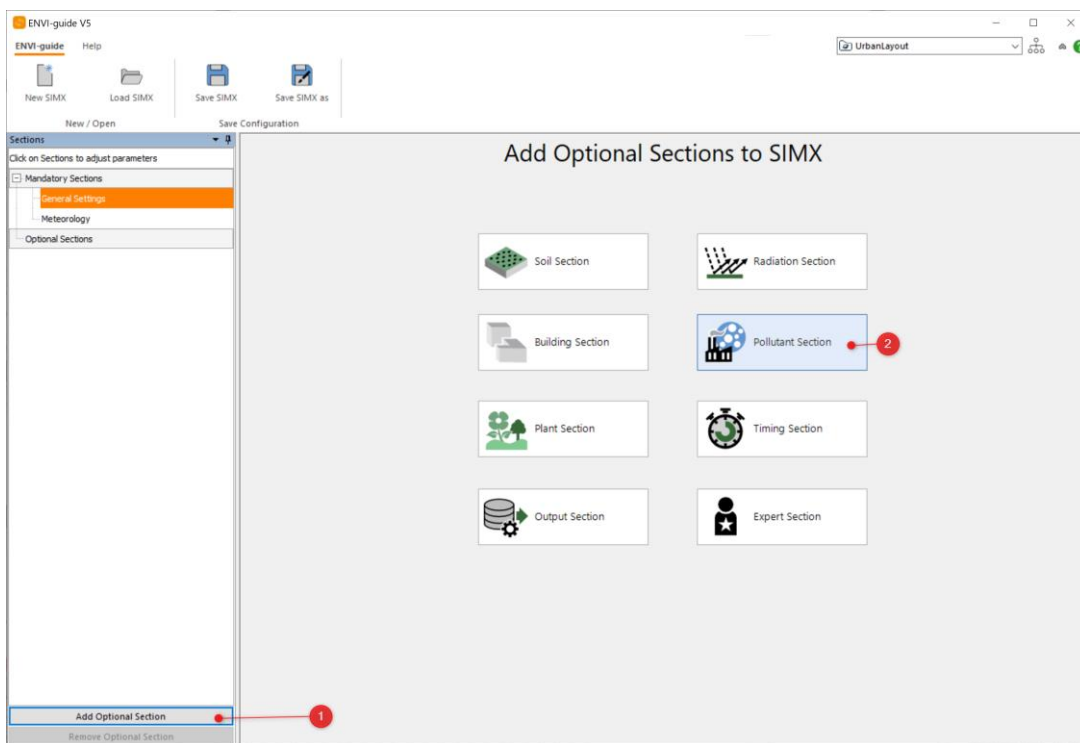
STEP 3: CREATE THE SIMULATION FILE (.SIMX)



When creating the SIMX-File with the ENVI-Guide, follow the usual steps to create the basic data for your simulation (refer to our homepage for various video tutorials on this).

ENVI-guide

Then, select “Add Optional Section” ① and select “Pollutant Section” ②



Pollutants / Sources Settings

General Pollutant Simulation Settings

The pollutant emission rates are taken from the emission profiles defined in the Database Manager. In Single pollutant mode, only the user-defined pollutant is used. In MultiPollutant Mode, all other emission profiles such as NO, O3 etc. are used as well.

Pollutant Operation Mode

Single Pollutant 1 Multi Pollutant

Using the additional Active Chemistry module requires Multi Pollutants mode. Active chemistry supports reaction terms of NO, O3 and NO2.

Active Chemistry (NO-O3-NO2)

Dispersion only Dispersion and Active Chemistry 2

(Active Chemistry is not supported in LITE)

The update interval for emission rates can be adjusted in the Timing Section.

Background Concentration

Set the background concentration of different pollutants if needed. Similar to the CO2 background concentration in the atmosphere (~400 ppm) that can be changed in the Plants Section, there might be a specific pollutant background concentration for your simulated area. Default values are set to 0 in order to assume a clean atmosphere for the simulation start.

NO background level (µg/m3): 0.00 6

NO2 background level (µg/m3): 0.00

Ozone background level (µg/m3): 0.00

PM 10 background level (µg/m3): 0.00

PM 2.5 background level (µg/m3): 0.00

User specified pollutant (µg/m3): 0.00

User Pollutant / User Emission Profile Settings

Name of User Pollutant Source: My Pollutant 3

Enter a name to identify the pollutant in the output files.
You need to specify a user-defined pollutant in Single pollutant mode!

Chemical species or type of pollutant: SPRAY (Water Spray) 4

Additional settings only needed for Particles (PM or Water Spray)

Particle diameter (µm): 10.00 5

Particle density (g/cm³): 1.00

1. Choose the operation mode for the simulation: **Single Pollutant** or **Multi Pollutants Mode** ①
Also, define if you want to use **Active Chemistry** or not ②.

Obviously, active chemistry will only be available if you have *Multiple Pollutants* selected. If Chemistry is activated, the released pollutants can react with each other and create new products. At the moment, this is limited to the NO-NO₂-Ozone reaction cycle including vegetation emitted Isoprene if defined in the plant database.

2. Define the User-defined pollutant.
If you use **Single Pollutant Mode** the **only pollutant present in your simulation** will be the User-Defined pollutant. It is therefore essential that you specify its properties here (③, ④, ⑤) and provide hourly emission data in the Database Manger.

If you use **Multi Pollutant Mode**, you can still specify your own User Pollutant as an additional 6th pollutant adding to the 5 fixed pollutants in ENVI-met.

Do not specify a pollutant that is already included in the 5 pre-defined pollutants (NO, NO₂, PM10, PM2.5 and Ozone) because ENVI-met will treat this pollutant independently of the others and will not understand that e.g. PM10 is defined in the fixed pollutants AND in the user defined pollutants.

3. Give your user defined pollutant a name ③. You can select a name of your choice, it has no influence on the simulation process.
4. Define the chemical species or type of pollutant to be emitted as User Source ④.
If a source of the **Particulate Matter** type or **Water Spray** is selected as type of pollutant, you also need to give the particle diameter and density ⑤.
For **Particles**, the density normally should be set to 1 g/cm³ unless specific other data are available.
For **Water Spray**, the density settings are ignored, and the density of water is always set to 998.2 kg/m³.
5. Finally, save the .SIMX file as usual when leaving the ENVI-Guide..

STEP 4: RUN THE SIMULATION



The simulation will be executed as usual. Depending on your sources settings, the simulation time will increase because additional calculations with often small time steps are required.

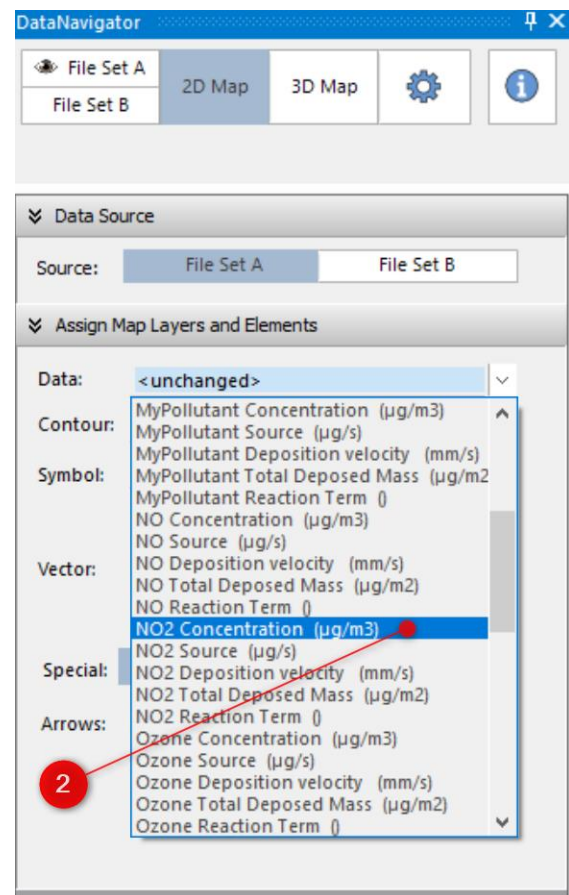
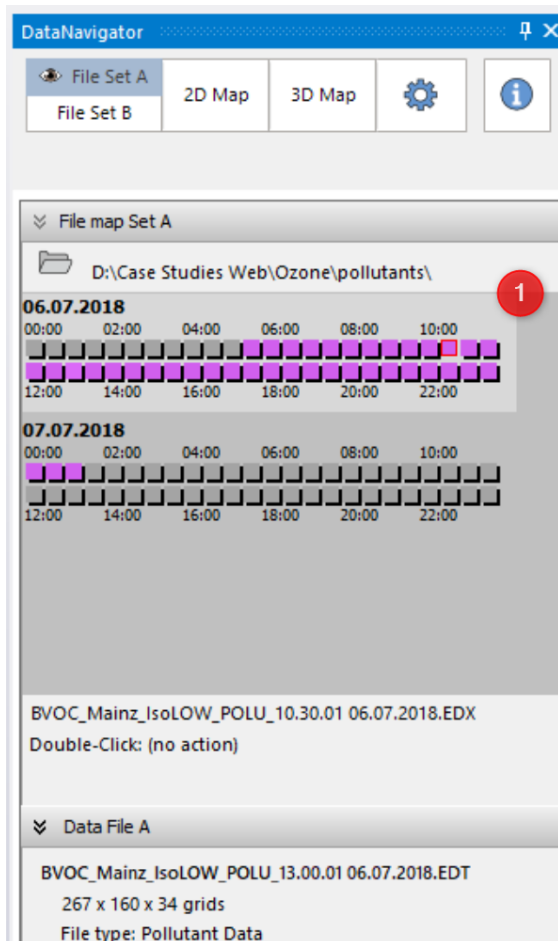
STEP 5: VISUALIZE THE RESULTS



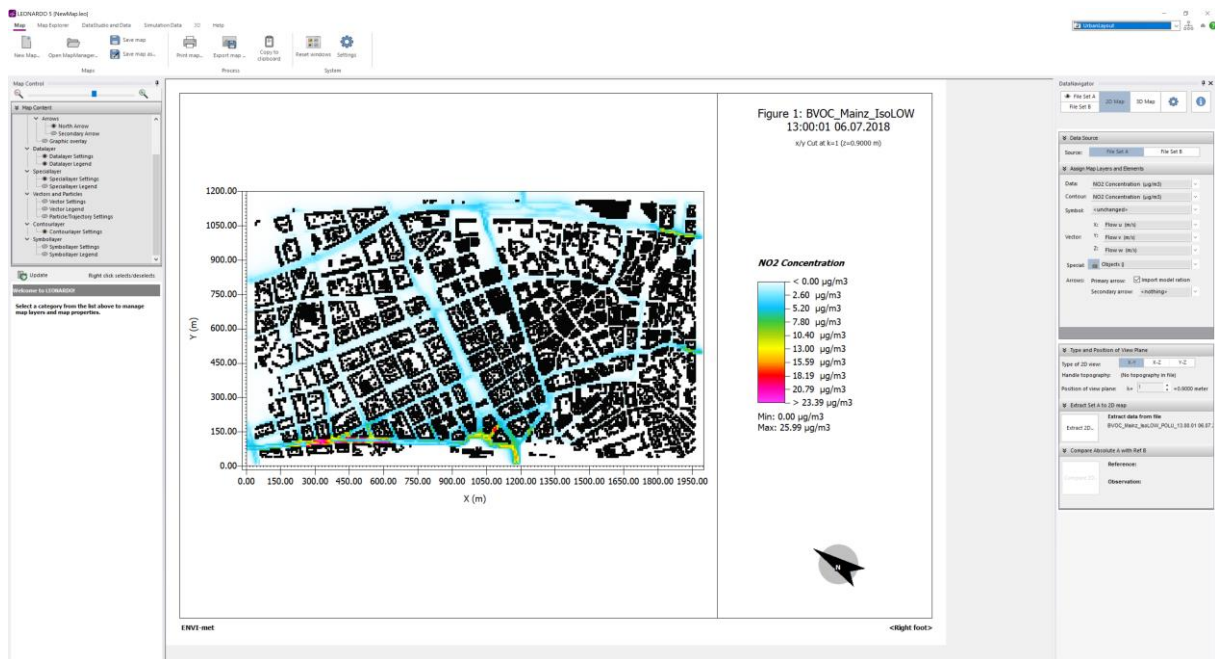
The pollutant specific data are all stored in the **Pollutants** folder of your simulation output folder.

In *Leonardo*, use the DataNavigator to select a file from the pollutant output folder ①.

Then, select the information you want to visualize ②. Depending whether you have run a Single Pollutant mode simulation or a Multiple Pollutants mode simulation, you have different data for one or 6 different species in your file. Here we use the example from our website (see last section) and selected the NO₂ concentration to be shown.



Then follow the usual steps for preparing a LEONARDO 2D or 3D map



SAMPLE DATA

On our ENVI-met technical model page you can find a pre-defined case study for using air pollutants including input & output files: <https://envi-met.info/doku.php?id=examples:airpollution>