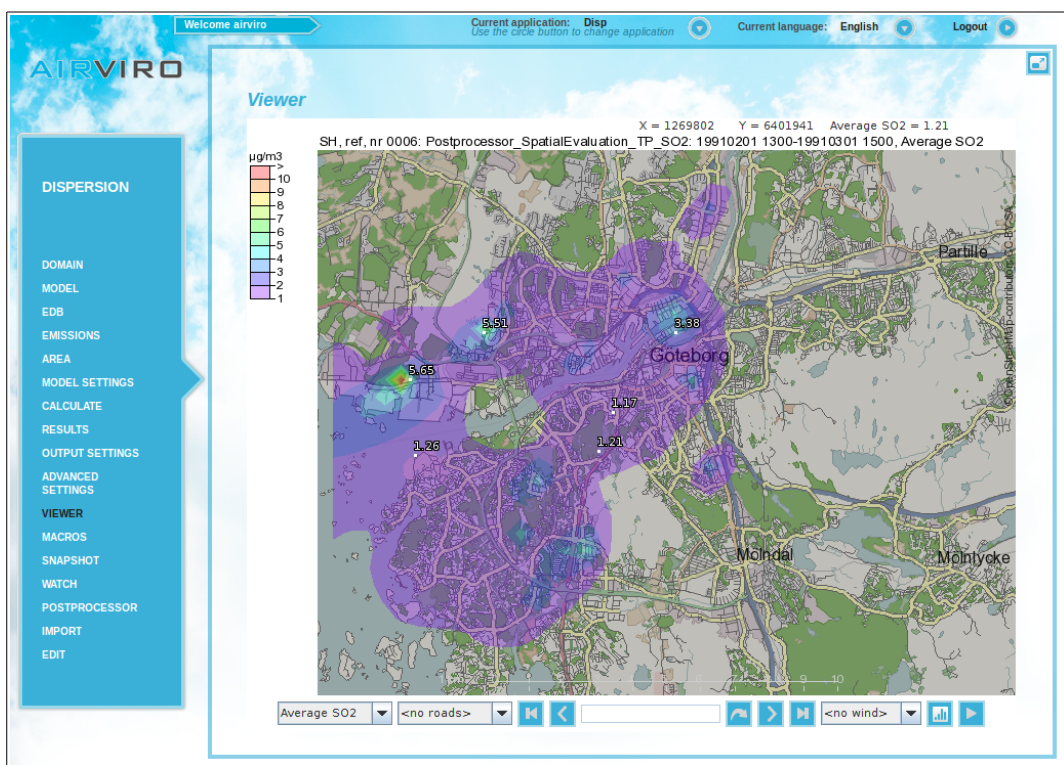


Airviro User's Reference

Working with the Dispersion Module



How to simulate pollutant dispersion

How to simulate pollutant dispersion

Amendments

Version	Date changed	Cause of change	Signature
3.11	Ago2007	Upgrade	GS
3.12	January2009	Upgrade	GS
3.13	January2009	Upgrade	GS
3.20	May 2010	Upgrade	GS
3.21	Dec 2010	Upgrade	GS
3.21	June 2012	Review	GS
3.22	April 2013	Release	GS
3.23	Jan 2014	Upgrade	GS
3.23	January 2014	Review	GS
3.23	June 2015	Review	GS
4.00	Sept 2015	Upgrade	GS
4.00	Aug 2018	Review	GS
5.00	Oct 2019	Review	GS
5.00	Feb 2021	Review	DC

Contents

2.1 Introduction.....	7
2.1.1 Why You Need to Use Dispersion Models.....	7
2.1.1.1 What's the Use of Dispersion Simulations.....	7
2.1.1.2 How Can Airviro Help?.....	7
2.1.2 Model Assumptions.....	8
2.1.3 Brief description of the available models.....	9
2.1.4 How does Dispersion Module work?.....	17
2.1.4.1 Getting started.....	17
2.1.5 Guidance for the beginner:.....	19
2.1.6 Overview of the Dispersion Module Main Window.....	20
2.1.6.1 Changing Weather Conditions – Model settings.....	20
2.1.6.2 Specifying Emissions.....	21
2.1.6.3 Submitting a Dispersion for Calculation.....	22
2.1.6.4 Viewing Existing Dispersion Calculations.....	22
2.1.6.5 Finding Local Values.....	22
2.1.6.6 Using Receptor Points.....	22
2.1.6.7 Map Functions.....	23
2.1.6.8 Menu options in the main Dispersion window.....	23
2.1.6.9. What is the resolution of the simulation grid?.....	28
2.2 The Gauss Model.....	33
2.2.1 Examples Based on the Case Weather Condition.....	33
2.2.2 Ordering a Calculation.....	35
2.2.3 Examples Based on Emission for a Specific Hour.....	41
2.2.4 Using Scenario Weather Conditions.....	44
2.2.5 Using the Advanced Result Options.....	48

2.3 The Street Canyon Model.....	52
2.3.1 Street Canyon Overview.....	52
2.4 The Grid Model.....	56
2.4.1 Looking at Dispersions for a Specific Time Period.....	56
2.4.2 Looking at Seasonal Dispersions.....	59
2.5 The Heavy Gas Model.....	61
2.5.1 Simulation of a Continuous Liquid Leak.....	62
2.5.2 Simulation of a Tank Burst.....	63
2.6 Introduction to the Receptor Model.....	65
2.6.1 Where is this pollution coming from?.....	65
2.6.2 To ensure a proper use of the Receptor module:.....	66
2.6.4.1 Source Location.....	70
2.6.4.2 Emission Estimation.....	73
2.7 Aermod Model.....	77
2.8OSPM Model.....	81
Appendix 2A: The Wind Model - Calculation of the Wind Fields.....	85
2A.1 Principles of Wind Field Calculations.....	85
2A.1.1 Wind Model Equations.....	86
2A.1.2 Initialization.....	86
2A.1.3 Stability and Turbulence Estimation - Preprocessing of Meteorological Data.....	87
2A.1.4 Mesoscale Interpolation.....	91
2A.1.5 Procedures for Solving the Wind Model Equations.....	93
2A.1.6 Numerical Methods.....	93
2A.1.7 Reference Literature.....	94
Appendix 2B: Results from Verification Studies.....	95
Appendix 2C: The Gauss Dispersion Model.....	98

2C.1 Introduction.....	98
2C.2 When to Use the Gauss Plume Model.....	98
2C.2.1 Source Type Definitions.....	100
2C.3 Plume Rise Formulation.....	101
2C.4 Building Downwash.....	106
2C.5 The Gaussian Plume Model.....	108
Appendix 2D: The Eulerian Advection-Diffusion Grid Model.....	110
2D.1 Introduction.....	110
2D.2 Formulation of the Grid Model.....	110
2D.2.1 Numerical Methods.....	111
2D.2.2 The Numerical Grid, Coordinate Transformations.....	113
2D.2.3 The Turbulent Exchange Coefficient.....	113
2D.2.4 The Three-Dimensional Wind Field.....	117
2D.2.5 Source Initialization.....	117
2D.3 Particle Deposition.....	118
2D.3.1 Dry Deposition of Gases and Non-Gravitating Particles.....	118
2D.3.2 Wet Deposition.....	120
2D.3.3 Particle Settling.....	121
Appendix 2E: Street Canyon Model.....	122
2E.1 Model Description.....	122
Appendix 2F: The Heavy Gas Model.....	125
2F.1 Continuous Release or a Leakage From a " Small " Hole.....	126
2F.2 Instantaneous Release or a Tank Rupture - Emission Only from the Liquid Phase.....	129
2F.3 Substances and Coefficients.....	130
2F.3.1 Comments.....	130
Appendix 2G: Scenario Calculations.....	131

2G.1 General Principles.....	131
2G.3 General Applications Using the Scenario Feature.....	133
Appendix 2H: A Method to Locate Dominating Sources.....	134
Appendix 2I: A Method to Estimate Emissions from Dominating Sources.....	137
2.I.1 Monitor Option for a Time Varying Background Concentration.....	139
Appendix 2J: The OSPM Model.....	140
Appendix 2K: Inhomogeneous Grid.....	146
Appendix 2L: References.....	156
Appendix 2M: Noise.....	161

2.1 Introduction

2.1.1 Why You Need to Use Dispersion Models

2.1.1.1 What's the Use of Dispersion Simulations

Given a situation at a specific place, the results will be more easily understood by non-experts if they are well presented. The weaknesses of measured data are mainly that:

- They only represent individual locations (even if the results from various measurement stations sometimes may be generalised to larger areas)
- They describe present and historical conditions, but say nothing about future air quality.

2.1.1.2 How Can Airviro Help?

Taking measurements continuously, everywhere would certainly make it possible to gain full knowledge of the ambient air quality (except for future conditions). But even taking small steps in this direction would lead to exorbitant costs. Simulation models offer a more economical way to describe air quality conditions over large areas, as well as forecasting future conditions. Moreover, the important aspect of planning can be addressed: "What happens if...".

Airviro is a powerful tool for data analysis and dispersion modelling. To study air quality conditions, the use of measured and simulated contamination levels using Airviro Indico Presentation and a set of dispersion models would be highly beneficial. Simulations give an overall description, while data should be used to verify the validity of the model results. Although dispersion models can be tested in many ways, we recommend verifying their results at a few key spots, for each application area. Validating simulation results from measurements gives confidence in forecast simulations.

The Airviro Receptor model illustrates the advantage of using an integrated system, where measured and simulated values are statistically evaluated over a given period of time. Simulation models are of great value for interpreting measured data.

2.1.2 Model Assumptions

The following sections describe the Airviro dispersion models. But first we must mention the main assumptions that motivate the choice of a particular method. To describe current or future pollution levels within an urban area, the following is necessary:

- Emission data must be close to reality, both in magnitude and in time/space variation. Sophisticated dispersion models will give poor results if emissions are erroneously specified. This motivates the complexity and versatility of the Airviro emission database (see Users Reference Volume 3: Working with Emission Database (EDB)).
- Realistic descriptions of the local wind fields and other meteorological conditions are of the utmost importance. Meteorological parameters used in simulations should refer to locally determined conditions as much as possible, preferably through profile measurements in masts. The wind model must be able to realistically cope with calm situations with weak vertical mixing (stable stratification), i.e. when critical pollution levels are likely to occur.
- For small scale urban applications (that is, a few tenths of a kilometre), the use of the Gauss plume model has many advantages: it is simple and fast to run, it is well tested and the empirical coefficients have been documented, compared and discussed in books and papers for many years.
- For other large-scale applications, such as simulations over a large city or a regional area, simulations in a complex topography, or event simulations where pollutants are accumulated over a period of time, the advection/diffusion grid model should be used instead of the Gauss model. The grid model also provides an easy way to simulate deposition and sedimentation of particles.
- Model results should be presented in a form that can be directly compared with national standards. This means that the simulations should cover not only specific weather conditions (present, historical or forecast), but also longer periods (e.g. seasons or years). The seasonal calculations include mean as well as extreme concentrations.
- If you don't know where pollution is coming from, then you can use Receptor Model /

Source Location. The Receptor Module will calculate a probability field that will pinpoint any dominating sources, based on the concentrations of monitored time series.

If you know where pollution is coming from, then you can use Receptor Model / Emission Estimation. The Receptor Module will calculate the estimated emission for one or various sources, based on the concentrations of monitored time series. However, the quality of the input data for estimating emissions must be even higher than that required for locating the dominant sources.

2.1.3 Brief description of the available models

- Airviro Gauss.

This model is used to simulate the distribution of ground concentrations of pollutants over urban or industrial areas, with a typical scale of one or a few tenths of a kilometre. The minimum application area is given by the fact that the Gauss model coefficients are not valid near to the source (distances less than approximately 25 metres). The maximum application area is given by the fact that more or less stationary conditions should prevail during the time it takes for an air parcel to be advected through the area. One hour mean values are simulated, as it is known that the wind can be more or less constant during such a period (daily averages would not be sufficient). For wind speeds of 2 - 5 m/s, an air parcel would travel 7 - 18 km within one hour. This limitation of the Gauss model should be taken into account when performing simulations at scales larger than 20 km.

To avoid using this model over extended areas, producing unrealistically long plumes, the plume length in the Airviro Gauss model is limited. The maximum plume length is defined in a resource file and it depends on the actual stability and persistence of different weather conditions.

The Gauss model does not resolve individual buildings. Instead, surface structures such as houses and trees are entered into the model through local roughness values and, consequently, through the wind field. Buildings create a rougher surface; more friction means a lower wind speed, which in turns, influences the dispersion, and so on. (1)

The default height for calculations is 2 m, but any other value can be specified. The height at which the simulated concentration values are valid will match the calculation height above ground in open areas (i.e. in the countryside). Over a city, the simulation will reflect concentrations at the specified height above the roof height. Therefore, the default setting will provide concentrations at roof height, not at street level. The blocking effects of buildings surrounding a street, resulting in higher concentration levels, can be studied separately. (2)

Note that the Gaussian dispersion model simulates steady states of pollution concentrations. Accumulation of polluted air over a period of several days cannot be studied with this model.

Those mentioned above are the restrictions of the Lagrangean/Gaussian model, but there are additional advantages to be gained from its use. Technological articles (e.g. Hanna et al, 1982) mention, among other things, the following points:

- It produces results that match experimental data just as well as any other model.
- It is fairly easy to perform mathematical operations with this equation.
- It is conceptually appealing.
- It is consistent with the random nature of turbulence.

Usually, Gaussian plume models are applied to horizontally homogeneous wind fields, while Airviro takes into account a realistic wind field. This means that the effect of topography, roughness distribution and horizontal variations in surface heating/cooling will be used in the dispersion calculation.

- US/EPA AERMOD.

Use AERMOD when you consider a steady-state plume model that incorporates air dispersion based on planetary boundary layer turbulence structure and scaling concepts, including treatment of both surface and elevated sources, and both simple and complex terrain.

- Airviro Grid.

The Eulerian Advection-Diffusion grid dispersion model included in Airviro is specially designed to meet the requirements given by areas with complex topography, e.g. valleys. This model has some additional advantages when compared to the standard Gaussian dispersion approach:

- It allows transient simulations and can simulate accumulation of pollutants during calm wind events.
- Boundary conditions (such as deposition) as well as particle characteristics (such as settling) may be introduced.

With regard to individual buildings, surface roughness and height for calculations, the considerations are the same as those already explained for Airviro Gauss. (SH), see paragraphs (1) & (2)

The Grid model should be used on larger areas. It is not suitable for small scale calculations and therefore a minimum length of 25 m has been set for the sides of the grid boxes used for calculations.

- Airviro Street Canyon.

The Street Canyon model is a small-scale model that simulates concentrations at street level on a single road flanked by buildings on each side. The simulated concentrations are visualised on a vertical plane perpendicular to the direction of the street, creating the illusion of looking down the canyon.

This model simulates the dispersion of pollutants generated by traffic on a particular road. Be aware that traffic on other roads may very likely contribute to a substantial part of the pollution levels, especially if the traffic in the simulated road has a low intensity and larger streets around it. Is it possible to separately estimate the percentages of street pollution generated by local traffic and those generated by emissions elsewhere?

A feasible way to do this using Airviro is:

- Run the street canyon model with traffic emissions only for the road segment under

study

- Run a citywide simulation using Gauss or Grid model, including all sources except those generated by traffic on the road segment under study. The concentration at roof level above road segment under study can then be considered as a result of non-local sources. Due to the above mentioned eddy circulation , the same contribution from non-local sources down in the street canyon itself can be expected. This concentration level can be considered as a background value to be added to the street canyon model results.
- Airviro Heavy Gas. (HG)

The heavy Gas model describe the emission and dispersion of gases heavier or denser than air.

Dense gases are gases that are heavier than the ambient air because:

- They have a molecular weight higher than air
- They are stored as condensed gases at low temperatures
- They are stored as condensed gases at high pressures
- A combination of the above

These gases are usually toxic, in addition, a condensed gas is very cold when it is released into the atmosphere. Therefore, it may cause frostbite to people who come into contact with a cloud of released gas. The gas cloud can also have suffocating effects even if it is non-toxic or very cold.

This model should be applied to industrial and transportation environments, where storage and transportation of this type of gasses; like chlorine, ammonia and propane, takes place. Accidents, or scenarios, that can be currently simulated by this model, are:

- The rupture of a storage container, causing a large hole in a pressurised container. It is assumed that the emission generated by this type of accident is due to a rather violent boiling off of the condensed liquid. The release time is assumed to be short. This scenario is therefore classified as an instantaneous

liquid phase release. A dense puff is eventually formed and carried away by the wind while collapsing gravitationally. After a while, the puff is diluted and it behaves like a neutral density puff.

- Leakage from a small hole, which could be a leak from a valve or a small broken pipe. The release time is assumed to be quite long. The release can be originated from both the liquid and the gas phase of the pressurised storage vessel. This scenario is called a continuous liquid or gas phase release. A heavy gas continuous plume or slab is formed. The slab stretches out in the wind direction and gets wider due to the gravitational spread perpendicular to the main slab direction. When the plume density match the density of the ambient air, the plume behaves like a neutral Gaussian plume.

It is clear that a simulation includes the application of several models. Emission models, models for initial cloud forming, gravitational phase models and models for the neutral plume phase.

- Airviro Receptor

The receptor model actually consists of two different models:

- The source location model that is used to find the most likely position for a dominant source.
- The emission estimation module that is used to estimate the emission from dominating sources.

The Source Location Model

The method assumes time series of:

- Hourly meteorological data: wind speed and direction at one or two levels, temperature difference in the surface layer and standard deviations of wind direction and vertical wind speed
- Hourly concentration values of a pollutant from one or - preferably - several monitor stations (point or open path measurements)

- Plume height: Specified by user.
- Trajectory: Since a homogeneous wind field is used, the plume axis will be a straight line.
- Vertical wind profile: The free wind speed (V_g) is estimated by the extrapolation at 25m with the help of the power law. The wind direction is twisted according to empirical data (Holtslag).

Concentrations are evaluated where monitoring stations are located. If an open path instrument is used, the simulated concentration level is calculated at 10 points along the path. The mean value is found after averaging the concentration values.

Time series data are given for $j=1, \dots, m$ hourly values, and the measured concentrations are given at $k=1, \dots, l$ number of monitoring stations. The simulated concentration at the monitoring station k is written as a result of the i th source and the j th hour of meteorological data (S_{ijk}). For each fictitious source i , the correlation between simulated and measured time series is calculated as, $corri = corr(S_i, C)$, where each of the two series - the simulated (S_i) and the measured (C) - include data from $j=1, \dots, m$ hours and $k=1, \dots, l$ stations.

The calculations are repeated for all fictitious sources. The horizontal distribution of the correlation coefficients is displayed as isolines multiplied by 100.

The Emission Estimation Model

The method assumes the same kind of data as for the localization of sources:

- Hourly meteorological data: wind speed and direction at one or two levels, temperature difference in the surface layer and standard deviations of wind direction and vertical wind speed
- Hourly concentration values of a pollutant from one or - preferably - various monitoring stations (point or open path measurements)

The user assumes the presence of one or various area sources, specifying its emissions

either with a constant or a time variation. You also have to specify the plume height along with the location and geometry of each source. Normally the background concentration, i.e. the part of the registered levels that are not explained by the specified sources, is assumed to be constant in time. There is, however, an option that lets you use a variable background concentration.

As a starting point, each specified area source will be divided into 9 sub areas and the emission represented by 9-point sources. All sources are assumed to emit with a normalized, default emission rate, as an average over the specified period. During this period, each area source may have a unique variation specified by a normalized EDB formula. For the chosen time period, the model will evaluate simulated concentration time series for all receptor points/open paths.

For each specified source, we will then have a simulation time series S_i consisting of data from $j=1, \dots, m$ hours and $k=1, \dots, l$ stations, assuming a normalized emission rate from each source. The emission estimation method consists of a multiple regression analysis of the simulated source contributions (predictors) with the actual, measured data C (as predictand).

- Austal 2000

Austal 2000 from the Environment German Federal Agency is an air dispersion model that uses Lagrangian particle tracking and contains its own diagnostic wind field model. It also considers the influence of topography on the wind field.

- US/EPA Calmet/Calpuff

Calmet

It is a diagnostic meteorological model in 3D that also can accept pre-processed wind fields.

Calpuff

Provides a complete spatial variation of wind and stability, hour by hour. It is applicable at distances under 300 km

- Danard Wind Model

The wind model is used to generate wind fields for some of the Airviro dispersion models including Airviro Gauss, Airviro Grid and Airviro Street Canyon. When selecting the Danard wind model it is executed alone. The wind model is based on a concept where mesoscale winds are generated by using:

- Horizontal momentum equation
- Pressure tendency equation
- First thermodynamical equation

The concept assumes that small-scale winds can be seen as a local adoption of large scale (free) winds.

- Emissions (EM)

The emission model performs emission calculations and stores the emissions in dispersion result. This is very useful if the emission calculation requires a lot of time. It is typically used for ship emission calculations.

- Ensemble (ZE)

Used to run time series simulations in parallel for several (at least two) dispersion models. The available models are: Airviro Gauss, Airviro Grid, US/EPA AERMOD, US/EPA Calmet/Calpuff and Austal 2000. The results can be compared using the Post Processor.

- Noise Screening ISO-9613-2 (NS)

The model was related to rather simple terrain conditions. Only flat terrain and no obstructions like buildings or vegetation were considered for now, more complex terrain and conditions were not part of the actual implementation. The model was however implemented in such a way that it easily allows later developments for more advanced applications.

The model is based on specifications laid out in the document ISO 9613-2. The model was intended for the attenuation of sound from point sources. Extended noise sources, therefore, such as road and rail traffic or an industrial site (which may include several installations or plants, together with traffic moving on the site) are represented by a set of sections (cells), each having a certain sound power. Attenuation calculated for sound from a representative point within a section is used to represent the attenuation of sound from the entire section. A line source may be divided into line sections, an area source into area sections, each represented by a point source at its center. A group of point sources may be described by an equivalent point source situated in the middle of the group.

Concerning meteorological conditions, downwind propagation conditions for the method in this part of ISO 9613 are as specified in 5.4.3.3 of ISO 1996-2:1987.

2.1.4 How does Dispersion Module work?

This Airviro module runs on any PC or other devices running Internet Explorer 6 or later, Firefox or any other mozilla based browsers. After logging in to Airviro with your user-ID and password, a domain must first be selected, and then DISPERSION module should be chosen from the available modules. All data processing is made on the Airviro server and afterwards the results are sent to the web browser.

Airviro version v5.00 or higher dont need Java Runtime Environment to run.

2.1.4.1 Getting started

Once Airviro has been properly installed on the *server*, you can begin using it by typing the correct URL in your web browser over Intranet/Internet. The web interface controls all web modules, including **Dispersion**, Indico Administration, Indico Report, and Indico Validation among others.

After logging in to Airviro with your user name and password a domain must first be selected and then DISPERSION module should be chosen from the available modules .

At the top of the window, in Current application the name of the module actually selected is shown. Clicking on the [down arrow] button beside the name other modules can be selected.

Clicking on the Current language [down arrow] button it is possible to select the language to work with. This functionality is not yet implemented.

By clicking on the [down arrow] **Logout** button, the current module is closed and the Airviro login page is displayed instead.



Figure 2.4.1.1 Logging



Figure 2.4.1.2 Domains and Modules.

2.1.5 Guidance for the beginner:

This user manual consists of two parts:

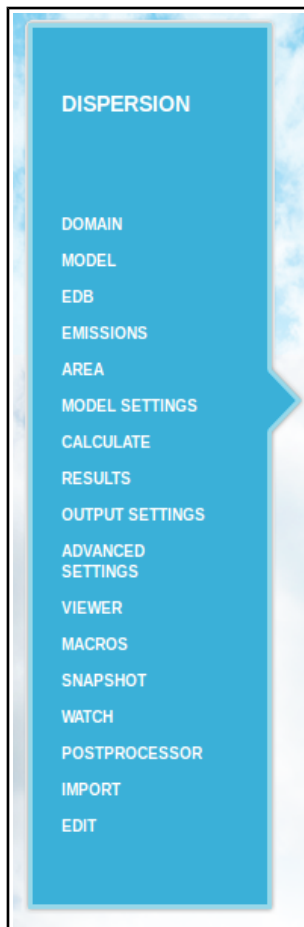
- A first part consisting of an introduction - section 2.1-2.4 - on how to operate the menus and sub windows , and how to run some simple examples.
- A reference part collected in the appendices where some of the models are described and references are available.

The beginner is advised to read the first, instructive part and does one or more of the illustrated examples before penetrating the reference part.

The following sections provide an introduction to the dispersion simulation functionalities and how to use the menus and subwindows . The subwindows where you specify and display your dispersion calculations are almost identical for different models, although the actual dispersion calculations that take place differ strongly in complexity and computer time demanded (being Canyon model calculation the fastest and Grid model the most time consuming).

The examples shown here are based on real data from the Airviro (Göteborg) Reference System, included in all Airviro installations.

2.1.6 Overview of the Dispersion Module Main Window



The main menu window consists of a menu to the left and a working area to the right.

The left menu allows you to:

- Select the domain.
- Select a dispersion model.
- Select the EDB that contains the emissions
- Choose hypothetical or observed weather conditions.
- Post process the results.
- Save macros for later viewing of results.
- Import external result fields.
- Define and display concentrations for receptors points or lines
- The operation of the different menus are explained below, but more detailed explanations on how to use them are given through worked examples in the following sections.

2.1.6.1 Changing Weather Conditions – Model settings

Weather plays a crucial role in dispersion calculations. You can select weather conditions by clicking on **MODEL SETTINGS**. Here you can select hypothetical or observed weather conditions as a basis for dispersion calculations. Different weather options are available, depending on the model that has been chosen, Usually it is possible to run most models over a time period. For some models such as Airviro Gauss and Airviro Grid a scenario calculation can be performed as well. A scenario contains a number of hours that represents a whole time period.

When working with a time period it is possible to verify if meteorological data is available for the selected period.

Airviro can work with multiple meteorological masts configurations and if there is more than one mast configuration , it is possible to select which one have to be used for the simulation. New mast configurations can be added using the IADM module.

Figure 2.1.6.1 Model settings for Gauss model

HINT: from version 3.20 you can select Case with Edb. This selection allows you to run case using an emission database.

2.1.6.2 Specifying Emissions

The **EMISSIONS** window will display different sets of options depending on the selected model. Most models use emissions from an EDB and you can select the sources using the EDB search criteria page (for a detailed explanation see the User reference manual for EDB). There are some exceptions : For Gauss model and simulation type case you can specify point, area or road sources. For heavy gas you specify the type of leakage. For Airviro Street Canyon model the traffic in the street canyon is described. For OSPM a

number of roads can be picked from an EDB and an OSPM simulation is performed for each one of them.

Starting with version 3.23, Gauss and Grid models support ship sources.

For Gauss model, the input texts labelled name, info and info2 can be used as search conditions when searching in the EDB.

You must select a substance from the substance list and then select if only emission, only resuspension or emission & resuspension will be searched for.

Use the **[APPLY]** button to save your settings.

2.1.6.3 Submitting a Dispersion for Calculation

Once you have specified the source emissions, go to **CALCULATE**. Specify the calculation height, and a name for your dispersion calculation before pressing the **[APPLY]** button.

It is possible to follow the progress of a simulation using **WATCH** in the main menu.

2.1.6.4 Viewing Existing Dispersion Calculations

RESULTS allow you to access all completed calculations, either those stored under separate projects or those that were executed by other users. You can also delete unwanted result files (but only those from your own calculations, unless you have the necessary privileges to delete results from other users).

2.1.6.5 Finding Local Values

Use the left mouse button to find out the concentrations at specific locations on the map. Click on different points on the map to view the pollution levels at those locations.

2.1.6.6 Using Receptor Points

RECEPTORS, under **SNAPSHOT**, allows you to extract values at pre defined locations. When pressing this option on the main menu, a new window pops up, displaying simulated concentration levels at certain locations. Go to **EDIT, Receptors** on the main menu to specify these locations. All point definitions must begin with **#POINT** followed by a coordinate, and all station references must begin with **#STN** followed by a station key. A typical receptor definition could be:

```
#STN GM5
```

```
#POINT 2483280 5690787
```

```
#POINT 2484182 5691447
```

```
#POINT 2484141 5691401
```

The receptor points defined by each user are saved on the server in a user's personal folder.

2.1.6.7 Map Functions

The **Area** subwindow contains the usual map functions. You can switch between different maps if more than one is installed in your system, **Zoom in/out**, **Clear** the map, restore the original zoom with **Area Reset**, etc (for more details, see User Reference for EDB, section 1.2.6)

Note that the wind model resolution may be different for different maps.

2.1.6.8 Menu options in the main Dispersion window

Domain Here you select a Domain to work with. As in the EDB module,

Model Here you select a dispersion model

Edb Here you select an EDB to work with. As in the EDB module.

Emissions Here you select the substance used for the simulation . It is also possible to specify what type of sources should be included and to specify search conditions on the sources.

Area Here you specify the geographical area for the simulation including the size and number of grid cells.

Model Settings Here you select the time period, the mast configuration and other model specific settings.

Calculate Here you start the simulation.

Results Here you select a result for viewing. Results matching certain filtering conditions can be displayed.

Output Settings Here you configure how the result will be displayed on the map:

- **Grid rectangles**

Displays a rectangle for each grid square.

- **Grid points**

The same as for **Grid rectangles** but the middle point of each square is displayed instead.

- **Calculation area**

It shows the boundary around the area that you have chosen to do the simulation for.

- **Description**

It shows a description of the current result at the top of the map. Information such as the weather conditions and a description of your calculation will be displayed

- **Color legend**

It can be used when the output configuration format has been set to grid or isocolour. Grid colours and levels used will be displayed on the left side of the map.

- **Output configuration**

Here you can specify how the result field will be displayed. There are four different presentation styles: Isolines, Grid, Isocolour and Class enumeration. The isoline frequency is adjustable. Levels and colours for coloured grid and Isocolour are configurable. The colour palette can be modified. There are 31 levels available.

- **Output window**

Here you configure the window size for **Snapshot Map, PDF. etc**

Advance Settings

A mathematical or logical formula can be applied to the result field. The name of the variable is x1. The syntax is the same as for the formulas used in Indico Presentation except for the time lag which can't be used. To smooth out transients in the result the Smoothing option can be used. Running averages can be calculated when viewing a time series result. The calculated average is shown for the last time stamp in the averaging period. Finally a constant background level can be added to the field.

Viewer Here the simulation result selected can be viewed on a map

Macros

- **Load:** Loads a macro. This could include the simulation result plus the settings to display it. The user interface for settings will be set as it was when the macro was saved.

- **Save:** The selected result and its display settings can be saved in a macro to be used another time.
- **Delete:** It delete a macro.

Snapshot

Clicking on a **Snapshot** option **PDF** or **MAP** whatever is currently displayed on the map will be copied to a new pop up window . This new window is independent of the Dispersion module and can be moved to another part of the screen to be used for reference or comparison. The snapshot window size is specified in **OUTPUT SETTINGS**.

Text , **ESRI Grid** and **EXCEL** opens a new window displaying the calculation area coordinates, calculation description, and concentration values for the calculation grid in text format.

Receptors calculate the concentration value at previously specified locations on the map. The output is displayed in a text window.

Shape and Iso Shape are used to export the result to a zip file containing shape files.

KML generates a KMZ file that contains an isocolour image, projected on WGS84 coordinates. This image (PNG) along with the legend (PNG) are packed together with a KML file. This is the KMZ file. The KMZ file can be viewed in Google Earth.

Animation opens a windows displaying an animated GIF.

Watch

This pops up a new window displaying the progress of dispersion calculations. Calculations can also be cancelled here.

Postprocessor

This option requires that you have first selected a dispersion result as input. The main functionality is based on the processing of results from time period calculations, but there are also some actions that may operate

on other types. The outcome of these operations is a new dispersion result.

With the Model Postprocessor, it is possible to make a statistical study or to create new time series. It is also possible to export the simulated concentrations to the time series database for a station defined in Indico, in order to make comparisons between measured and simulated concentrations.

Available processing types:

- Spatial evaluation. It is used to calculate statistical parameters from time series fields, such as averages, percentiles etc. Running averages and peak-to-mean correction factors may be applied to compensate for different averaging times. Results are saved as simulation type "Combined".
- Time series export. A time series representing the location of a station can be exported to the Indico's Time series database for comparison with monitoring data.
- Model adjustment. An Indico macro is used to adjust the selected result with measured values. The macro may contain more than one measurement station. The result is adjusted in such a way that the location of the station in the result will have the same value as the Indico time series. An interpolation is performed for the values between the stations. Results are saved as simulation type "Combined".
- Split model result. A time series result can be divided into smaller parts. Each part is a new result. Results are saved as simulation type "Time period".
- Merge model result. A number of time series results are blended into one single result. Results are saved as simulation type "Time period".

- **Combine model result:** One or more model results are combined with an arithmetical expression. This works on time series fields as well as single field results.

Import

External results calculated by other models outside the Airviro system can be imported from a file as long as it is in any of the supported import formats. External results are saved as simulation type **“Extern”** .

Edit

Project

It is possible to have up to 26 projects and these can be created here. Once a project has been created it, only the user who created it can delete it. Projects are useful for saving dispersion results in an organized manner.

Receptor

Here you can edit receptors. For a detailed explanation see section 2.1.6.6 Using Receptor Points

2.1.6.9. What is the resolution of the simulation grid?

There are three spatial resolution scales in the horizontal: the wind field, the emissions and the grid where the concentrations are evaluated.

- **Wind field:** It has one unique resolution that depends upon the input of topographic and physiographic information, usually given on a 250x250 m or 100x100 m grid. Wind model calculations will always produce an output wind field with this resolution. If you zoom in over a smaller area, a grid net will be displayed. However, the wind field will be linearly interpolated from the original grid.
- **Emissions:** Point sources are always introduced in the dispersion calculation with

their actual locations. Area and road sources are collected on an emission grid before being entered into the dispersion model, and the grid size will be adjusted to fit in the simulation domain. This means the smaller the simulation (map) area the higher the resolution for emissions. The resolution of the emission grid is twice that of the concentration grid.

- Concentrations: They are evaluated on a grid, which is adjusted to the size of the map. This means that the more you zoom in on a particular area, the better the resolution (in metres) you will get out of the dispersion results. The best available resolution is 25x25 m for all models but the grid model whose best resolution is 100x100 m.
- Go to the **Area** option in the main menu to change the calculation area and the grid size.

Can sources outside the calculation domain influence the results?

No. What you see on the map is all you get out of the simulation. Any impact from sources outside the area must be considered as a background influence.

What is the accuracy of the simulated concentrations?

The accuracy of simulated results depends on the quality of the input data: emission data, description of the surface and meteorological data. There are also uncertainties built into the model assumptions. It is impossible to give a quantitative answer valid for every application. A general rule of thumb says that a model result should be within a factor of two from measured (good quality) values. Various Airviro applications have shown a much better accuracy after a careful inspection of the input data. Therefore, it is recommended to verify the model results with a few monitored values, both as long-term averages and for special events.

Getting Started

The best way to get familiar with the Dispersion module is by carrying out lots of

examples. In the following sections, you will learn to perform simple (but realistic) simulations based on the Göteborg reference database, which are designed to guide you through the main features of the different dispersion models. The first example is explained in great detail but many of the more trivial details have been omitted in the rest of the examples .

. All beginners are advised to work with these examples. . Keep in mind that it is always possible to refer to the appendices to find out details on how the calculations are performed, model assumptions, etc.

After logging in to Airviro with your user-ID and password, select the domain **REF**, and then the **DISPERSION** module from the available modules.

Once the Dispersion module has been loaded, click on the **Model** menu option. The following models are available in Airviro but your installation may not have them all :

- Airviro Gauss (SH)
- Airviro Grid (EU)
- Airviro Street Canyon (CN)
- Airviro Heavy Gas (HG)
- Airviro Receptor (RD)
- OSPM Street Canyon / SMHI Open Road(OS)
- US/EPA AERMOD (AM)
- US/EPA Calmet/Calpuff (CP)
- Austal2000 model (AU)
- Emissions (EM)

- Enable (ZE)
- Noise Screening ISO-9613-2 (NS)

Click on the model name that you are going to use and then click on **[Apply]**.

Then, select a personal edb. The procedure is the same as in the EDB Module.

Click on **Area** in the main menu and select the calculation area.

Inhomogeneous Grid

You can design an inhomogeneous grid. To make a grid with variable density (inhomogeneous) a quad tree structure will be used, i.e. a base grid consisting of quadratic grid squares that are then recursively divided into four when necessary.

For example, in rural areas it is necessary to know the midpoint of a 10x10m grid. Instead, an urban grid is 1x1m.

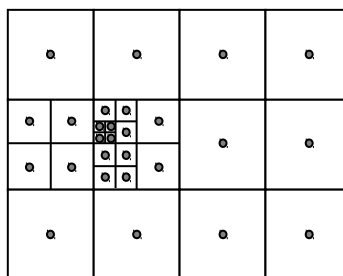


Figure 2.1.6.9.1. Quad tree

Appendix 2K contains a detailed description of the principles for indexing the grid.

Define an inhomogeneous grid with the following input data (*Figure 2.1.6.9.2.*):

Road Sources: maximum grid near source, distance from source where base grid is used and emission threshold.

Point Sources: maximum grid size near source, distance from source where base grid is used and chimney height maximum.

Inhomogeneous grid

Use quad grid

Road Sources

Max grid size near source [m]
10

Dist. from source, base grid is used [m]
500

Emission threshold
1

Point Sources

Max grid size near source [m]
10

Dist. from source, base grid is used [m]
500

Chimney height maximum [m]
20

GRID PREVIEW

Apply your choices

APPLY

Figure 2.1.6.9.2. Inhomogeneous grid.

Clicking on the Grid [UPDATE] button to generate the information in Figure 2.1.6.9.3.



Figure 2.1.6.9.3. Result. Quad grid

2.2 The Gauss Model

2.2.1 Examples Based on the Case Weather Condition

Case is the simplest type of weather condition that can be configured . It only uses homogeneous wind fields and does not take into account neither topography nor physiography . Wind speed and wind direction can be selected along with one of six stability options.

The first example deals with the impact of three stacks with different fundamental characteristics, emitting NO_x.

The screenshot shows a 'Settings' window with a light blue background and a cloud pattern. It contains several sections:

- Type:** A dropdown menu set to 'Case' and a 'Mast' dropdown set to 'Default'.
- Case:** A 'Stability' dropdown set to 'VERY_STABLE', a 'Wind direction' input field with '225', and a 'Wind speed' input field with '2.0'.
- Height [m]:** An input field with the value '2'.
- Emission model:** Two radio buttons, 'Normal' (selected) and 'Resuspension'.
- Apply your choices:** A blue 'APPLY' button and a grey 'CHECK METEO DATA AVAILABLE' button with a green checkmark icon.

Figure 2.2.1.1. Case weather condition.

The ground level concentrations caused by these emissions will be calculated for a particular meteorological event. A fairly stable event will be used with wind coming from the SW at 2.5 m/s.

Start by defining the weather conditions : click on **MODEL SETTINGS** in the main menu. Then select the **Case** option from the drop-down list labelled **Type** , Select a **Stability** condition - for this example we will simulate stable conditions so choose **MOD.STABLE**. Then enter a wind speed and direction. Wind direction has to be in

degrees so for SW you should enter 225. Assign a wind speed of 2.5 m/s (referring to conditions 10m above ground level).

The calculation **height** is the level above ground where the concentration values are evaluated. Default height is 2 m,

This window also allows you to select the emission model (normal or resuspension) and saving either the concentration field or receptor points layers, or both of them.

Click on [**APPLY**], to save your settings

You will study the impact of three point sources with the following characteristics:

	source 1	source 2	source 3
Substance:	NO _x	NO _x	NO _x
Chimney height: (m)	50	20	150
Outer diameter: (m)	7	7	7
Inner diameter: (m)	6	6	6
Exhaust gas temperature: (°C)	175	10	175
Exhaust gas velocity: (m/s)	20	5	20
House width: (m)	0	0	0
House height: (m)	0	0	0
Absolute emission: (g/s)	25	25	25

Create these point sources using the EDB module. Back in Dispersion, click on Emissions in the main menu and under the Sources section, uncheck all checkboxes except Point, in order to use only point sources emissions.

Figure 2.2.1.2 .1 Emission subwindow

2.2.2 Ordering a Calculation

Once you have specified weather and emission data, go to the **CALCULATE** menu. From the drop-down list, select under which project your results will be saved (the project list is shared with the rest of the Airviro users).

The screenshot shows the 'Calculate' dialog box. It has a title bar 'Calculate' and a light blue background with a cloud pattern. The dialog is organized into sections: 'Project' with a dropdown menu showing 'A: Proj 1'; 'Description' with a text area containing 'My first simulation'; and 'Host' with a dropdown menu showing '[any]' and a table with columns 'Host', 'State', 'Load', and 'NtSims'. Below the table are three buttons: 'NESTING...', 'PARALLELIZATION...', and 'HYPE EDIT pm10'. There is also a checkbox for 'HYPE Use Pobs Tabs'. At the bottom, there is a section 'Apply your choices' with an 'APPLY' button.

Figure 2.2.2.1. Calculate

To create a new project name go to **Edit** > Project In the main menu.

Assign a meaningful **Description** , as this text will be used to identify your simulations in the Results window.

If there is more than one host, you can select on which one to perform the dispersion calculation.

Nesting

Nesting allows a calculation in a smaller area (local) using background concentrations from a larger area (urban) without calculating the emissions in the smaller (local) area twice. An overview of the nesting operation can be found in *Figure 2.2.2.2*.

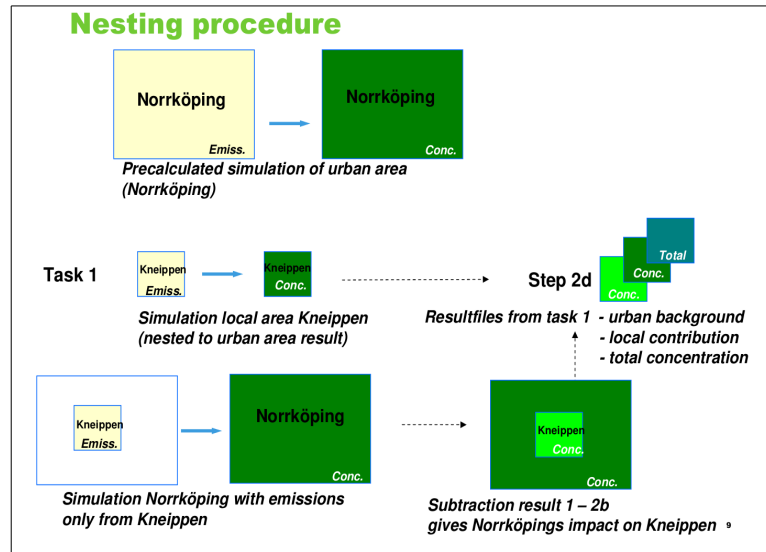
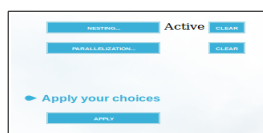


Figure 2.2.2.2. Nesting operation.

The final result from the nesting operation contains the total concentration as well as the local contribution and the urban contribution.

Typically, nesting is used when a result with very high resolution is required, for example 20x20 meters in the local area, and at the same time, concentrations from a simulation with a much lower resolution, for example 500x500 meters. Doing this using nesting saves a lot of calculation time compared to running a simulation with a resolution of 20x20 meters over the whole area.

The **[NESTING]** button opens a window where you can select a field from an existing dispersion result to nest. See Figure 2.2.2.3.



Use the **[CLEAR]** buttons to delete the selections made in both **NESTING** and **PARALLELIZATION** menus.

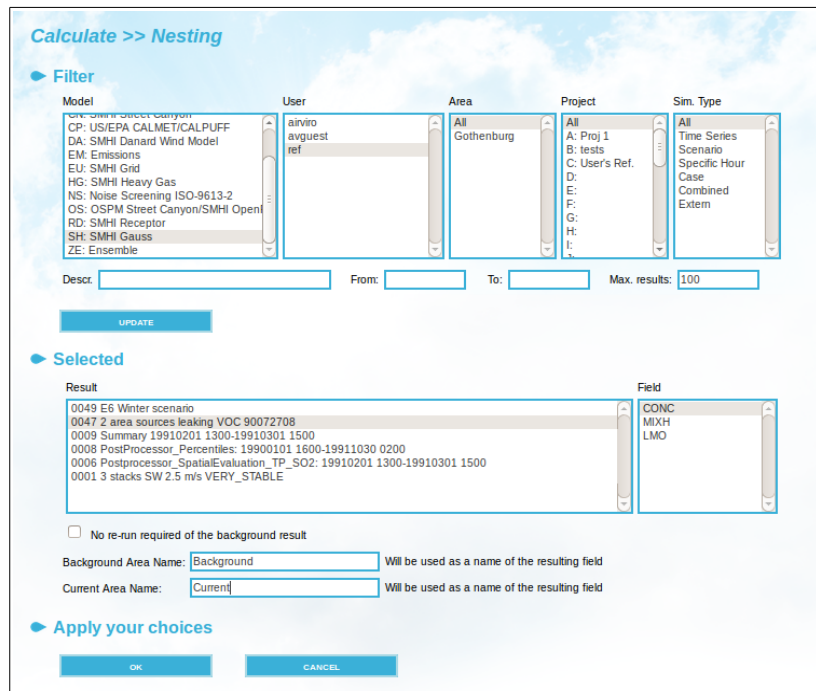


Figure 2.2.2.3. Calculate. Nesting.

Nesting results are shown in Figure 2.2.2.4. Observe that we have three concentration fields: Total, CURRENT and BACKGROUND. BACKGROUND and CURRENT fields were those selected in the nesting menu.

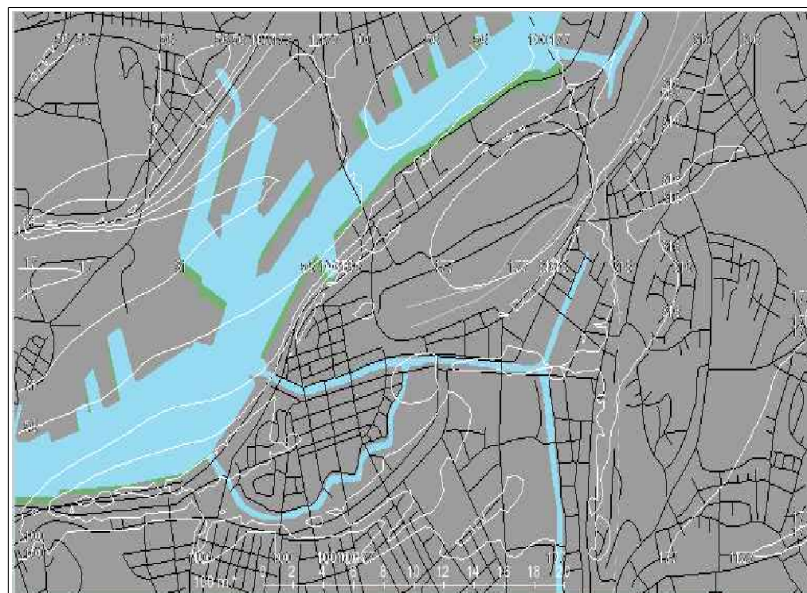


Figure 2.2.2.4. Nesting. Result.

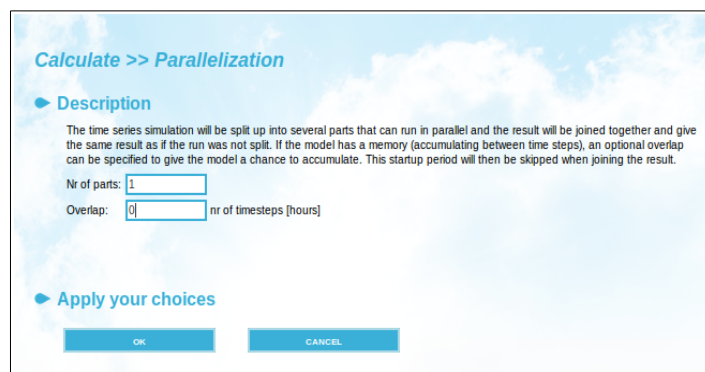
The nesting operation can be performed in any number of steps, in which case the last result will contain the contribution from each step, as well as the total.

Parallelisation

The time series simulation is divided into several parts that will be executed in parallel, then the partial results are joined together in order to produce the same result as if the calculation had not been divided.

If the model has a memory (accumulating between time steps), an optional overlap can be specified to give the model a chance to accumulate. This startup period will then be skipped when joining the result.

You must manually specify the number of parts and number of overlapping time steps (hours).



Calculate >> Parallelization

Description

The time series simulation will be split up into several parts that can run in parallel and the result will be joined together and give the same result as if the run was not split. If the model has a memory (accumulating between time steps), an optional overlap can be specified to give the model a chance to accumulate. This startup period will then be skipped when joining the result.

Nr of parts:

Overlap: nr of timesteps [hours]

Apply your choices

Figure 2.2.2.5 Parallelization

An overview of the parallelization operation can be found below:

- Time series simulation
- At start time it is possible to specify: Number of parts, Overlap (hour)

Example: A Grid model run over 4 weeks is divided into 4 parts with a 24 hour overlap to minimize loss of accumulated concentrations.

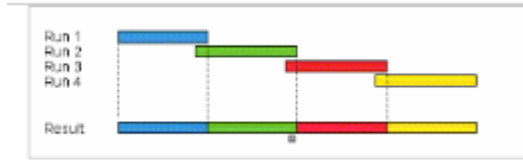


Figure 2.2.2.6 Example Parallelization

Displaying Results

Once the calculation is complete you can display the result. Go to the RESULTS menu. All results that have been saved are listed grouped under four identification lists: User, Area, Project and Simulation type. This gives you the option to display results created by other users, under different map areas or in other projects. Click on [UPDATE] to list the results that match your selections. Use the [DELETE] button to remove unwanted results, use [ctrl] key for multiple selections .

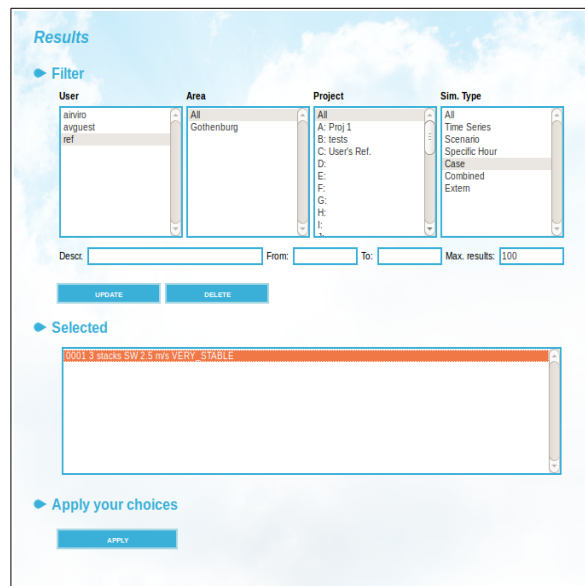


Figure 2.2.2.7 Results.

HINT: All examples discussed in this section are saved under the user name ref and project C: Users Ref.

To display the results of a recent calculation, first select the relevant user name, map area, project and simulation type, then press [UPDATE] to update the listed results. You will identify the result by the description you entered for that particular calculation .

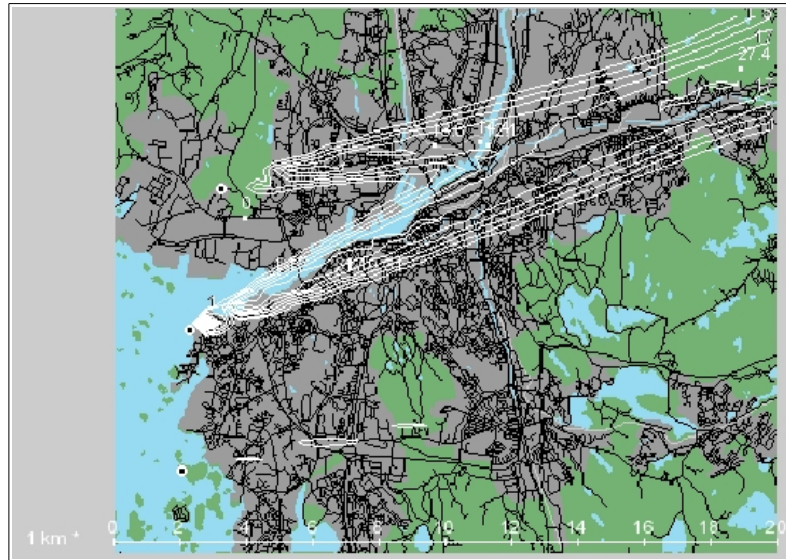


Figure 2.2.2.8 Three stacks with a 2.5 m/s SW wind.

The concentration isolines will be displayed on the **VIEWER** sub-window, A result may contain more than one result field, these fields can be displayed along with a wind field if this was also generated by the calculation .

HINT: The **Output configuration** option in the **OUTPUT SETTINGS** menu gives you three ways of spacing the isolines: either with a logarithmic spacing (by default), a linear spacing or just one single line.

HINT: Tick off the **Description** checkbox in the **Output Settings** menu, to display the calculation description on the map.

The three plumes created by our three sources are displayed in *Figure 2.2.2.8*. If you have repeated this example on your Airviro system, you can use the left mouse button to view the NO_x concentrations at different locations (for a detailed explanation see section 2.1.6.5 Finding Local Values). (The unit is always $\mu\text{g}/\text{m}^3$). The **Receptors** menu under **SNAPSHOT**, can be used to display simulated concentrations at certain pre-defined locations . For a detailed explanation on how to configure receptor points see 2.1.6.6 Using Receptor Points.

Result View Control

The Viewer starts by displaying on the map the first concentration field in the result. Different models creates different results. For Airviro models the wind field can also be displayed over the map, showing how the wind changes direction over the simulated area. Show this field by selecting the **Wind10m** drop-down option in the Viewer. Other fields generated by Airviro models are Emissions, Monin Obukhov length (LMO) and Mixing height (MIXH).

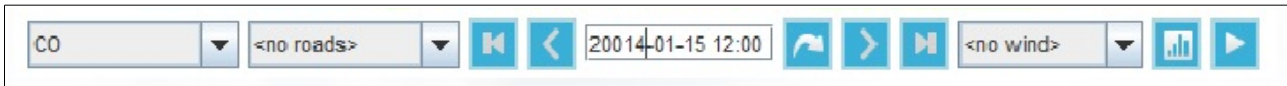


Figure 2.2.3.2. Result View Control (top left corner of the map)

The map tools available are the same as those explained for EDB maps, zoom in/out, reset, etc (for more details see EDB User manual, section 1.2.6 Choosing a Geographical Area).

Show date : Shows the date and time of the dispersion field currently displayed on the map. Use the [**<**] / [**>**] buttons to view the previous / next time frame. Use the [**|| <**] / [**> ||**] to see the first / last time frame.

Show Stats: (Graph icon) Displays the maximum, minimum, average and sum in a text box on the *bottom* right corner of the map. It will also mark the location on the map of the maximum value in the field with a white circle.

Play (Play icon) : It plays an animated sequence of the dispersion evolution for the calculation period.

When hovering the mouse pointer over the map, the coordinates and distance to the point of maximum value are displayed on the map. Clicking on left mouse button the concentration and the coordinates of that location are displayed.

2.2.3 Examples Based on Emission for a Specific Hour

A summertime event will be simulated for a specific hour . You will study the impact of two sources of volatile hydrocarbons (VOC) emissions, the first one generated by gasoline storage and the second from a petrochemical industry. These can be treated as area

sources.

An area source might consist of a number of vents on top of an industrial building or an area where petrol evaporates from leaking tanks (for a detailed definition of area sources see the EDB user manual). It is not possible to calculate the plume rise for emissions generated by area sources, so the dispersion will be that of a neutral pollutant which is just exposed to advective transport and diffusion.

Specify two area sources in the EDB module , as follows:

	source 1	source 2
Substance:	VOC	VOC
Plume height: (m)	5	20
Absolute emission: (g/s)	25	25

Back in Dispersion module, start by selecting Domain, Gauss Model and EDB personal .

Go to the **AREA** menu and select the area you want to cover for the simulation by dragging the mouse pointer over the map while pressing the [ctrl] key as explained in EDB User's manual (the area source in the Northwest in *Figure 2.2.3.1.* is source 1).

Now go to the **MODEL SETTINGS** menu. Select Type: **Specific hour** and select the time period 900727 07 to 900727 08 (note that you are only allowed to change the **To** time, the **From** time is automatically set to one hour less).

When you are done with model settings, go to the **Emission** menu and select a substance, **VOC** in this case. press the [**SUB AREA**] button and select a map area to include the emission sources that will be used in the dispersion calculation, this would normally be the whole simulation area but in some special cases it could be set to a smaller area within the simulation area, then click [**OK**]. Save your settings pressing [**APPLY**]

Now go to the **Calculate** menu, write a meaningful description and press [**APPLY**] to start the calculation. Once the calculation has finished and the result is available, you can watch it using the **VIEWER** (Figure 2.2.3.1), you will notice a much stronger impact close to the area sources as compared to the point sources in 2.2.1 *Examples Based on the Case Weather Condition*, although all examples have used the same emission rate (25 g/s). The low emission height together with the absence of a plume rise explains the high concentrations found close to the area source.

The number of grid points at which concentrations are evaluated will usually be the same, regardless of whether you have zoomed in or not (see 2.1.3 *Comments on Resolution and Precision in the Dispersion Module*). This means that by zooming in a small and fairly square area, you will increase the spatial resolution of the dispersion calculation, which may be very valuable when looking at conditions close to, for example, a road. The resolution in the description of the emission source location is high. But remember that the wind field will always have a rather low spatial resolution, typically calculated on a 250 or 500 m grid.

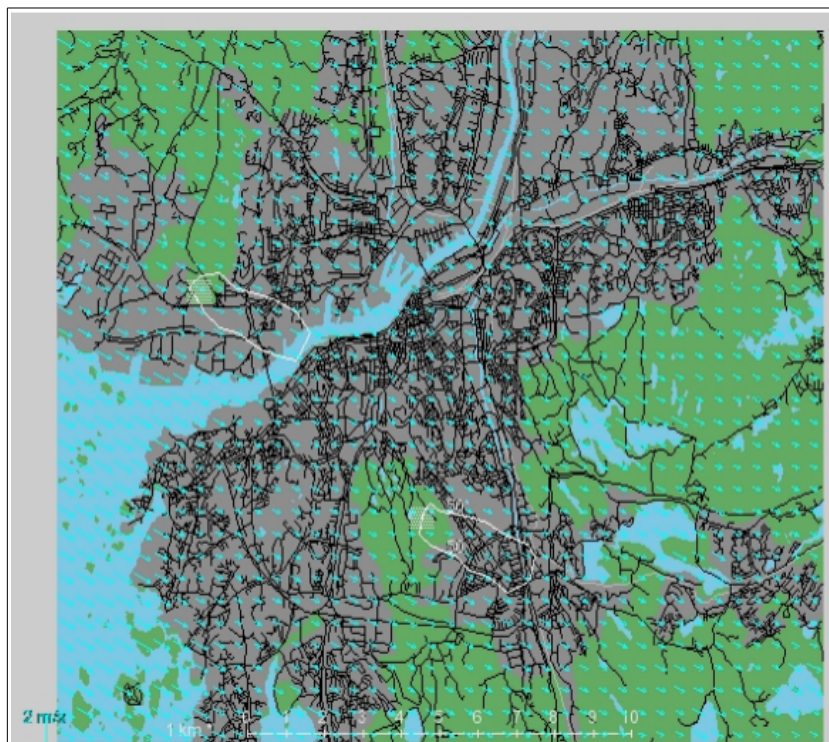


Figure 2.2.3.1 Two area sources along with the wind field.

Figure 2.2.3.1. shows only the 50 $\mu\text{g}/\text{m}^3$ isoline, which clearly shows the directions of the plumes. This is done by selecting **Single 50**, under **Output configuration** in the **OUTPUT SETTINGS** menu.

HINT: If you want to get a better idea of the grid resolution, go to the Output settings menu and tick off the grid rectangles and/or grid points checkboxes to display them on the map area.

2.2.4 Using Scenario Weather Conditions

The E6 is a very busy motorway that passes through Göteborg in a north-south direction. This road has been entered into the reference database as several different joined segments where each section reflects the traffic flow between two junctions. You can look at these road segments using the EDB module . All these E6 segments have names containing the text E6 (for example “Kungsbackaleden E6”, “Kungälvsleden E6”, etc).

Using this information from the global reference EDB we can find out the impact of this single road on the seasonal mean levels of NO_x during wintertime in the central areas of the city.

Firstly, go to the EDB menu and select the **EDB personal** (the EDB option is the only option available),

Now go to the **Model Settings** subwindow to select the weather conditions . Choose Type: **Scenario** and select the **WINTER** scenario option.

Hint: use the IADM Module to set up new climatology, ie. winter or summer. In **Mast**, you can generate climatologies with a name, time period and number of wind direction.

Figure 2.2.4.1. Emission

Then go to the **Emission** subwindow, uncheck all check-boxes for sources except for road sources. Enter search conditions to match all road sources containing the string E6. Use the settings shown in the subwindow above. It is advantageous to use EDB search macros for complicated queries. See *Saving your search criteria in User's Reference Volume 1: Working with the Emission Database (EDB)*.

Now go to **CALCULATE** and start the calculation. In the calculation description you can include information about which EDB sources are included and the specified weather conditions. Monitor the progress of your calculation by starting the **Watch** sub window in the main menu. Season scenario calculations are based on the classification of each season into a number of scenario classes (usually 180). For a description of the calculation principles, refer to *Appendix 2G: Scenario Calculations*.

HINT: the **WATCH** window allows you to monitor the progress of your calculations, this is particularly useful if you have submitted more than one calculation. You can also cancel unwanted calculations in progress.

Once the calculation has finished, you can view it using the **VIEWER**, you will see that you have actually obtained several results for both the mean conditions and also some of the percentile conditions. Select the **MEAN** result field.

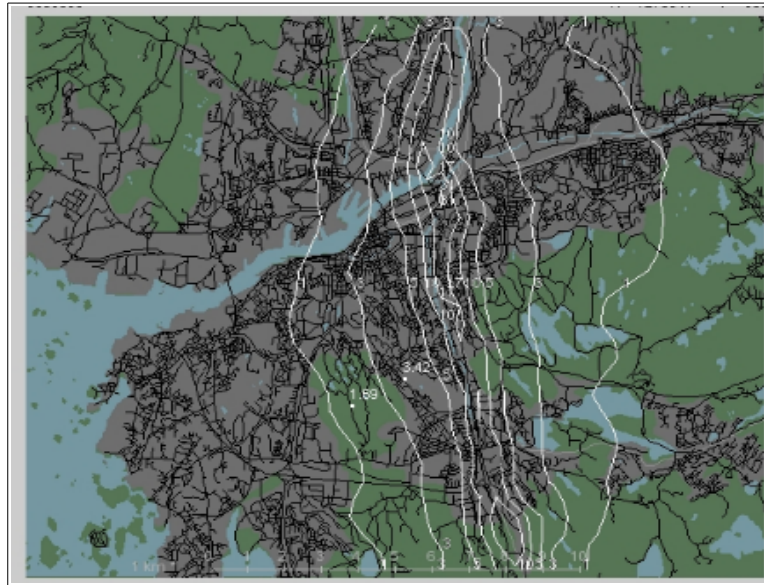


Figure 2.2.4.2 The E6 through Göteborg with winter mean weather conditions.

The result in Figure 2.2.4.2. illustrates the effect of the inhomogeneous wind fields for the rather hilly Göteborg. You will find higher concentration in the valleys running east/west (such as the river valley), and lower values up on the hills. Note also that the winter mean calculation produces nearly symmetric concentrations west of the road, although most of time the winds tend to blow from west to east. The relatively large importance of the sparse occurrence of east winds can be explained by a low degree of mixing during this weather situation. Thus the east wind situation is connected to stable and low wind speed situations while the frequently occurrence of south west winds show higher wind speeds and more vertical mixing.

A different way of presenting this dispersion result, is to display it as a coloured grid. All calculations are actually performed over a grid. You can show the grid rectangles on the map by selecting **Grid rectangles** under **OUTPUT SETTINGS** on the main menu. The size of the grid rectangles is determined by how much you have zoomed in/out on the map.

Go to the **OUTPUT SETTINGS** subwindow.

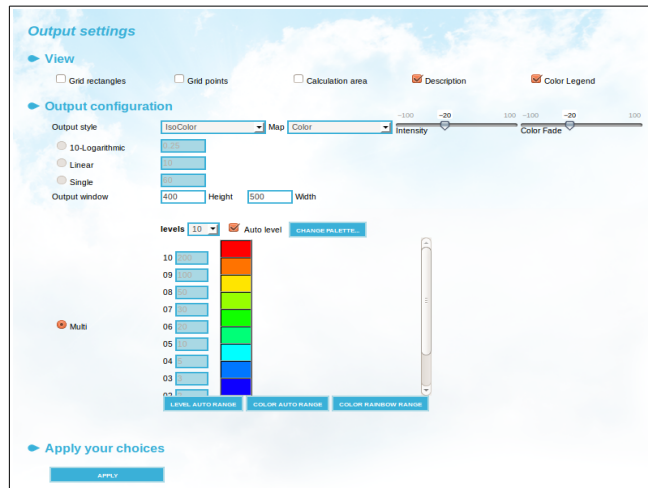


Figure 2.2.4.3 Output settings.

Select the **Output style** option called **Grid** (if you select **Class enumeration** then a class number is displayed instead of a colour). It is best if you also select the **Map** option **B/W** (Black and White) , as the grid doesn't stand out so clearly on a coloured map.

Intensity values range from -100 (darker) to +100 (lighter), keeping the colour but changing the intensity of the background map. **Colour Fade** reduces the colours saturation of the background map. colour fade values range from -100 (full grayscale) to +100 (full colour).

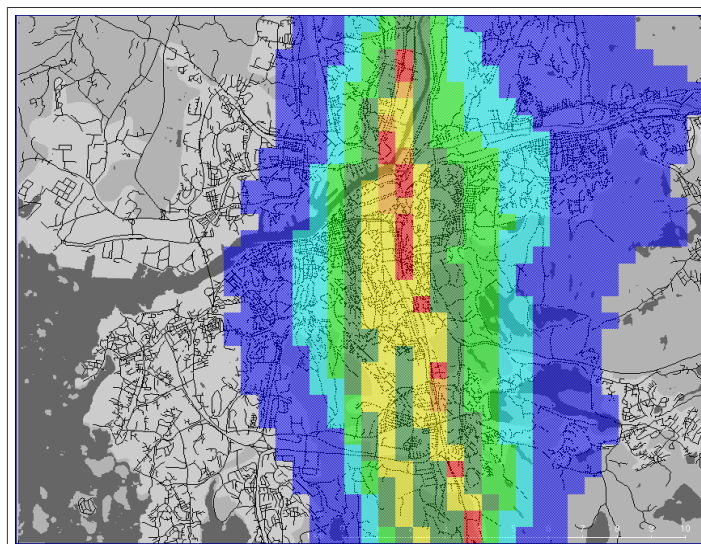


Figure 2.2.4.4 E6 through Göteborg with winter mean weather conditions, shown as a grid.

You can change the colours displayed as well as the class limits by clicking on **Multi** and

configuring the columns for limits and colours at your discretion. To change a colour, just click on it and a select a different colour from the colour palette that shows up.

Tick off the **colour legend** checkbox to display a colour legend on the map.. Press **[APPLY]** and return to the **VIEWER** to see the effect of your new settings.

2.2.5 Using the Advanced Result Options

This function allows you to perform some very useful operations with your dispersion result files created in previous examples. The function is activated within the **ADVANCE SETTINGS** menu, and does not alter the saved result files in any way.

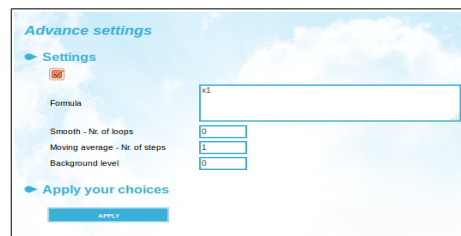


Figure 2.2.5.1. Advance settings.

2.2.5.1 Example Formula 1: conversion of NO_x to NO₂

The photochemical reactions that regulate the fractions of NO and NO₂ (together constituting the total amount of NO_x) are fast and complicated, which means that a realistic simulation including these reactions is a rather advanced task. However there is an empirical way of estimating the NO₂ content from the total NO_x levels. This is important, since NO₂ is the more harmful of the two oxides.

The prerequisites that allow you to use this conversion are:

- Historical NO_x and NO₂ measurements should be available for a station that is located in a representative place (i.e. a station that is situated at a height - e.g. above roof level - equal to where the simulated NO_x values are evaluated) and from a time period that is similar to the one simulated

- The simulation should be a long term value, e.g. winter or summer season (the conversion is not valid for individual hourly NO_x concentrations).
- You have used the multiple regression formulas to find long term correlations between NO₂ and NO_x levels (see Appendix 1 - Model Describing the Conversion of NO_x to NO₂ in Users Reference Volume 2).
- The conversion is performed when you first select a NO_x result file and specify a conversion formula. In the example here, a regression relation is used that was found from monitored data (station Femman at roof level) in Göteborg. Convert the NO_x field simulated in *2.2.3 Using Scenario Weather Conditions*, i.e. the effect of the European highway passing Göteborg:
- On the **Formula**, insert the **NO_x→NO₂** formula and enter the following algorithm:

$$0.73*x1*\exp(-0.00452*x1+0.003014*0.0001*x1*x1)$$

- Click on the **[Apply]** button

The displayed isolines will now be considerably lower than those from the NO_x result file itself. The formula is based on the existence of a statistical relation between the NO₂/NO_x ratio to the absolute NO_x level, in that the ratio normally is higher for low NO_x concentration values.

2.2.5.2 Example Formula 2: Adding Various Result Files Together

This function allows you to build up your own combination of various different concentration fields. These results do not need to have been created over the same sub area, as long as they share the same map. A prerequisite is of course that the same weather conditions should prevail for the results that are summed together.

The function is particularly useful in the calculation of Air Pollution Indices, which are built up by different pollutant concentrations summed together, but with different weight functions. Another interesting use of this function is the calculation of the difference between two result files, e.g. to answer questions like "How much better will the situation

be if I take this action....?”

Initially, you have to select the files from the **Results** subwindows, by clicking on first one result and then clicking **Update**. A list of chosen files is built up in the box under **Select**. Now click on **Advanced settings** to access the advanced result viewer.

The default operation when more than one file is selected is pure addition, which serves our purpose. The result presented on the screen is hence the way to combine effects of different sources (which can not be done directly in one simulation). The functions available in the Combination formula box are the same as those available in the Indico package (see *Exploiting the Mathematical Functions for Calculation Parameters* in *User's Reference Volume 3: Working with Indico Presentation*).

As an exercise, assume that the VOC emissions of the area sources in 2.2.3. *Examples Based on Emission for a Specific Hour* contained a toluene fraction of 11%. Plot the toluene concentration levels for the simulated summer mean values.

Hint: Also you can use the **Postprocessor** subwindow, select **Combined result model** as working tool. Remember: select one result first and then press **Update**, followed by another result and Update again. A list of chosen files is built up in the box under **Selected**. To replace a selected result, first highlight the result to be replaced by clicking on it in the Selected results box, then choose the result to replace it with from the list under **Description** and click on **Replace**. Instead, if you click on **Remove** the result will be taken away from the Selected results list. The **Remove All** button clears all the results in the Selected box.

2.2.5.3 Smoothing of Concentration Isolines

The need might arise for you to smooth your isoline pattern, without actually changing the levels (although this should be used with care). The smoothing function consists of a Laplacian filter, whose effect increases as more iteration are asked for. Test the output by a stepwise increase in the number of iterations.

2.2.5.4 Moving average

Many air quality standards use moving averages. If you do not want to apply a moving

average, make sure that the size of the filter is 1 time step, otherwise it might be applied by mistake.

2.2.5.5 Adding a Background Value ($\mu\text{g}/\text{m}^3$)

When you know that your calculated concentration values only contribute to a part of the measured levels, you may add a constant background value. The background concentrations refer to sources not included in the simulation, within, or more commonly outside the calculation domain.

The background level will simply be added to your simulated concentrations. The isoline pattern remains exactly the same.

2.3 The Street Canyon Model

2.3.1 Street Canyon Overview

The street canyon model allows you to perform small scale simulations of the situation at a particular street segment surrounded by buildings. The model output is presented as concentration isolines on a vertical plane perpendicular to the street (for more details, see *Appendix 2E: Street Canyon Model*).

Firstly, go to the **MODEL** menu and select **Canyon model (CN)** . Now you can define your street canyon. The definition of the weather conditions in **MODEL SETTINGS** is identical to that of the Gauss model (*2.2 The Gauss Model*). Other model specifications are assigned in the **Emission** subwindow.

The screenshot shows the 'Emission' window for the Canyon model. It is divided into several sections:

- Parameters:** A list of substances including NOx tunga, NO2, Naphthalene, Ni, Nitrate, Nucleation10^13, Number of hours, O3, PAC, PAH, PAN, PCB, PCDD, PCDF, and PFC. 'NO2' is currently selected.
- House information:**
 - Distance between houses [m]: 20
 - Height of houses [m]: 30
- Street information:**
 - Street width [m]: 14
 - Street direction [degrees]: 45
 - Position of street: 1275083 9608125
- Emission:**
 - Fixed (selected) / Roadtype (unselected)
 - Emission [microg/m.s]: 1000
- Apply your choices:** An 'APPLY' button.

Figure 2.3.1.1. Emission window for Canyon model.

Start by choosing a substance from the list . Then filling in the rest of the inputs with the correct information.

Distance between houses and **Street width** are not the same – **Distance between houses** includes pavements and verges while **Street width** is just the part of the canyon base where the substances are emitted, e.g. the width of the part of the road where

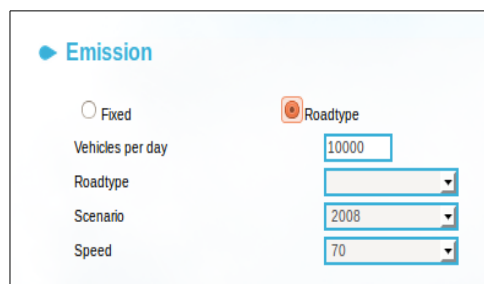
vehicles actually drive on.

Height of houses is the height of the buildings on each side of the canyon. The model assumes that the buildings on both sides of the canyon are approximately the same height.

The **Street direction** is the angle of the street in degrees measured clockwise from north.

Now click on the **Position of street** button and define the position of the canyon on the map. This is necessary in order to estimate the local wind at roof level for the dispersion calculation.

To specify the emission from the street you have two options, Fixed and Roadtype. **Fixed:** you only specify a constant emission value (in microg/m,s). **Roadtype:** you can link the emission to the road types defined in the EDB. In this case, select a road type from the **Road types** drop down list, enter the average number of vehicles per day, scenario and finally select the average speed of the vehicles in the canyon from the **Speed** drop down list.



Emission

Fixed Roadtype

Vehicles per day: 10000

Roadtype: [Empty dropdown]

Scenario: 2008

Speed: 70

Figure 2.3.1.2. Emission from road types

Select the option that you want and click on the **[Apply]** button to save settings.

Example: Use of the EDB emission model in a Street Canyon simulation

This example studies midday VOC levels on Linnégatan, one of the main streets in the city center of Göteborg. The pollution levels along sidewalks are of particular interest, as many outdoor restaurants and cafes can be found in the vicinity. The traffic volume is about 21 000 vehicles per day, with a heavy traffic percentage of about 5%. The case will be studied with a typical north westerly wind of around 1-3 m/s during the summer season.

Assign the following weather settings in **MODEL SETTINGS**:

Type (Model Settings): Time period

Period: 9908021-99080220

If you want to know the wind speed and direction for the chosen hours, use the Indico Presentation module and see those values for Lejonet station. You will see that at 5 pm, at a height of 8 m the wind speed is around 2.8 m/s with a direction of 325° (315° would be true northwest). At 8 pm, the wind has calmed down to about 0.6 m/s

In the **Emission** subwindow, specify the following input data:

Substance:	VOC
Distance between houses (m):	30
Height of houses (m):	25
Street width (m):	15
Street direction (Degrees):	0
Road types	City centre 5% 5 s
Vehicles per day	21000
Speed	40 km/h
Scenario	1999

The position of the Linnégatan model section is shown as a white rectangle on the map to the left of the result shown in *Figure 2.3.1.3*. Use the mouse pointer to locate the position where you want your simulation to be performed.

Street Canyon calculations are submitted for batch execution in exactly the same way as for the Gauss model. Don't forget to give a proper calculation description.

The finished dispersion result consists of several fields, one for each hour chosen. When you select the result, a Result View Controller automatically appears where you can use the arrows under **Time** to step through the hours. If you activate **Wind**, you will also be able to see how the wind field changes from hour to hour.

The result file will be displayed as in *Figure 2.3.1.3*. Note the higher pollution levels along the western side of the street, i.e. on the side to which the vertical eddy circulation will bring the traffic exhausts. Use the left button mouse to find VOC levels along the pavement (about 5 m from edge of the buildings, 1-2 from ground). Percentile values for season calculations can be viewed using the sub-menu shown below.

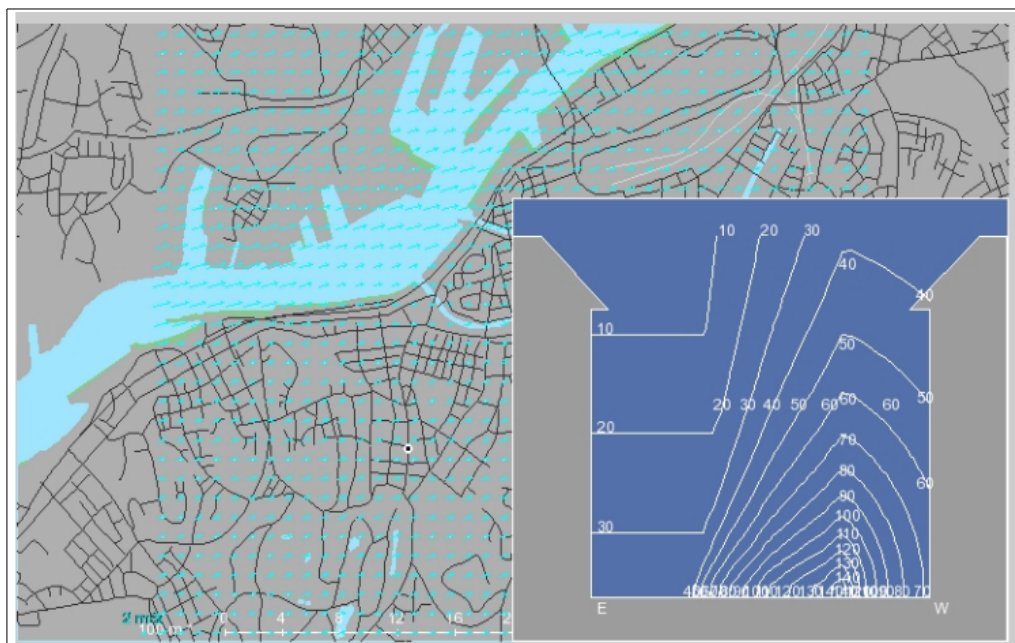


Figure 2.3.1.3. Canyon model of a typical summer west wind across Linnégata.

2.4 The Grid Model

Although the examples for the Gauss model have shown that different types of sources can be entered manually for each individual simulation, you will probably use the emission database for most of our dispersion calculations. Advantages of using EDB are:

- all source characteristics are already stored in the Airviro and ready for use in dispersion models
- the EDB allows time and temperature dependence of the emission rate
- the EDB includes an emission model for road traffic, which means that you use traffic intensity as input instead of absolute emissions (the former being considerably easier to conceive)

When using the Grid model, you can only use input data from an EDB. If you want to perform a simulation from a single source, you must first create the source in an EDB and then specify search conditions that will only match that source.

Start by clicking on **MODEL** in the main menu, select the **Airviro Grid (EU)** model from the list and press **[APPLY]**, then click on **EDB** in the main menu and select an edb. . For this example you should use the global EDB for the REF database (the global EDB is usually the default EDB in Dispersion module).

2.4.1 Looking at Dispersions for a Specific Time Period

This example examines how the dispersion pattern changes from hour to hour when a dramatic change in wind direction occurs. The wind data to be used can first be examined using the Indico Presentation module .

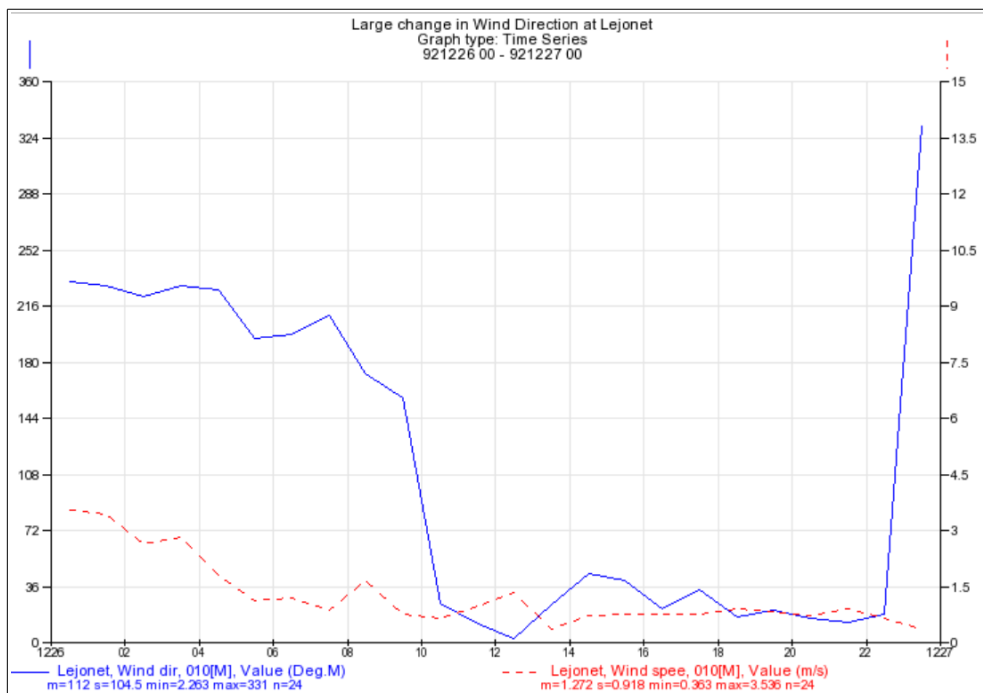


Figure 2.4.1 Wind speed and direction at Lejonet 921226

Select Lejonet station and wind direction and wind speed parameters. Then select a time period from 921226 00 to 921227 00 and display the graph. You will see that at 10 o'clock the wind was blowing from the south east but at 11 o'clock it had changed to a northerly wind. During the entire period the wind speed is very low.

Now some EDB sources are needed as input to a dispersion calculation with these weather conditions. Ideally it is best to use a large point source in order to be able to clearly see the effects of the changing wind direction. Start the EDB module and examine the global EDB. Search for all point sources that have Large NO_x as search key 1. This should show one large source in the north eastern part of the map, which is an incinerator, and a cluster of smaller sources in the western part of the map, which are part of the Shell oil refineries. The total emission is 1144 tons per year emitted at a constant rate throughout the whole year. Such high emission levels produce illustrative plumes in the dispersion models. Note however that the two largest NO_x sources found by the search have a stack height of over 100m. This means that under certain weather conditions all or part of the emission plume breaks through the inversion layer and can be transported outside the map area.

Now return to the Grid model and set up the parameters for the dispersion calculation.

Under **MODEL SETTINGS** choose Time period and enter the period 921226 07 to 921226 15.

Under **EMISSIONS** only EDB is available; select it, then select Large NO_x as search key 1.

You are now ready to submit the calculation. Click on **CALCULATE** and enter a proper description including the date and times for the time period selected. Start the calculation.

The final dispersion result consists of several fields, one for each hour chosen. When you select the **Results** option and click on **Viewer**, the Viewer page is shown and you can use the arrows under **Time** to move through the calculations. If you **select Wind10m** then you will also be able to see how the wind field changes from hour to hour. Looking more carefully at the wind field, you will see how it is affected by both topography and ground type.

To get a clear overview of the areas with the highest concentration and the way the plume moves around, tick off the **Grid** rectangles checkbox in **Output Settings**.

The following pictures show some of the resulting fields. If you have not run the simulation yourself then you can find it under the user **ref** for the project Users Ref. so that you will be able to look at the whole time series of simulations.

The first dispersion result in *Figure 2.4.2* shows the dispersion pattern at the beginning of the period. Emissions reach values above 30 µg/m³ for the left plume and above 20 µg/m³ for the right plume. The south-south-westerly wind is stronger over the sea and rotates to the right over land. Variations in the wind field over land can be seen between built up areas and open fields.

The second picture in *Figure 2.4.2* shows the dispersion pattern at midday. The wind has completely changed direction and is now blowing from the north. The wind is a bit stronger and there has been no build up of emissions yet so the emission levels are lower (the highest value is just above 10 µg/m³). Small variations in the wind field can be detected.

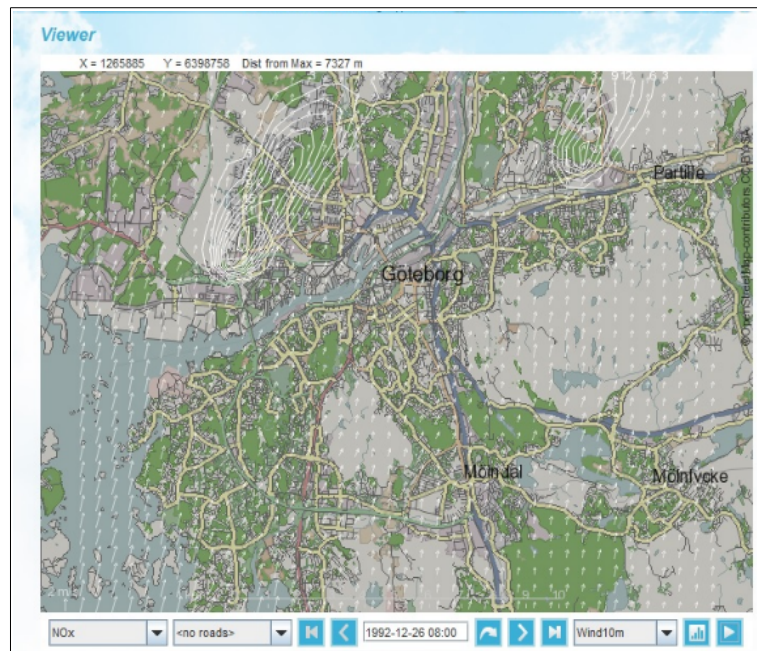


Figure 2.4.2 Two NO_x plumes and wind field at 921226 08 and 921226 12

2.4.2 Looking at Seasonal Dispersions

Seasonal calculations for the Grid model are very complicated and only accept input data from the EDB. Therefore these calculations can take a very long time, at least several hours and even longer if you have many point sources in your EDB.

The output from the Grid model can be compared with that of the Gauss model. In 2.2.3 *Using Scenario Weather Conditions* a simulation of the E6 motorway emissions was performed for a winter scenario. The same simulation can be run with the grid model and the mean wind field is shown in *Figure 2.4.3*.

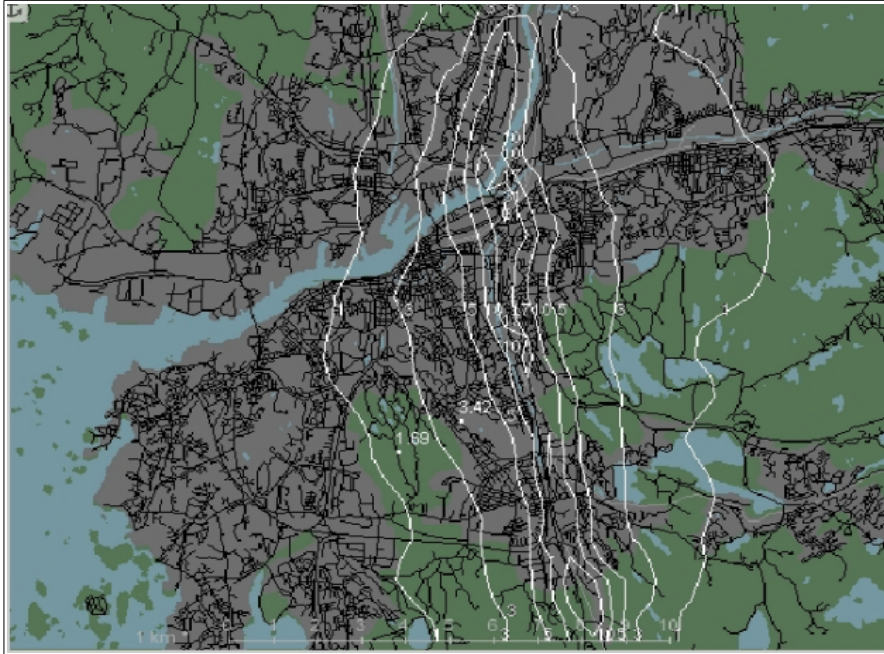


Figure 2.4.3 Winter mean scenario for the E6 motorway.

2.5 The Heavy Gas Model

The Heavy Gas model is unlike any other models. As its name implies, it cannot be used with the ordinary substances used as input for the other models, instead has its own predefined substance list where properties such as molecular mass and specific heat capacities are taken into account.

The Heavy Gas model has been developed mainly for estimating emissions of pressure-condensed gases generated by accidents, such as a leaking pipe or a fractured tank. The emission can be specified in one of three ways:

- As a tank burst (where all the gas is emitted instantaneously)
- As a continuous liquid leak (where the leaking liquid evaporates to gas)
- As a continuous gas leak
- As a continuous pipeline leak

The weather input for this model is either a hypothetical **case** or a **specific hour** extracted from the time series database. A wind field is then calculated and this together with some information about the amount of gas and size of the leakage are used as initial conditions for the model. The output produced is, however, a time series and the result can be presented for up to 20 predefined time steps.

The initial wind field is used for the entire calculation period (usually less than one hour). A major difference compared to the **case** alternative when using a **specific hour** is that the topography and surface roughness are taken into account in the calculation.

The Heavy Gas model should be used to give an indication of where the worst concentrations would be and how high this concentration level might be. It should be used for small areas (ideally a couple of square kilometres only). The initial dispersion process (the heavy gas phase) should be studied by zooming in as close to the source as possible.

Change to the Heavy Gas model by selecting **Heavy Gas** in the **Model** sub-window .

2.5.1 Simulation of a Continuous Liquid Leak

The following simulation describes a tank accident at midday in the town centre. Chlorine leaks out of the tank over a period of about half an hour. An hour after the leakage began; the chlorine has been carried away by the wind.

Figure 2.5.1 Emission-Heavy Gas.

First, set up the weather conditions in **MODEL SETTINGS**. Select **Specific hour** and specify the date and time 920524 12-13.

Go to **AREA** and zoom into an area of approximately 2*2 km in the middle of the map, use the scale shown at the bottom of the map to get an idea of the size of the area you have selected.

Now go to **EMISSIONS**,. Select a substance from the list and finally click on [**Pos. of source**] and, locate the source somewhere towards the top right corner of the map. Then enter the criteria for the heavy gas emission. Enter the following information:

Events: Continuous Liquid Leak

Substances: Chlorine

Tank data

Pipe length: 0.1 m

Pipe diameter: 0.025 m

Tank contents: 10000 kg

Save your settings by clicking on the [**APPLY**] button.

Now go to **CALCULATE** and start the calculation. The calculation will be ready very quickly.

Figure 2.5.2 shows the concentration levels (in mg/m³) 10 minutes after the leakage began. The dispersion has now reached a steady state and the plume stops growing. However, 45 minutes after the accident, all of the chlorine has leaked out and the plume starts drifting away.

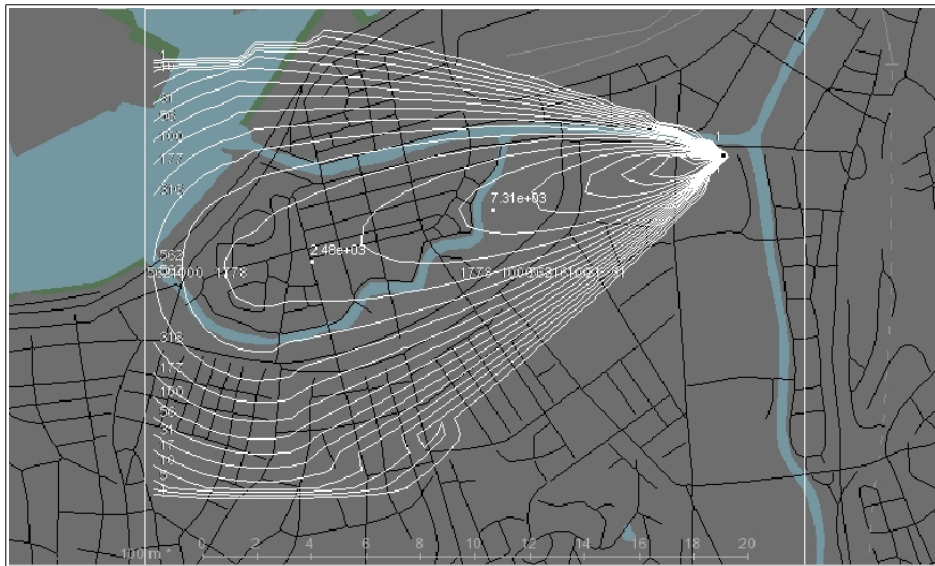


Figure 2.5.2. Simulation of Chlorine levels from 920524 12, 10 minutes after a leak began.

2.5.2 Simulation of a Tank Burst

With a tank burst, the emission takes place instantaneously and the highest concentration levels immediately begin to drift away from the source location. The tank burst concentration is rather idealised and is always shown as concentric circles. For this simulation example a Case weather condition will be used.

Go to **MODEL SETTING**, select **Case** type option and **NEUTRAL_POS** stability condition. Assign a wind direction of 225 degrees and a wind speed of 3 m/s. Keep the same map area as for the continuous liquid leakage.

In the **EMISSIONS** subwindow specify the following settings:

Events: Tank burst

Substances: Ammonia

Tank Data:

Pipe length: 5 m

Pipe Diameter: 0.025 m

Tank contents: 10000 kg

Click on the [**Apply**] button to save your settings .

Now start your calculation. It should be ready almost immediately.

Figure 2.5.3 shows how much the centre of the plume has moved just 10 minutes after the burst. An hour after the burst the plume has completely left the area.



Figure 2.5.3 Ammonia leakage from a tank burst, 10 minutes after the burst.

2.6 Introduction to the Receptor Model

Suppose you have at your disposal:

- A number of high quality time series measurements of a particular pollutant
- Simultaneous time series measurements from a meteorological mast situated in the same area

Using the Indico Presentation Module , you have looked at the concentration time series, and have noticed occasional events where the pollution levels rise in a way that clearly demonstrates an impact of some source or sources. Your immediate question is likely to be:

:

2.6.1 Where is this pollution coming from?

Although you have your suspicions, you need more objective proof to support your ideas.

A first indication can be found by performing one or various Breuer plot diagrams in the Indico Presentation Module, which will inform you about the wind direction correlating to high pollution levels at a certain monitoring station (see section *Using the Polar Diagrams* in *User Reference Volume 3: Working with Indico Presentation.*). However, these diagrams will only tell you the directions to a potential source, not the distance. You will be able to get a more detailed analysis of the measured data through the use of the Receptor module described in this volume. The Receptor Model implies the combined use of measured concentration values at certain locations (receptor points/open paths) and simulated concentrations evaluated at the same locations. This manual assumes that you are already familiar with using Indico Presentation Module .

A requirement for working with this module is that at least a few months of hourly concentration measurements and simultaneous data series from a meteorological mast measurement station should be available. The need for high quality data cannot be overstated, as in the end it might be difficult for the user to discover erroneous results that depend on non-representative input data. This is the reason why only meteorological mast data is accepted.

Although the Receptor model may in principle be used on any scale, we recommend that it be only applied to small areas, typically not more than a few square kilometres. This is partly because of the fact that the model assumes a homogeneous wind field, i.e. the dynamic wind model is not used. The Receptor model is easier to use for ground level sources than for stack emissions due to the fact that the height of the source is specified before execution. The height is vital to estimate the location.

2.6.2 To ensure a proper use of the Receptor module:

Use only high quality data, checked and controlled in the Indico Presentation Module

Small scale applications are recommended.

It is better to use the model for ground level sources (sources where the emission height is lower than the height of the wind sensors on the meteorological mast)

You may also consider the case where you have a rather clear picture of the dominating pollution source /sources location/locations.

The next important question is:

What are the emission rates for these dominating sources?

The Receptor Model will calculate the estimated emission for one or various sources, based on the monitored time series concentrations.

Note, however, that the quality requirement of the input data to perform the emission estimation is even higher than that required to locate the dominating sources.

The use of the Receptor model will be explained by discussing some examples built on synthetic data, since the actual Reference database lacks relevant data. More detailed descriptions of the calculation methods are given in *Appendix 4A: A Method to Locate Dominating Sources* and *Appendix 4B: A Method to Estimate Emissions from Dominating Sources*.

2.6.3 Overview of the Receptor Model Settings Window

Click on **MODEL** in the main menu and select the **Receptor model (RD)** option from the list. You will use the emission database for our dispersion calculations.

Then, go to **MODEL SETTINGS**. For **Period**, specify the start and stop dates to search for data in the database. Select the set up of your meteorological mast from the MAST list, for example, default or temporary mast.

The screenshot shows the 'Settings' window for the Receptor Model. It features several sections: 'Mast' with a dropdown menu set to 'Temporary mast'; 'Format' with a dropdown menu set to 'European'; and 'Period' which includes 'From' and 'To' date pickers. The 'From' date is '000101 14' and the 'To' date is '020101 16'. Below the date pickers are navigation buttons (<<, >>) and a 'PRESENT' button. There are also sets of minus/plus buttons for Year, Month, Week, Day, and Hour for both 'From' and 'To'. Below the Period section is a 'CRITERIA' button. Further down are sections for 'Meteorology' and 'Concentrations', each with 'DATABASES' and 'VARIABLES' buttons. At the bottom is an 'Apply your choices' section with 'APPLY' and 'CHECK METEO DATA AVAILABLE' buttons.

Figure 2.6.3.1. Model Settings subwindow

Click on the **[Criteria]** button to set restrictions and conditions for the variables.

Settings >> Criteria

▶ Period Selection

Period within year

Active From 0101 To 1231

Period within day

Active From 01 To 24

Weekday All Day type All

▶ Condition Selection

STATUS CONDITIONS CLEAR ALL

▶ Apply your choices

OK CANCEL

Figure 2.6.3.2. Criteria subwindow

Now select the meteorological data that will be used as a basis for your calculations by clicking on the **[DATABASE]** button below the **Meteorology** heading to open the **Time Series** subwindow. Select one meteorological mast from the stations list and parameters wind speed, wind direction, temperature, standard deviation of vertical wind direction and vertical wind speed, and temperature difference. It is possible to also include wind speed and direction from a second level if this data is available. The data resolution is fixed to hourly values.

Note that for any given hour, data will only be used if it is present for **all** chosen parameters.

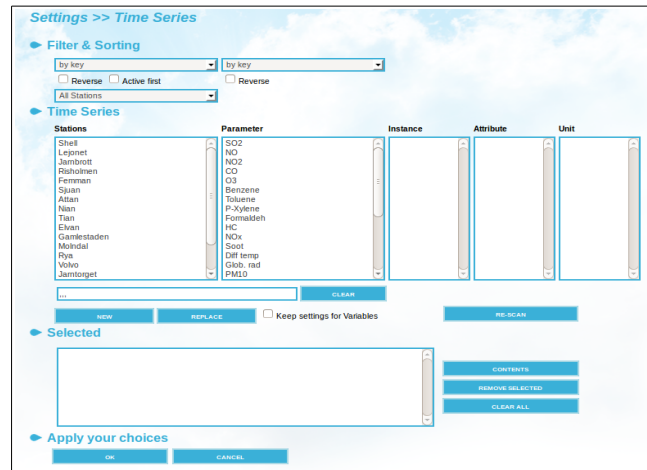


Figure 2.6.3.3. Time Series subwindow

The button next to [DATABASE] is [VARIABLES], and can be used to apply mathematical transformations or criteria to the meteorological data selected.

Concentrations works in the same way as **Meteorology**. Repeat the same steps explained above.

Note: Use the on-line help if you have difficulty specifying the meteorological data.

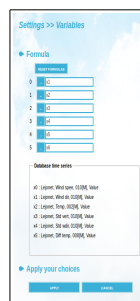


Figure 2.6.3.4 Variables subwindow

Before saving your settings by clicking on the [APPLY] button, verify the meteorological data selected by clicking on the [CHECK METEO DATA AVAILABLE] button

2.6.4 Overview of the Receptor Model Emissions Window

In the main menu click on **EMISSIONS** to open this subwindow. Depending on whether

you are performing a **source location** or **source estimation**, different information will be required about the sources that you are searching for.

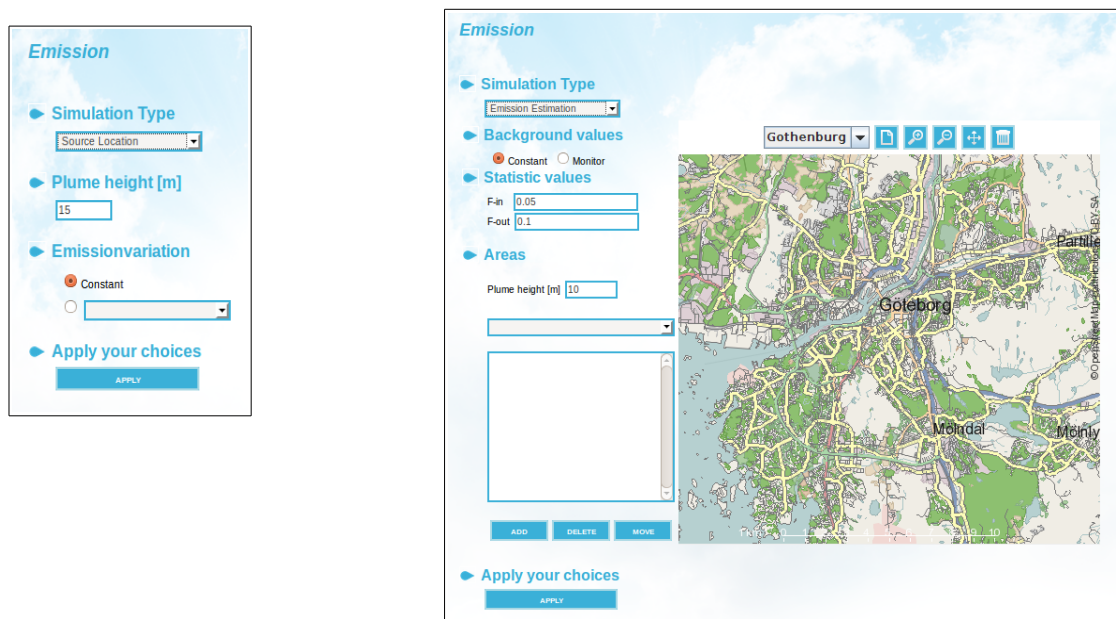


Figure 2.6.3.5. Emission subwindows. Source location(left) and emission estimation (right)

2.6.4.1 Source Location

How to Locate Dominating Sources. An Example.

As previously mentioned, high quality data cannot be overstated to use with this application. In this example, we will first locate the sources and then perform the actual emission estimation, partly built on synthetic data. The reason for this is the lack of good, high quality data in the Reference database.

The Data

The basis for the calculation consists of meteorological data from *Lejonet* station in Göteborg, for the period 890201 to 890501. We have taken the Gaussian dispersion model and calculated the impact of two area sources on two DOAS stations, each of them taking measures along two paths (see Fig. 2.6.4.1.). In order to achieve more realistic time series, we have further introduced:

- an instrument noise randomly distributed within the interval $\pm 2 \mu\text{g}/\text{m}^3$
- a randomly distributed background level in the interval 0 to $20 \mu\text{g}/\text{m}^3$, equal for all 4 DOAS paths, and with a long term average of $10 \mu\text{g}/\text{m}^3$.

Time series of measured toluene concentrations have been saved in the Reference database. Stations are named EMEST1 and EMEST2. The area sources emit at a constant rate of 10 and 5 g/s, respectively. Both emissions have plume heights of 10 m.

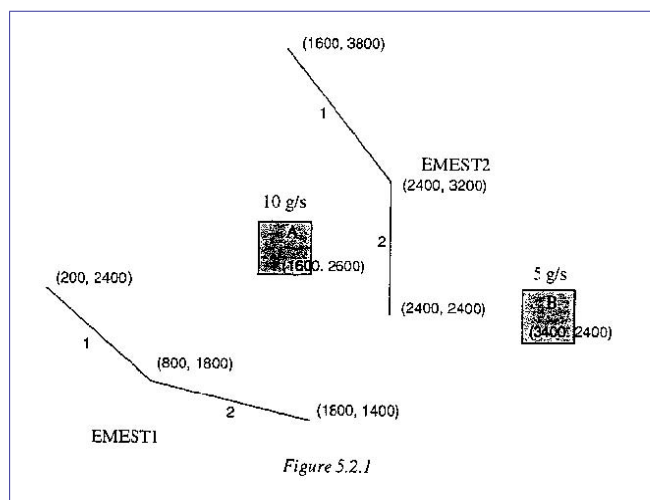


Figure 2.6.4.1. DOAS stations.

The calculation

It is advisable to study the concentration data in the Indico Presentation Module before using the Receptor Model. You may see the variation around the background level of 10

$\mu\text{g}/\text{m}^3$ plus some peaks where it is obvious that some dominating sources affect the DOAS stations.

Assign the following data to Model settings :

Meteorology:

Database Lejonet, Wind speed, 010, Value

Lejonet, Wind dir, 010, Value

Lejonet, Temp, 002, Value

Lejonet, Std vert, 010, Value

Lejonet, Std vdir, 010, Value

Lejonet, Diff temp, 008, Value

Concentration:

Database: EMEST1, Toluene, 001, Value EMEST1, Toluene, 002, Value EMEST2,
Toluene, 001, Value EMEST2, Toluene, 002, Value

Period From: 890201 To: 890501

When it comes to specifying the emission (EMISSIONS sub window), there are several ways to do it. First you must guess the height at which the largest concentration is found (plume height). Then you can specify an emission variation, which you can assume to be either constant or based on a formula previously defined in the EDB (monthly or daily variation).

For this example though, specify a plume height of 10m with a constant emission variation.

Go to **AREA** and zoom in on a relevant sub area around the two DOAS stations.

Now go to **CALCULATE** and start the calculation. Remember to give a meaningful description.

This calculation can take quite a long time, depending on the speed of your computer. You can watch its progress by clicking on **WATCH**. When it has finished, click on **RESULTS**, select the finished calculation and click on the [**APPLY**] button.

Now go to **VIEWER**. The result should look like Fig. 2.6.4.2. as long as the sub-area that you selected is similar to the one shown in the figure. Using the left mouse button (Local value) you can easily find the two area sources, although their physical size is not so easily deduced. If these area sources were real industries, it should be possible for you to combine your knowledge of the area with the isoline pattern to actually be able to guess the dimensions of the two dominating sources.



Figure 2.6.4.2. Result

2.6.4.2 Emission Estimation

Once you have found one or more dominating sources, you can proceed to estimate their respective emission rates. We will use the same data as in section 2.6.4.1 for this exercise.

Use the result of the source location calculation as a good background on which to base your estimate and keep the same settings used previously.

Setting a grid output in **Output Settings**, could help you to guess the size and location of the sources more easily. Select the **Grid** option in the **Output style** drop-down list, change the map to **B/W** and select the **Multi** option. Experiment with different settings until you get a satisfactory grid on the map.

Now go to **EMISSIONS**, change the simulation type to the **Emission estimation** and define the location and size of the dominant sources, in order to estimate their individual emissions.

Currently you can only locate area sources. Use the result of a source location calculation to guess the approximate location of the source(s) that you are looking for. This way you can locate one or more sources on the map. Define your areas on the map by clicking on the [**ADD AREA**] button and dragging the mouse pointer over an area while pressing the [ctrl] key. Each emission source thus defined can be linked to an emission variation described by a time variation formula previously defined in the EDB module. It is also necessary to assign to each source an estimated plume height.

For this example use 10 m plume height for both sources and the **STANDARD** formula (which gives a constant emission that does not have any time or temperature variations). Keep the isoline or grid pattern given to the source location calculation to locate two area sources.

You should also make some assumptions about the background concentration. Generally, it can be assumed to be constant, but in cases where you expect the pollution monitors to be influenced by a remote source, you can try the **monitor** option. The latter method will only use the hourly values when there are at least one monitor that is assumed to be

influenced and at least one that is not influenced by the local sources you have defined. The influence/not influence question is determined from the simulated concentrations. The average value of the non-influence monitor stations is used to create a background time series that will be subtracted from all monitor time series before they are used as input to the regression analysis. Select a constant emission rate for this exercise.

The statistical parameters **F-in** and **F-out**, are used by the regression model. See section *Exploiting the Regression Models in User's Reference Volume 3 Working with the Indico Presentation* for more information about the station database. Default values **Fin** = 0.05 and **Fout** = 0.10 are recommended for use with the regression model.

The emission estimate will be presented on the map, showing the specified area sources with their respective emission rate (g/s) followed by the standard error (the standard deviation of the estimated error). If you have chosen to show the description, the background value will also be displayed.

Hint: Repeat the emission estimation for different source locations/sizes, emission variation and/or the emission heights. A higher multiple correlation coefficient will indicate that your assumptions are improving. For the same number of sources, the individual standard error can also be used as a sign that your assumptions for each individual source are improving.

With the monitor option for background values, you will have both the average of the monitored time series as well as the background value that comes out of the regression.

Repeat the estimation but now using the **Monitor** option instead of **Constant** for the background levels. As before, use the result of the source location calculation as a background on which to base your estimate.

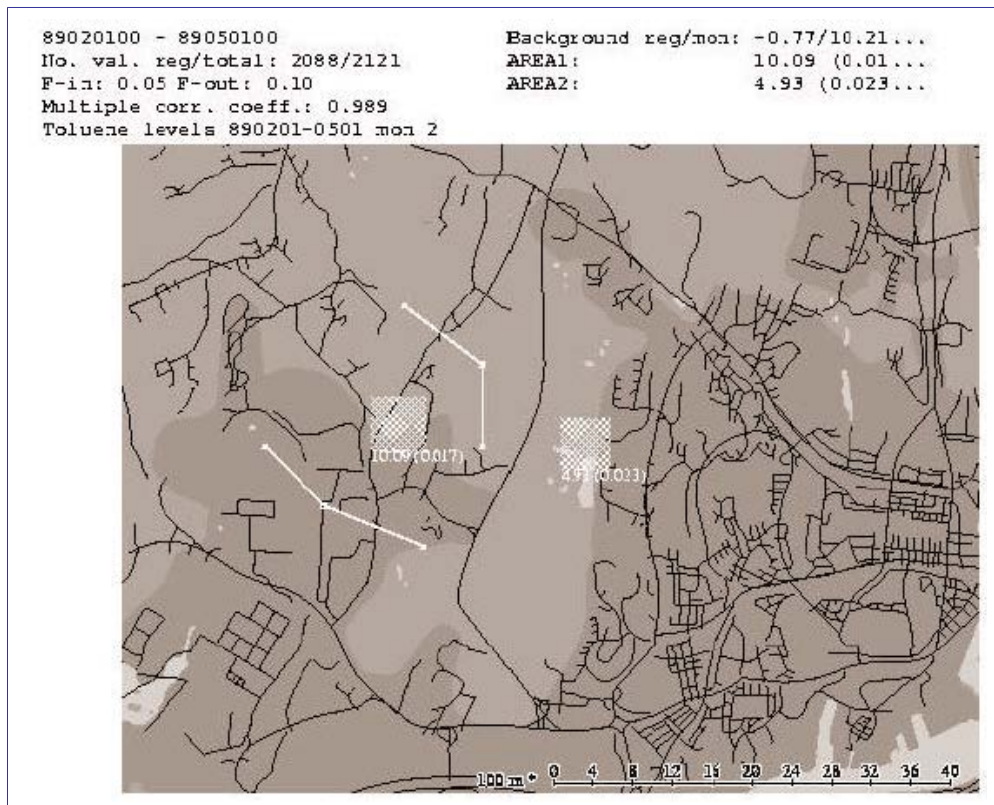


Figure 2.6.5.1. Result

Ticking off the **Description** option in **OUTPUT SETTINGS** further additional information will be displayed in the **VIEWER**:

The monitor option does not accept all time series data. The number of data values used out of the total available is given next to No. val. reg/total.

Beside Background reg/mon are two background values. The first is the background value detected by the regression analysis and the second is the average of the monitored time series.

Dates, statistical parameters, correlation coefficient and calculation description are also displayed along with the estimated emission and standard error for each area sources.

For more information on the monitor option See *Appendix 4B: A Method to Estimate Emissions from Dominating Sources*

2.7 Aermod Model

This is a steady-state plume model that incorporates air dispersion based on the planetary boundary layer turbulence structure and scaling concepts, including treatment of both surface and elevated sources, and both simple and complex terrain.

The **US EPA** document **AERMOD: DESCRIPTION OF MODEL FORMULATION. EPA-454/R-03-004 September 2004**, 91 pages, describes the topographical correction of the plume concentration, the formulas used for the plume rise (pp 51 – 53) and the segments used by the model to describe the influence of surrounding building for point sources.

The Aermod model can only use data from EDBs. If you want to make a simulation for a single source, then you have to create the source in an EDB. For point sources the **Aermod** sub window allows to specify the number of segments around the source and the characteristic of the surrounding buildings (width, height, etc)

This Manual contains an example that you can follow step by step. Begin by selecting the **Aermod model** option from the list in the **Model** sub-window. Then go to the **EDB** sub window and select the global EDB for the REF domain (the global EDB is usually the default EDB in Dispersion module).

In **Model Settings** See *Figure 2.7.1*. Select a time period from 900812 01 to 900812 03

The calculation **height**, is the level above ground where the model evaluates the concentration values, is 2 m by default,

Click on the **[APPLY]** button to save your settings

Settings

- Type**
 - Time period: [dropdown]
 - Mast: [Default] [dropdown]
- Format**
 - Format: [European] [dropdown]
- Period**
 - From: [900101 14] [calendar icon] [navigation icons] [150405 02] [calendar icon] [Present]
 - Year: [-] [Year] [+]
 - Month: [-] [Month] [+]
 - Week: [-] [Week] [+]
 - Day: [-] [Day] [+]
 - Hour: [-] [Hour] [+]
- Height [m]**
 - [2] [input field]
- Emission model**
 - Normal Resuspension
- Apply your choices**
 - [APPLY] [CHECK METED DATA AVAILABLE]

Figure 2.7.1. Aermod. Model Settings

In **EMISSIONS** you can use the [**Apply&show emissions**] button to check the availability of emissions for the calculation. See Figure 2.7.2. Emission.

Emission

- Name Information**
 - Name: [input field]
 - Info: [input field]
 - Info2: [input field]
- Substance**
 - [NOx] [dropdown]
- Sources**
 - Point Area Road Grid Ship Treat ships as points
- Restrictions**
 - [POINT, AREA & GRID] [ROAD] [SEARCHKEY] [SUB AREA] [CODES] [ADDITIONAL INFO]
- Correction**
 - Point: [1] [input field] Area & Grid: [1] [input field] Road: [1] [input field]
- Variation**
 - Scenario: [2007] [dropdown]
- Apply your choices**
 - [CLEAR ALL] [APPLY] [APPLY & SHOW EMISSIONS]

Figure 2.7.2. Aermod. Emissions.

Now you are ready to start the calculation. Go to **Calculate**, enter a meaningful description including date and times for the time period selected and start the calculation.

The dispersion result consists of several fields, one for each hour of the simulated period.

Once you have selected the result in the **RESULTS** window, go to **VIEWER** and use the [**<**] / [**>**] buttons to view the previous / next time frame. See *Figure 2.7.3. Viewer*. This calculation used the source Point_1_ExampleAermod.

To get a clear overview, go to **OUTPUT SETTINGS** and select the Isocolor option from the *Output style* list , B/W (black and white) for *Map* and *Intensity* 50. See *Figure 2.7.4 Output Settings*.

If you have not run the simulation yourself then you can find it under the user **ref** for the project Users Ref.

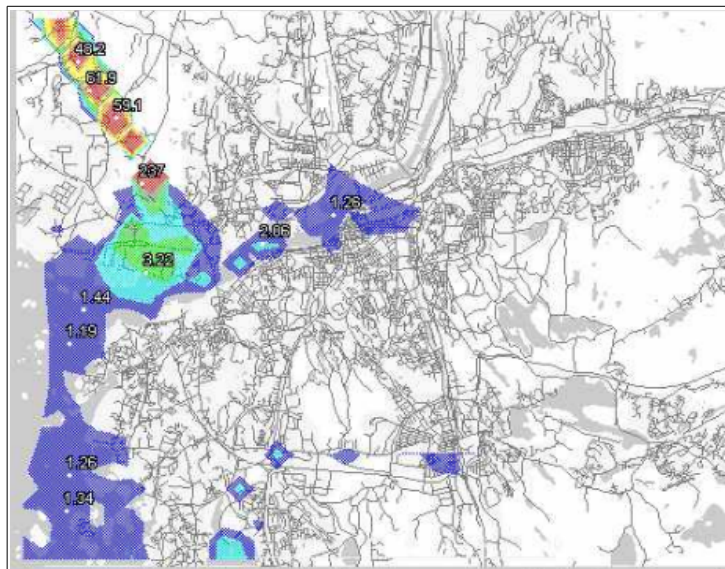


Figure 2.7.3. Viewer.

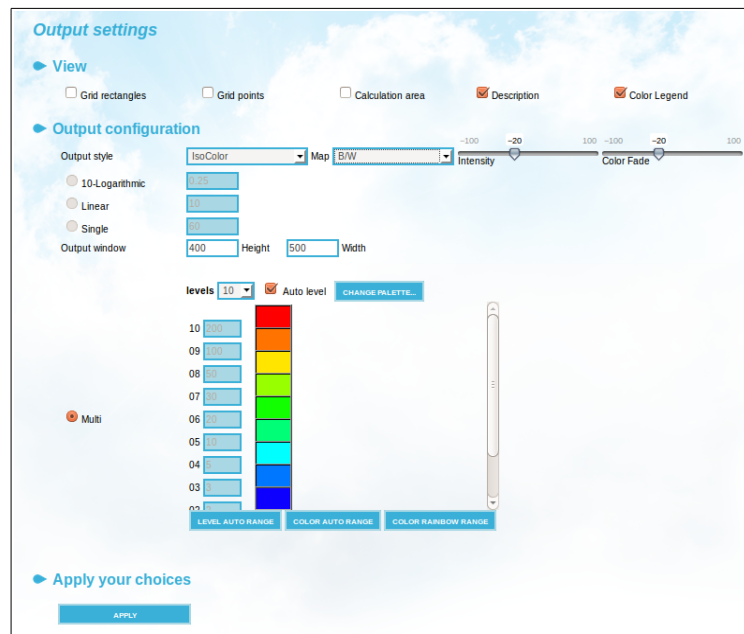


Figure 2.7.4. Output Settings.

2.8 OSPM Model

The OSPM street canyon model allows you to perform small scale simulations of situations at a particular street segment surrounded by buildings. A model description summary is presented in *Appendix 2J: OSPM Street Canyon Model*.

However, the Airviro Open Road model is applied to open streets without buildings.

Building heights are defined in the EDB module as road properties (**Roads - Height**)

Go to **MODELS** and select OSPM Street Canyon / Airviro Open Road (OS)

In **MODEL SETTINGS** . See *Figure 2.8.1*. Select a time period from 900812 01 to 900812 03.

This model does not use the calculation **height**, instead these are specified per road. Click on **[APPLY]** to save your settings.

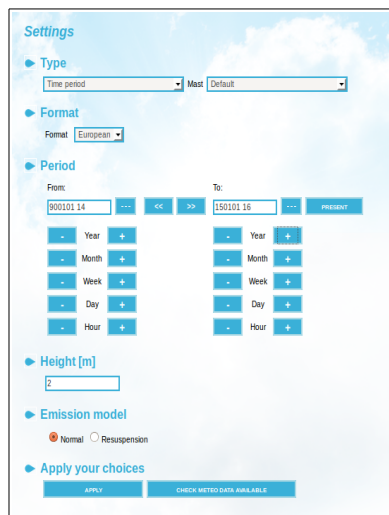


Figure 2.8.1. OSMP. Model Settings

Now click on **Area** in the main menu and select the calculation area and the grid . For this model, the grid is only used by the Danard Wind model to give the wind data for each street. *Figure 2.8.2*.

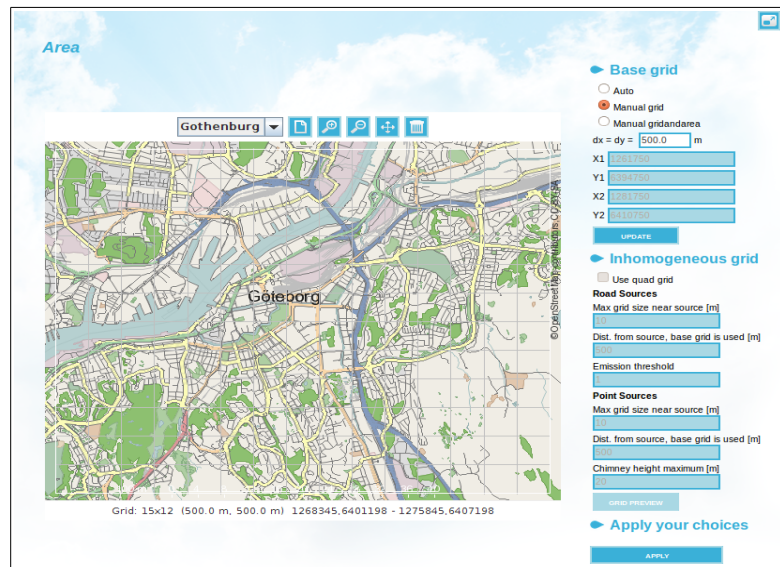


Figure 2.8.2. Area

In **EMISSIONS** a number of roads from the EDB can be picked out to perform an OS simulation for each of them (see Figure 2.8.3).

The following characteristics will apply to all roads selected for the simulation:

- **Receptor Height:** The receptor height in meter is required. Example: 2 meters.
- **Distance from (open) road:** In case the road is an open road (no surrounding buildings) the model requires the distance from the road to the receptor. Example: 3 meters from road. This distance is not used if the road is a street canyon.
- **Substance:** A substance for the calculation must be selected. Example: NO_x
- **ADT:** if checkbox [Select all sources] is ticked off, an ADT must be specified

To select the emission sources to be included in the calculation, place the mouse pointer over a source and press the left mouse button. A small window displaying the name of the road selected will pop up requesting to include it in the simulation, or to exclude it if you click on a source that has been previously included. Roads included in the simulation are highlighted on the map.

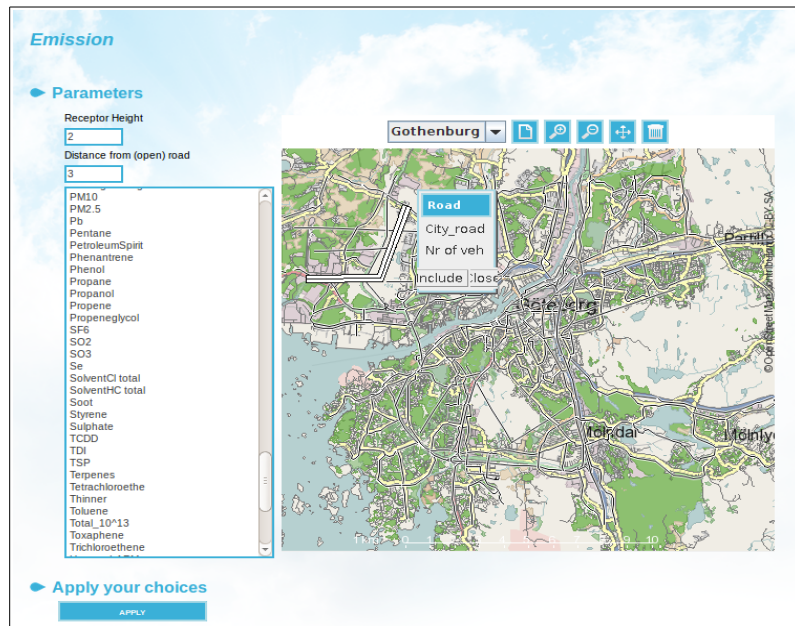


Figure 2.8.3. Emission. OS Model.

Now you are ready to start the calculation. Proceed in the same way as with the previous Models. Click on **CALCULATE**, enter a meaningful description including date and times for the time period selected and start the calculation.

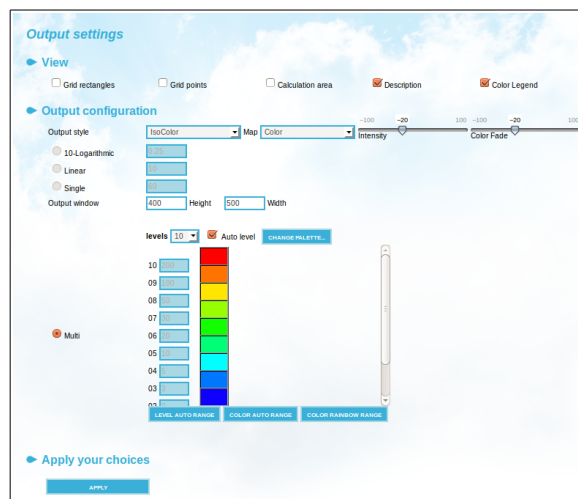


Figure 2.8.4. Output Settings

The dispersion result consists of several fields, one for each hour of the simulated period. Once you have select the result in the RESULTS window, go to VIEWER and use the [**<**] / [**>**] buttons to view the previous / next time frame. Select **Wind10m** to see how the wind

field changes from hour to hour, and if you look closely, you will see how it is affected by both topography and ground type.

Use the **OUTPUT SETTINGS** window to configure the options that best suit you for viewing the result. Set colour grid and tick off description for this example. See *Figure 2.8.4*.



Figure 2.8.5. Viewer. Example Open Road.

The **Viewer** window will display the source description. Local concentrations in $\mu\text{g}/\text{m}^3$ can be viewed by clicking on the map. These are given for both sides of the road e.g, E means concentration on the east side of the road and W means the concentration on the west side.

The results will be displayed as in *Figure 2.8.5*. Note the pollution levels on the roads through the time.

Appendix 2A: The Wind Model - Calculation of the Wind Fields

2A.1 Principles of Wind Field Calculations

The wind field calculation is based on the concept first described by Danard (1976), where mesoscale winds are generated by using:

- horizontal momentum equation
- pressure tendency equation
- first thermodynamic equation

This concept assumes that small-scale winds can be seen as a local adaptation of large scale winds (free winds) due to local fluxes of heat and momentum from the sea or earth surface. Any non-linear interaction between the scales is neglected. Danard assumes that the adaptation process is very fast, 1.5 hours for model resolutions of 10*10 km. It is also assumed that horizontal processes can be described by non-linear equations while the vertical processes can be parametrised as linear functions.

The large-scale winds as well as vertical fluxes of momentum and temperature are estimated from profile measurements in one or several meteorological masts (called principal masts). When the topography is relatively smooth, without dominating ridges or valleys, the free wind is assumed to be horizontally uniform.

For deep bi-directional valleys, this is not possible. A channel flow approach has been used to include the governing effects on the free winds in deep valleys.

For larger areas or for areas with complex meteorological situations (such as sea breeze) information from more than one meteorological station is needed. The data from these stations will then pass through a mesoscale interpolation before entering the wind model.

2A.1.1 Wind Model Equations

The equations below are solved in a terrain-following coordinate system (s-coordinates):

$$\frac{\partial V_s}{\partial t} = -V_s \cdot \nabla V_s - (g \nabla Z_s + RT_s \nabla \ln p_s) - fK \times V + F + K_m \nabla^2 V \quad (\text{eq 2A.1.1})$$

$$\frac{\partial \Theta_s}{\partial t} = -V \cdot \nabla \Theta_s + K_t \nabla^2 \Theta_s + Q/C_p \quad (\text{eq 2A.1.2})$$

$$\frac{\partial p_s}{\partial t} = \frac{g}{R \Theta_s T_s} \int_0^H \frac{\partial \Theta}{\partial t} dz \quad (\text{eq 2A.1.3})$$

V = horizontal wind

$$\Theta = T_s \left(\frac{P_0}{P_s} \right)^{\frac{R}{C_p}} = \text{potential temperature}$$

$$F = c \cdot C_D \cdot \frac{V^2}{H} = \text{friction, the drag coefficient: } C_D = \left[\frac{\kappa}{\ln \left(\frac{z}{z_0} \right)} \right]^2$$

p = air pressure

H = boundary layer height

(all variables with index s refer to the surface, i.e. 10m above ground)

2A.1.2 Initialization

Boundary scaling parameters are determined from one or several profile measurements in the area, giving estimates of boundary layer heights (H), diabatic heating (Q) and potential temperature distribution at ground level (Θ_s). Physiographical information (surface characteristics) is used for area interpolation of H , Q and Θ_s . A free wind, i.e. an estimate of a wind at the location of the mast, at the level (H) where the wind is not affected by surface fluxes of heat and momentum, is estimated based on the profile measurements and extrapolation procedures suggested by Holtslag (1984). The free wind field is

estimated according to one of the methods suggested in *2A.1 Principles of Wind Field Calculations*.

When the free wind field is estimated, the initial surface pressure field is determined in accordance with a geostrophic balance. The initial wind field at the surface is estimated by running the first equation (eq 2A.1.1) until two successive estimations of the average wind component do not differ by more than 2%.

2A.1.3 Stability and Turbulence Estimation - Preprocessing of Meteorological Data

Stability and turbulence conditions in the boundary layer are evaluated at the locations of the principal masts, i.e. where information about both the temperature gradient and the wind speed at one or two levels is available. These data are used to calculate the boundary layer scaling parameters.

The two most important parameters for the scaling of the atmospheric stability and turbulence in the surface boundary layer are the vertical heat flux H and the friction velocity u_* . From these, a characteristic length scale - the Monin-Obukhov length - can be calculated:

$$L = \frac{\bar{T}}{g} \cdot \frac{u_*^3 \rho c_p}{\kappa H}$$

where \bar{T} is the average surface air temperature, g is gravity, κ is the von Kármán constant and ρc_p is the specific heat capacity. In the Airviro dispersion models, the Monin-Obukhov length is extensively used as a discriminator of different meteorological regimes.

The determination of the Monin-Obukhov length (L) follows the profile method discussed by Berkowicz and Prahm (1982). With measured values of potential temperature difference (Δq) close to the ground and wind speed (U) at one or two levels, L is calculated with an iterative method. The local value of the ground roughness length z_0 must also be estimated (see tabulated values in textbooks, i.e. Panofsky and Dutton (1984), Table 5.1 and 5.2). For the case that two temperatures and one or two velocities are available, it is

possible to calculate two other scaling variables u_* (friction velocity) and T_* (temperature scale):

$$u_i = \frac{\kappa(U_{z2} - U_{z1})}{\ln\left(\frac{z_{u2}}{z_{u1}} - \Psi_{m2} + \Psi_{m1}\right)}$$

$$T_i = \frac{H}{\rho c_p u_i} = \frac{\kappa \Delta \theta}{\ln\left(\frac{z_{t2}}{z_{t1}} - \Psi_{H2} + \Psi_{H1}\right)}$$

The measurement heights are z_{u2} , z_{u1} , z_{t2} and z_{t1} . If wind is measured at only one level (z_{u2}), z_{u1} is set equal to z_0 and U_{z1} to 0.

The similarity functions Ψ_m and Ψ_H are functions of (z/L) . For stable conditions ($L > 0$) we can use (Dyer, 1974):

$$\Psi_m = -5 \cdot \frac{z}{L}$$

$$\Psi_H = -5 \cdot \frac{z}{L}$$

For unstable conditions ($L < 0$) Paulson (1970) proposes

$$\Psi_m = \ln\left[\left(\frac{1+x^2}{2}\right)\left(\frac{1+x}{2}\right)^2\right] - 2 \arctan(x) + \frac{\pi}{2}$$

where

$$x = \left(\frac{1-16z}{L}\right)^{\frac{1}{4}}$$

and

$$\Psi_H = 2 \ln \left[\frac{1}{2} \left(1 + \sqrt{1 - \frac{16z}{L}} \right) \right]$$

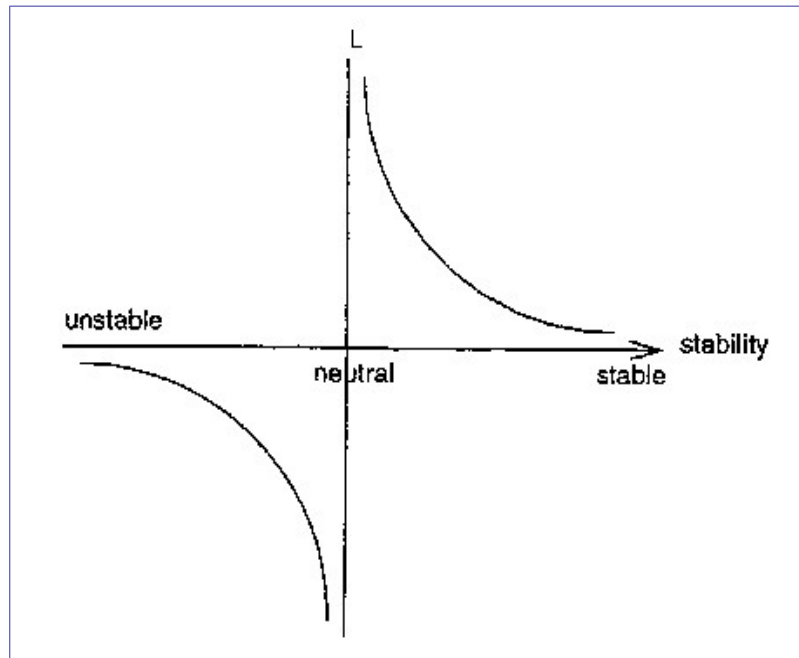


Figure 2A.1 The Monin-Obukhov length (L) behaviour as a function of stability

The calculated Monin-Obukhov length (L) is further used to classify the stability conditions. The depth of the boundary layer (Z_i) is a key parameter, since the pollutants released close to ground will be more or less trapped inside this layer.

For stable conditions ($L > 0$) the Zilitinkevich (1972) estimation is used:

$$Z_i = 0.4 \sqrt{\frac{u_*}{f}} L$$

Note that this estimate relies on the assumption that the inversion is caused by local cooling. It will not serve for cases with (relatively high) inversions that are caused by advective air masses.

The corresponding expression for neutral and unstable conditions ($L < 0$) is (Panofsky and Dutton, 1984):

$$Z_i = 0.3 \left(\frac{u_c}{f} \right)$$

In practice the mixing height (mixh) often cannot be set equal to the boundary layer height (Z_i). Here the relationship between these two heights is given in a resource file, by default

$$\text{mixh} = 50 + 10 \times \sqrt{Z_i + 1}$$

Note that mixh varies with latitude.

This empirical relationship was obtained from a data set for Göteborg (see Indic, 1990).

It is possible as an option to set Z_i equal to mixh.

Direct measurements of velocity fluctuations may be used to evaluate the dispersion coefficients σ_y and σ_z that regulate lateral and vertical diffusion of the Gaussian plume. The measured quantities are the standard deviation of wind direction (σ_α) and vertical wind speed (σ_ω). The technique suggested by Draxler (1976) is as follows:

$$\sigma_y = \sigma_\alpha X \left[\frac{1}{1 + 0.9 \sqrt{\left(\frac{X}{UT_i} \right)}} \right]$$

where T_i is a Lagrangean time scale (see Table below for values recommended by Draxler). X is downstream distance (from source), U is horizontal wind velocity and σ_α the standard deviation of the horizontal wind direction.

A similar expression is used for the vertical coefficient:

$$\sigma_z = \frac{\sigma_\omega}{U} X \left[\frac{1}{1 + 0.9 \sqrt{\left(\frac{X}{UT_i} \right)}} \right]$$

where σ_ω is the standard deviation of the vertical wind speed.

Figure 2A.2 Recommended values of T_i according to Draxler (1976). Sources higher than

30 m is considered as elevated.

	Surface Sources		Elevated Sources	
	Stable	Unstable	stable	unstable
Lateral	300	300	1000	1000
Vertical	50	100	100	500

Note that all calculated meteorological parameters should be valid for simulation of one-hour values.

2A.1.4 Mesoscale Interpolation

Influence areas

Each grid point belongs to an influence type area and within each area the meteorological conditions are considered as similar. The influence areas can be for example sea, sea close to land, land, land close to sea, urban, suburban, and so on. There can be several different areas of the same kind, for instance two separate valleys. These influence areas must exist in a resource file with names and type numbers.

Observation places

Each observation place must be classified with an influence type number. The influence from every observation place on different areas must be stated with a real number $a_k(n)$, in the interval $[0,1]$, where k is an index for the observation and n the area type number.

Analysis method

Each grid point should be classified with an influence type number, say nn (see appendix D3.3 in Airviro Specification Part II). For every grid point (i,j) a value is interpolated from all observation places:

$$value(i, j) = \frac{\sum_k obs(k) \times weight(i, j, k)}{\sum_k weight(i, j, k)}$$

where $weight(i, j, k) = \frac{\alpha_k(nn)}{d_k(i, j)}$

$d_k(i, j)$ = distance between the grid point (i,j) and the observation place k.

The resulting grid will thus be affected by the observations and to some extent also by different influence type areas.

L, u*, T*, w*, Zi and mixh

The Monin-Obukhov length (L), friction velocity (u*), temperature scale (T*), convective velocity scale (w*), boundary layer (Zi) and mixing height (mixh) are treated in much the same way.

The input data is taken from the pre-processor: L, u*, T*, w* and Zi. It is also possible for Zi to be measured directly, for instance from a SODAR.

If no principal station data exists within a zoomed area then data from the nearest principal mast is used as a constant value over the whole area.

The *heat island effect* is taken into account by adjusting stable conditions to neutral conditions over urban areas. The areas of distribution for which this correction is to be applied is defined in a grid - see appendix D3.3 in Airviro Specification Part II.

The location of this effect is given in a grid, see appendix D3.3 in Airviro Specification Part II.

Surface temperature

The input data are temperature observations from all places with at least two sensors in the vertical and in addition an analysed grid of L, u*, T* and Zi from above.

With this information a surface temperature T_s and T_{init} , which assumes a surface zero heat flux flow, is computed for every observation.

If data are missing within a zoomed area the main mast is used as a constant value over the whole area.

Free wind

Input data are all wind observations and the analysed grid of L.

The wind speed is first extrapolated to 150 m with help of a power law and the wind direction is twisted according to empirical data (Holtslag).

If there is no principal mast data available within a zoomed area then data from the nearest principal mast is used as a constant value over the whole area.

2A.1.5 Procedures for Solving the Wind Model Equations

When the initial conditions have been estimated, the thermodynamic equation (eq 2A.1.2) is used to estimate the local change in potential temperature. The tendency equation (eq 2A.1.3) is then utilised in order to estimate the local pressure tendency due to temperature effects. The change in local pressure at ground level is then used in the momentum equation (eq 2A.1.1) to estimate a wind tendency. Thereafter, the slightly changed wind field is used to estimate the tendency in the surface potential temperature and so on.

By iterating in this way, a quasi-steady-state condition will usually be reached within a few time steps (10-20). According to the initial assumptions, the length of the time steps is not allowed to be larger than a few seconds, in order to have an adaption process within a few minutes. The Danard concept is to some extent a contradiction. The concept is a diagnostic model, trying to identify the small scale variations in the wind field due to the forcing at the surface. To do this, prognostic equations are applied, but for periods so short that the large scale transient effects such as wind rotation due to the Coriolis force are filtered out. Consequently, it is not possible to describe the evolution of a sea breeze, but to diagnose the sea breeze if relevant input data (mast data) is present.

It is worthwhile observing that the model is not mass conservative.

2A.1.6 Numerical Methods

Generally, centred time and space differences have been used, i.e. the so-called leapfrog method. An upstream formula has been used for the advection terms.

The length of the time steps (Δt) depends upon the resolution of the grid (Δs) and the wind speed of the large scale wind (V_g), i.e.

$$\Delta t = \frac{(0.125 \times \Delta s)}{V_g}$$

Normally, for a 250*250 m resolution and a typical large-scale wind of 5 m/s, the length of the time steps would be 6 seconds. The total number of time steps has a maximum of 40. Consequently, the total adaption period in this case is 4 minutes.

2A.1.7 Reference Literature

Readers that are interested in further theories and experience with the Danard model are referred to: Danard (1977), Mass (1981), Mass (1984), Alpert and Getenio (1988). Numerical methods can be found in: Richtmeyer and Morton (1967), Messinger and Arakawa (1976).

Appendix 2B: Results from Verification Studies

During the period 25 January to 25 March 1990 a wind experiment was performed in the Göteborg area. A total of 18 wind monitoring stations were located in the area. At these locations wind speed and direction were measured at approximately 10 m from ground level. Data was collected as 15 minute average winds once per hour during the period. In addition, at one location (Jarnbrott) a 130 m mast was equipped with wind and temperature sensors at 4 levels (2m,10m,60m and 130m). The size of the area was 20*16 km and the locations of the stations can be seen in Figure 2B.1 (the Jarnbrott station is station no. 01).

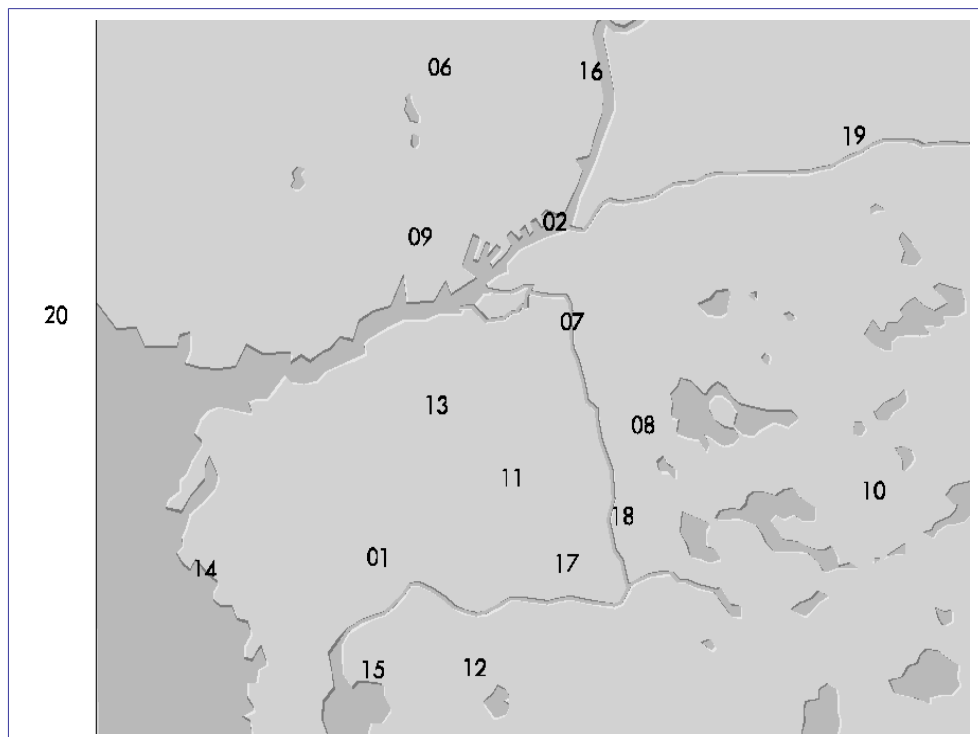


Figure 2B.1

The area was digitised as a grid with a resolution of 500*500m. Input data for the estimation of boundary scaling parameters, mixing height, large scale winds, etc. was taken from the profile measurements at Jarnbrott.

In *Figure 2B.2* a scatter diagram is presented showing the measured winds compared to

the modelled winds at all locations during the two-month period.

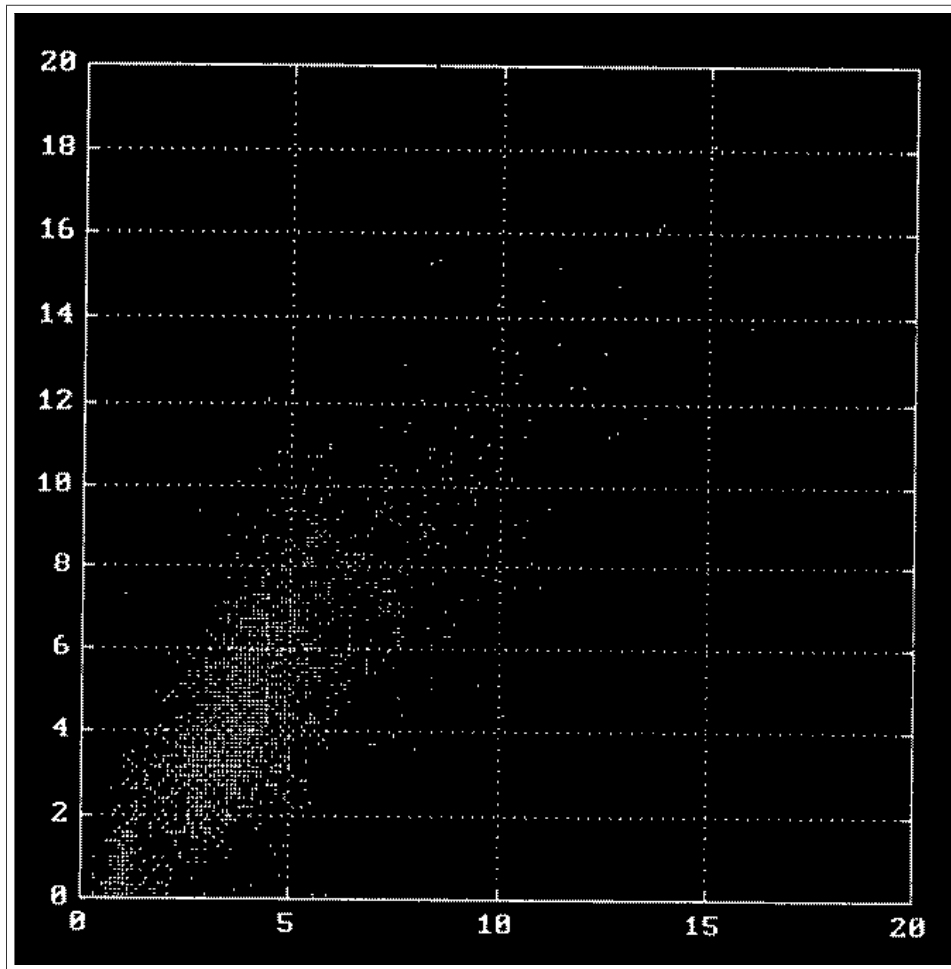


Figure 2B.2

In *Figure 2B.3* the very complex situation from March 3 at 05.00 is presented. The temperature and wind profiles taken from the mast at Jarnbrott are shown in *Figure 2B.3* below. A strong surface temperature inversion had developing during the night, and the upper wind at 130m (wind speed: 6 m/s) cannot penetrate through the inversion layer. The 10 metre winds are weak and are mainly driven by the fluxes from the ground, and the wind pattern in the area is extremely complex. This is a typical situation during winter time when severe pollutant episodes occur.

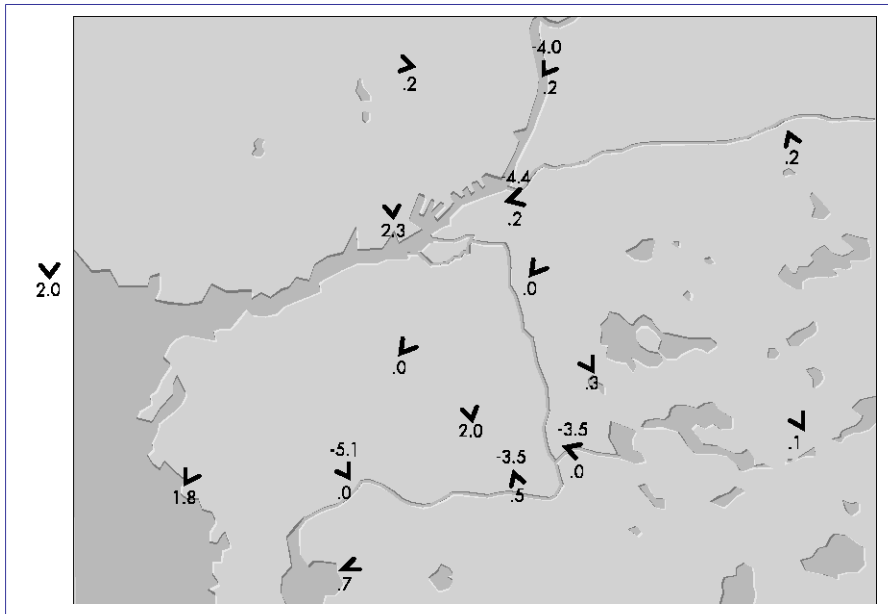


Figure 2B.3

In *Figure 2B.4* the model simulation is presented. Although, the resolution is low (500*500m) it is clearly shown that the main part of the local wind pattern is described.

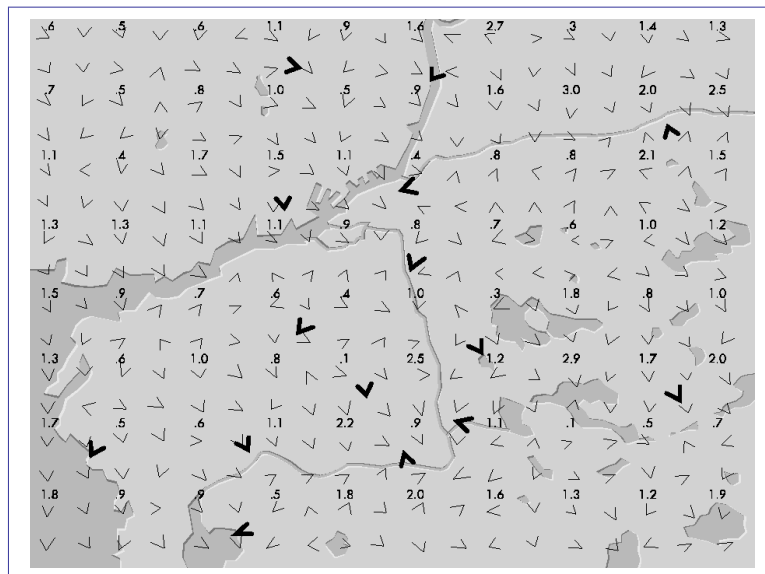


Figure 2B.4

Appendix 2C: The Gauss Dispersion Model

2C.1 Introduction

This section describes the details of the dispersion calculation. The user can choose to use it during different weather conditions. The Reference database includes

- Winter season: A statistical sample of meteorological data between 1st October and 31st March is extracted from the time series database (see *Appendix 2G: Scenario Calculations*)
- Summer season: A statistical sample of meteorological data between 1st April and 31st September is extracted from the time series database (see *Appendix 2G: Scenario Calculations*)
- Case: A homogeneous wind field is used that is not based on any real data. Instead the user can enter the wind speed and direction to use as well as stability conditions
- Specific hour: The weather data for a specific date and time is fetched from the time series database

When using a scenario (season) or specific hour the wind field for the whole area is calculated for the whole map area when you start the dispersion calculation. For this reason you cannot view the wind field until the calculation is finished.

2C.2 When to Use the Gauss Plume Model

This model is used to simulate the distribution of ground concentrations of pollutants over urban or industrial areas with a typical scale of one or a few tenths of kilometres. The size of the application area is limited from below by the fact that the Gaussian model coefficients are not valid close to the source (distances below approximately one hundred metres). The upper limit is given by the fact that more or less stationary conditions should prevail for the time it takes for an air parcel to be advected through the area. One hour

mean values are simulated, as it is known that the wind may be more or less constant during such a period (daily averages would not be sufficient). With wind speeds of 2 - 5 m/s, an air parcel would travel 7 - 18 km within one hour. This limitation of the Gaussian model should be kept in mind while performing simulations on scales larger than 20 km.

In order to avoid using the model on too large areas, producing unrealistically long plumes, the Gauss model plume length is limited. The maximum plume length is defined in a resource file and depends upon the actual stability and the persistence of different weather conditions.

The Gaussian dispersion model does not resolve individual buildings. Instead, surface structures like houses and trees enter the model through local roughness values and, as a consequence, through the wind field (see *Appendix 2A:*). Buildings give a more rough surface, more friction means a lower wind speed which influences the dispersion, and so on.

The default setting for the calculation height is 2 m, but the user may specify any other value. The height at which the simulated concentration values are valid will coincide with the calculation height above ground in open areas (i.e. in the countryside). Over a city area, the simulation will reflect the concentrations at the specified height above the roof height. The default setting will thus give concentrations at roof height, not the concentration at street level. The blocking effects of buildings surrounding a street which lead to raised concentration levels can be studied separately (see *Appendix 2E: Street Canyon Model*).

Note that the Gaussian dispersion model simulates steady states of pollution concentrations. Accumulation of polluted air throughout various days can not be studied with this model.

These are the restrictions of the Lagrangean/Gaussian model, but there are advantages to be gained from using it too. State-of-the-art articles (e.g. Hanna et al, 1982) mention, among other things, the following points:

- It produces results that agree with experimental data just as well as any other model.

- It is fairly easy to perform mathematical operations on this equation.
- It is appealing conceptually.
- It is consistent with the random nature of turbulence.

Usually Gaussian plume models are applied to a horizontally homogeneous wind field, while the model in the Airviro system will take into account a realistic wind field. This means that the effect of topography, roughness distribution and horizontal variations in surface heating/cooling will enter the dispersion calculation.

Clearly there are important cases where this model is not an ideal choice such as when the accumulation of pollutants is of interest and a time sequence must be simulated, or when the horizontal scales grow from an urban/local to a regional scale. Neither will dispersion in an application area with complicated topography - e.g. a narrow and deep valley - be satisfactorily treated with the Gaussian approach. For these cases, the Airviro user should choose the advection/diffusion Grid model.

2C.2.1 Source Type Definitions

The source emission data may be described as coming from a point, a line or an area or from the EDB. From a model point of view, these are all treated as point emissions (except EDB sources): A line (normally a road) will be treated as a number of points distributed along a line segment, and an area as a number of points equally distributed over a horizontal rectangle.

If the emission input data is obtained from the EDB, then the area, line and grid layers are collected together on an emission grid at ground level.

A source specified as a point emitter can take into account an initial plume rise calculation, relevant for chimney stacks. Area sources assume a dispersion height prescribed by the user, while the line source is assumed to be identical with traffic emissions from roads (at ground level).

2C.3 Plume Rise Formulation

The plume rise calculation closely follows the recommendation made by Hanna et al. (1982) (if no other reference is given).

The release height, i.e. stack height, is denoted by h_s . Due to momentum (exhaust gas velocity) and buoyancy (exhaust gas temperature excess over the ambient temperature), the plume will rise considerably over the release height. Close to the source (typically within a horizontal distance of a few hundred metres), the rise and curve of the plume is more or less independent of turbulence and stratification in the ambient air. Later the atmospheric conditions become more important, and the final plume height is determined by the stability conditions.

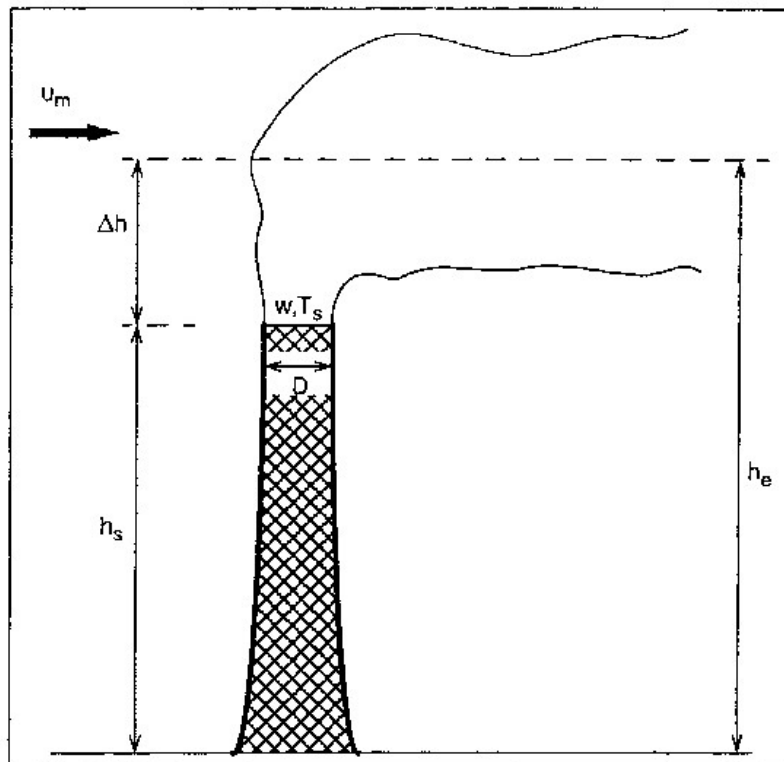


Figure 2C.1 Explanation of the different heights relevant for the plume rise calculation

The initial plume path height may be described as a function of horizontal distance from the source (X):

$$h_p = h_s + \left(\frac{3M_0}{\alpha^2 u_s^2} X + \frac{3F_0}{2\beta^2 u_s^2} X^2 \right)^{\frac{1}{3}}$$

where M_0 is the initial momentum flux and F_0 the initial buoyancy flux. Note the convention of the way the initial exhaust gas volume flux is defined: $V_0 = w$ (internal stack radius)², which means that a factor π is left out. Hence we get

$$M_0 = wV_0$$

$$F_0 = \frac{g}{T_s} (T_s - T_a) V_0$$

where w is the (vertical) exhaust gas velocity, g is the gravitational acceleration, T_s is the exhaust gas temperature and T_a the air temperature (in °C).

The wind speed at stack height is denoted u_s . The empirical coefficients α and β are set to:

$$\alpha = 0.4 + \frac{u_s}{w} \quad \text{and} \quad \beta = 0.4$$

Here initial values of both momentum and buoyancy are used. When the plume mixes with ambient air, its lifting potential will decrease and it will reach the final plume height somewhat later than what the curve expression above states.

The final plume height ($h_e = h_s + \Delta h$) depends on stability conditions. For stable situations ($0 < L < 100$) the so called bent-over plumes are to be expected:

$$\Delta h = 2.6 \left(\frac{F_0}{u_m s} \right)^{\frac{1}{3}}$$

where s describes the atmospheric stability:

$$s = \frac{g}{T_a} \left(\frac{\Delta T_a}{\Delta z} + 0.01 \right)$$

The wind speed (u_m) - an average between stack height and final plume height - is estimated iteratively. The power law is used for extrapolating the wind velocity from the 10 m wind field value, using a power $p = 0.55$.

For very low wind speeds, the expression above may yield unrealistically large plume heights, hence an upper limit is calculated as (originally from Briggs, recommended by Omstedt, 1984):

$$\Delta h = 5 \frac{F_0^{0.24}}{s^{0.375}}$$

$$h_{e''} = h_s + \Delta h$$

where $h_{e''}$ is the intermediate final plume height.

The calculated plume height might be higher than the mixed layer depth $mixh$, i.e. the plume penetrates the inversion. For a calculated plume height ($h_{e''}$) within 2 times the mixed layer depth, a fraction ($frac$) of the plume is reflected back below the inversion:

$$frac = \frac{mixh}{h_{e''}} - 0.5$$

Weil and Brower (1982) suggests this fraction should be treated as a plume that is dispersed within the mixed layer at a height (Δh) above the stack height (h_s). The calculated plume height then becomes:

$$h_e' = h_s + (0.62 + 0.38(1.0 - \text{frac}))(h_e'' - h_s)$$

The fraction that remains above the inversion will not influence ground pollution concentrations.

The final plume rise for unstable situations ($-68 < L < 0$) is calculated with the break-up model. The final rise is not limited by any inversion, instead its buoyancy is diluted by ambient turbulence until it stops rising (originally from Briggs, recommended by Omstedt, 1984):

$$\Delta h = 4.3 \left(\frac{F_0}{u_m} \right)^{0.6} H_i^{-0.4}$$

with

$$H_i = \frac{gH}{c_p \rho T_a}$$

and u_m as the mean wind speed between stack height and plume height. As before the vertical wind profile is extrapolated with the power law, here with $p = 0.15$.

As an upper limit of the plume rise during unstable conditions, we use an expression suggested by Weil (1979):

$$\Delta h = 0.62(\text{mixh} - h_s)$$

where, as before, h is the mixed layer depth and h_s the stack height. The plume height in this case is:

$$h_e' = h_s + \Delta h$$

There remain the neutral or near neutral cases ($L > 100$ or $L < -68$), when stronger wind speeds may be expected. For this stability class, the plume rise may be influenced by

downdraft (effects of lee wake of the stack or of nearby buildings). The plume rise will also be lowered due to entrainment caused by downwind turbulence created by the stack (or the plume jet itself). According to Nielsen et al. (1986):

$$\Delta h = 1.54 \left(\frac{F_0}{u_p \cdot u_*^2} \right)^{\frac{2}{3}} (h_s + \Delta h_d)^{\frac{1}{3}} + \Delta h_e + \Delta h_m$$

where u_p - the wind speed at plume height - is extrapolated with a power $p = 0.25$, and u_* is the friction velocity.

The plume downdraft Δh_d is:

$$\Delta h_d = -2 \left(\frac{w}{u_s} - 1.5 \right) D$$

for wind speeds $u_s > 1.5w$, D being the outer diameter of the stack. The plume entrainment is calculated as

$$\Delta h_e = -0.25 \sqrt{DX_i}$$

with $X_i = 6.49 F_0^{\frac{2}{5}} \Delta h_d^{\frac{3}{5}}$ (Nielsen et al, 1986). The initial momentum contribution is

$$\Delta h_m = 3D \left(\frac{w}{u_s} - 1 \right)$$

and

$$h_e' = h_s + \Delta h$$

For all stabilities, the plume height to be used is then the minimum value of the plume path h_p (close to the stack) expression and the final plume height estimate h_e' , that is:

$$h_e = \min(h_p, h_e').$$

2C.4 Building Downwash

So far it has been assumed that the plume is not influenced by nearby buildings or other obstacles. We repeat that the downdraft Δh_d calculated earlier was due to a pressure drop created by the stack itself.

Concerning the building downwash, Briggs (1973) has given some recommendations that are found in most textbooks, e.g. in Lyons and Scott (1990). These recommendations have been followed:

We consider only *neutral* or *near neutral* cases ($L > 100$ or $L < -68$), i.e. downwash will not occur for stable or unstable conditions.

Define a typical height H_b and a typical width W_b of the building/buildings near to - or under - the stack. What is to be considered as "near" is within a radius of about twice the stack height. For buildings with asymmetric dimensions in the horizontal, an equivalent "width" may be estimated as

$$W_b = \sqrt{\text{length} \cdot \text{width}}$$

(actually it is the crosswind width that is of importance, but we simplify this to a typical width). The relevant length scale (ξ) to estimate the downwash effects of the building is the minimum of the height and width scales

$$\xi = \min(H_b, W_b)$$

There are 3 regimes where the buildings influence the final plume height h_e :

a) The case when the plume is totally out of the building influence

$$h_s + \Delta h_d > H_b + 1.5\xi \Rightarrow h_e = h_e' \text{ (from above, } \min(h_p, h_e') \text{)}$$

b) The case when the plume is within the influence area but still higher than the building:

$$H_b + 1.5\xi > h_s + \Delta h_d > H_b \Rightarrow h_e = 2(h_s + \Delta h_d) - (H_b + 1.5\xi) \quad (1)$$

c) The case when the plume is below the building height:

$$h_s + \Delta h_d < H_b \Rightarrow h_e = h_s + \Delta h_d - 1.5\xi \quad (2)$$

The plume is considered to be trapped within the cavity if the calculated h_e is less than 0.5ξ , and the final plume height will then be set to $0.5H_b$. The lateral spreading will be proportional to ξ (think of the plume as being spread out over the lee wall of the building).

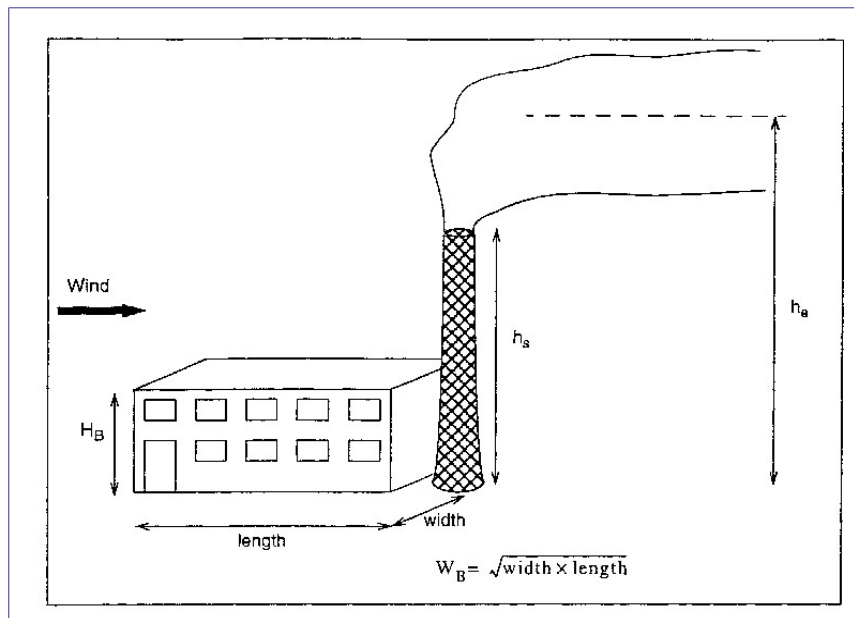


Figure 2C.4 Parameters of interest in the downwash calculation. Find $\xi = \min(H_B, W_B)$.

¹ It is important to note that the calculation of these h_e 's does not take the initial thermal and momentum lift into account. The transition from a plume height affected by thermal and momentum lift to a moderate plume height affected by buildings can therefore cause drastic changes in the calculated concentration field.

² It is important to note that the calculation of these h_e 's does not take the initial thermal and momentum lift into account. The transition from a plume height affected by thermal and momentum lift to a moderate plume height affected by buildings can therefore cause drastic changes in the calculated concentration field.

2C.5 The Gaussian Plume Model

If we deal with a point, road or area emission, the model will treat this as one or several point emissions, to be dispersed at a certain plume height. Depending on the emission characteristics, we estimate an initial distribution of the pollutant defined by the initial (at $X = 0$) values of σ_y and σ_z :

For point sources introduced either as individual point sources or through the EDB, σ_y and σ_z at

$X = 0$ are set equal to external stack radius.

For area sources introduced individually the initial σ_y is set to where

$$dxy = \sqrt{\text{width}^2 + \text{length}^2}$$

σ_z is defined with an empirical formula for neutral conditions, $\sigma_z = 0.22 \times dxy^{0.78}$.

For road sources introduced individually, σ_y is set to $0.5 \times \Delta xy_{em}$ and in open areas σ_z is set to $0.53U$.

However, in built up areas where the Gauss model calculations are evaluated at roof level, σ_y is set to $0.667 \times \text{building height}$. The building heights are specified in a grid, see appendix D3.3 in Airviro Specification Part II.

For area sources, road sources and grid layers specified in the EDB the emissions will enter the model in the form of an emission grid with grid size Δxy_{em} . The corresponding values of σ_y will be $0.5 \times \Delta xy_{em}$ and σ_z is set to $\max[(0.53U), (0.667 \times \text{building height})]$.

The advection of the polluted air will follow trajectories in the wind field. For each emission point source, such a trajectory will constitute the plume centre line, i.e. a transformation to a Lagrangean coordinate system is taken place. The Gaussian plume equation is solved along the trajectories (the standard plume model may be found in any textbook on

atmospheric dispersion, e.g. Hanna et al., 1982):

$$C = \frac{Q}{2\pi\sigma_y\sigma_z U} e^{-\frac{y^2}{2\sigma_y^2}} \left[e^{-\frac{(z-h_e)^2}{2\sigma_z^2}} + e^{-\frac{(z+h_e)^2}{2\sigma_z^2}} + e^{-\frac{(z+h_e-2h)^2}{2\sigma_z^2}} \right]$$

where the second and third exponential term within the brackets are due to reflection at the ground and top of mixed layer, respectively. For low inversions, there will be repeated reflections between the ground and the top mixed layer. The desired ground concentrations are found by setting $z = 2m$. The contribution of each individual (trajectory) plume is summed up on a calculation grid, where the grid concentrations are the true contributions over each grid area, according to the curvature of the plume concentration distribution as a function of the distance from the plume centreline.

Appendix 2D: The Eulerian Advection-Diffusion Grid Model

2D.1 Introduction

The grid dispersion model built into the Airviro system is especially designed to meet the requirements given by areas with a complicated topography, e.g. valleys. The model has some additional advantages compared to the standard Gaussian dispersion approach:

- It allows transient simulations and may simulate accumulation of pollutants during calm wind events.
- Boundary conditions (such as deposition) as well as particle characteristics (such as settling) may be introduced

The Grid model does not resolve individual buildings. Instead, surface structures like houses and trees enter the model through local roughness values and, as a consequence, through the wind field (see *Appendix 2A*). Buildings give a more rough surface, more friction means a lower wind speed which influences the dispersion, and so on.

The default setting for the calculation height is 2 m, but the user may specify any other value. The height at which the simulated concentration values are valid will coincide with the calculation height above ground in open areas (i.e. in the countryside). Over a city area, the simulation will reflect the concentrations at the specified height above the roof height. The default setting will thus give concentrations at roof height, not the concentration at street level. The blocking effects of buildings surrounding a street which lead to raised concentration levels can be studied separately (see *Appendix 2E: Street Canyon Model*).

2D.2 Formulation of the Grid Model

The model is based on the three-dimensional advection-diffusion equation for a certain substance. The substance could be one passive substance, a scalar, a chemically active substance or a suspension of non-interacting particles with or without deposition properties. The model for a chemically active substance includes in principle an advection-

diffusion equation for each of the interacting "mother" and "daughter" substances. The model equation (see for example Pielke (1984)) is:

$$\frac{\partial c}{\partial t} + u \frac{\partial c}{\partial x} + v \frac{\partial c}{\partial y} + w \frac{\partial c}{\partial z} = \frac{1}{\rho} \frac{\partial}{\partial z} \left[\rho \left(K_z \frac{\partial c}{\partial z} + w_s c \right) \right] + \frac{Q}{\rho}$$

where

c = concentration, mass of substance per unit volume/ mass of air per unit volume

ρ = density of air, a function of height

u, v, w = wind velocity components

x, y, z = cartesian coordinates

w_s = particle settling velocity

Q = sources

K_z = vertical turbulent diffusion coefficient

The equation is solved on a three-dimensional numerical grid. Two successive coordinate transformations of the equation are made. The necessary three-dimensional wind field (u, v, w) is produced by using the two-dimensional surface wind field, obtained with the wind model, combined with similarity profiles for the boundary layer and the constraint of mass continuity.

2D.2.1 Numerical Methods

The transport equation is solved with an operator splitting method and the method of fractional steps (Yanenko (1971), Marchuk (1975)). A decomposition of the operators in the transport equation is made (see also McRae et al (1982)). Note that here the operators have not been specified in the time domain:

$$C^{t+1} = A_x A_y A_z (A_c A_c) A_z A_y A_x C^{t-1}$$

A_x and A_y are the advection operators in the horizontal directions. A_z is the vertical

advection, diffusion, deposition and (chemically passive) source/sink operator. A_c is the chemical sink/source operator. Principally, we then have the following principal sequence of equations to solve:

$$\begin{aligned}c^* &= A_x c^{t-1} \\c^{**} &= A_y c^* \\c^{***} &= A_z c^{**} \\c^t &= (A_c A_c) c^{***} \\c^{t+*} &= A_z c^t \\c^{t+**} &= A_y c^{t+*} \\c^{t+1} &= A_x c^{t+**}\end{aligned}$$

We are then solving several one-dimensional equations instead of one three-dimensional equation. The basic idea with the splitting method is that it is faster to solve the sequence of triangular matrices associated to the equations above, than to invert one big ordinary non-triangular matrix, associated with the nonsplitted transport equation.

To solve the equations on the numerical grid, they are reformulated with different numerical techniques depending on the type of operator. In the horizontal directions a finite element scheme is used, a Galerkin method with linear basis functions (see Long and Pepper (1976), McRae et al (1982), Chock and Dunker (1983), van Stijn et al (1987), Pielke (1984), for example). In the vertical direction and for the chemical step a finite difference scheme is applied. In the time domain, a Crank-Nicholson approach is used in the horizontal directions and for the chemical operator. Fully implicit time stepping is applied in the vertical direction.

In the horizontal a non-linear implicit numerical filter is applied after each time step in order to remove the shortwave noise produced by the approximate numerical method (Pielke (1984)). The filter also acts, together with numerical diffusion, as an artificial horizontal diffusion. Modelling of the physical horizontal turbulent diffusion has so far not been introduced in the model.

Boundary Conditions

At the lateral boundaries constant inflow and gradient outflow conditions are applied (see

Pielke (1984)). A Laplace filter is applied at the lateral boundaries at each time step.

At the lower boundary, the surface, zero wind velocity is introduced through the wind fields. The dry deposition rate is the lower boundary condition for the turbulent transport.

2D.2.2 The Numerical Grid, Coordinate Transformations

Basically the transport equation is formulated in a Cartesian co-ordinate system. In order to make the dispersion calculations in a proper topographical and numerical space environment, two space co-ordinate transformations are made.

The topographical properties of the terrain in the calculation domain are considered by a co-ordinate transformation to a terrain following co-ordinate system (Pielke (1984)). The vertical co-ordinate is transformed according to,

$$\sigma = z_{top} \frac{z - z_g}{z_{top} - z_g}$$

where z_{top} is the top level of the calculation domain, z_g is the terrain height and z is the vertical co-ordinate.

The staggered vertical co-ordinate σ is transformed to a logarithmic-linear vertical co-ordinate.

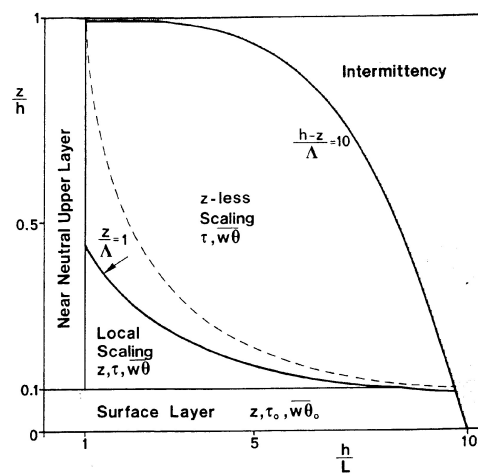
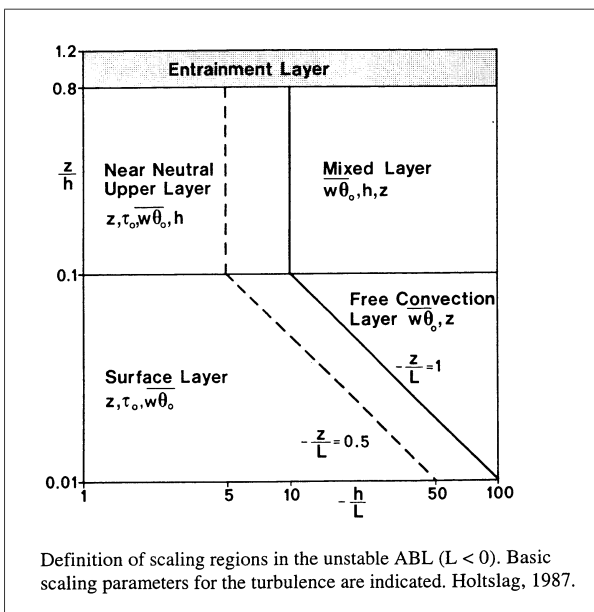
$$\zeta(\sigma) = a \left[\frac{1}{\kappa} \ln \left(\left(\frac{\sigma + z_0}{z_0} \right) + \frac{\sigma}{b} \right) \right] + 1$$

where z_0 is the surface roughness. For a certain choice of the coefficient a , the ζ -system is adjusted in an iterative way so that the distance between the grid points ($\Delta\zeta$) is close to unity for a given number of grid points. With the coefficient b , the resolution of the log-lin scale is tuned. The spacing between the levels is normally much finer close to the surface than higher up.

2D.2.3 The Turbulent Exchange Coefficient

The parametrisation of the vertical turbulent exchange coefficient, K_z , is prescribed according to the following classification:

A state of the art classification of the different scaling regions in the boundary layer, shown below for unstable and stable conditions, respectively, has been introduced. z is distance above ground, h is the height of the boundary layer and L is the Monin-Obukhov length scale, a parameter characterising the atmospheric stability. Negative L 's means unstable conditions while positive L 's indicate stable conditions. Stable conditions do suppress the vertical turbulent mixing, typical clear night conditions. Small values of h/L indicate close to neutral conditions; i.e. the vertical turbulent mixing is of mechanical origin only, thermal forces are absent.



Stability regimes under **unstable** conditions
(from Holtslag, 1987)

Stability regimes under **stable** conditions
(from Holtslag, 1987)

Models for the formulation of the vertical turbulent exchange coefficient, K_z , were discussed in Yamartino et al (1992). The formulas for K_z in the different regimes indicated in the diagrams above are:

Unstable case $L < 0$:

Surface layer ($z/h \leq 0.1 ; -z/L \leq 1.0$):

$$K_z(z) = \frac{u_i \kappa z}{\Phi_h\left(\frac{z}{L}\right)} \quad \text{where } \Phi_h\left(\frac{z}{L}\right) = 0.74 \left(1 - 9 \frac{z}{L}\right)^{-\frac{1}{2}}$$

Free convection layer ($z/h \leq 0.1$; $-z/L > 1.0$):

$$K_z(z) = w_i z$$

Near neutral upper layer ($z/h > 0.1$; $z/h \leq 1.0$; $-h/L > 1.0$; $-h/L \leq 10.0$):

$$K_z(z) = \frac{u_i \kappa h}{\Phi_h\left(\frac{0.1h}{L}\right)} \quad \text{where } \Phi_h\left(\frac{z}{L}\right) = 0.74 \left(1 - 9 \frac{z}{L}\right)^{-\frac{1}{2}}$$

Mixed layer ($z/h > 0.1$; $z/h \leq 1.0$; $-h/L > 10.0$):

$$K_z(z) = 0.1 w_i h$$

Entrainment layer ($z/h > 1.0$; $-h/L > 1.0$; $-h/L \leq 10.0$)

$$K_z(z) = \frac{0.01 u_i \kappa h}{\Phi_h\left(\frac{0.1h}{L}\right)} \quad \text{where } \Phi_h\left(\frac{z}{L}\right) = 0.74 \left(1 - 9 \frac{z}{L}\right)^{-\frac{1}{2}}$$

Entrainment layer ($z/h > 1.0$; $-h/L > 10.0$)

$$K_z(z) = 0.01 w_i h$$

Stable case $L >= 0$:

Surface layer ($z/h \leq 0.1$ & ; $h/L \geq 0.0$; $(h-z)/\Lambda \leq 10.0$):

$$K_z(z) = \frac{u_i \kappa z}{\Phi_h\left(\frac{z}{L}\right)} \quad \text{where } \Phi_h\left(\frac{z}{L}\right) = 0.74 + 4.7z/L$$

Near neutral upper layer ($z/h > 0.1$; $z/h \leq 1.0$; $h/L > 0.0$; $h/L \leq 1.0$):

$$K_z(z) = \frac{u_{*l} \kappa h}{\Phi_h\left(\frac{0.1h}{L}\right)} \quad \text{where } \Phi_h\left(\frac{z}{L}\right) = 0.74 + 4.7z/L$$

Local scaling layer ($z/h > 0.1$; $h/L > 1.0$; $z/\Lambda \leq 1.0$; $(h-z)/\Lambda \leq 10.0$):

$$K_z(z) = \frac{u_{*l} \kappa z}{\Phi_h\left(\frac{z}{\Lambda}\right)} \quad \text{where } \Phi_h\left(\frac{z}{\Lambda}\right) = 0.74 + 4.7z/\Lambda$$

Z-less scaling layer ($z/\Lambda > 0.1$; $h/L > 0.0$; $(h-z)/\Lambda \leq 10.0$):

$$K_z(z) = 0.4 u_{*l} \Lambda$$

Intermittency layer ($(h-z)/\Lambda > 10.0$; $z/h \geq 0.0$; $h/L \leq 1.0$):

$$K_z(z) = \frac{0.1 u_{*l} \kappa h}{\Phi_h\left(\frac{0.1h}{L}\right)} \quad \text{where } \Phi_h\left(\frac{z}{L}\right) = 0.74 + 4.7z/L$$

Layer above ($h/L > 0.0$; $h/L \leq 1.0$; $z/h > 1.0$):

$$K_z(z) = \frac{0.1 u_{*l} \kappa h}{\Phi_h\left(\frac{0.1h}{L}\right)} \quad \text{where } \Phi_h\left(\frac{z}{L}\right) = 0.74 + 4.7z/L$$

where:

u_* = friction velocity (= $\sqrt{\tau_0/\rho}$)

u_{*l} = local friction velocity (=)

q_* = convective velocity scale (=)

q_{*l} = local convective velocity scale (=)

$\overline{w\theta_0}$ = surface heat flux

T = air temperature

g = gravitational acceleration

ρ = density of air

c_p = specific heat of air at constant pressure

κ = von Kármán's constant (= 0.4)

z = height above the surface

L = Monin-Obukov length scale ($= -(u_i^3 / (\kappa \frac{g}{T} \overline{w\theta_0}))$)

Λ = local Monin-Obukov length scale ($= -(u_i^3 / (\kappa \frac{g}{T} \overline{w\theta_0} (1 - z/h)))$)

h = height of the mixed layer

2D.2.4 The Three-Dimensional Wind Field

The construction of the surface (two-dimensional) wind field is described in section *Appendix 2A: The Wind Model - Calculation of the Wind Fields*. In order to create a three-dimensional wind field, necessary for the three-dimensional grid model, vertical profiles of the horizontal wind field are calculated at every grid point. The profiles are calculated according to methods described in an other section. The profiles are distributed through the boundary layer up to the height $mixh$, the mixed layer height. Above $mixh$ a free wind field, constant up to the top of the calculation domain, is assumed to prevail.

Having obtained horizontal wind fields (u,v) at each level in the numerical calculation grid, the vertical, cartesian, wind components are calculated with the use of a numerical formulation of the mass continuity equation,

$$\frac{\partial w}{\partial z} = \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y}$$

or, if the horizontal wind fields are evaluated directly in the ζ -system:

$$\frac{\partial w^\zeta}{\partial \zeta} = \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y}$$

where q^ζ = vertical velocity in the ζ -system.

2D.2.5 Source Initialization

The Grid model only uses the EDB to obtain emission input data. Points sources are treated individually and all other sources are collected together onto an emission grid.

The sources are distributed in the three-dimensional grid according to type and size. The

source strength for a line source passing through a grid square is transformed to an equivalent grid area source.

A point source from a stack is initially modelled by a series of puffs until the puff radius has grown to the size of two grid sizes. The puff is then released as a three-dimensional source into the grid equation solver. Plume rise and downdraft is calculated according to *2C.3 Plume Rise Formulation*.

2D.3 Particle Deposition

2D.3.1 Dry Deposition of Gases and Non-Gravitating Particles

The process of dry deposition of gases and very small non-interacting particles is determined by the turbulent conditions in the boundary layer. The transport through the air down to the ground is therefore incorporated in the turbulent diffusion term in the transport equation. The transport from the lowest part of the surface layer to the ground, the boundary condition in the diffusion term, is in principle modelled as:

$$\left(K_z \frac{\partial c}{\partial z} \right)_{surface} = v_g c_{surface}$$

v_g is called the dry deposition velocity. When dry deposition is calculated in the grid model, direct estimates of the dry deposition velocities for the different land use classes, i.e. sea, field, wood etc., are given by default in a resource file. Discussions about dry deposition can be found in Hanna et al (1985), McMahon and Denison (1979) and Sehmel (1980).

It is also possible to calculate the dry deposition velocities to adapt to the actual local conditions.

For gases the deposition velocity is modelled as:

$$\frac{1}{v_g} = r_a + r_b + r_c$$

r_a is the atmospheric resistance for the surface layer (the aerodynamic resistance), r_b is resistance in the laminar boundary layer and r_c is the canopy resistance.

The aerodynamic resistance is a function of the surface layer conditions. During unstable conditions ($L < 0$) r_a is calculated as:

$$r_a(z_r) = \frac{0.74}{\kappa u_i} \left[\ln \left(\frac{x-1}{x+1} \right) - \ln \left(\frac{y-1}{y+1} \right) \right]$$

where

$$x = \sqrt{1 - 9 \frac{z_r}{L}} \quad \text{and} \quad y = \sqrt{1 - 9 \frac{z_0}{L}}$$

For stable stratification ($L > 0$) we have:

$$r_a(z_r) = \frac{1}{\kappa u_i} \left[0.74 \ln \frac{z_r}{z_0} + \frac{4.7}{L} (z_r - z_0) \right]$$

and for neutral conditions

$$r_a(z_r) = \frac{0.74}{u_i} \ln \frac{z_r}{z_0}$$

The resistance in the laminar boundary layer, r_b , depends also on the physical properties of the air and the gas:

$$r_b = \frac{2}{\kappa u_i} \left(\frac{\nu}{D} \right)^{\frac{2}{3}}$$

where

ν = kinematic viscosity of air

D = molecular diffusion coefficient of gas

The canopy resistance, r_c , is a function of the type of gas and vegetation, the vegetation density and the type of ground surface:

$$r_c = \left(\frac{1}{R_m} + \frac{LAI}{r_{st} + r_{mes}} + \frac{LAI}{r_{cut}} \right)$$

where

R_m = ground surface resistance

$r_{st}+r_{mes}$ = foliage resistance, a stomatal and mesophyll part

r_{cut} = cuticle resistance

2D.3.2 Wet Deposition

The wet deposition depends on the concentration of the actual gas and on the precipitation. The most important processes for the small particles are the condensation of water on the particles, i.e. forming cloud water drops, and the impaction against falling water drops under the clouds. The uptake of gases in the cloud drops depends on the chemical processes in the drops. The interchange of gases between the drops and the surrounding is relatively fast and therefore in equilibrium.

The wet deposition is modelled as

$$-\lambda c$$

where λ is a washout coefficient defined as:

Both I and c are functions of the height, λ is in addition a complicated function of the

$$\int_0^h \lambda(z)c(z)dz = dep = W_r c_0 P$$

precipitation. On the right hand of the equation we have W_r , the washout ratio, P , the precipitation and c_0 the ground level concentration. These variables are better known than the complicated washout coefficient and the approximation:

$$\int_0^h \lambda(z)c(z)dz = \bar{\lambda} \bar{c} mixh = \bar{\lambda} c_0 mixh \Rightarrow \bar{\lambda} = \frac{W_r P}{mixh}$$

which for the moment is applied in the grid model. $Mixh$ is the mixed layer height. Estimations of the washout ratio are for example found in Hanna et al (1982) and McMahon and Denison (1979).

2D.3.3 Particle Settling

Particles in a certain size range are affected by gravitational forces, without being big enough to be ballistic. For the dispersion treatment of this class of particles, a settling or sedimentation velocity (w_s) coupled with the vertical concentration gradient, is used (see formula in *2D.2 Formulation of the Grid Model*).

Principles and estimates of the settling velocity can be found for example in Hanna et al (1982).

It is important to note that if the actual particle size distribution is wide, many dispersion calculations have to be done with each calculation representing a narrow size range. The reason is that the settling velocity is a very strong function of the particle diameter.

Appendix 2E: Street Canyon Model

Both the Gaussian and the Grid model calculate roof level concentrations. The Street Canyon model is a small-scale model that allows you to simulate the street level concentrations on a single road that has a row of buildings on each side. The simulated concentrations are visualised on a vertical plane perpendicular to the street direction, the buildings creating the effect of looking down a canyon.

This model simulates the dispersion from traffic on a particular road. The user must be aware of the fact that traffic on other roads may very likely contribute to a substantial part of the pollution levels, especially if the simulated road has a low traffic intensity and is surrounded by larger streets. Will it then be possible to separately estimate the fractions of street pollution created by the local traffic and those created by emissions on other places?

A feasible way to do this within Airviro is to

- run the street canyon model with traffic emissions only from the particular road/road section of interest
- run a city-wide simulation with the Gaussian or the Grid model, including all sources except for the traffic on the particular road/road section of interest. The concentration at roof level above the street of interest can then be considered as a result of non-local sources. Due to the eddy circulation mentioned earlier, the same contribution from non-local sources down in the street canyon itself can be expected. This concentration level can be considered as a background value to be added to the street canyon model results.

2E.1 Model Description

The basis of the model originates from studies in San José, USA, where measurements indicated a vertical eddy circulation when the wind was blowing perpendicular to the street directions. The eddy circulation implies higher pollution concentrations on the upwind side

of the road, because this air includes the traffic exhausts. The leewind side shows much lower concentrations, as the air originates from the roof level. The model is referred to as the "Stanford model".

The "Stanford model" expressions have been tested and modified in a study in Scandinavia (SNV, 1977) and the expressions for the CL (concentration leewind side, i.e. on the upwind side of the street) and CW (concentration wind side) are used in the Airviro model. The original CL equation is modified with a wind polynomial reported in Laurin and Omstedt (1987):

$$CL = \frac{K \cdot Q \cdot (0.27u - 0.016u^2 + 0.65)}{(u + 0.5) \cdot \left[(x^2 + y^2)^{\frac{1}{2}} + L_0 \right]}$$

$$CW = \frac{K \cdot Q \cdot (H - z)}{W \cdot (u + 0.5) \cdot H}$$

where

K is an empirical constant, set to 14

Q is the traffic emission in g/m,s

u is the wind component perpendicular to the street axis, in m/s

L₀ is a length scale of the individual cars, set to 2 m

W is width between the houses, H the typical house height in m

x, z are horizontal and vertical distances from the street emission segments

The vertical section between the houses is divided into a grid, on which the concentrations are evaluated. The road, i.e. the part of the street width (RW) where the emission takes place, is divided into a large number of road segments with equal emission rate.

The wind component is taken from the wind field calculation, evaluated at the street location. If the wind is blowing within 22.5 degrees from the principal street direction, the concentration will be found as an average of CL and CW, being symmetric.

For cases with a wind direction more perpendicular to the street direction, we will find an asymmetry with higher concentrations on the side of the road from where the roof wind

blows (lee side) and lower values on opposite side (called wind side). The concentrations on the wind side of the emitting road are calculated according to CW. At the other gridpoints, a summation of CL values takes place, implying a contribution from each emitting segment situated upwind the gridpoint itself (no impact from an emitting segment situated downwind).

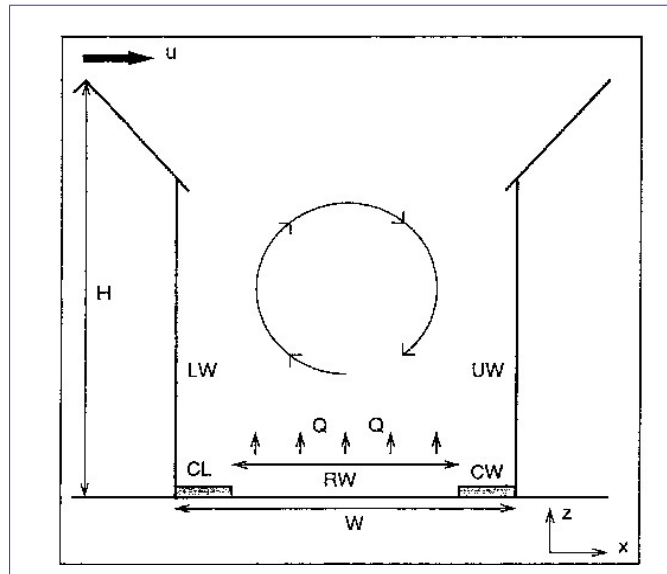


Figure 2E.1 Principles of air circulation within a street canyon (the upwind side is denoted UW, the leeward side LW, see also notations for house height (H), street width between houses (W) and road width (RW)).

Appendix 2F: The Heavy Gas Model

The Heavy Gas Model is a model system for the description of the emission and dispersion of gases heavier or denser than the ambient air.

Dense gases are gases that are heavier than the ambient air because of the following reasons:

- they have a higher molecular weight than the air at the air temperature
- they are stored as condensed gases at a low temperature
- they are stored as condensed gases at a high pressure
- a mixture of the above mentioned reasons

These types of gases are often toxic and in addition a condensed gas is very cold when it is released. It can therefore cause frostbite in people who come into contact with a released condensed gas cloud. The gas cloud can also have suffocating effects even if it is neither toxic nor very cold.

The model should be applied to industrial and transportation environments, where storage and transportation of this type of gas, for example chlorine, ammonia and propane, takes place.

The accidents or scenarios that can currently be simulated in the model are,

- The rupture of a storage container, resulting in a "large" hole in a pressurised container. The emission from this type of accident is assumed to be quite a violent boiling off of the condensed liquid. The release time is assumed to be short. This scenario is therefore classified as an instantaneous liquid phase release. A dense puff is eventually formed which is transported away with the wind while collapsing gravitationally. After some time the puff is diluted and it behaves as a neutral density puff.
- A "small" hole leakage, which could mean a leakage from a valve or a broken small

pipe. The release time is assumed to be quite long. The release can originate from both the liquid and the gas phase of the pressurised storage vessel. This scenario type is called a continuous liquid or gas phase release. A heavy gas continuous plume or slab is formed. The slab stretches out in the wind direction and gets wider because of the gravitational spread perpendicular to the main slab direction. When the plume density is the same as the ambient air, the plume behaves as a neutral Gaussian plume.

It is clear that a simulation includes the application of several models. Emission models, models for initial cloud forming, gravitational phase models and models for the neutral plume phase. A description of appropriate models for each release scenario is given below.

2F.1 Continuous Release or a Leakage From a” Small” Hole

The initial phase - Emission from the liquid phase

A choked two-phase outflow is assumed (Winter et al (1984)). The emission or outflow is:

$$q_0 = A c_f G_0 \quad (\text{kg/s}) \quad (\text{eq 2F.1.1})$$

A is the prescribed outflow area, c_f a frictional coefficient and G_0 is the maximum of the function:

$$G(p) = \sqrt{\frac{p_0 - p}{v(p)}} \quad (\text{kg/s, m}^2) \quad (\text{eq 2F.1.2})$$

where p_0 is the pressure in the vessel, p is the pressure which maximises the function, normally higher than the atmospheric pressure. $v(p)$ is the specific volume of the binary substance at the outlet section. A two-phase momentum jet forced by the momentum outflow and the difference between the pressure p and the atmospheric pressure p_a , is assumed to stagnate and form a continuous initial heavy (or neutral) gas cloud. The momentum of the jet is constant and equal to the initial momentum:

$$M = q_0 u_0 + A(p - p_a) \quad (\text{N}) \quad (\text{eq 2F.1.3})$$

where u_0 is the outlet velocity.

$$u_0 = \frac{q_0}{Av(p)} \quad (\text{m/s}) \quad (\text{eq 2F.1.4})$$

The jet formulae are (Winter et al. (1984)):

- massflow $q(x) = C_1 \sqrt{M \rho_a x}$ (kg/s) (eq 2F.1.5)

- centre velocity $u_c(x) = C_2 \sqrt{\frac{M}{\rho_a} \frac{1}{x}}$ (m/s) (eq 2F.1.6)

C_1 and C_2 are coefficients and ρ_a is the air density. The initial cloud is formed when the centre velocity is of the same order as the ambient wind velocity. The distance x to the cloud forming and the massflow at that distance are given by equations 2F.1.6 and 2F.1.5 respectively. The relation between $q(x)$ and q_0 gives the dilution. The size of the cloud, the average concentration and the average temperature are then easily calculated. The cloud is assumed to be rectangular, with the wind parallel side decided by the local trajectory length and the height is half the wind perpendicular side. The duration of the release is found by the assumption that the whole tank contents are released.

Emission from the gas phase

The gas outflow is tested for critical flow conditions. If the relation between the outlet pressure, the atmospheric pressure p_a , and the tank pressure p_0 :

$$\frac{p_a}{p_0} \leq \left(\frac{2}{\kappa + 1} \right)^{\frac{\kappa}{\kappa + 1}}; \kappa = \frac{c_p}{c_v} \quad (\text{eq 2F.1.7})$$

is less than a certain gas specific value, the outflow is critical. c_p and c_v are the specific heat capacities at constant pressure and constant volume, respectively. Critical conditions give the velocity and mass flux at the outlet:

- velocity $u_0 = \sqrt{2 \frac{\kappa}{\kappa + 1} p_0 v_0}$ (m/s) (eq 2F.1.8)

$$\text{- mass flux } q_0 = \sqrt{2 \frac{\kappa}{\kappa+1} \left(\frac{2}{\kappa+1} \right)^{\frac{2}{\kappa-1}} \frac{p_0}{v_0}} \quad (\text{kg/s}) \quad (\text{eq 2F.1.9})$$

otherwise, if condition 2F.1.7 is not fulfilled,

$$\text{- velocity } u_0 = \sqrt{2 \frac{\kappa}{\kappa+1} \left[1 - \left(\frac{p_a}{p_0} \right)^{\frac{\kappa-1}{\kappa}} \right]} \quad (\text{m/s}) \quad (\text{eq 2F.1.10})$$

$$\text{- mass flux } q_0 = \sqrt{2 \frac{\kappa}{\kappa-1} \left[\left(\frac{p_a}{p_0} \right)^{\frac{2}{\kappa}} - \left(\frac{p_a}{p_0} \right)^{\frac{\kappa+1}{\kappa}} \right] \frac{p_0}{v_0}} \quad (\text{kg/s}) \quad (\text{eq 2F.1.11})$$

Using the jet equations 2F.1.3 to 2F.1.6, with the initial values given by equations 2F.1.8 to 2F.1.11, gives the necessary initial cloud characteristics, the height and length of the cloud are calculated as described earlier. The duration of the release is found by the assumption that the release is over when the tank liquid temperature has decreased to its boiling point.

The gravitational dispersion phase

If the initial cloud is denser than the ambient air, it will collapse because of the gravitational forces. When the cloud height decreases, the sides of it will travel outwards, forming a pancake-like cloud. Simultaneously the cloud will be advected by the wind in the wind direction. A ground based plume, stretching out in the wind direction and widening primarily by gravitational outflow, will be formed. Mixing and entrainment will take place at the top and the edges of the plume. The mixing processes used are the same as described in Eidsvik (1980). The gravitational widening velocity is:

$$u_g = C_3 \sqrt{gH \frac{\rho_p - \rho_a}{\rho_p}} \quad (\text{m/s}) \quad (\text{eq 2F.1.12})$$

C_3 is a coefficient, here equal to 1.0. g is the gravitational acceleration, H is the cloud height and ρ_p is the plume density.

The neutral dispersion phase

When the plume is diluted so much that the plume density is close to the density of the

ambient air, the gravitational effects are small and the plume behaves like a traditional non-buoyant ground surface plume. At this stage a Gaussian plume model, described on another place in this documentation, takes over and proceeds with the dispersion calculations (see appendix E3.2 in Airviro Specification Part II).

2F.2 Instantaneous Release or a Tank Rupture - Emission Only from the Liquid Phase

In this case a large hole is assumed to be formed. An accident could for example be caused by overfilling the storage container. A slight temperature rise could cause a fracture in a weak section of the container. A violent boiling creating an intense two-phase jet could be the result.

The hole area is assumed to be (Winter et al. (1984)):

$$A = C_4 V^{\frac{2}{3}} \quad (\text{m}^2) \quad (\text{eq 2F.2.13})$$

where V is the tank volume and C₄ is a coefficient. The emission phase follows equations 2F.1.1 to 2F.1.6. The whole tank contents are assumed to boil off. The maximum distance between the source and place where the initial cloud is formed is (Winter et al. (1984)):

$$x = C_5 \left(\frac{u(x) T \tau}{\rho_a} \right)^{\frac{1}{4}} \quad (\text{m}) \quad (\text{eq 2F.2.14})$$

C₅ is a coefficient, T is the mass contents in the tank, u(x) is the jet velocity when the jet pressure has decreased to the atmospheric pressure and t is the release duration time. The volume of the initial cloud is:

$$Vol = C_6 \left(\frac{u(x) T \tau}{\rho_a} \right)^{\frac{3}{4}} \quad (\text{m}^3) \quad (\text{eq 2F.2.15})$$

The initial cloud is assumed to be cylindrical with equal radius and height. The gravitational and the neutral dispersion phase are in principle the same as in the earlier description.

2F.3 Substances and Coefficients

Currently the following substances can be handled by the model:

Ammonium, chlorine, sulphur dioxide, propane, vinyl chloride and methane.

The coefficient values are (Winter et al. (1984)):

$$C_1 = 0.284$$

$$C_2 = 7.21$$

$$C_3 = 1.0$$

$$C_4 = 0.05$$

$$C_5 = 2.9$$

$$C_6 = 1.7$$

2F.3.1 Comments

It should be noted that if the stored gas is condensed by refrigeration, the release phase is expected to be much less violent than if it is pressurised.

It is important to note that small-scale topography and big roughness elements, like houses, are not considered in the model. The influence of these disturbances has to be estimated by the user. Further, the emission phases in the model are constant in time while they in reality are very transient. A general treatment of Vapour Cloud Dispersion Models is given in Hanna & Drivas (1987) and is recommended.

Appendix 2G: Scenario Calculations

2G.1 General Principles

The Airviro dispersion models, except the Heavy Gas Model, simulate hourly concentration values. To make it possible to simulate the long-term impacts of emissions, without running through a very long time series, a certain technique is applied, which we call scenario calculations. The procedure involves a statistical approach. By extracting a representative sample of the joint variation of weather and emissions, expected mean values and extreme values of air quality can be simulated based on a limited number of representative hours.

The basic principles are as follows: When you select weather conditions in the main menu of the dispersion module you will activate the **Weather** sub-window. Choose **Scenario....** A list box will appear and you can select one of a number of headings. In the REF database you will find the alternatives WINTER and SUMMER.

Scenarios are defined in special configuration files. The Airviro configuration files are described in detail in the on line manual (**man**) pages. A statistical sample of a winter and a summer period in Göteborg is specified within these files. The statistical sample is given as specific dates and hours, (representing different weather and emission events) and a figure describing the frequency of occurrence.

When a dispersion calculation is performed based on the WINTER scenario, the weather information given in the time series database is extracted, using the dates/hours defined in a configuration file as the selection criteria, and the frequencies of the various weather classes will be used to estimate mean values and extremes (95-99 percentile).

2G.2 A Standard Method to Generate a Statistical Sample: klmstat

Airviro includes a utility program to prepare statistical samples based on long term

measurements of weather. A description of the program (**klmstat**) can be found in the man-pages.

In order to achieve a climatology that reflects the local variation of weather conditions, more than one, preferably five years of monitoring data of the horizontal wind vector, the air temperature and the vertical temperature gradient from a meteorological mast is necessary. However, additional measurements of the standard deviation of the horizontal wind direction and of the vertical wind velocity, the solar radiation and the precipitation, are recommended.

Provided that the monitoring data (described above) is available in the time series database, the **klmstat** program will extract a representative sample using the following technique:

The data is classified according to the wind directions. The user can decide how many classes to use. As default, it is recommended to use 60, i.e. each class representing a sector of 6°. All events falling into a specific sector will be classified according to the atmospheric stability conditions (as default discriminated by 6 intervals of Monin-Obukhov lengths). When all data have been sorted, frequencies of all classes will be estimated and the median values (of the Monin-Obukhov lengths) of each class (in this case 360 classes) are identified, including the specific date/hour when each class example occurred.

The number of classes required depends on the characteristics of the area. In several countries, regulations concerning the number of classes exist. The number of classes should be chosen as a balance between the required quality of the calculation results and the computing time. An objective method to decide the number of cases is as follows:

1. Start with 60 wind direction classes. When the representative dates/hours have been determined, plot the diurnal distribution of hours extracted in the sample (do not forget to weight the hours together with the estimated frequencies). Examine the distribution, and be sure that the distribution is fairly uniform. If not, extend change the number of classes. This test is important in order to find a representative joint distribution of weather and emissions.

2. Use the dates/hours determined as selection criteria for extracting the **sample mean**

and **95/99 percentile** for the concentration of one or several gases (measured at one or several ambient air monitoring stations). Compare these figures with the mean and 95/99 percentile (from the identical ambient air quality stations) determined from the entire period. If the estimated figures based on the sample deviates from the figures based on the entire period you need to change (probably increase) the number of classes.

Experiences from Stockholm and Göteborg indicate that 60 wind direction classes and 6 stability classes are sufficient.

2G.3 General Applications Using the Scenario Feature

The Scenario concept can be used in a variety of ways. In fact any set of data can be used as long as the date, hour, and frequency of occurrence are defined in the configuration files, and the weather information is available in the time series database. Some examples of how the Scenario feature could be used:

- **Episodes of SO₂**. Examine the last 5 years measurements of SO₂ and determine the 100 worst cases. Note the corresponding dates/hours and assume frequencies to be equal. Include this information in the configuration files in order to define an episode scenario. The Scenario calculation can then be used to illustrate what cost effective measures could be taken to improve the situation at these worst cases.
- **Validation of an intensive measurement period**. In order to validate the EDB/dispersion model the user could use a large amount of passive samplers during a given period, such as a month. By defining the period as a Scenario, it would be possible to run validation calculations in a simple way.
- **Special events**. If there are certain events, such as: Olympic games, world cup arrangements, national conference week, international exhibition, train strike etc., it would be possible to define the period of the event as a Scenario and combine it with a modified EDB, in which the emission consequences of the event are described, in order to simulate the environmental impact of the event.

Appendix 2H: A Method to Locate Dominating Sources

The method assumes time series of

- hourly meteorological data: wind speed and direction at one or two levels, temperature difference in the surface layer and standard deviations of wind direction and vertical wind speed
- hourly concentration values of a pollutant from one or - preferably - several monitor stations (point or open path measurements)

A grid consisting of 24 x 20 cells is created over the calculation domain. A fictitious source is located in each cell center (i.e. 24 x 20 sources). All sources emit according to

$$q_h = a_h q_0 \quad (h=1,\dots,24)$$

where a_h is the emission amplitude as specified by a normalized EDB formula and q_0 is a constant emission rate (the absolute value is not of interest, only variations).

We start with the first fictitious source ($i=1$) and simulate the effect on concentration levels at the monitoring locations. The initial diffusion - in the Gaussian plume model named σ_{yinit} and σ_{zinit} - depends on the horizontal dimensions of the calculation grid (dxy):

$$\sigma_{yinit} = 0.5dxy \quad \text{and} \quad \sigma_{zinit} = 0.22dxy^{0.78}$$

The simulation uses the Gaussian plume model, using the mast data for the estimation of stability, mixing height and plume growth coefficients σ_y and σ_z . The evaluation of the wind at plume height uses the following assumptions:

- **Plume height:** Specified by user.
- **Trajectory:** Since a homogeneous wind field is used, the plume axis will be a straight line.
- **Vertical wind profile:** The free wind speed (V_g) is estimated by the extrapolation at

25m with the help of the power law. The wind direction is twisted according to empirical data (Holtslag).

The estimation of the wind at plume height will thus relate to the measured wind at measurement level (z_u) and the free wind (V_g) at the height of the mixed layer ($mixh$). With wind measurement at only one level, we get:

- Plume height $< z_u$: adiabatic wind profile (logarithmic)
- Plume height $> z_u$: power law between z_u and $mixh$, based on measured wind and V_g . If plume height is within surface layer ($0.1h$), no turning of wind direction from measured level to plume height is assumed. With plume height above surface layer, the wind direction will turn linearly between measured wind vector and V_g .

With wind data from two levels, the power law is used for estimating wind speed at

plume heights in between the two measurement levels. For plume heights above the upper wind measurement level, the power law is used in the same way as described for the one-level measurement.

The concentrations are evaluated at the locations of the monitoring stations. If an open path instrument is used, the simulated concentration level is calculated at 10 points along the path. The mean value is found after averaging the concentration values.

Time series data are given for $j=1, \dots, m$ hourly values, and the measured concentrations are given at $k=1, \dots, l$ number of monitoring stations. The simulated concentration at monitoring station k is written as a result of the i th source and the j th hour of meteorological data (S_{ijk}).

For each fictitious source i , the correlation between simulated and measured time series is calculated as

$$corri = \text{corr}(S_i, C)$$

where each of the two series - the simulated (S_i) and the measured (C) - include data from $j=1, \dots, m$ hours and $k=1, \dots, l$ stations.

The calculations are repeated for all fictitious sources. The horizontal distribution of the correlation coefficients is displayed as isolines multiplied by 100.

The interpretation of the result should concentrate on the isoline pattern that is the distribution of maximum and minimum, and not in absolute values. This is because one dominating source will yield a higher correlation coefficient compared to the case of various and geographically dispersed sources, even if each of the latter sources is as strong a contaminator as the first, single source.

Appendix 2I: A Method to Estimate Emissions from Dominating Sources

The method assumes the same kind of data as for the location of sources:

- hourly meteorological data: wind speed and direction at one or two levels, temperature difference in the surface layer and standard deviations of wind direction and vertical wind speed
- hourly concentration values of a pollutant from one or various monitor stations (point or open path measurements). It is preferable to have more than one time series for concentration values.

You assume the presence of one or various area sources, specified with either a constant or a time variation in its emissions. You also have to specify the plume height together with the location and geometry of each source. Normally the background concentration, i.e. the part of the registered levels that are not explained by the specified sources, is assumed to be constant in time. There is, however, an option that permits a time varying background concentration and this is explained in section *2.I.1 Monitor Option for a Time Varying Background Concentration*.

As a starting point, each specified area source will be divided into 9 sub areas and the emission represented by 9-point sources. The initial diffusion is as described in *Appendix 2H: A Method to Locate Dominating Sources* (where d_{xy} is now the dimensions of the 9 sub areas). All sources are assumed to emit with a normalized, default emission rate, as an average over the specified period. During the period, each area source may have a unique variation specified by a normalized EDB formula. For the chosen time period, the model will evaluate simulated concentration time series at all receptor points/open paths. The simulation uses exactly the same wind evaluation and Gaussian plume model concept as described in Section *Appendix 2H: A Method to Locate Dominating Sources*.

For each specified source, we will then have a simulation time series S_i consisting of data

from $j=1, \dots, m$ hours and $k=1, \dots, l$ stations, assuming a normalized emission rate from every source. The emission estimation method consists of a multiple regression analysis of the simulated source contributions (predictors) with the actual, measured data C (as predicted). The regression method - stepwise regression with a forward selection principle - is described in *Exploiting the Regression Models in User's Reference Volume 3: Working with Indico Presentation*.

The multiple regression analysis will optimize the expression

$$C = b_0 + b_i S_i + \varepsilon$$

where b_0 is interpreted as the average background value, b_i gives the amplitude of source i and ε is the (minimized) estimation error. The stepwise regression uses two criteria, F_{in} and F_{out} , that regulate the risk (in percent) that you are willing to take in the inclusion or exclusion of an individual source i . The user can specify the values to use in the emission estimation but something similar to $F_{in}=0.05$, $F_{out}=0.10$ is recommended.

If the impact of the sources to be estimated gives a clear signal in the measured concentration data, the method will yield a coefficient b_i that scales the normalized simulation time series. If, as an example, the normalized emission is set to 1 g/s, the b_i value will express the estimated emission for source i directly in g/s.

The multiple correlation coefficient is the overall measure of how good your source assumptions are. You should always try to maximize it, either by running individual or combined source estimates. Typical tests are:

- changing area sizes (e.g. half, double)
- changing plume heights
- changing emission variations (EDB formulas)

For a given number of sources, the source specific standard error (the standard deviation of the estimated error in the emission estimation for a particular source) is the measure of how good a source estimate is. Assuming the estimate error to be normally distributed, it gives the interval of 67% confidence level (twice times the estimate error gives 95% confidence level).

The part of the monitored pollution data that is not explained by the impact of the specified sources will fall into the (constant) background value.

2.1.1 Monitor Option for a Time Varying Background Concentration

When the user suspects that there are some remote sources that contribute considerably to the monitored pollution levels, the monitor option may be used as a complement to the normal assumption of a constant background level. This calculation will be able to pick out the part of the contamination levels that are spatially uniform over the calculation domain, i.e. one should think of a remote source that gives a time varying but equally distributed impact on the all the pollution monitors included in the model. The method uses only a minor part of the time series, which is why this method should be thought of as only complementary.

With the monitor option activated, only hourly values where

- at least one monitor is influenced by the local sources defined by the user
- at least one monitor is not influenced by any of the local sources defined by the user will be used. The average value of the monitored data with no influence is assumed to be the background value of that particular hour, and this value will be subtracted from all monitored data values before the data is used as input to the regression analysis.

The model output will then consist of

- two numbers of hourly values, one showing the number of hours where data exists
- and the other showing the number of hourly values where the restrictions above are fulfilled.
- two background values, one being the average of the monitor time series, and the other being the output from the regression analysis.

Appendix 2J: The OSPM Model

- Description

OSPM is a practical street pollution model

Concentrations of exhaust gases are calculated using a combination of a plume model for the direct contribution and a box model for the recirculating part of the pollutants in the street.

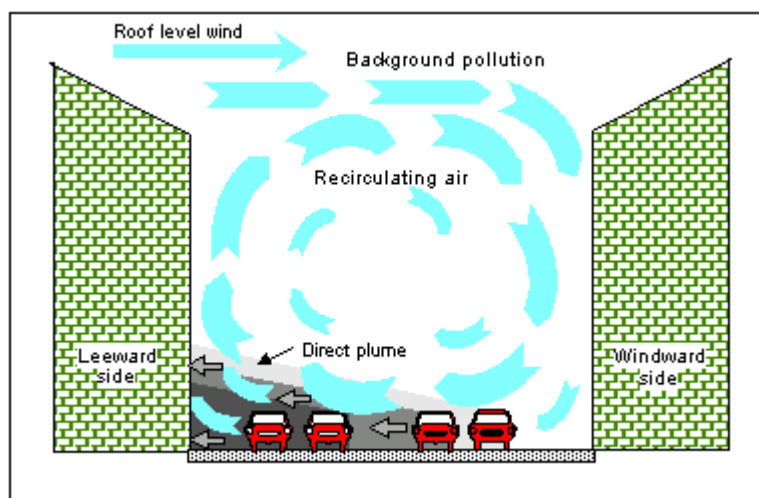


Figure 2J.1. Description

The direct contribution is calculated using a simple plume model. It is assumed that both the traffic and emissions are uniformly distributed across the canyon. The emission field is treated as a number of infinitesimal line sources aligned perpendicular to the wind direction at the street level. The cross wind diffusion is disregarded. The wind direction at the street level is assumed to be mirror reflected with respect to the roof level wind. The plume expression for a line source is integrated along the path defined by the street level wind. The length of the integration path depends on the extension of the recirculation zone.

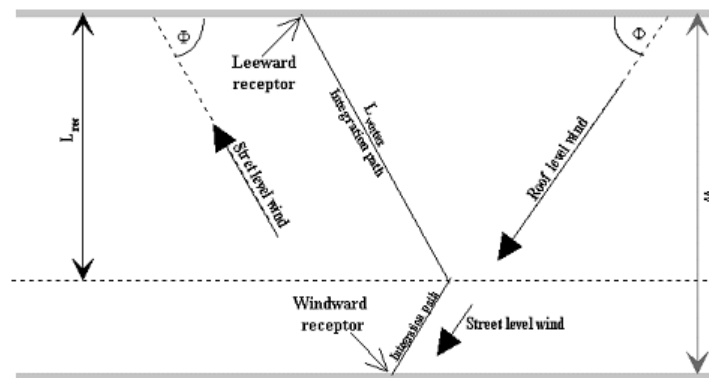
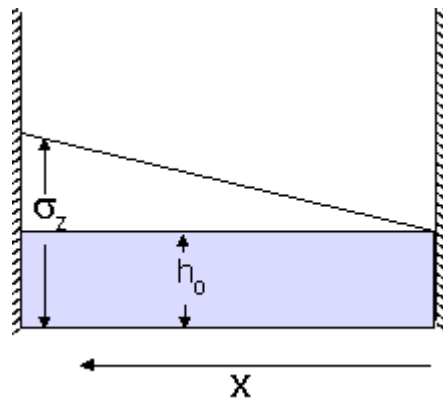


Figure 2J.2. OSPM

- The length of the vortex, calculated along the wind direction, is 2 x the upwind building height. For roof-level wind speeds below 2 m/s, the length of the vortex decreases linearly with the wind speed. The buildings along the street may have different heights, affecting thereby the length of the vortex and subsequently the modelled concentrations.
- The upwind receptor (lee-side) receives contribution from the traffic emissions within the area occupied by the vortex (the recirculation zone), the recirculated pollution and a portion of the emissions from outside of the vortex area.
- The downwind receptor (wind-side) receives contributions from the recirculated pollution and the traffic emissions from outside of the recirculation zone only.
- As the wind speed approaches zero or is parallel with the street, concentrations on the both sides of the street became equal.
- The vertical dispersion is modelled assuming a linear growth of the plume with the distance from the source.



Vertical dispersion:

$$\sigma_z(x) = \sigma_w \frac{x}{u_b} + h_0$$

The direct contribution:

$$C_d = \sqrt{\frac{2}{\pi}} \frac{Q}{W \sigma_w} \ln \left[\frac{h_0 + (\sigma_w / u_b) W}{h_0} \right]$$

Initial dispersion:

$h_0 = 2 \sim 4$ m (depends on wind speed)

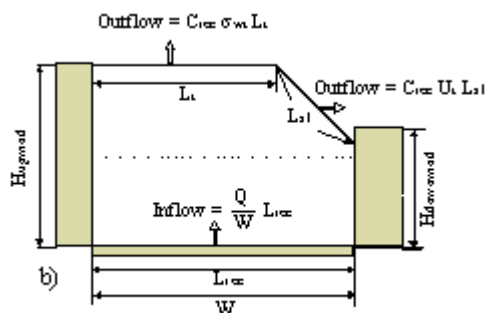


Figure 2J.3. Recirculation

The contribution from the recirculation part is calculated using a simple box model.

It is assumed that the canyon vortex has the shape of a trapeze, with the maximum length of the upper edge being half of the vortex length. The ventilation of the recirculation zone takes place through the edges of the trapeze but the ventilation can be limited by the presence of a downwind building if the building intercepts one of the edges. The concentration in the recirculation zone is calculated assuming that the inflow rate of the pollutants into the recirculation zone is equal to the outflow rate and that the pollutants are well mixed inside the zone.

Traffic Created Turbulence

The turbulence within the canyon is calculated taking into account the traffic created turbulence. The traffic induced turbulence plays a crucial role in determination of pollution levels in street canyons. During windless conditions the ambient turbulence vanishes and the only dispersion mechanism is due to the turbulence created by traffic. Thereby, the traffic created turbulence becomes the critical factor determining the highest pollution levels in a street canyon.

Street geometry

The model can be used for streets with irregular buildings or even buildings on one side only but it is best suited for regular street-canyon configurations. The model should not be used for crossings or for locations far away from the traffic lanes.

Wind meandering

Concentration distribution of pollutants in the street is calculated taking into account wind direction fluctuations. For each calculation hour, the resulting concentrations are averaged over a wind direction sector centered around the hourly mean wind direction. The width of the averaging wind sector depends on the roof level wind speed and increases with the decreasing wind speed. For calm conditions the averaging sector approaches 360°, which results in uniform concentration distribution across the street.

NO₂ chemistry

The NO₂ concentrations are calculated taking into account NO-NO₂-O₃ chemistry and the residence time of pollutants in the street.

The presence of NO_2 in ambient air is mainly due to the chemical oxidation of the emitted NO by background ozone. Under sunlight conditions, photo dissociation of NO_2 leads to partial reproduction of NO and O_3 .

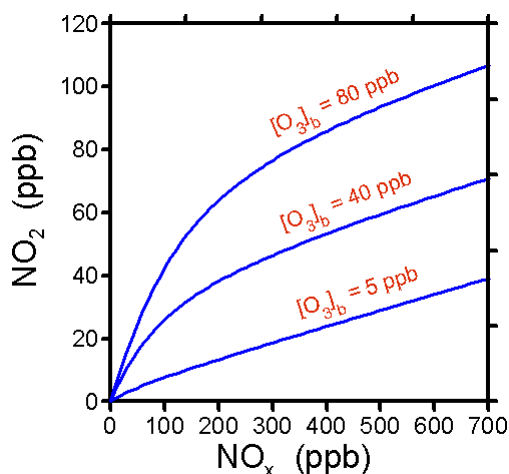


Figure 2J.4. Relation NO_2 - NO_x

The relationship between NO_2 and NO_x concentrations in the ambient air is non-linear and depends on the concentrations of ozone. The time scales characterising these reactions are of the order of tens of seconds, thus comparable with residence time of pollutants in a street canyon. Consequently, the chemical transformations and exchange of street canyon air with the ambient air are of importance for NO_2 formation.

- **Model structure**

The model is designed to work with input and output in the form of one-hour averages.

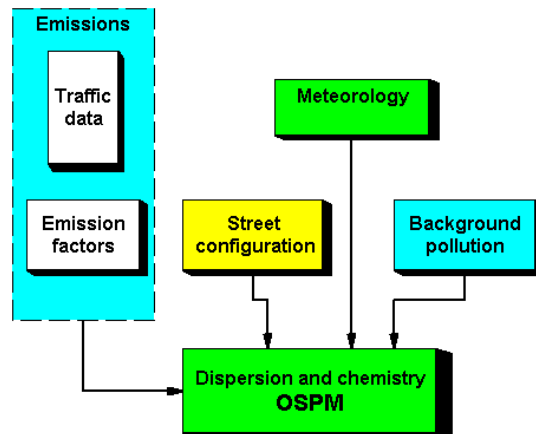


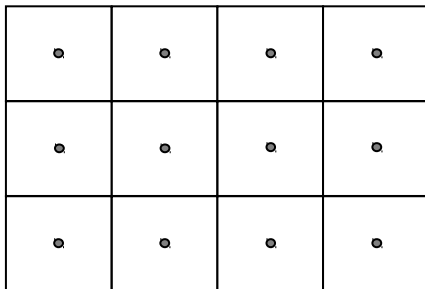
Figure 2J.5. Model structure

The required input data are hourly values of wind speed, wind direction, temperature and global radiation. The two last parameters are used for calculation of chemical transformation of NO-NO₂-O₃. The model requires also hourly values of urban background concentrations of the modelled pollutants. Beside the hourly input parameters, the model requires also the data on the street geometry and the traffic in the street.

Appendix 2K: Inhomogeneous Grid.

Definitions

A grid is a number of points, where each point can be said to represent the value for a square area or at least for the point itself:

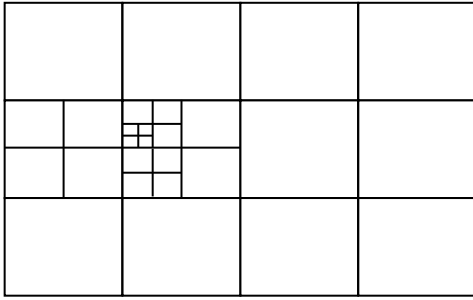


Concept

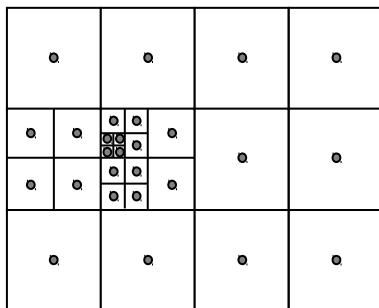
To achieve a grid with variable density (inhomogeneous) a quad tree structure will be used, i.e. a base grid consisting of quadratic grid squares that are then split into four recursively when needed. The main reason for choosing this structure above for example a triangular grid with free grid nodes is performance. The operation that is most time critical is to generate some kind of iso-lines/iso-surfaces from data in a grid, and an algorithm that draws iso-surfaces from a random set of dots (there are such algorithms) is not assumed to be fast enough for the requirements.

Grid definition

Example of a simple quad-tree grid structure, base grid is 4×3 squares:



Square geometry means fast calculations and possibilities to take short cuts when calculating value in a random receptor point or rendering for example iso-surfaces. When using a grid structure in a dispersion simulation, the structure of the receptor points is not essential. Example of receptor points from a quad-tree grid:



ASCII-representation of the grid (only list the splitted cells and their contents). Please note that the grid representation starts at lower left corner. All grid cells are referred to by x, y coordinate pairs, where 0,0 is the lower left corner. All indexing starts at 0. Cells should appear in order:

grid: 4 3

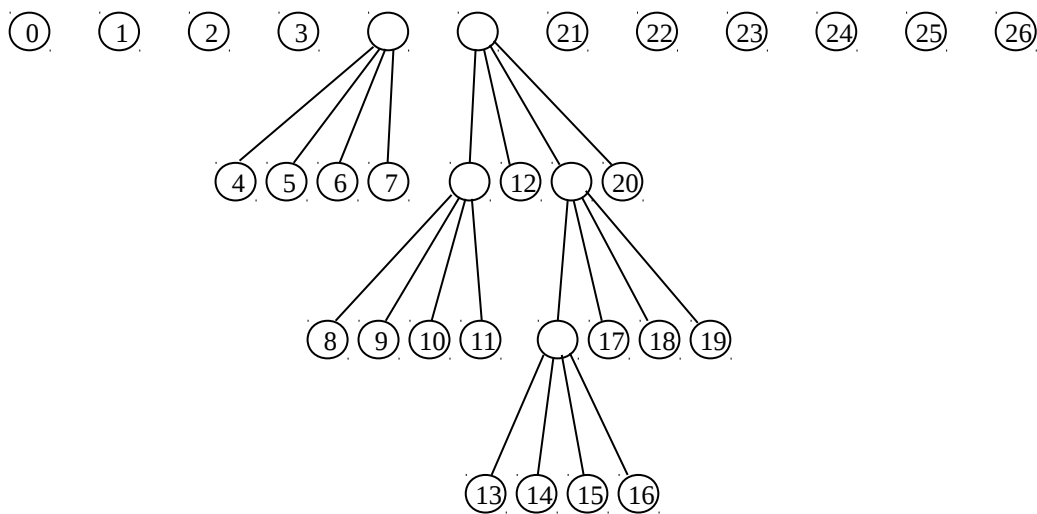
div: 2

area: 1261750 6395000 1281750 6410000

0,1 1,1{0,0 0,1{0,0}}

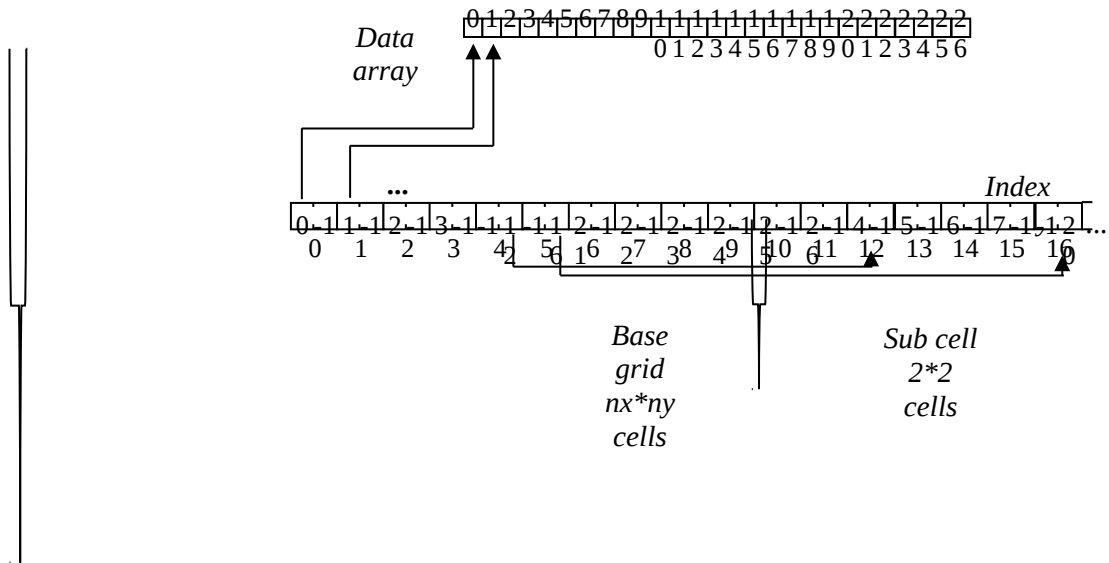
Quad tree as index

The resulting quad tree grid structure can also be viewed as an index to the cells. The base grid is the top nodes and the leaves of the search tree are the cells. The cells (leaves) are numbered in order if we traverse the tree depth-first:



Quad tree index representation

This tree structure can be represented internally as an array of integer tuples (index array): One data pointer into an array of data, and one index pointer into another position in the index array. At the start of the index array, there should be $n \times n$ tuples representing the base grid. Each tuple can then contain either a data pointer if it marks a leaf in the tree, or an index pointer if it marks a node in the tree:



Quad tree index ASCII representation

The above index structure can be dumped as a series of pairs of integers:

grid: 4 3

div: 2

area: 1261750 6395000 1281750 6410000

nrdata: 27

nridx: 32

0 -1 1 -1 2 -1 3 -1 -1 12 -1 16 21 -1 22 -1 23 -1 24 -1

25 -1 26 -1 4 -1 5 -1 6 -1 7 -1 -1 20 12 -1 -1 24 20 -1

8 -1 9 -1 10 -1 11 -1 -1 28 17 -1 18 -1 19 -1 13 -1 14 -1

15 -1 16 -1

or if indexing starts at 1 (Fortran):

grid: 4 3

div: 2

area: 1261750 6395000 1281750 6410000

nrdata: 27

nridx: 32

1 0 2 0 3 0 4 0 0 13 0 17 22 0 23 0 24 0 25 0

26 0 27 0 5 0 6 0 7 0 8 0 0 21 13 0 0 25 21 0

9 0 10 0 11 0 12 0 0 29 18 0 19 0 20 0 14 0 15 0

16 0 17 0

Grid data

The data can be given as a sequence of numbers. As given by the depth-first search of the tree, the order of data for the example above is:

23		24		25	26
6	7	18	19	20	22
		15	16		
		13	14	17	
4	5	10	11	12	
		8	9		
0	1	2	3		

Example of data in grid:

3		5			1	7
3	6	33	22	23	11	21
		1516	11			
9	1	1718	14			
		1 2				
		3 4				
19		17			16	12

Quad tree data in tree formatted ASCII:

19 17 16 12

{9 1 3 6} {{1 2 3 4} 14 {{ 17 18 15 16} 11 33 22} 23} 11 21

3 5 1 7

Quad tree data in unformatted ASCII:

19 17 16 12

9 1 3 6 1 2 3 4 14 17 18 15 16 11 33 22 23 11 21

3 5 1 7

or:

19

17

16

12

9

1

3

6

1

2

3

4

14

17

18

15

16

11

33

22

23

11

21

3

5

1

7

Grid generation

For the grid generation, a grid density function is used. The grid density function gives the maximum grid cell size at a certain distance from a source. A density function could be a gaussian function, like this:

$$F(L) = C_1 - C_2 * e^{-C_3 * L * L}$$

The function can be defined by the user by three parameters:

D_{base}	The grid size (m) of the base grid (implicit).
D_0	The grid size (m) at the source ($L=0$).
L_{base}	The distance from the source at which the function should give the base grid size (return to base grid).

The constants of the function can then be calculated as:

$$C_1 = D_{base} * 1.1$$

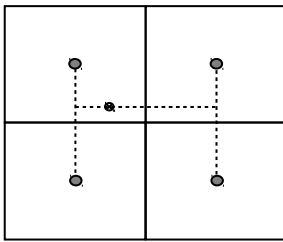
$$C_2 = C_1 - D_0$$

$$C_3 = -\ln((C_1 - D_{base}) / (L_{base} * L_{base}))$$

Fundamental problems

Receptor points

One essential operation of a grid is to extract the value in a receptor point. This is currently done by bi-linear interpolation between the closest four points:



If the grid is arranged as a quad-tree structure, this method is not directly applicable any more.

Re-sampling of fields

If the destination field has a higher resolution (smaller grid squares) than the source grid, it is possible to use a receptor point based re-sampling of the source field:

If the destination field has lower resolution (bigger grid squares) than the source field, or a similar resolution but is badly aligned with the source field, the receptor method is not very accurate any longer. Instead, an area-based method should be used:

Presentation of inhomogeneous fields

Grid generation

A homogeneous weight grid is created. The values in the grid could be for example the maximum size of a grid square. The criteria used for determining this grid can be for example: Emission, emission gradient, distance to certain source types, etc. It is probably necessary to have a user interface where you can change criteria and a couple of parameters and where you have a visual feedback on how the grid will look. A total number of grid points should also be shown along with some guidelines for what is a reasonable maximum number of grid points.

Storage

A new storage format will be added to the field database in Airviro. A meta data file will contain the grid definition. The grid definition serves as a geo-spatial index to the data file. Packed/unpacked storage should be available.

Grid definition

The grid definition needs to contain the following info:

nx, ny	Base grid dimensions.
xb, yb, xe, ye	Coordinate bounds for the base grid (meter).
bgrid[nx*ny]	Matrix of grid nodes
nodes[nrnodes]	Array of grid nodes.

Data

Data will be stored in a plain array. This makes it possible to do operations on data in a similar way as grid data in other formats.

Appendix 2L: References

Alpert, P., Getenio, B. (1988)

One-Level Diagnostic Modelling of Mesoscale Surface Winds in Complex Terrain. Part I: Comparison with Three-Dimensional Modelling in Israel.
American Meteorological Society, 116, 2025-2046.

Bahe, F.C., Schurath, U. and Becker, K.H. (1980)

The frequency of NO₂ photolysis at ground level, as recorded by a continuous actinometer.
Atmospheric Environment, 14, 711-718

Berkowicz, R. and Prahm, L. P. (1982)

Evaluation of the Profile Method for Estimation of Surface Fluxes of Momentum and Heat
Atmos. Environ. 16, 2809-2819

Briggs, G. A. (1973)

Diffusion Estimation for Small Emissions
ATDL Contribution File No. 79, Atmospheric Turbulence and Diffusion Laboratory

Businger, J. A., Wyngaard, J. C., Izumi, Y. and Bradley, E. F. (1971)

Flux profile Relationships in the Atmospheric Surface Layer.
J. Atmos. Sci., 28, 181-189

Cats, G. J. (?)

A Simple Method for Quick Estimates of Frequency Distributions of Air Pollution Concentrations from Long-Term Average Concentrations
Proceedings....

Chock, D.P. and Dunker, A.M. (1983)

A comparison of numerical methods for solving the advection equation.
Atmospheric Environment, Vol., 17, No.1, 11-22.

Danard, M. (1977)

A Simple Model for Mesoscale Effects of Topography on Surface Winds.
Monthly Weather Review.,99,831-839

Draxler, R. (1976)

Determination of Atmospheric Diffusion Parameters.
Atmospheric Environment, 10, 95-105

Dyer, A. J. (1974)

A Review of Flux-Profile Relationships.
Boundary Layer Meteorology, 7., 363-372

Eliassen, A. and Saltbones, J. (1983)

Modelling of the long range transport of sulphur over Europe: A two-year model run and some modelling experiments.
Atmospheric Environment, 17, 1457-1473

Hanna, S. R., Briggs, G. A. and Hosker Jr, R. P. (1982)

Handbook on Atmospheric Diffusion
Technical Information Center, U. S. Department of Energy

Hanna, S.R. and Drivas, P.J. (1987):

Guidelines for use of Vapor Cloud Dispersion Models,
American Institute of Chemical Engineers, ISBN 0-8169-0403--0

Hildemann, L.M., Russel, A.G. and Cass, G.R. (1984)

Ammonia and nitric acid concentrations in equilibrium with atmospheric aerosols: Experiment vs theory.
Atmospheric Environment, 17, 1737-1750

Holtslag A. A. M. (1984)

Estimates of Diabatic wind Speed Profiles from Near Surface Weather Observations.
Boundary Layer Meteorology 29, 225-250.

Iversen, T., Saltbones, J., Sandnes, H., Eliessen, A. and Hov, O. (1989)

Airborne transboundary transport of sulphur and nitrogen over Europe - model description and calculations.
EMEP MSC_W report 2/89

Laurin, S. And Omstedt, G. (1987)

En modell för beräkning av bilavgaser i gatumiljö.
Swedish Meteorological and Hydrological Institute (SMHI)

Long, P.E.Jr. and Pepper, D.W. (1976)

A comparison of six numerical schemes for calculating the advection of atmospheric pollution.
Am. Meteor. Soc. Symp. Atmos. Turbul. Diffus. Air Qual. Prepr. 3rd, 181-187.

Lyons, T. J. and Scott, W. D. (1990)

Principles of Air Pollution Meteorology
Belhaven Press, London

Marchuk, G.I. (1975)

Methods of Numerical Mathematics.
Springer, New York.

Mass, C. (1981)

A Single-Level Numerical Model Suitable for Complex Terrain.
Proceedings or the Fifth Conference on Numerical Weather Prediction, American Meteorological Society, Boston, MA 02108, 316-319.

Mass, F. (1984)

Simple Mesoscale Models for Operational Use in Regions of Topographic and Thermal Forcing: Development and Evaluation.
Contractor report CR 84-05, Naval Environmental Predictions Research Facility, Monterey, CA 93943

McMahon, T. A. and Denison, P.J. (1979):

Empirical Atmospheric Deposition Parameters-A survey,
Atmospheric Environment, 13: 571-585.

McRae, G.J., Goodin, W.R. and Seinfeld, J.H. (1982)

Numerical solution of the atmospheric diffusion equation for chemically reacting flows.
Journal of Computational Physics, 45., 1-42

Messinger, F., and A. Arakawa. (1976)

Numerical Methods Used in Atmospheric Models.
Vol I. GARP Publications Series No. 17.

Nielsen, S. P., Gryning, S. E., Karlberg, O., Lyck, E. and Thykier-Nielsen, S. (1986)

A Field Experiment to Validate Atmospheric Dispersion and Dose Models.
Risø National Laboratory, DK-4000 Roskilde, Denmark

Omstedt, G. (1984)

An Operational Air Pollution Model using Routine Meteorological Data.
SMHI Report RMK 39, Norrköping, Sweden.

Panofsky, H. A. and Dutton, J. A. (1984)

Atmospheric Turbulence.
John Wiley & Sons, Inc.

Paulson, C. A. (1970)

The Mathematical Representation of Wind Speed and Temperature Profiles in the Unstable Atmospheric Surface Layer.
J. Appl. Meteorol., 9, 857-861.

Pielke, R.A. (1984)

Mesoscale meteorological Modelling.
Academic Press, ISBN 0 12 554820 6.

Richtmeyer, D. R., Morton, K. W. (1967)

Difference Methods for Initial-Value Problems.
Interscience, 405 p.

Russel, A., McRae, G. J. and Cass, G. C. (1983)

Mathematical modelling of the formation and transport of ammonium nitrate aerosol.
Atmospheric Environment, 17, 949-964

Sehmel, G.A. (1980):

Particle and Gas Dry Deposition - A review,
Atmospheric Environment, 14:983-1012.

Sherman, C. A. (1978)

A Mass-Consistent Model for Wind Fields over Complex Terrain.
J. Appl. Meteor., 17, 312-319.

SNV

Bilavgaser i Gatumiljö - Modell och Modell Test
SNV PM 891, Bilagor 1977

Stelson, A. W. and Senfeld, J. H. (1982)

Relative humidity and temperature dependence of the ammonium nitrate dissociation constant.
Atmospheric Environment, 16, 983-992

van Stijn, Th.L., van Eijkeren, J.C.H. and Praagman, N. (1987)

A comparison of numerical methods for air quality models.
Scientific Reports/ KNMI WR 876, ISSN 0169 1651.

Winter, S., Nyrén, K. and Karlsson, E. (1984):

Uppkomst och utbredning av explosiva eller giftiga gasmoln,
FOA rapport E 40011.

Yamartino, R.J., Scire, J.S., Carmichael, G.R., Chang, Y.S. (1992):

The CALGRID Mesoscale Photochemical Grid Model – I. Model Formulation
Atmospheric Environment, Vol 26A, No 8, pp. 1493-1512.

Yanenko, N.N. (1971)

The Method of Fractional Steps.
Springer, New York.

Zilitinkevich, S. S. (1972)

On the Determination of the Height of the Ekman Boundary Layer.
Boundary Layer Meteorol., 3, 141-145

Appendix 2M: Noise

Scope

To formulate and implement a screening type sound propagation model. The model does not, for now, consider more complex terrain and/or geographical conditions. The model will be implemented in such a way that it allows later developments for more advanced applications.

Introduction

This model is, as a start, relates to rather simple terrain conditions. Only flat terrain and no obstructions like buildings or vegetation are considered.

The model is based on specifications laid out in the document ISO 9613-2. Some theory and definitions concerning the sound propagation are given below.

- *Definition - Equivalent continuous A-weighted sound pressure level, L_{AT} :*

Sound pressure level, in decibels, defined by:

$$L_{AT}=10*\log(((1/T)\int p_A^2(t)dt)/p_0^2) \quad (1)$$

$p_A(t)$ = is the instantaneous A-weighted sound pressure i Pa

p_0 = is the reference pressure ($20*10^{-6}$ Pa)

T = is the specified time interval, in seconds

The A-frequency weighting is that specified for sound level meters in IEC 651. (The time interval T should be long enough to average the effects of varying meteorological parameters. The situation considered here is short-term downwind overall averages).

The equivalent continuous downwind octave-band sound pressure level, $L_{fT}(DW)$:

- *Definition - Sound pressure level, in decibels, defined by:*

$$L_{fr}(DW) = 10 \cdot \log\left(\frac{1}{T} \int p_f^2(t) dt\right) / p_0^2 \quad (2)$$

where $p_f(t)$ is the instantaneous octave-band sound pressure downwind in Pa, and the subscript f represents a nominal mid-band frequency of an octave-band filter.

- *Definition - Insertion loss (of a barrier):*

Difference, in decibels, between the sound pressure levels at a receiver in a specified position under two conditions;

- a) with the barrier removed, and
- b) with the barrier present

and no other significant changes that affect the propagation of sound.

Source description

The equations to be used are for the attenuation of sound from point sources. Extended noise sources, therefore, such as road and rail traffic or an industrial site (which may include several installations or plants, together with traffic moving on the site) shall be represented by a set of sections (cells), each having a certain sound power and directivity. Attenuation calculated for sound from a representative point within a section is used to represent the attenuation of sound from the entire section. A line source may be divided into line sections, an area source into area sections, each represented by a point source at its centre.

A group of point sources may be described by an equivalent point source situated in the middle of the group, in particular if

1. the sources have approximately the same strength and height above the local ground plane,
2. the same propagation conditions exist from the sources to the point of reception, and

3. the distance d from the single equivalent point source to the receiver exceeds twice the largest dimension H_{\max} of the sources ($d > 2H_{\max}$).

If the distance d is smaller ($d \leq 2H_{\max}$), or if the propagation conditions for the component point sources are different, the total sound source shall be divided into its component point sources.

Meteorological conditions

Downwind propagation conditions for the method specified in this part of ISO 9613 are as specified in 5.4.3.3 of ISO 1996-2:1987, namely,

- Wind direction within an angle of $\pm 45^\circ$ of the direction connection the centre of the dominant sound source and the center of the specified receiver region, with the wind blowing from source to receiver, and
- Wind speed between approximately 1 m/s and 5 m/s, measured at a height 3 m to 11 m above the ground.

The equations for calculation the average downwind sound pressure level $L_{AT}(DW)$ in this part of ISO 9613, including the equations for attenuation given below, are the average for meteorological conditions within these limits. The term average here means the average over a short time interval, as described earlier.

These equations also hold, equivalently, for average propagation under a well-developed moderate ground-based temperature inversion, such as commonly occurs on clear, calm nights.

Basic equations

The equivalent continuous downwind octave-band sound pressure level at a receiver point, $L_{rT}(DW)$, shall be calculated for each point source, and its image sources, and for the eight octave bands with nominal mid-band frequencies from 63 Hz to 8 kHz, from equation (3) below:

$$L_{rT}(DW) = L_W + D_C - A \quad (3)$$

- L_w = is the octave-band sound power level, in decibels, produced by the point sound source relative to a reference sound power of one pW.
- D_c = is the directivity corrections in decibels, that describes the extent by which the equivalent continuous sound pressure level from the point source deviates in a specific direction from the level of an omni-directional point source with power L_w . For an omni-directional point sound source radiating into free space, $D_c = 0$.
- A = is the octave-band attenuation that occurs during propagation from the point sound source to the receiver.

The attenuation term A is given by:

$$A = A_{div} + A_{atm} + A_{gr} + A_{bar} + A_{misc} \quad (4)$$

- A_{div} = is the attenuation due to geometrical divergence
- A_{atm} = is the attenuation due to atmospheric absorption
- A_{gr} = is the attenuation due to ground effect
- A_{bar} = is the attenuation due to a barrier
- A_{misc} = is the attenuation due to miscellaneous other effects

The *equivalent continuous A-weighted downwind sound pressure level* shall be obtained by summing the contributing time-mean-square sound pressures calculated according to (3) and (4) for each point sound source, for each of their image sources and for each octave band, as shown in equation (5):

$$L_{AT}(DW) (i,j) = 10 * \log(\sum(\sum 10^{0.1(L_{fT}(i,j) + A_f(j))}) \quad (5)$$

- $A_f(j) = 20 * \log(Ra(f(j))) + 2.0$
- $Ra(f(j)) = 12200^2 * f(j)^4 / ((f^2 + 20,6^2) * (f^2 + 12200^2) * \sqrt{f^2 + 107.7^2} * \sqrt{f^2 + 737.9^2})$

Where the inner summing is over octave-band (j) and the outer summing is over sources (i). A_f denotes the standard A-weighting (see App 1)

Calculation of the attenuation terms

Geometrical divergence (A_{div})

The geometrical divergence accounts for spherical spreading in the free field from a point sound source, making the attenuation, in decibels, equal to

$$A_{div} = 20 \cdot \log(d/d_0) + 11 \quad (6)$$

d = is the distance from the source to receiver in metres

d_0 = is the reference distance, 1 m

Atmospheric absorption (A_{atm})

The attenuation due to

Atmospheric absorption A_{atm} , in decibels, during propagation through a distance d , in metres is given by:

$$A_{atm} = \alpha \cdot d \quad (7)$$

α = atmospheric attenuation coefficient, decibels/m, for each octave band at the mid-band frequency. The atmospheric attenuation coefficient depends strongly on the frequency of the sound, the ambient temperature and relative humidity of the air, but only weakly on the ambient pressure.

Table 1. Atmospheric attenuation coefficient α for octave bands of noise

Ground effect (A_{gr})

Ground attenuation, A_{gr} , is mainly the result of sound reflected by the ground surface interfering with the sound propagation directly from the surface to receiver. The downward-curving propagation (downwind) ensures that this attenuation is determined primarily by the ground surfaces near the source and near the receiver. This method of calculating the ground effect is applicable only to ground which is approximately flat, either horizontally or with a constant slope. Three distinct regions for ground attenuation are specified:

- a) the source region, stretching over a distance from the source towards the receiver of $30h_s$, with a maximum distance of d_p (h_s is the source height, and d_p the distance from source to receiver, as projected on the ground plane);
- b) the receiver region, stretching over the distance from the receiver back towards the source of $30h_r$, with a maximum distance of d_p (h_r is the receiver height);
- c) a middle region, stretching over the distance between the source and receiver regions. If $d_p < 30h_s + 30h_r$, the source and receiver regions will overlap, and there is no middle region.

According to this scheme, the ground attenuation does not increase with the size of the middle region, but is mostly dependent on the properties of source and receiver regions.

In this work we have used what in ISO 9613 is called "Alternative method of calculation for A-weighted sound pressure levels". Under the following specific conditions,

- only A-weighted sound pressure level at the receiver position is of interest
- the sound propagation occurs over porous ground or mixed ground most of which is porous
- the sound is not a pure tone

and for ground sources of any shape, the ground attenuation may be calculated from equation (8):

$$A_{gr} = 4.8 - 2 \cdot h_m / d \cdot (17 + 300/d) \geq 0 \quad (8)$$

- h_m is the mean height of the propagation path above the ground, in metres
- d is the distance from the source to receive, metres

Negative values for A_{gr} shall be replaced with zeros.

When the ground attenuation is calculated using equation (8), the directivity correction D_c in equation (3) shall include a term D_Ω , in decibels, to account for apparent increase in sound power level of the source due to reflections from the ground near the source.

$$D_\Omega = 10 \cdot \log^* \left(1 + \frac{d_p^2 + (h_s - h_r)^2}{d_p^2 + (h_s + h_r)^2} \right)$$

- h_s = is the height of the source above the ground in metres;
- h_r = is the height of the receiver above the ground, in metres;
- d_p = is the source -to-receiver distance projected onto the ground plane, in metres.

The model

Calculate the terms,

$D_C = (A_{div} + A_{atm} + A_{gr})$, for each source – receptor and source strength $L_w(f)$ for each midband frequency, $L_{rT}(DW)(i,j)$.

Calculate,

$$L_{AT}(DW)(i,j) = 10 * \log(\sum(\sum 10^{0.1(LfT(i,j)+Af(i))}))$$