

GRAL Manual

GRAL Graphical User Interface 22.09

The following persons contributed to this report:

Authors:

Mag. Dr. Dietmar Öttl
Ing. Markus Kuntner

Map sources

GIS-Styria
TIRIS Maps Tyrol

Content

1. Changes to previous versions	7
1.1. Changes from version 20.09 to 21.09	7
1.2. Changes from version 20.01 to 20.09	10
1.3. Changes from version 19.1 to 20.01	11
1.4. Changes from version 18.1 to 19.01	12
1.5. Changes from version 17.9 to 18.1	13
1.6. Changes from version 17.8 to 17.9	13
1.7. Changes from version 17.1 to 17.8	14
2. Note – Compatibility to previous program versions	15
3. Introduction.....	16
4. Installation of GRAL.....	17
5. Basic concepts for a new project.....	19
5.1. The Main Window	24
6. Create or open a project.....	25
6.1. GUI Settings.....	26
6.2. Show Log Files	27
7. Save a project	28
8. GRAL Settings - Main control parameters for GRAL	29
8.1. Column „GRAL General“.....	30
8.2. Column „Concentration grid“	36
8.3. Column „GRAL Internal flow field grid“	37
9. Domain.....	43
10. Definition of source groups.....	44
11. GIS Interface	47
11.1. The Toolbox.....	48
11.2. The Lock-symbol.....	49
11.3. The object manager	49
11.3.1 Removing maps or items from the drawing stack.....	50
11.4. Layout manager	51
11.5. Add maps.....	55
11.6. Import of already geo-referenced maps	55
11.7. Geo-referencing of pixel graphics	56
11.7.1 Geo-referencing using a map scale and one reference point	56
11.7.2 Geo-referencing using two reference points.....	58
11.8. Adjustment of geo-referenced base maps	59
11.9. Store and recall map details.....	60
11.10. Definition of the GRAL model domain.....	61
11.11. Editing emission sources and obstacles	62

11.11.1	Point sources	62
11.11.2	Line sources and emissions from road traffic	64
11.11.3	Area sources.....	69
11.11.4	Tunnel portals	72
11.11.5	Edit deposition settings	74
11.11.6	Buildings and obstacles	76
11.11.7	Removing obstacles at the edge of a GRAL domain	77
11.11.8	Receptor points.....	78
11.11.9	Walls.....	80
11.11.10	Vegetation.....	81
11.11.11	Time series for exit temperature and exit velocity.....	83
11.12.	Selection of items (sources, buildings, receptors)	84
11.12.1	Item Context Menu (edit, copy and paste an item)	84
11.13.	Select – Search items.....	85
11.14.	Import of sources, receptor points, buildings, wind data.....	87
11.14.1	Import of sources, receptor points, buildings	87
11.14.2	Import of wind roses.....	89
11.15.	Export sources, rec. points, buildings or contour lines.....	91
12.	Number of concentration and flow field cells.....	92
13.	Select source groups and a pollutant for dispersion calculations.....	93
13.1.	Emission modulation	94
13.1.1	Diurnal and seasonal emission modulation	94
13.1.2	Emission modulation on an hourly basis for a time series or the transient GRAL mode	96
13.2.	Wet deposition settings	98
13.3.	Decay rate	99
14.	Meteorological data.....	100
14.1.	Import data	100
14.2.	Classify meteorological data.....	103
14.3.	Analyze meteorological data	104
15.	Starting GRAL and analyzing results	107
15.1.	Starting GRAL	107
15.2.	The “Project locked” function.....	109
15.3.	GRAL simulations in complex terrain	110
15.4.	Check the rasterized GRAL buildings and walls	111
15.5.	Analyzing GRAL results.....	112
15.5.1	Evaluation for any pollutants except odour.....	112
15.5.2	Evaluation of odour simulations	115
15.6.	Visualizing concentration and other kinds of maps.....	117
15.7.	Further analysis tools	118
15.7.1	Convert a *.grz or a *.con file to an ASCII file.....	118
15.7.2	Mathematical raster operations (e.g.NO2 conversion)	118

15.7.3	Source apportionment at a location.....	119
15.7.4	Extract concentration values	120
15.7.5	Create animated GIF file	121
15.7.6	Show vertical profiles	121
15.7.7	Show vertical profile along a section line.....	122
15.7.8	Generate a concentration time series for a raster cell	123
15.7.9	Generate a meteorological time series for a raster cell	124
16.	Flow field simulations with GRAMM	125
16.1.	Define grid.....	125
16.2.	Define model domain	126
16.3.	Generate GRAMM topography	126
16.4.	Consideration of land-use data	127
16.5.	Adjusting GRAMM control parameters.....	130
16.6.	Check the GRAMM geometry	133
16.7.	Starting GRAMM.....	134
16.8.	Visualizing GRAMM computations online	135
16.9.	Post processing GRAMM wind fields: The Re-Order function.....	137
16.10.	Evaluation of GRAMM results	138
16.11.	Import of existing wind field libraries	142
16.12.	The Match-to-Observation Function.....	143
16.13.	Exporting GRAMM flow fields for sub-domains.....	150
17.	Miscellaneous.....	151
17.1.	Horizontal and vertical slices for GRAMM and GRAL flow fields	151
17.1.1	Horizontal slice	151
17.1.2	Vertical slice	152
17.1.3	3D view of sections for GRAMM and GRAL surfaces and wind vectors.....	153
17.2.	Measure distances.....	154
17.3.	Measure areas	154
17.4.	Absolute heights	155
17.5.	Cell heights	155
17.6.	Import GRAL topography	156
17.7.	Edit and visualize the GRAL topography	156
17.8.	Moving (archiving) a project to a different directory	157
17.9.	Archiving a project after finishing all work	158
17.10.	Invalid numerical inputs.....	158
18.	Further map features.....	161
18.1.	North arrow	161
18.2.	Scale bar.....	161
18.3.	Saving a map as pixel graphics	162
18.4.	Copy a map to the clipboard	162

19. Appendices	163
19.1. Methodology „All in all out stables“	163
19.2. Methodology “compost works”	163
19.3. File formats.....	165
19.3.1 ESRI-ASCII format.....	165
19.3.2 Topography and landuse data	166
19.3.3 Meteorological files *.met.....	166
19.3.4 Emission_Mod_Diurnal.txt and Emission_Mod_Seasonal.txt	166
19.3.5 DepositionSettings.txt	166
19.3.6 Emissions_timeseries.txt.....	167
19.3.7 Precipitation.txt	167
19.4. Mouse- and key-references.....	168
19.5. The match to observation algorithm	171
19.6. GUI Error messages.....	173

1. Changes to previous versions

1.1. Changes from version 21.09 to 22.09

New features

- The application has been transformed to .NET6
It provides the following benefits:
 - Improved High-DPI support for scaled high resolution screens
 - Better performance: reading *.wnd files improved by a factor of 2, redraw of huge concentration maps about 30% faster
 - publishing of single files including the entire required framework
 - native Windows-ARM support
 - improvements due to future developments by Microsoft
- The routine for reading in the wind fields no longer breaks if a wind field is missing or cannot be read, but searches for other existing wind fields and reads them if possible.
- During the match process the original meteorological file and a preview of the matched wind rose at a wind measurement point can be displayed, but now it is also possible to write a temporary *.met file, which can be analyzed during the match process in the Meteorology tab.
- In the Layout Manager, the contour line file and the layer name can be checked and changed (see chapter 11.4)
- New dialog for the statistical conversion equations from NO_x to NO₂
- When evaluating the concentrations, the output window is no longer closed automatically. The average emission modulation for each source group and the output files are displayed.
- When displaying the total emissions, the evaluation of the emission in the transient mode is now performed for the available time period

Bug fixes

- The wind roses for stability classes are determined the same way as the wind roses of the wind speed with classified wind data and applied bias correction (rounding adjusted)
- The display of vertical slices of 3D concentration fields (transient mode) has been fixed
- Enumeration issues have been fixed when selecting objects using the left or right mouse button
- Files in the Meteorology folder are no longer deleted automatically
- Duplicate entries are fixed when pasting coordinates to receptor and point sources
- Bugs in the control view when showing emissions are fixed

1.2. Changes from version 20.09 to 21.09

New features

- Enable the creation of concentration time series for arbitrary raster cells as post processing based on the *.grz files (Menu bar “Analyze results” – “Generate time series for several evaluation points”)

- Enable the creation of GRAL meteorological time series for arbitrary raster cells based on the *.gff files (Windfield analysis – Windrose button or menu bar “Windfield analysis” – “Wind statistics at a point”)
- New features for the “Match-to-Observation” function including an automatic tuning option
- Support for various wind rose sector widths in the wind rose view and new wind rose design. Improved quality of wind evaluations (Velocity Classes, Stability Classes,..) copied to the clipboard
- Further improvement of the dual screen usability
- Support for the new GRAL options (deactivation of GRAL online functions and the prognostic sub domain reduction, see Reduce the size of prognostic sub domains)
- New option for displaying GRAL or GRAMM wind vectors in a relative height above terrain or at a height above sea (absolute height)
- Item dialogs are no longer TopMost windows but child forms of the Domain window. This means that these dialogs do not lose their focus when the mouse pointer leaves the dialog. In addition, the dialog will be minimized with the domain window. Furthermore, the dialogs are clipped to the Domain window. Each item dialog got an additional cancel button to avoid unintentional modifications. The question whether a new shape of a source should be applied is no longer asked for new sources, where no shape is available yet.
- The usage of the unit kg/h as input for the emission rates of line sources in the user dialog has been enabled. The values are still saved in kg/h/km and converted within the dialog using the total length of the line source
- Improved performance for the high percentile calculation
- Further progress bars showing the real progress and additional cancel buttons for further long taking operations
- New GUI settings option: enable or disable automatic scaling when loading or drawing vector maps (chapter 6.1).
- GUI settings: it is possible either loading the entire wind file (default) or to discarding lines with wind speed 0 and wind direction 0 or shuffle the wind direction for lines with wind speed 0.
- GUI settings: you can now select whether GRAL concentration files and GRAMM wind fields should be deleted or moved to the recycle bin when they are deleted (chapter 6.1). This option is enabled (move to the recycle bin) by default!
- The GRAMM grid heights are displayed either for each cell (as used up to V 21.09) or per corner point (as used in GRAMM) using the new menu entry “View → Show cell heights for → “
- Now you can write meteorological files *.met from the GUI. Therefore, for example, an imported *.akt or *.akterm file can be saved as a *.met file or an imported subset of a large *.met file (e.g. one or more months or days) can be saved as new *.met file.

Changes to previous versions

Bugfixes

- Fixed the reuse of shape objects (problem, when using shape files with multiple parts)
- Fixed rounding errors in the wind rose visualization
- Reset the new option for the conversion from kg/h to kg/h/km when selecting a new line source using the slider
- Reset the cell heights view when creating a new GRAL domain area
- Fixed a bug when a wall has only 2 edge points (such walls were ignored when reloading the project)
- Correction of the cell index calculation (elimination of rounding errors from double to int)
- Fixed invalid reset cancellation tokens (avoid NullPointerException)
- Fixed a bug in the GRAMM terrain mesh generation (one offset error); use a new terrain interpolation for the GRAMM terrain mesh, fix incorrect NoData handling
- Disabled controls in the “Special Settings” dialog if a project is locked
- Fixed a bug, if no item is selected in the most recent files dialog
- Fixed the display of the adaptive roughness settings when loading a project
- Further improvements for the Linux version
- Fixed a rare error when reading the *.scl files or writing *.wnd files for the GRAMM export function
- The unit of odor sources at the item info form (MOU/h instead of OU/h) has been fixed
- Removed double clicks when setting edge points
- It is possible that an area source is not included in the file cadestre.dat if too large grids are set for area sources or the area sources are strongly divided. To avoid this problem, the raster size is decreased during rasterization until at least two subareas are generated per area source.
- Fix an incorrectly displayed sum emission in the deposition dialog
- Enable all supported separator characters when showing the file emissions_timeseries.txt or when summing up the total emissions
- A possible division by 0 has been fixed in the emission modulation preview window
- Disable the selection of source groups for the receptor evaluation, because the evaluation delivers incorrect results when some source groups are not included
-

1.3. Changes from version 20.01 to 20.09

New features

- Delete or edit item data (name, height, source group number, emission rate) within the spreadsheet search and filter dialog (chapter 11.13)
- Geo referenced item information is shown in an own layer (chapter 11.12)
- A wind speed cumulative distribution diagram has been added (chapter 14.3)
- The legend and the info box within the wind rose form can be moved (chapter 14.3)
- Ascending or descending line sources have been enabled (chapter 11.11.2)
- Shape file types PointZ and PolylineZ are supported
- It is possible to show solely already existing weather situations in the dispersion situation dialog
- Multiple height slices can be selected for the evaluation of GRAL results
- Enable access to special GRAL options from the GUI (wrench button)
- The building rasterization process can be controlled by the user (Building cell coverage
-)
- The evaluation routines support large domain ranges by using jagged arrays
- Enable a progress bar for the mathematical raster operations
- Support for spatially-varying surface roughness values ("Adaptive surface roughness"**Adaptive surface roughnessAdaptive surface roughness**)

Bugfixes

- Fixed the reading of the changed file format for the GRAL receptor meteorology file data (GRAL_Meteozeitreihe.dat)
- Fixed the shape import for receptors
- The anemometer height from *.met files is copied to the match to observation dialog
- Fixed the 3D view of GRAL topographies with receptors
- Fixed several tooltips
- Fixed the GRAMM start process for the new *.exe GRAMM core for multiple instances
- Fixed an error when creating the file lln.dat with Taiwanese date format

Changes to previous versions

- Fixed a crash when the object manager was started but no layers were available
- Improved reading of background maps when the absolute path in the bitmap world file has been changed
- Fixed an error in the color and value settings for contour lines in the layout manager when a vertical scroll bar is present
- Fixed the selection of traffic situations in the NEMO line source dialog and a bug in the NEMO interface
- Fixed the windrose dialog for tabbed *.met files
- Fixed the meteorological analysis tool for empty or invalid *.met files
- Enabled large arrays for the evaluation of result files (> 2 GByte)
- The evaluation of DayMax values has been fixed
- Commas or semicolons at time series names are replaced by an underline character

1.4. Changes from version 19.1 to 20.01

- Support for the new high-performance .NET Core 3.1 GRAL and GRAMM cores (see chapter 4)
- Optional new contour line drawing mode “Spline Line” (chapter 11.4)
- Shape line export for contour lines (spline mode only, chapter 11.15)
- Support for timeseries of exit temperature and exit velocities for point sources and portal sources in the transient GRAL mode (chapter 11.11.11)
- New options and improvements for wind rose visualization (chapter 14.3)
- Show wind roses in the GIS – domain window (chapter 11.14.2)
- Creation of animated GIF files (chapter 15.7.5)
- Additional options for the matching algorithm (chapter 16.12)
- New dialog to set and reload viewframes (chapter 11.9)
- Improved multiple display support
- Copy & paste for objects; copied objects can be rotated in 10 degree steps using the keys “R” and “L” while the object is floating (chapter 11.12.1)
- Delete and add edge points for buildings, area sources and line sources using the context menu (chapter 11.12.1)
- If an object is selected, an additional dialog appears if several objects overlay each other

- Information of selected objects is shown in tooltips instead of windows
- Long taking evaluations got a progress bar showing the real progress and a cancel button
- Some time-consuming functions are started in own threads to reduce the impact to the UI thread
- Vertical profiles: the forms with the vertical graph is refreshed at each new point instead of creating new forms; this makes it easier to check multiple points
- Support for variable stretching factors for the flow field calculation (chapter 8.3)
- Support for new GRAL result file formats
- Support for source group dependent decay rates (chapter 13.3)
- Object manager: toggle the visibility of a layer using the space key and start the layout manager for a selected layer using the return key
- Item forms: the windows close button and an OK button have been added
- Optimized buildings rasterization algorithm (improved accuracy)

1.5. Changes from version 18.1 to 19.01

- Transient dispersion simulations are enabled (see chapt. 8.1)
- Vertical concentration profiles can be visualized in the transient GRAL mode (see chapt. 15.7.6)
- Wet deposition can be considered in the transient GRAL mode (see chapt. 13.2)
- A decay rate for bioaerosols can be considered (see chapt. 13.3)
- Influence of vegetation can be considered (see chapt. 11.11.10)
- Percentiles of odour concentrations can be computed
- Numerical values can be assigned to receptors and displayed
- The GUI settings can be stored in the user directory (see chapt. 6.1)
- Copying the computation cores GRAL and GRAMM to the project folder is optional (see chapt. 6.1)
- More extensive implementation of the local culture for numeric inputs (see chapt. 17.10)
- The file "emissions_timeseries.txt" can be generated and visualized (see chapt. 13.1.2)
- Adapted input dialogs for line sources and tunnel portals (see chapt. 11.11.2)

1.6. Changes from version 17.9 to 18.1

- Preview of hourly and monthly modulations in the sources tab (chapt. 13)
- Double click on items in the “Used source groups” listbox at the sources tab not mandatory anymore
- 3D view for GRAL topography enabled (chapt. 17.7)
- New edit tools for the GRAL topography (chapt. 17.7)
- Support for the new high-performance .NET Core 2.0 GRAL and GRAMM cores
- Additional validation check and progress information while importing topography and landuse files
- The German Klug-Manier stability classes in *.akterm files are slightly different assigned to GRAL classes in order to get a better representation of Obukhov lengths in the GRAL simulation.
- Font size is made adjustable and the automatic scaling can be locked in the wind-rose chart
- Fine tuning to optimize geo-referenced maps has been implemented (see chapt. 11.8)
- Bug fixes
 - The GUI has written invalid decimal separators in the “Topography.txt” file in some language configurations
 - Some crashes at the 3D view were fixed
 - Shape file import and export for area sources produced errors sometimes
 - In some cases the pie diagram for source group apportionment did not work correctly
- The pollutants SO₂, NO₂, NH₃, and NMVOC have been erroneously taken from NEMO (= road traffic emission model).

1.7. Changes from version 17.8 to 17.9

At the main form and the GIS interface

- Just another serious error was found and fixed regarding the GRAMM export function (see chapt. 16.13)
- An error when using the match-to-observation function repeatedly, without closing the dialog window, has been fixed.
- An error occurring when using “Mean, Max, Daily Max” and “Receptors” in cases with only one or two computed dispersion situations has been fixed.
- The relative or absolute height and the vertical extension of line sources can additionally be defined by the user. Please make sure that all source groups to be used for line sources are defined before creating any line source. Only explicitly defined source groups are available in the editing modus for line and portal sources.

- A Douglas-Peucker point reduction is available at the shape-import of line sources.
- The encoding for strings at the shape-file import can be selected.
- The memory management is optimized.

1.8. Changes from version 17.1 to 17.8

At the main form and the GIS interface

- Automated computation of wind statistics at all receptor points is enabled.
- Improved match-to-observation algorithm: fine tuning based on error estimates implemented; new weighting factor for wind direction
- New “flat terrain option”, which improves GRAMM flow-field simulations in nearly flat or completely flat terrain
- Additional optional proxy data for buildings introduced: address, house number
- New files for the initially seasonal and daily emission modulation introduced: "Emission_Mod_Diurnal.txt" and "Emission_Mod_Seasonal.txt"
- The topography height is displayed on the top right corner of the GIS interface: when available the GRAL otherwise the GRAMM model cell height.
- New option to import the GRAL topography at the GIS window; improved (faster) import algorithm
- Optionally, absolute building and point source heights can be used. Improves handling of building roofs in complex terrain, i.e. flat roof top instead of terrain-following roof top.
- New object “Wall” has been introduced: Allows for easily digitizing walls, handled as buildings in the model with a width equal to the horizontal raster size of the microscale flow-field model.
- The editing of source groups for line- and tunnel-portal sources has been facilitated.
- The object manager is actualized, if new items are loaded. Layout managers are closed automatically, if the redraw order in the object manager is changed, to avoid wrong assignments between layout and object manager
- A serious error was found and fixed regarding the GRAMM export function (see chapt. 16.13)

2. Note – Compatibility to previous program versions

The current Graphical User Interface will work with the GRAMM versions 19.01 and GRAL versions 20.01 and upwards.

Existing projects starting with version 17.08 can be analyzed with the new GUI. Older projects need to be opened with the GUI version 18.01 or 19.01, the GIS window (domain window) must be opened, and any setting needs to be changed (e.g. the drawing order in the object manager). This procedure writes setting files in a more modern format, which can also be read by the current GUI. After this process, the current GUI can also load older projects.

New projects need to be launched using the actual GRAMM and GRAL versions. GUI versions up to 20.06 are not compatible with some of the latest output-formats.

3. Introduction

The following User Guidebook is primarily intended to explain the Graphical User Interface (GUI) for the Lagrangian Particle Model GRAL (Graz Lagrangian Model) and the non-hydrostatic mesoscale model GRAMM (Graz Mesoscale Model).

You will find more information on good modeling practices and further recommendations when using GRAL/GRAMM in the documentation reports in the section “Recommendations for using GRAL and known limits of application”. It is highly recommended to go also through the validation studies documented there to get an impression on how to set the various parameters in specific applications.

It is explicitly noted that the Government of Styria, Section Air Quality Control, doesn't accept liability for software errors and resulting consequences or misuse of the models. It is the responsibility of the model user to check results for plausibility, which can be done by comparisons with results from observations or other models for instance.

4. Installation of GRAL

GRAL consists of several files and is distributed as a zipped archive. Create any directory on your computer (application folder), name as desired, and save the packed archive there. Unzip the files, and best create a link to GRAL_GUI.exe to the desktop, such that there appears a GRAL-symbol for launching the Graphical User Interface (GUI).

It is recommended to create a new directory each time a new version of GRAL is distributed, although it is OK to overwrite existing files as well.

Microsoft Windows

The GRAL graphical user interface (GUI) is developed for the Microsoft .NET6 framework. If you are using the default GUI version for Windows (delivered as a trimmed single file), you do not need to install the .NET6 framework.

If you want to use the framework-dependent version, you must install a current .NET6 or higher framework.

The GRAL GUI creates no registry entries in the Windows system files, so delete the GRAL folder to remove GRAL from your machine.

Linux

The Linux version of the GUI is developed for the MONO 4.5 framework. Almost all functions of the windows version are available, except the 3D visualization and the clipboard functions.

Computation Cores GRAL and GRAMM

The computation cores GRAL and GRAMM are compiled for the .NET6 framework, they are platform independent. The Windows and Linux versions are additionally delivered as a trimmed single file, so that the .Net6 framework does not need to be installed for these versions.

For Linux or advanced windows users, the operating system independent *.dll versions are recommended, because you can use the latest .NET version and benefit from new .NET optimizations. The .NET5 framework is not installed by default. Installation guides are available at the Microsoft homepage for Windows, Linux and macOS.

The .NET6 computation cores are launched from the GUI using the *.exe file at Windows, the *.dll file at Windows or Linux or - for all supported operating systems - from the console, using the keyword dotnet (e.g. "dotnet GRAL.dll").



Create a project on Windows and start the calculation with Linux

It is possible to use the Windows GUI, but to start the calculation in Linux.

Due to the different file systems, you need to adjust the following control files using paths:

Files “windfeld.txt”, “ggeom.asc” (if a path is visible in the 1st line), “GFF_FilePath.txt”: add a new (2nd) line with the Linux path to the used GRAMM windfield data.

Files in the application folder

File name	Description
 NETCoreGRAL	Directory for the .NET Core GRAL version
 NETCoreGRAMM	Directory for the .NET Core GRAMM version
AllSituations.met	Met data for an artificial 360° wind rose
<i>DefaultPath</i>	<i>Most recent project path</i>
DepositionSettings.txt	Presets for the deposition dialog
Emission_Mod_Diurnal.txt	Presets for diurnal emission modulations
Emission_Mod_Seasonal.txt	Presets for seasonal emission modulations
GRAL.nemo5.0.0	Settings for the optional NEMO application
GRAL_GUI.exe	GUI application for Windows .NET6
GRAMM.exe	GRAMM computation core for Windows10 .NET6
GRAL.exe	GRAL computation core for Windows10 .NET6
Landuse_Default.txt	Presets for the landuse dialog
<i>RecentFiles.txt</i>	<i>Most recent files for the open project dialog</i>

The files “RecentFiles.txt” and “DefaultPath” are created by the GUI, they are not part of the delivered zip file.

At the Linux version an additional shell file, called “GRAL_GUI.sh”, is delivered to create the console – command: “mono GRAL_GUI.exe”.

The folders NETCoreGRAL and NetCoreGRAMM contain the application files and start files (Windows: *.bat file, Linux: *.sh file) for the .NET Core versions of the computation cores.

5. Basic concepts for a new project

The following basic concepts should assist you to get an orientation on the GRAMM and GRAL systematic:

1. GRAL simulation

Dispersion simulation for flat terrain. If you take buildings into account, a microscale wind field model is used by GRAL to compute the flow around obstacles

The GRAL simulation can be performed in a steady-state mode (by default) or a transient mode.

2. GRAMM computation

Wind field simulation by the mesoscale model GRAMM at complex terrain

3. GRAL simulation in complex terrain

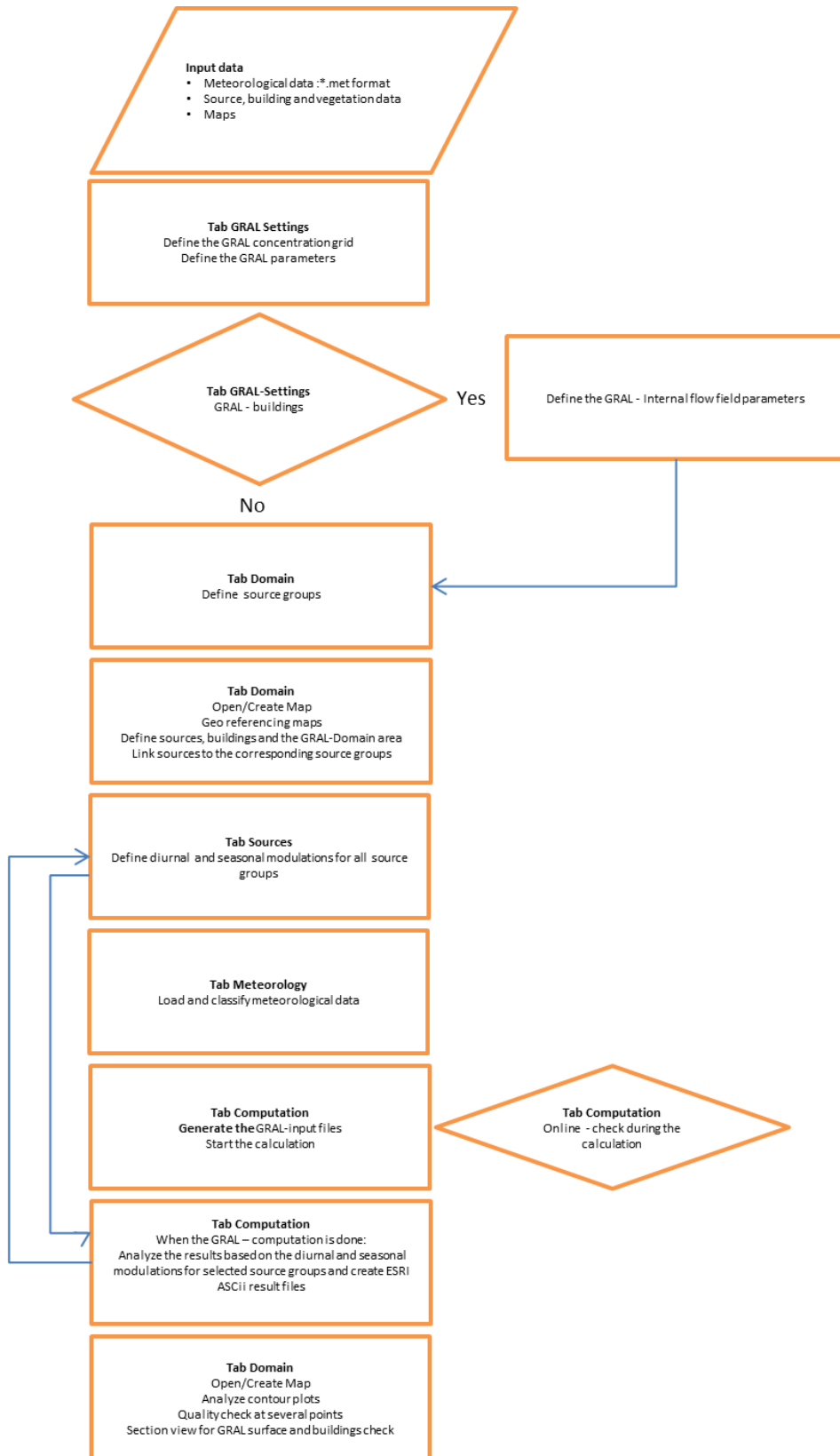
The 1st step is to calculate a GRAMM wind field, a match to observation post processing is recommended in most cases.

Based on that GRAMM wind field it is possible to start a GRAL simulation at complex terrain.

The internal microscale wind field model of GRAL uses the GRAMM flow field as a basis and computes the flow around obstacles and microscale terrain.

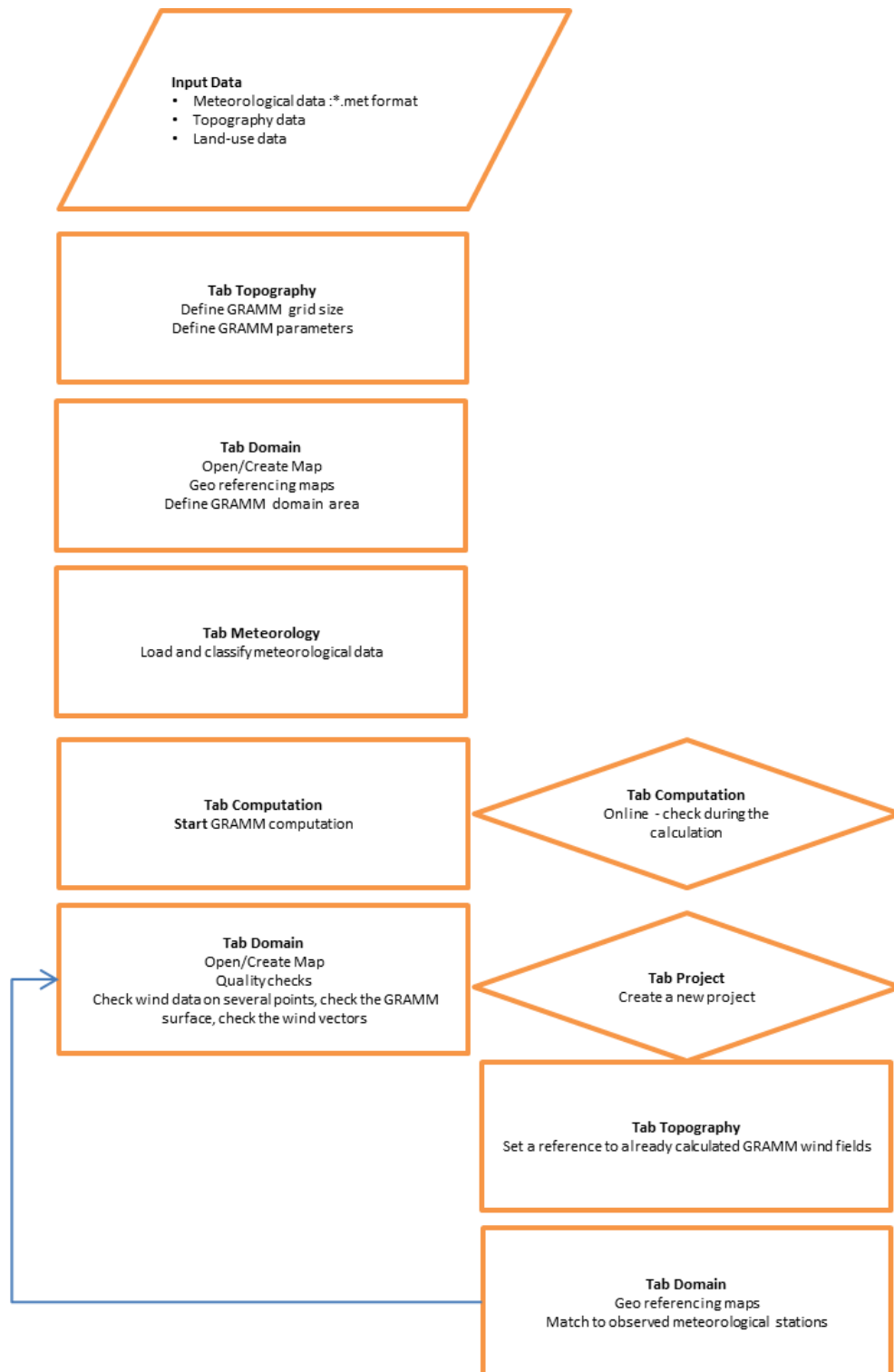
The GRAL simulation can be performed in a steady-state mode (by default) or a transient mode.

GRAL steady state simulation at flat terrain

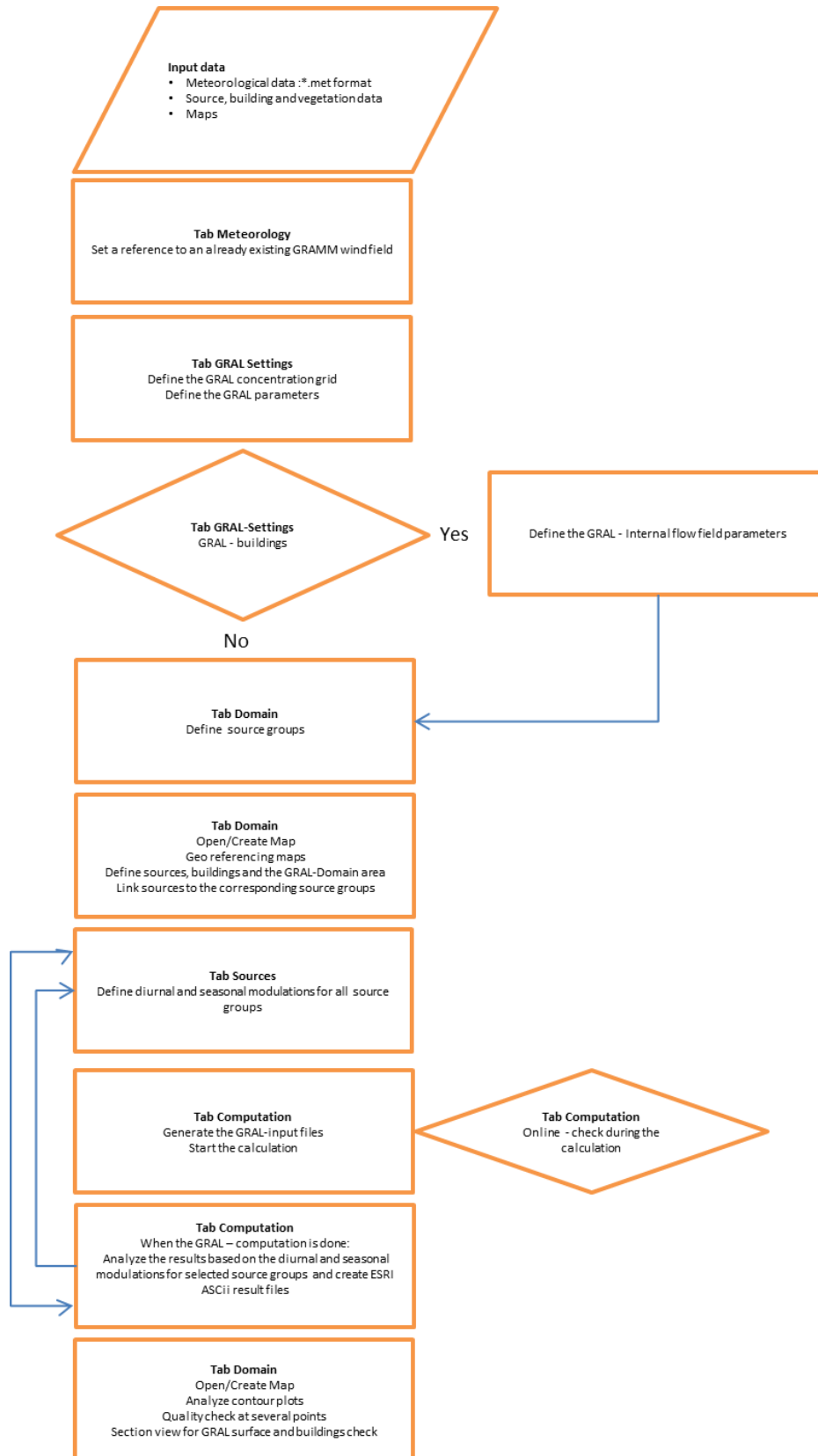


Basic concepts for a new project

GRAMM simulation

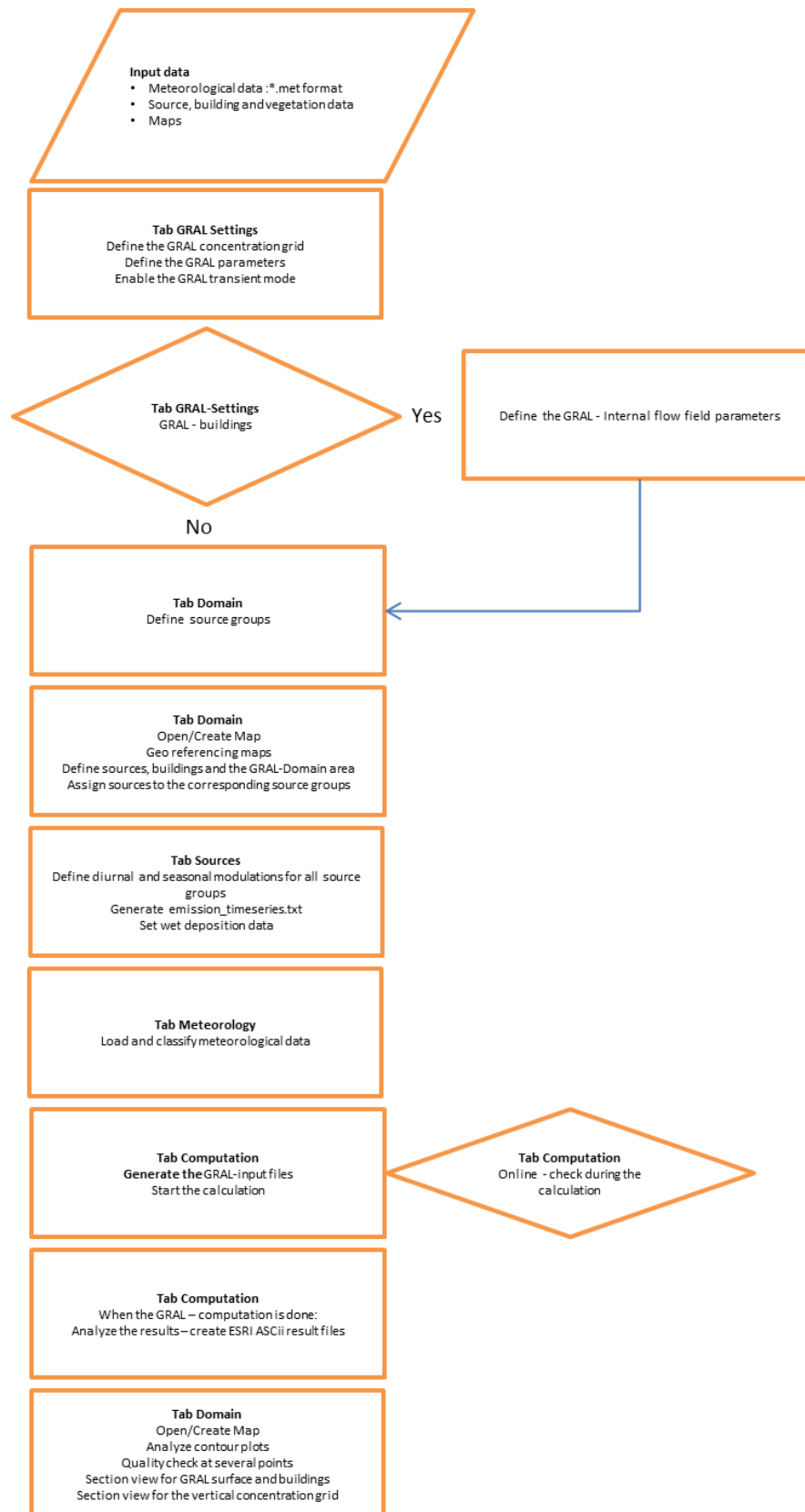


GRAL steady state simulation at complex terrain



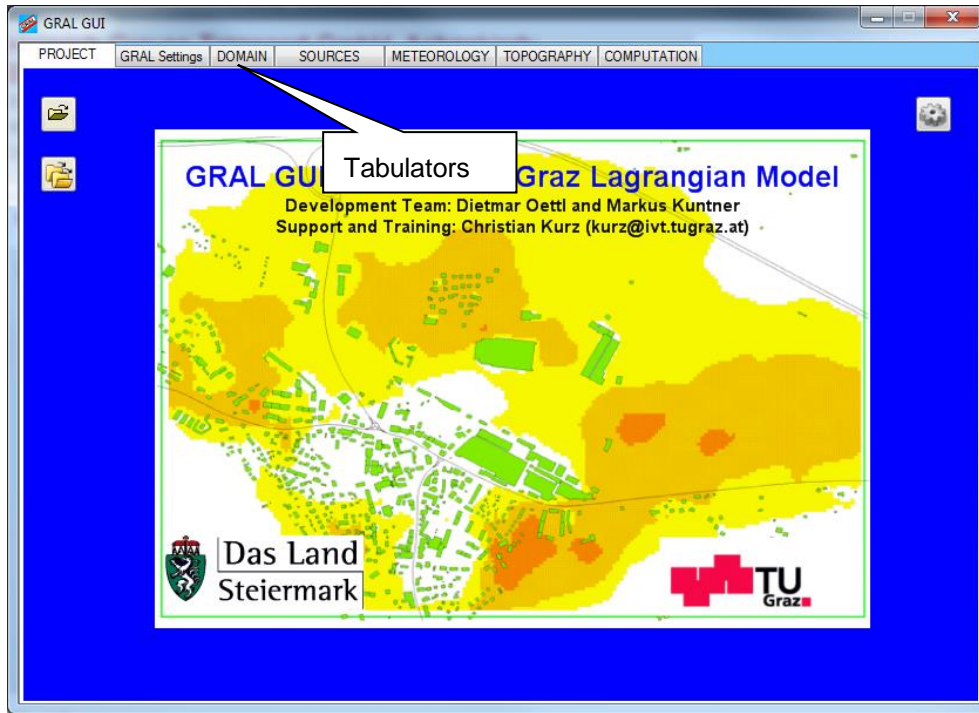
Basic concepts for a new project

GRAL transient mode simulation



5.1. The Main Window

The main window is the control center for GRAL or GRAMM projects.



The tabs contain the following settings

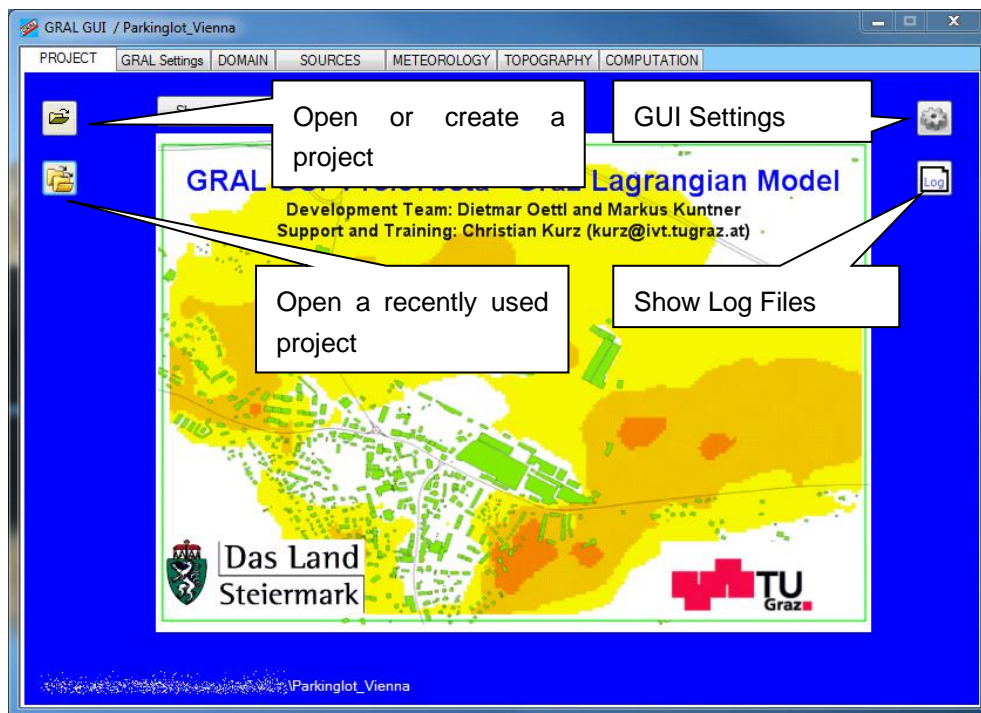
Project	Create or open a project (see chapter 6)
GRAL settings	Basic settings for the GRAL model (see chapter 8)
Domain	Define source groups and open a GIS window to digitize/import sources, buildings and the topography for the GRAL model (see chapter 10 to 12)
Sources	Select source groups and define emission modulations (see chapter 13)
Meteorology	Visualize and classify meteorological data (see chapter 14)
Topography	Define the basic GRAMM settings and import topographical and landuse data (see chapter 16)
Computation	Start the GRAL or GRAMM computation and analyze the results (see chapter 15 and 16.7)

6. Create or open a project

You can create a new project by clicking on an already existing or newly created folder. If you create a new project, the GUI automatically generates several subdirectories.

It is important to click on the main project folder and not on any of the subdirectories when opening an already existing project.

Otherwise, a new project will be created within the subdirectory.



We would suggest starting a new project with the “GRAL settings” tab on the left and continuing with the “Domain” tab to the right, though, other sequences are possible, too.

You can open recently used projects (up to 10 projects) using the “Open most recent projects” button.

Add comments to your project by pushing the “Show comments” button. This button appears as soon a project is opened.

By default, some information about your computer is displayed as a comment.

The button “Save comments” saves your comments in the project-folder.

The project-folder name appears at the title bar of the main menu, the complete project path is displayed at the bottom of the “Project” tab.

Please avoid any special characters, such as „ä“ „ö“ „ü“ as these may cause problems.

You must not use the separator characters comma and semicolon and the string combination “_@” in identifiers of sources, source groups and time series!

6.1. GUI Settings (Gear Wheel)

For some users, security policies do not allow programs to run outside certain folders.

To be prepared for such cases, there are the following GUI options.

Default path for the application settings

By default, the GUI settings are stored in the directory where the GUI is stored.

It is possible to store the GUI settings in your application data folder (Windows: My Documents, Linux: home directory). The settings are stored in a new folder called “GRAL_GUI_Settings”.

This option affects the files for the program settings (default project path, most recent files) and the default deposition and emission modulation files.

Copy GRAL or GRAMM computation cores to the computation folder?

If it is not allowed to launch applications within the project folder through the security policy on your computer, this setting gives you the option to start the calculation core in the program folder. The cores will not be copied to the project in this case, the path for the project is sent to the calculation core as a startup parameter.

This option needs a GRAL or GRAMM version 19.01 or higher.

File compatibility to GUI version 19.01

As of version 20.01, all item data (buildings, walls, vegetation) of the GUI are written to the Emissions folder. In the compatibility mode, some data will additionally be saved to the Computation folder.

Vector automatic scaling

If enabled, the GRAL and GRAMM wind vectors are scaled automatically if the GUI loads and displays the *.wnd or *.gff file. If disabled, all vectors get the same scale.

Meteo import: reject lines with wind speed and direction = 0 or shuffle wind direction

Using this option, you can load the entire meteorological file (default) or discard lines with wind speed 0 and wind direction 0 or shuffle the wind direction for lines with wind speed 0 and wind direction 0.

This setting affects the underlying meteorological import function and thus the wind visualization, the match to observation function and the meteorological classification.

Delete *.con/*.grz/*.gff/*.wnd files to the recycling bin

Select whether GRAL concentration files and GRAMM wind fields should be deleted or moved to the recycle bin when they are deleted. Deleting is faster, moving to the recycle bin is safer.

6.2. Show Log Files

Load and show the Log Files created by the GUI and/or the GRAL and GRAMM computation cores.

7. Save a project

There are two concepts for saving projects at GRAL

All settings and meteorological data at the main menu are saved by pressing the corresponding buttons at the “Computation” folder of the main form.

If your settings are not already saved and you try to close the project, a warning message will appear.

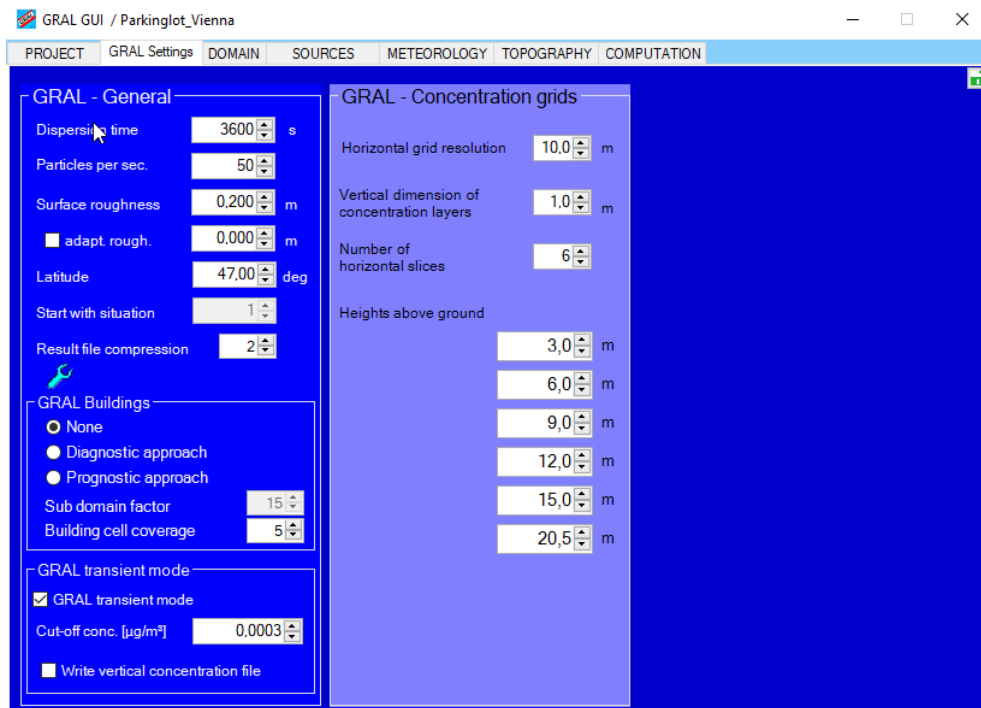
At the GIS form all changes are saved immediately, except modifications to the micro scale terrain (GRAL topography). There is no “save project” or “undo” button at the GIS form. Your changes at the GIS form are not saved if your project is locked (write protection - see chapter 15.2).

Modifications of the GRAL topography must be saved with the corresponding menu entry.

8. GRAL Settings - Main control parameters for GRAL

Define the main control parameters for GRAL computations in the tab “GRAL settings”.

The green lock on the upper right side decides, that the project isn't locked, that means no computation results are available and you can edit all parameters (further information on the lock function can be found in the chapter 15.2.).



8.1. Column „GRAL General“

Dispersion time

Set this value depending on the requested shortest averaging time for concentrations, e.g. 1800 s for half-hourly values or 3600 s for hourly values. The lower threshold of 300 s is due to validity of turbulence parameterizations in GRAL.

Particles per sec

Defines the total number of lagrangian particles released in each dispersion situation. The total number of particles emitted by the dispersion model is the product of the parameters “Particles per sec” times “Dispersion time”).

The higher the number of lagrangian particles, the smoother the concentrations fields (statistical errors are reduced with increasing number of particles).

Typical values are between 25 (for areas < 250x250m²) and 1000 (for areas > 20x20 km² and numerous sources).

If you need to calculate high stack sources or only a few or single weather situations, it is recommended to use a significantly higher number of particles.

In most cases (few sources, areas < 1 km²) 100 particles per second are a good choice.

Note that the calculation time increases linearly with the number of particles.

As of GRAL version 20.01, the GRAL calculation kernel writes the estimated statistical error at receptor points to the file “Receptor_Timeseries_Transient.txt”, which is also generated in steady state mode. The information is written as soon as the calculation is finished and gives you an indication if a sufficient number of particles have been used.

Surface roughness

Define the surface roughness length in the entire model domain.

Even if buildings are taken explicitly into account in GRAL simulations, it is necessary to take a mean roughness length in such cases, except the option “Adaptive Roughness” is used. Some hints about reasonable roughness lengths connected to land-use classes can be found in the GRAMM documentation, and more general in the GRAL recommendation guide.

If you use flow fields from GRAMM as a meteorological input, and GRAMM was fed with corresponding land-use data, the spatial roughness lengths are taken from the land-use file, and the value of this numeric field is not considered anymore.

Adaptive surface roughness

If this checkbox is activated, the user may create a file “RoughnessLengthsGral.dat” to set spatially defined surface roughness values (based on the flow field grid) for GRAL or to activate the “AdaptiveRoughness” option in GRAL.

When the “Adaptive Roughness” option is used without a user defined file, the surface roughness parameter specifies the minimum surface roughness and the value for “Adaptive Roughness” specifies the maximum surface roughness used by the adaptive algorithm. This algorithm estimates the local surface roughness based on building and vegetation heights.

Latitude

Set the average latitude of the model domain.

Start with dispersion situation

The number of the dispersion situation to be used to start the GRAL model (usually the first).

If you hit the “Pause” button in the tab “Computation”, this input field is updated with the number of dispersion situation interrupted by the user, such that a computation can be quickly continued by simply pressing the “Play” button in the “Computation” tab.

In the “Transient GRAL Mode”, the computation starts with situation 1 or the last stored intermediate state. Therefore, this setting is not available in the transient GRAL mode.

Result file compression

Mode 0 means no compression and is an option for compatibility to very old GRAL projects.

With options 1 to 3, all GRAL outputs per dispersion situation (all concentration *.con, odour *.odr and deposition *.dep files) are stored in one zip container using the filename extension “*.grz” (**g**ral **r**esults **z**ipped).

This option reduces the required disk space by a factor of 2 and the number of result files in order to the product of source-groups, horizontal slices and calculation type (odour, with/without deposition).

The creation of the compressed files is incredible fast and there is no influence on the calculation time.

When compressed result files are used, a compressed ggeom.asc file format is created. GRAMM and GRAL versions 17.1 or higher are needed to use the option 1.

The options 2 and 3 have been introduced with GRAL version 20.01 to create more error-proof files. Option 2 provides smaller result files, option 3 writes all cells of the grid in linear order.

Buildings

Buildings and obstacles (e.g. noise barriers, forests) can be considered in GRAL simulations either using a simple diagnostic or an advanced prognostic microscale wind field model.

In general, it is recommended to use the prognostic wind field model.

The diagnostic approach is an alternative for very large model domains, if the prognostic option would lead to very long calculation times and huge memory consumption.

More information is given some lines further by the description of the column „GRAL Internal flow field grid“

Sub domain factor

The prognostic wind field around obstacles is calculated in sub domains, overlapping sub domains are merged. Sub domains are defined by the height of the obstacles multiplied with the default factor of 15. This factor can be overruled to create larger sub domains. The maximum value for this factor is 1000. Outside the sub domains the wind field is always calculated diagnostically.

Building cell coverage

With this option you can control how buildings are rasterized. Each cell is divided into 9 sub-squares with one test-point in the middle of each sub square. A building cell is generated if at least the number of specified test points are inside the building shape. 1 means that a building cell is already created if at least one test point lies within the building shape, a number 9 means that almost the entire cell must be covered by the building to be defined as a building cell.

GRAL Transient Mode

Check this box for carrying out transient dispersion simulations with GRAL. The plumes of previous dispersion situations are stored in this mode and are considered again at the dispersion simulation of the next weather situations.

In this way, even puff releases with a minimum release time equal to the user-defined dispersion time (minimum 300 s) can be modeled. Emissions for each user-defined source group are

modulated in the GRAL calculation core as set by the user in the file “emissions_timerseries.txt” (see chapter 13.1.2).

The method can be used in combination with either classified or non-classified meteorological input data as well as classified GRAMM wind fields. By using classified meteorological input data, it is possible to speed up simulations when the microscale flow field model is invoked, too. In such cases it is recommended to use the option for storing the intermediate GRAL flow field files (.gff files) as these are then computed only once for each classified dispersion situation.

Note, that simulations times will increase when using GRAL in transient mode as the entire time series of dispersion situations (as defined in the file mettimeseries.dat) must be computed one after each other. The standard mode is therefore the steady-state option (box not checked).

The cut-off concentration is a threshold to reduce the number of transient particles and is used to speed up the computation. Cells below this threshold are not considered in the transient dispersion.

The checkbox “Write vertical concentration file” forces GRAL to write a text file with all internal stored horizontal mean concentration layers after the last dispersion situation is finished.

For computing wet deposition, the GRAL transient mode must be used.

Special Settings (Wrench Button)

Use these special settings if you know exactly what you are doing! The following options are available:

Write additional ASCII result files

This is a harmless option, GRAL additionally writes out ASCII results

Read temp. transient files when restarting with situation 1

This option is intended for users who continuously create new time series, calculate one such time series at a time and then continue the transient calculation with another time series in the future. This option will override the internal check of the time series order.

The optional interval for saving the transient concentration grid can be set by the user.

Reduce the size of prognostic sub domains

You can set a radius surrounding sources in [m]. Prognostic flow field calculations are performed around buildings defined by the SubDomainFactor but limited by this radius. Beyond this distance, the wind fields are always calculated diagnostically. This option is useful for large domain areas to reduce the memory consumption and speedup the calculation.

GRAL Settings - Main control parameters for GRAL

By default the prognostic sub-domains are created around buildings using the building height multiplied with the sub-domain factor. Now this can be displayed with the File “PrognosticSubDomainAreas.txt” (see Figure 1). If the new option is used, prognostic subdomains are created around sources at a defined distance only. An example is shown in Figure 2.

With reduced sub-domains, memory requirements and computation time are reduced. If large contiguous prognostic wind fields are required, as in urban areas, this option should not be used.

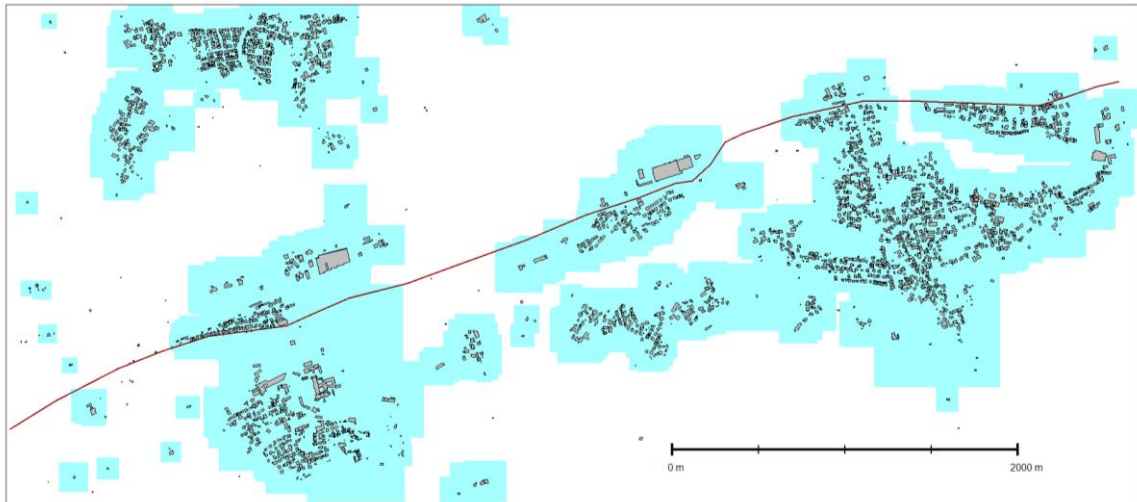


Figure 1: default sub domains

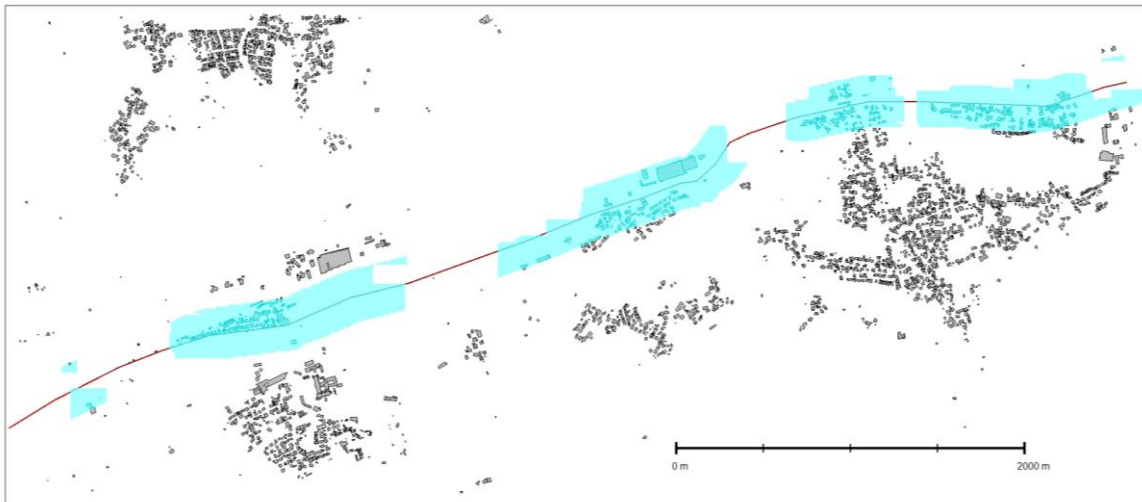


Figure 2: reduced sub domains 150 m around sources

Log-Level

You can define several logging levels (0 to 3) for GRAL calculations.

Keystroke when exiting GRAL

Exit GRAL immediately or wait for keystroke

Disable GRAL Online Functions

Optional deactivation of “GRAL Online” functions: Reduction of file accesses during the calculation

8.2. Column „Concentration grid“

Horizontal grid resolution

Define the horizontal grid size of the concentration grid.

If buildings or obstacles are resolved, grid sizes should preferably be in the range of 2 – 3 m.

Whenever the horizontal grid size in GRAL is changed, the model domain – if already defined – will be deleted for numerical reasons. Thus, the horizontal grid size must be set before defining the GRAL model domain.

Vertical dimension of concentration layers

You can define the vertical extension of the concentration grid here. A low vertical extension results in a higher statistical error. Values between one and five meters would be typical. The concentration is calculated in a layer defined by the height above ground ± 0.5 times the vertical dimension. The vertical dimension is also used for the concentration calculation at receptor points (see chapter 11.11.8).

Number of horizontal slices

You can define up to nine horizontal concentration grids.

Heights above ground

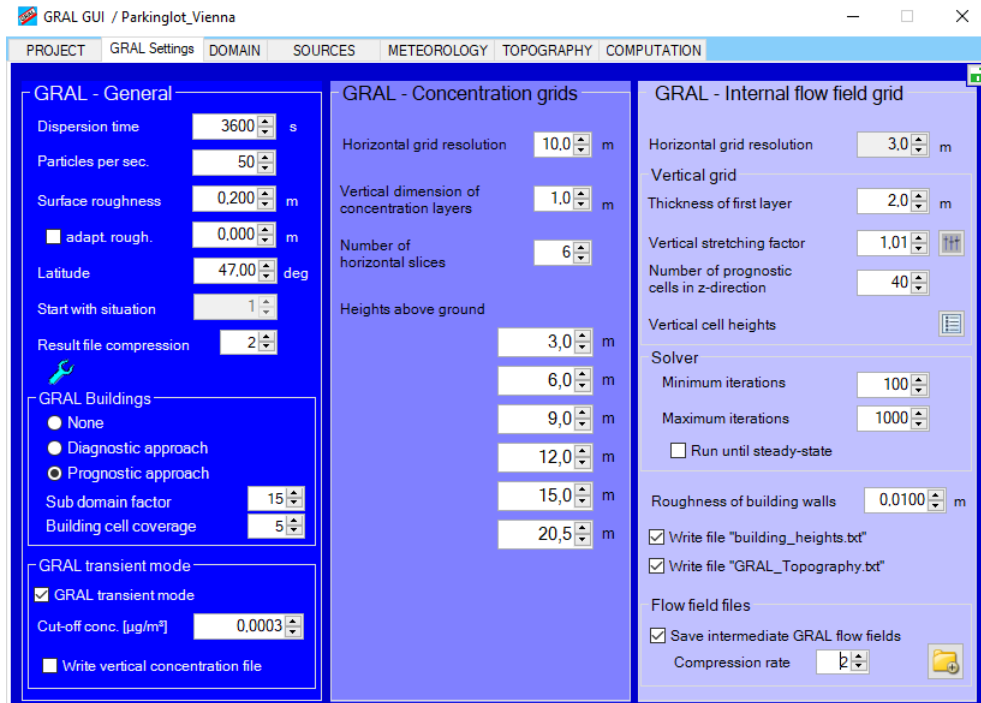
Set the height above ground for each horizontal concentration grid. The height is defined at the center of the vertical extension. The concentration is determined in a layer of $\pm 0.5 \cdot$ vertical dimension.

The lowest height above ground must be higher than 0.5 times the vertical dimension of concentration layers.

If you perform an odor simulation and you like to use the new concentration-variance model (see chapter 15.5.2) the lowest height above ground must be higher than 1.5 times the vertical dimension of the concentration layers, otherwise the new concentration-variance model is not available.

8.3. Column „GRAL Internal flow field grid“

If you include buildings, the „GRAL Internal flow field grid“ column will be displayed on the right side of this tab.



Whenever the diagnostic or prognostic buildings option is selected in the column „GRAL - General“, additional input fields appear on the right side to define the parameters for the microscale flow field model of GRAL.

Horizontal grid resolution

Define the horizontal grid size for the microscale flow field.

By default, the grid size of the flow field grid is set to the grid size of the concentration grid. When the grid size for the concentration grid in the “Project” tab is changed, the flow field grid is automatically adapted. In a few cases it might be advisable to use different grid sizes for the flow and concentration grids. For example, if high stack emissions are influenced by large buildings/obstacles, the flow field grid needs to be small, but the concentration grid should take much larger values to reduce statistical errors.

Vertical thickness of first layer

Define the cell height of the lowest layer of the flow field. Depending on the size of the model domain, typical values are 1 – 2 m.

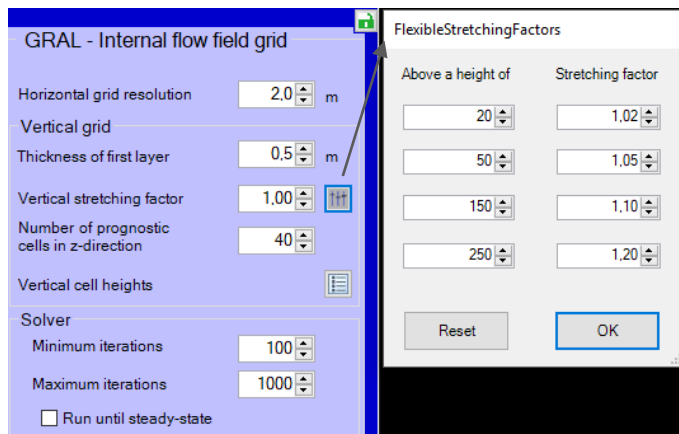
Vertical stretching factor

Define how cell heights increase with height above ground. A factor of 1.01 means a cell is 1 % higher than the previous lower one.

It should be noted that rather large cell heights may occur in the presence of mountains, thus it is recommended to use generally stretching factors close to 1.0 as long as the main memory or the calculation time does not set a limit.

Flexible stretching factors

This option enables faster flow field calculations (especially when using terrain) and/or higher accuracy.



These (default) settings specify that there is a stretching factor of 1 up to a height of 20 m above the lowest cell within the domain area, a factor of 1,02 up to a height of 50 m above the lowest cell, a factor of 1,05 up to 150 m and so on.

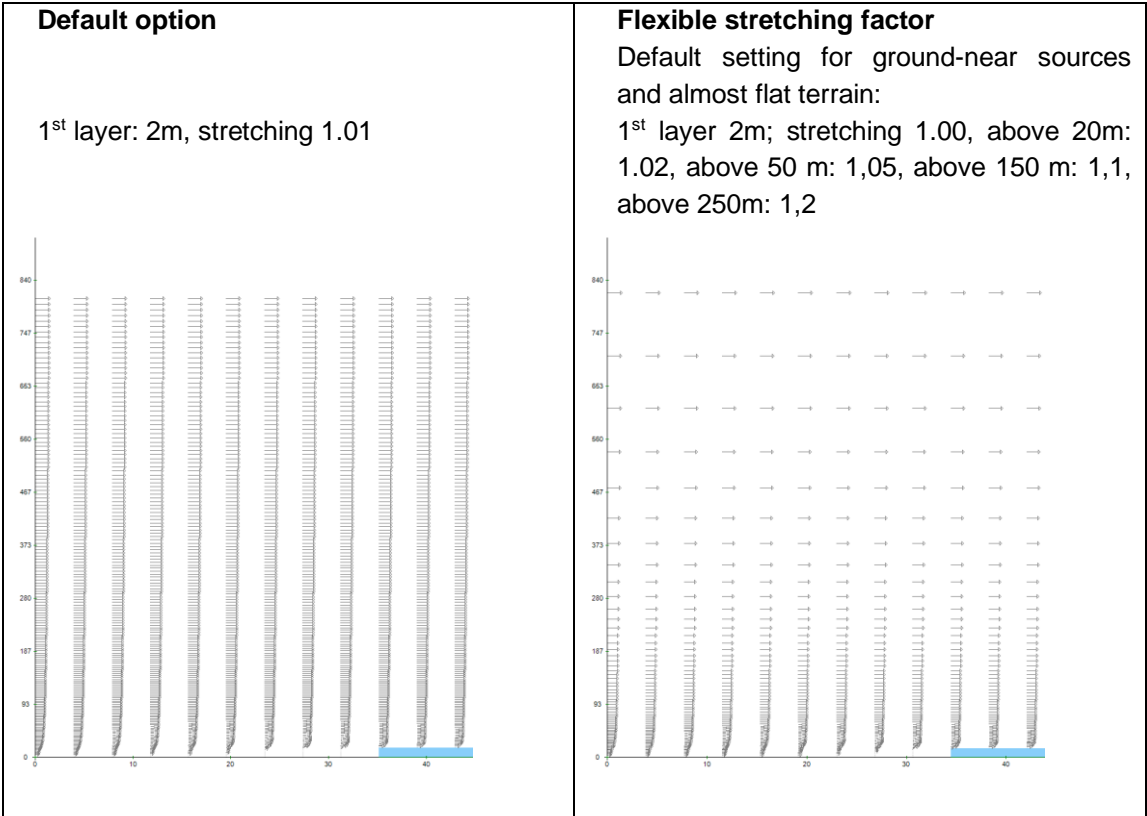
When calculating with terrain, one should be aware that the flow field grid and thus the vertical cell height starts at the lowest terrain cell within the GRAL domain.

This option allows a better accuracy near the ground and faster calculation times, because there are fewer cells in larger heights above ground. An example for wind vectors is shown in the next figure. The default option creates much more vertical cells, but the vertical resolution near the ground is lower compared to the option with flexible stretching factors.

If the flexible stretching factors are set, the button color turns to yellow.

The reset button restores the default option without flexible vertical stretching factors.

GRAL Settings - Main control parameters for GRAL



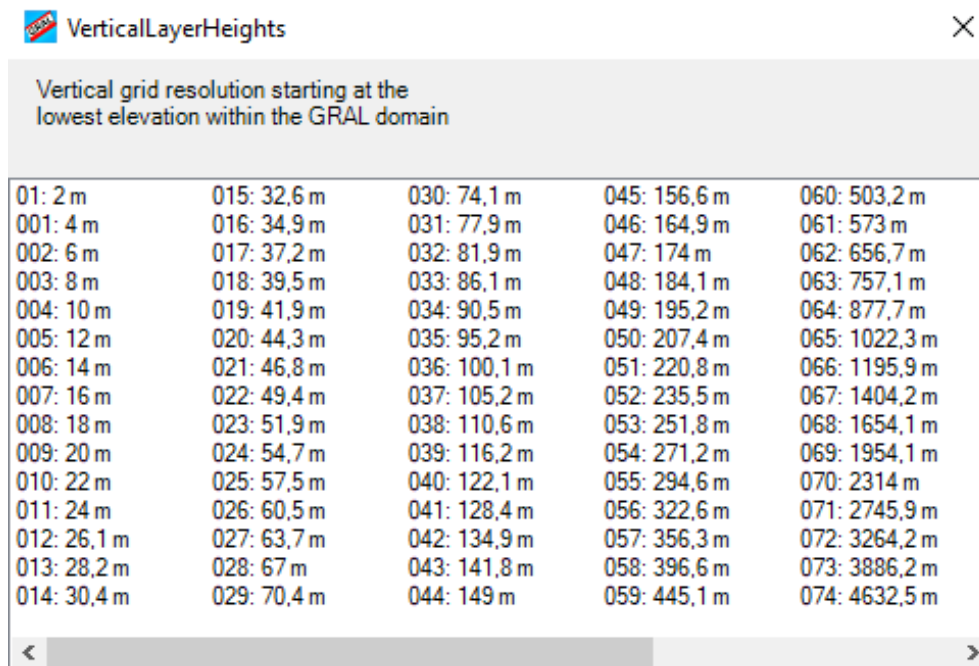
Example of vertical wind vectors for default and flexible stretching factors

Number of prognostic cells in z-direction – Vertical cell heights

The number of cells in z-direction defines the number of cells above ground used in the prognostic flow field model. Above these cells, the wind field is calculated diagnostically.

The total number of cells in the z-direction is automatically defined within the model (the user cannot change it). The number of prognostic cells in z-direction in combination with the vertical thickness of the first layer and the vertical stretching factor determine the relative model height as indicated in the box below. The model height should be high enough that about 90 % of the cross area in each vertical slice through the model domain is unobstructed.

The option to show the vertical cell heights is for user information only.



Minimum iterations

Define the minimum number of iterations for calculating the prognostic flow field. Increasing this number can significantly increase the computational times.

Maximum iterations

Define the maximum number of iterations that shall be performed. A reduction of this number can reduce the computational times significantly. Simulated flow fields, however, may then be far from meeting stationary conditions.

The model was validated with the default maximum of 500 iterations.

Run until steady state

Ticking this checkbox will override the “Maximum iterations” settings and flow field simulations are running until the internal steady-state condition is reached.

Note that in most cases, simulation times will increase dramatically if you enable this option.

Roughness of building walls

Define the roughness length of the walls of all obstacles within the model domain.

Write file “building_heights.txt”

When ticking this box, GRAL writes a file named ‘building_heights.txt’ in the sub-directory “Computation” if the calculation of the first dispersion situation is finished. This function is not activated for all other dispersion situations.

The file ‘building_heights.txt’ contains the exact building/obstacle heights as they appear in the model. Note, that they are usually different from the actual heights due to the vertical flow field grid. These building heights can be visualized in the same way as contour maps (for further details see sect. 15.6).

In addition, clicking on this box will force GRAL writing the file “PrognosticSubDomainAreas.txt” showing the prognostic sub-domains areas.

Write file “GRAL_Topography.txt”

Using this option, GRAL writes a file named “GRAL_Topography.txt” that contains the GRAL topography data used in the GRAL model. The topography can be visualized in the same way as contour maps (for further details see sect. 15.6).

Save intermediate GRAL flow files

Ticking this box forces the GRAL model to save the calculated flow fields as files in the subdirectory “Computation” as *.gff files.

This option is particularly useful if more than one simulation must be performed in the same project (e.g. simulations for several pollutants or for different emission rates).

This option saves the internal GRAL flow fields around obstacles. When you create a 2nd simulation, the flow fields are reused to save calculation time. If *.gff files are saved, it is possible to analyze the data of the flow field (see chapter 17.1).

Using that option saves an additional file for the vertical slices view (file: “GRAL_geometries.txt”). If you only want to force GRAL to create this file, disable this checkbox after the calculation of the first dispersion situation is complete and the file “GRAL_geometries.txt” has been written.

<i>If the model domain, the grid sizes of the flow field calculations, the GRAL topography or the building geometries are changed (changes are blocked by the project-lock function for already calculated simulations), all *.gff-files will be automatically deleted by the GUI as they do not fit anymore to the project data.</i>

Compression rate

There are two options for the compression rate for “.gff” flow field files:

0: compression is compatible to previous GUI versions prior to V 20.01.

1: compression is available from version 20.01; the files are significantly smaller compared to the option 0

Set a path for the flow field files

With the folder button you define a directory for saving and loading the *.gff files.

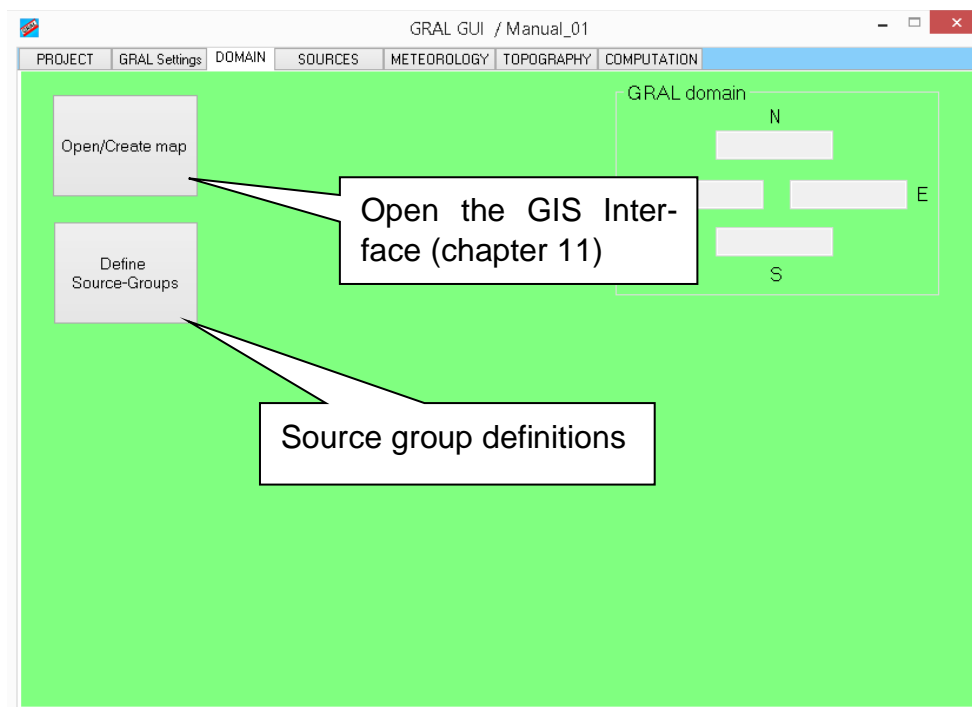
If you include *.gff files from another folder, make sure that all GRAL settings are unchanged. Otherwise GRAL deletes the current *.gff file, calculates a new flow field and overwrites the existing file.

9. Domain

On the “Domain” tab you can see an overview of the coordinates of the GRAL and GRAMM domain, the source groups can be defined (see chapter 10) and the GIS interface can be opened.

Define source groups before digitizing sources. This allows you to associate sources with source groups.

You can link sources with the same (diurnal and seasonal) emission modulation to the same source group. You will get results for all source groups, but results of a single source within a source group cannot be analyzed afterwards.

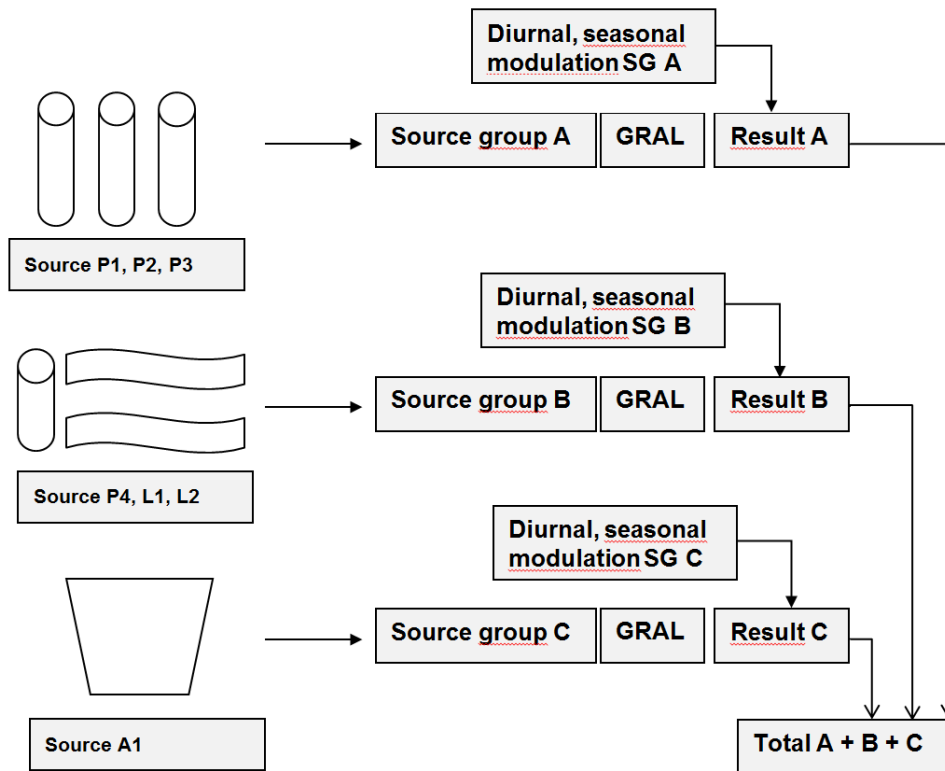


10. Definition of source groups

We suggest defining specific source group names at the beginning of a project, especially when more than one source group is needed.

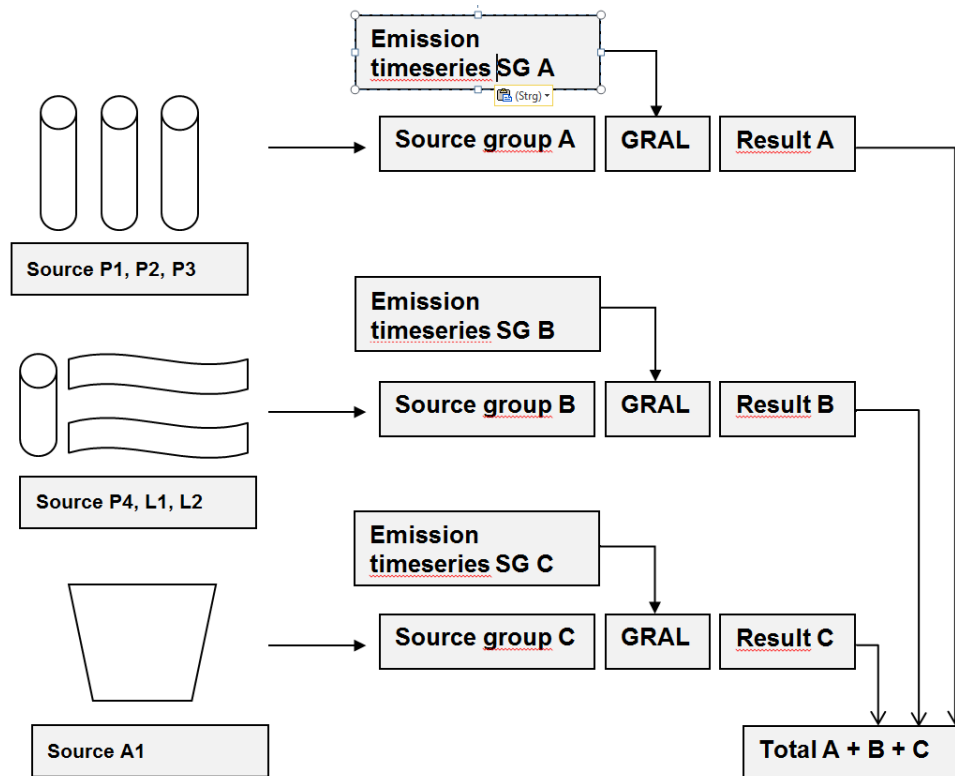
GRAL calculates concentrations fields for each defined source group (e.g. result A, B and C in the next figure) and the sum over all source groups to obtain the total concentration.

Source groups are needed to set individual diurnal and seasonal emission modulation for the final evaluation of the results (post processing at the steady state GRAL mode and preprocessing at the transient GRAL mode, see chapter 13). Each source group can be aligned to one diurnal and seasonal emission modulation or an emission time series.



Source group concept for the steady state GRAL mode

Definition of source groups



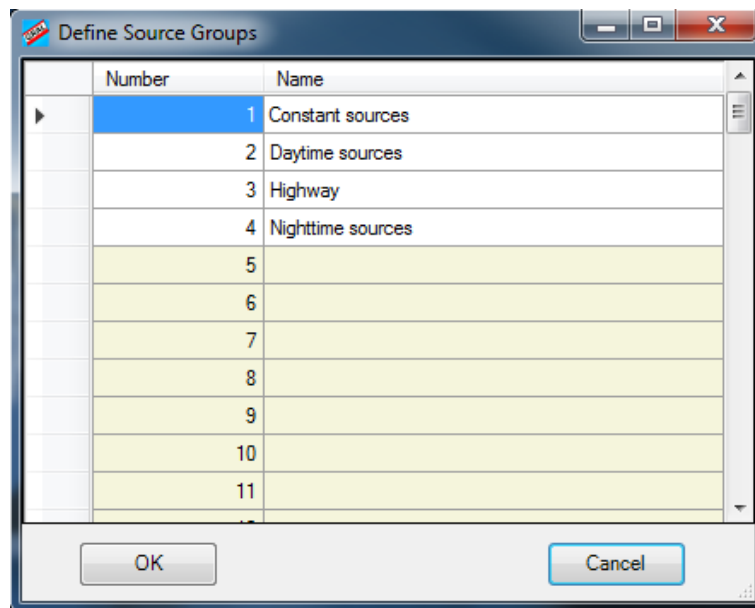
Source group concept for the transient GRAL mode

How to define source groups?

Edit the name nearby the source group number to add a source group. Unused source groups are highlighted with a beige color.

You can choose any name for the source-groups, but the comma is reserved and special characters (not allowed in file names) should not be used.

To remove a source group, delete the name or enter a space for the name. Be careful by removing source groups, because it is not checked programmatically, if sources are aligned to a deleted source group.



Source group dialog

11. GIS Interface

Launch the GIS interface from the tab “Domain”. You can use this interface to define locations, geometries, emission rates of emission sources as well as model domains or buildings, in addition many other features, especially analyzation tools, are provided.

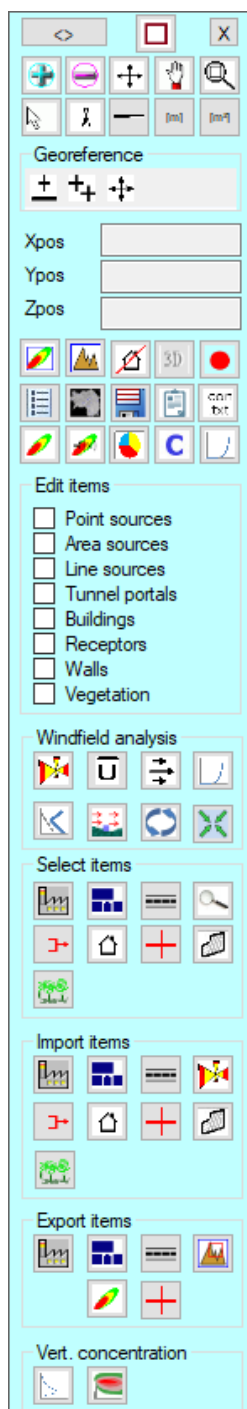
If the GIS interface window is open, any changes made parallel in the main window, which effect GIS specific features, are not automatically updated in the GIS interface. For instance, if the horizontal grid size of the GRAL concentration grid is changed, the model domain will be deleted, this won't be visualized automatically in the GIS interface. To do so, the GIS interface must be closed and re-opened again by pressing the button “Open/Create map” in the tab “Domain”.

11.1. The Toolbox

The Toolbox includes most tools to edit and digitize sources and buildings, to analyze computed and visualize data and to import and export datasets.

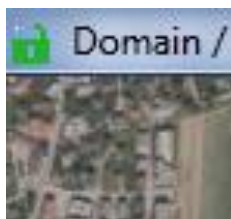
All options are available at the menu bar on the top of the GIS window and some options via shortcuts (take a look to the menu bar or to chapter 19.4).

Depending on the current program-settings and already calculated data, not all options are available.

	<div>Move and hide the toolbox, open the viewframe dialog</div> <div>Move / zoom map section North arrow / Scale bar / Measure section</div> <div>Georeference new maps (chap. 11.7)</div> <div>The x and y position of the mouse cursor</div> <div>Define the GRAMM and GRAL Domain (chap. 16.2 and 11.10), remove buildings (chap. 11.11.7), 3D view of GRAL and GRAMM surfaces; generate an animated *.gif file Object manager / load maps / save actual view / copy to the clipboard / convert *.con files Load contour maps, mathraster operations, get source apportion-</div> <div>Edit the emission and basic data for sources, buildings, walls and receptors (chap. 11.11)</div> <div>Analyze wind fields, section view of GRAMM and GRAL surfaces, buildings and wind vectors (chap. 17.1) Reorder and match GRAMM wind fields (chap. 16.12)</div> <div>Select or search, filter and sort the respective items (chap. 11.12)</div> <div>Import the respective items (chap. 11.14)</div> <div>Export the respective items (chap. 11.15)</div> <div>Vertical concentration profiles for transient computations</div>
--	---

11.2. The Lock-symbol

A small symbol on the left edge of the GIS form shows, either the source data can be changed, or the project is locked.



A green lock indicates, that the data can be changed, because no results are already computed.


A red lock indicates that results are available, and the project is protected.

This icon is not available at Linux, but the “GRAL locked” hint is displayed in the title bar of the GIS form.

You will find further information about the lock – function in section 15.2

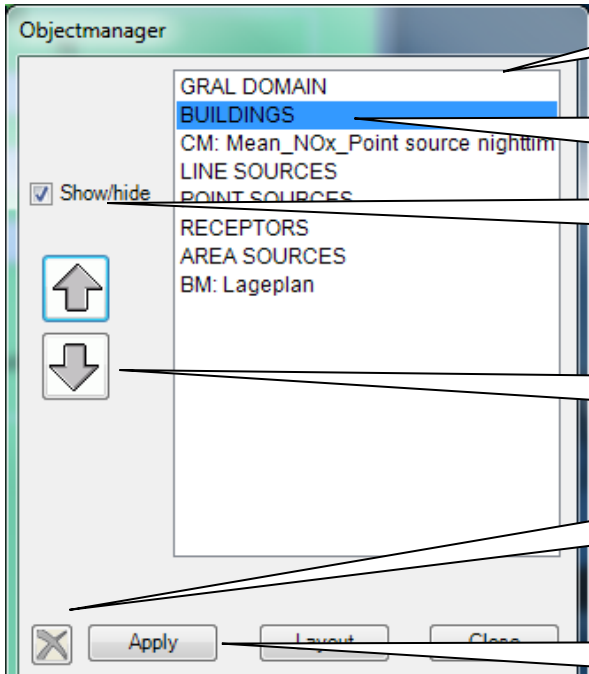
11.3. The object manager

Menu: “View – Object manager”

Toolbox:  symbol

The object manager affects the visibility of the selected layers.

The manager allows you to specify the following options



The layer - drawing list box (drawing stack)

Double click to open the layout manager to adjust the colors or labels (see chap. 11.4)

Show or hide the selected layer; hidden items are greyed

Set the drawing order of a layer – higher rows are drawn in the foreground

Delete the selected layer from the drawing list


Redraw the GIS map with the actual settings

The space key toggles the visibility of the selected layer(s). The return key opens the layout manager for the selected item. The delete key removes the selected layer(s) from the drawing stack.

If a filename is displayed in brackets next to the layer, this is the file associated with this layer.

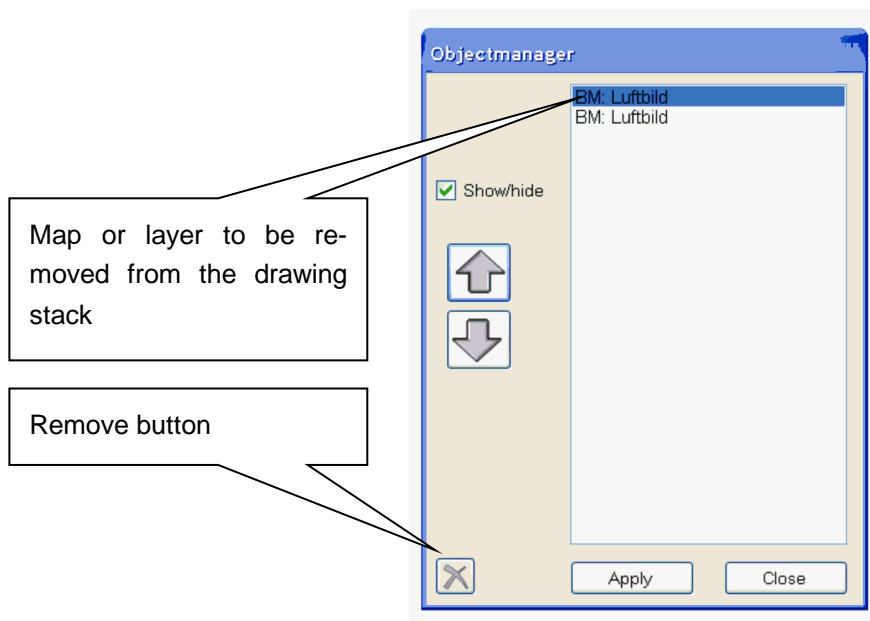
11.3.1 Removing maps or layers from the drawing stack

Menu: "View – Object manager"

Toolbox:  symbol

Open the Object manager, select the maps or layers to be removed and click the remove button as indicated in the figure below or press the delete key on your keyboard.

There must remain at least one geo-referenced map in the project to set the coordinate system and to enable correct drawing of all features.



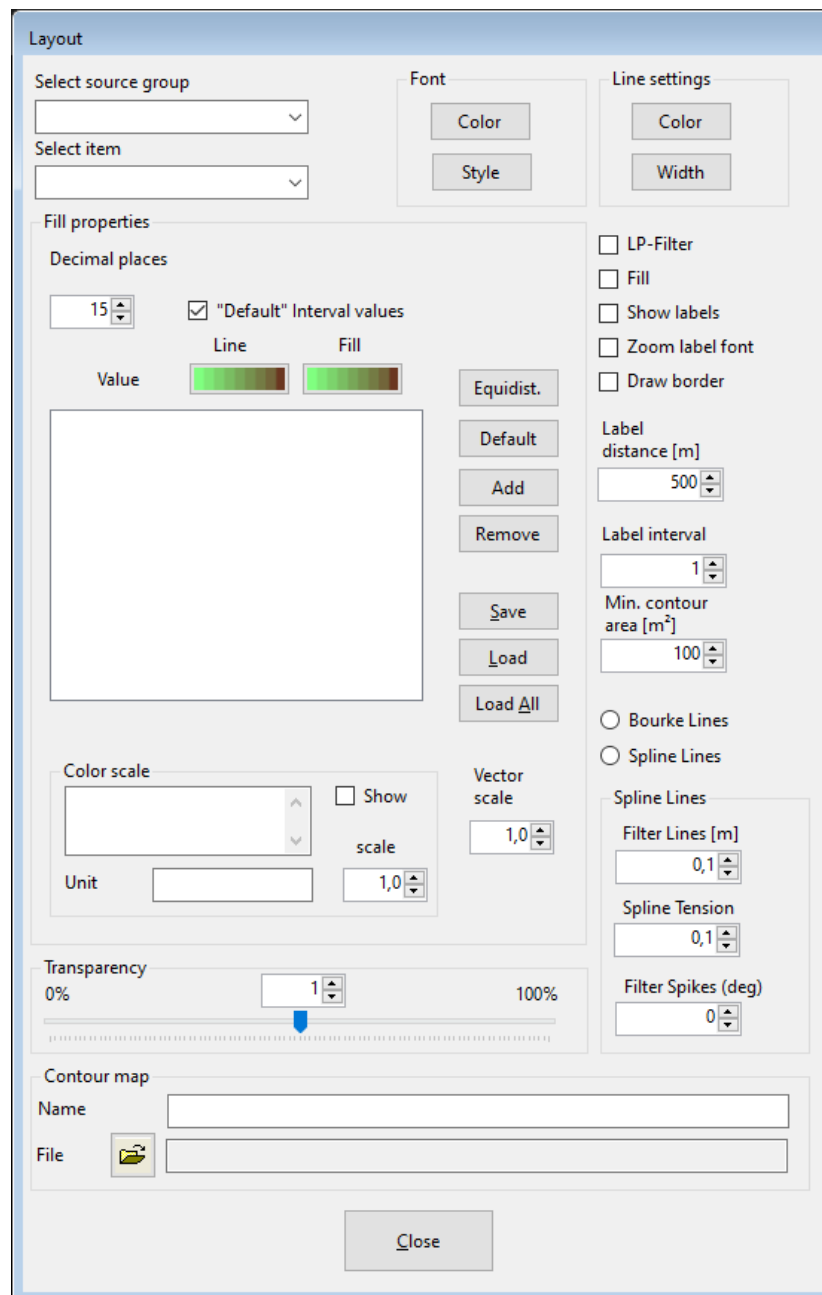
11.4. Layout manager


Using the layout manager you can adjust the appearance of layers or items.

You can start the layout manager by double-clicking a layer in the object manager (see chapter 11.3).

Depending on the type of layer, the layout manager provides different options for the graphical design.

Here you see all possible options of the layout manager. Depending on the layer type, individual labels and functions may change



Select source group	The visibility of sources can be filtered and defined by their source group
Select item	<p>Set the line and fill color depending on the parameter selected here.</p> <p>In the case of emission sources, pollutants can be selected to visualize the emission rates in the map by different colors. In case of line sources also traffic volumes and some more attributes can be selected.</p> <p>For buildings, the building height can be selected to represent the height in defined line and/or fill colors.</p>
Font Color / Style	Change the label font size, color and type using the “Fonts” and “Color” buttons
Line settings Color / Width	<p>Change the line color and width</p> <p>For the item “Contour lines”: set the contour line color (“Line properties” – “Color”) to snow white (the default-value) to show the contour lines in the desired color of the “Fill properties” – “Line” column. If you set another color at the “Line properties” – “Color” dialogue, the contour lines are drawn in that color.</p>
Decimal places	Number of decimals places for the labels
	Create a color gradient between the first and the last level - color for line and fill colors
Default Interval Values	The computed values when pressing the button “Default” are classified to the digits 1, 2, 5 and 8 (* 10 ^x).
Equidist.	Use this button to get equidistant labels starting from the lowest level. It is advisable to set the lowest level to the desired value prior to this function.
Default	Press the “Default” button to get the default label range
Add	Add a new value to the “Values” list box.
Remove	Remove the selected value from the “Values” list box.
Fill properties – List box	Double click to an item in the listbox to change the value and/or the line and the fill color of one item
Save	Use the “Save” button to save the color and label settings for further use
Load	Load saved settings for the values and line and fill colors
Load ALL	Load all saved settings
Color scale	<p>Enable the scale by clicking the checkbox.</p> <p>Change the position of color scale by clicking on the map to the top left corner of the legend. This option is only activated when the layout manager for the corresponding contour map is opened</p>
Smart draw	Enables a faster drawing of items
Fill	The drawn item is filled depending on the color in the “Fill” list box and the transparency option
Show Labels	This three-state box offers the following options. If the tick is set, labels are printed, if the square is selected, labels are drawn with a

	frame.
Zoom label font	When selected, labels are zoomed and the label size in pixels corresponds approximately the drawn height in m
Draw border	Draws a border around a contour map when the option “Spline line” is used
Label distance	Adjust the label to label distance <i>Hint: Narrow label distances will reduce the drawing performance.</i> When vectors are selected, the distance sets the interval between the vectors.
Label interval	Set the interval to define, how many contour lines are drawn between labeled contours. The labeled lines will be drawn with an increased width, like typically used on topographical maps.
Min Contour Area	If a contour line polygon does not exceed this value, it is not drawn to avoid “islands”.
Bourke lines	Default option; contour lines are drawn using the Bourke algorithm and filled using rectangles
Spline lines	Experimental option; contour lines are drawn using a marching squares algorithm and filled polygons
Filter lines [m]	Filter spline line vertices using a Douglas-Peucker filter
Spline tension	Set the tension of the used spline (0...1). If the value of the tension parameter is 0, straight line segments are used to connect the points
Filter spikes [deg]	Remove spikes from the contour line
Vector scale	Define the scale of a vector (e.g. wind vectors)
Transparency	Set the transparency of filled items using the slider or the numeric-up down box
Contour map	You can edit the name, as it appears in the Object Manager and check the folder and file name of the associated contour map raster file. You can use the “Open” button to assign a new file or a changed location to a raster file

Remarks:

By default, there is a tick in the LP-filter box, indicating that the contour map will be smoothed with a 5-point low-pass filter. This is recommended for most concentration maps. Uncheck this option when drawing topography maps or other types of maps, where filtering the data doesn't make a sense from the point of underlying physics.

Due to compatibility issues, the dimension of point sources and receptors has two special functions:

- a diameter of 0: the items are drawn with a fixed diameter in pixels
- a diameter of 1: the items are drawn with a zoomable diameter about 4 m

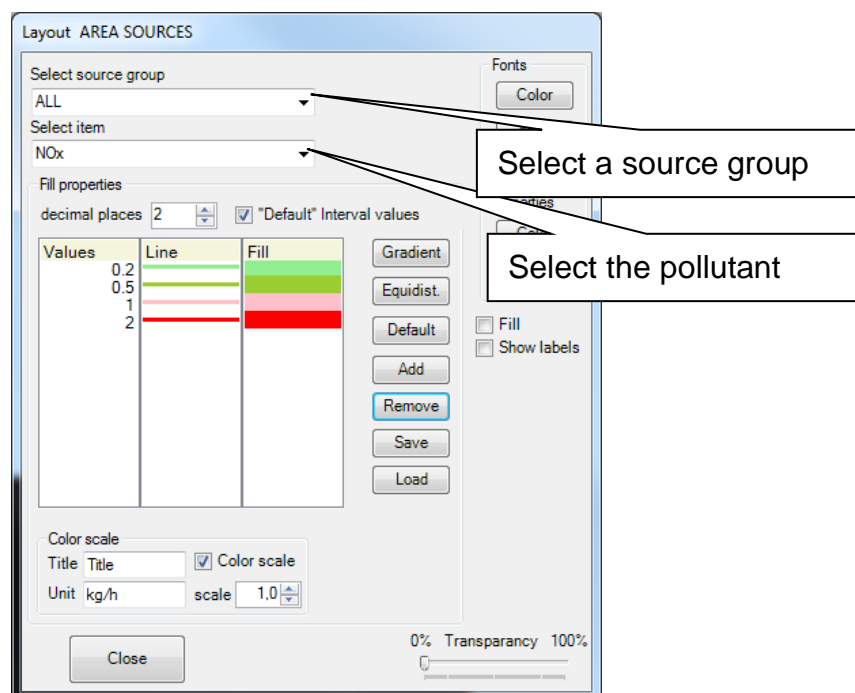
- a diameter ≥ 2 : the items are drawn with a zoomable diameter in m

Further functions depending on the item type

GRAL and GRAMM domain	show the GRAMM and/or GRAL grid
Bitmaps	set the opacity of bitmaps define a coordinate raster grid
Vector maps Line sources Buildings	set a smart draw option for a faster redraw of small vectors / line sources or buildings

For receptors, the display values can be selected for visualization.

Example: how to select a source group or a pollutant for the fill properties



11.5. Add maps

Menu: "File – Load base map"

Toolbox:  symbol

The number of maps is limited by the memory capacity. The opacity of maps is adjustable at the layout manager (see chapter 11.4).

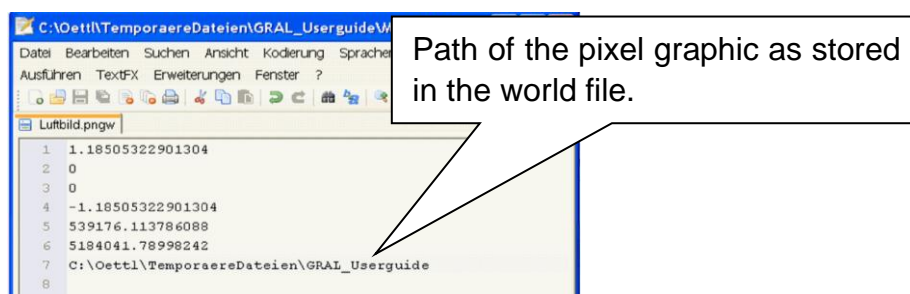
Any map to be geo-referenced must be in first order (on top) in the "Object manager" (to be opened with the button left to the "Open maps" button).

At the geo-referencing process, the maps will be scaled but not turned. So it is necessary to use north warded maps.

11.6. Import of already geo-referenced maps


Accepted formats are shapefiles (.shp) and geo-referenced pixel graphics (.jpg, .gif, .bmp, .tiff, .png). For the latter, a corresponding World-file is expected, which has the same name as the pixel file but an additional "w" is appended to the corresponding extension (e.g. .jpg-> .jgw / .jpgw or .gif-> .gfw / gifw). The GUI does currently not provide a coordinate transformation for maps.

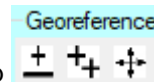
If a pixel graphic is geo-referenced within the GIS interface, the path of the pixel graphic is stored in the world file. If the project or the pixel graphic is moved to another folder, the GUI will not be able to open this map again but will ask for the new location of the graphic file. If the graphic file and the corresponding world file are stored in the same directory (preferably in the subdirectory "Maps" of the GRAL project), then, GRAL can automatically detect the location even though the whole project has been moved to a different directory.



11.7. Geo-referencing of pixel graphics

Menu: "File – Georeference map"

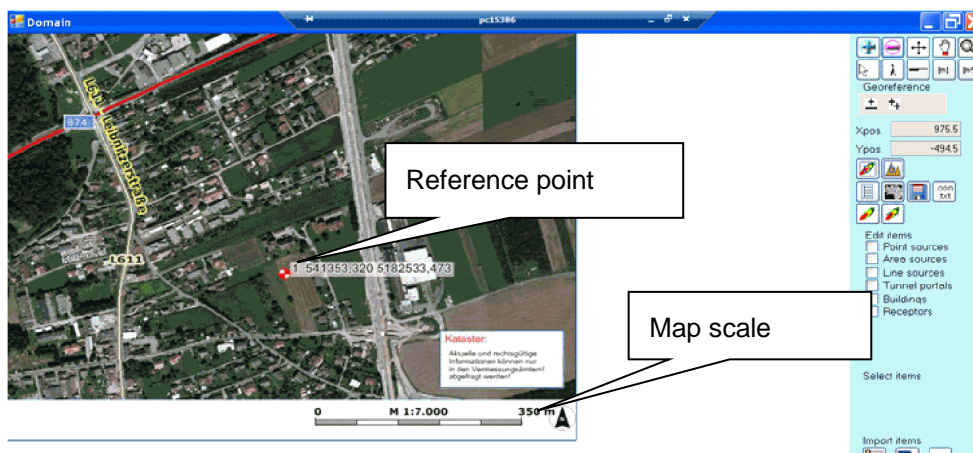
Toolbox:  symbol within the georeference group



Georeferencing means, that the position and extension of a map can be related to a real-world geographic coordinate system.

The GUI supports geo-referencing using two reference points or one reference point and a map scale.

During the geo-referencing process navigation of the map is supported either by the arrow-keys of the keyboard or by the mouse-buttons.

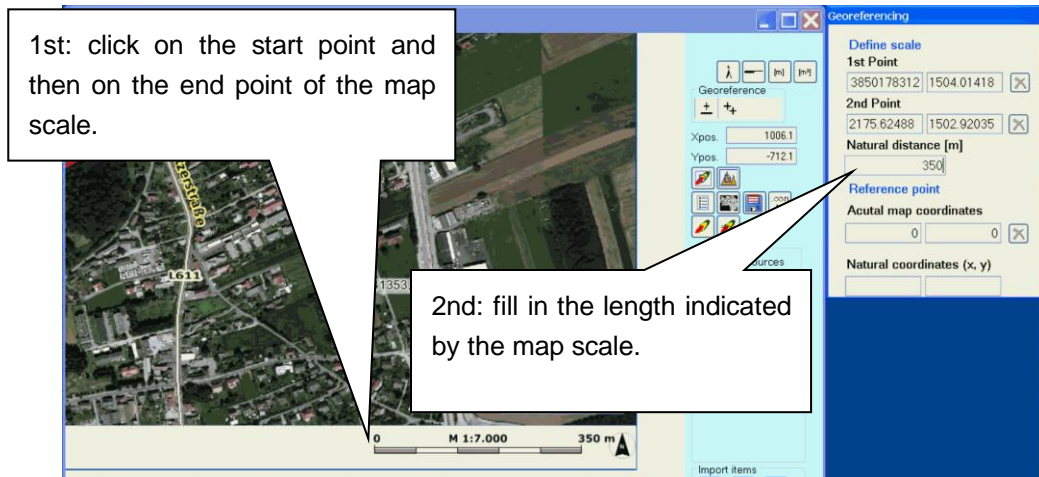


11.7.1 Geo-referencing using a map scale and one reference point

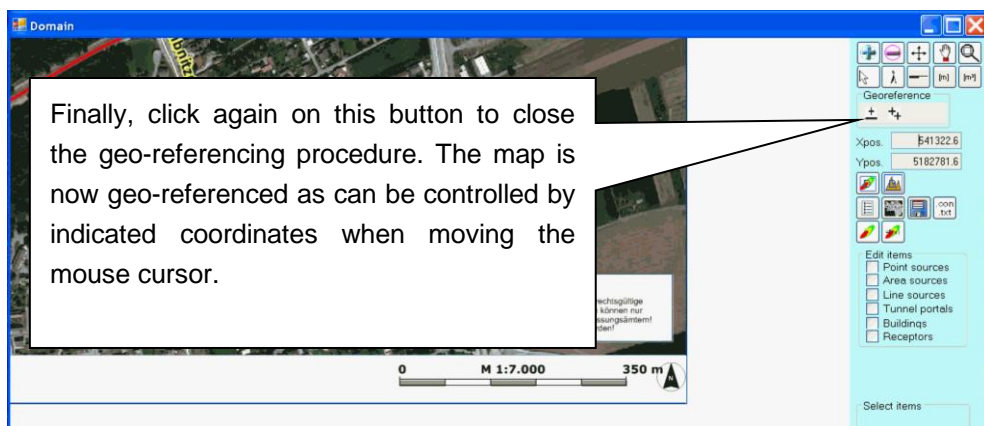
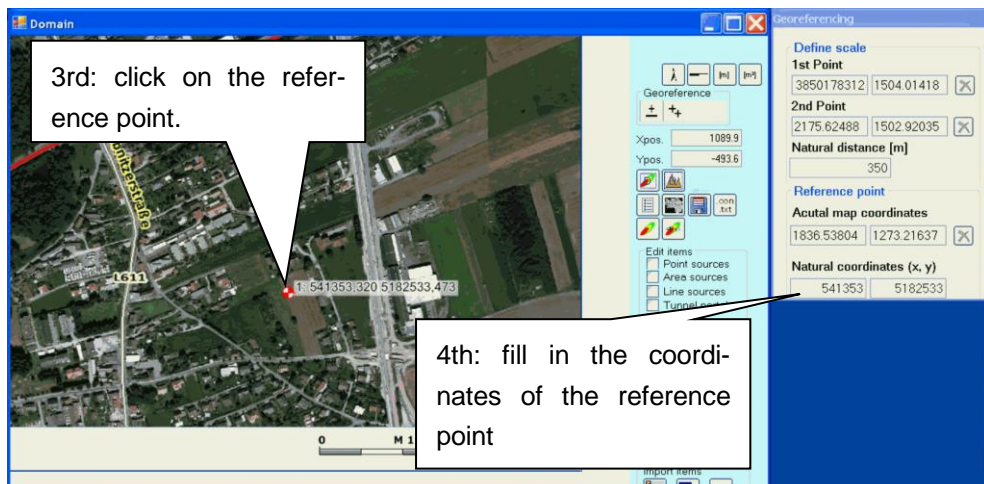
Menu: "File – Georeference map – Scale and one point"



GIS Interface



False values can be removed by clicking the „X“ button to the right of the erroneous input data. Then click again on the map to correct the data.



During the geo-referencing procedure, the map can be moved and zoomed either by using the following keys:

+ key: zoom in

- key: zoom out

Arrow keys: Move map in the various directions

or by using the scroll-button of the mouse for zooming


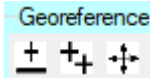
or pressing the middle mouse button for moving (see chapter 19.4)

Erroneous geo-referenced maps can be again being geo-referenced at any time. Wrong coordinates can be corrected by clicking on the button to the right of the coordinates. In this way the coordinates are reset to zero, clicking again on the map the coordinates are overwritten.

Automatically the top layer in the project is the one taken for the geo-referencing procedure, thus the map to be geo-referenced must be on top in the list of the "Object manager".

11.7.2 Geo-referencing using two reference points

Menu: "File – Georeference map – Two points"

Toolbox:  symbol within the georeference group 


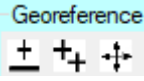
Click on the button as indicated in the figure below. The procedure is the same as outlined before, though, coordinates of two reference points must be filled in. Click a second time on the button to close the reference procedure.

A corresponding world-file is automatically generated by the GIS-Interface after each successful geo-referencing and stored in the subdirectory "Maps". Such geo-referenced pixel graphics can be further used in any other GIS application or GRAL project. To do so, click on the world-file (not on the pixel file) when adding the map.

World files use file extension like ".pngw", ".tfw", ".bmpw" or ".jpgw".

11.8. Adjustment of geo-referenced base maps

Menu: “File – Georeference map – Move and zoom base maps”

Toolbox:  symbol within the geo reference group 

It is possible to adjust the position and dimension of geo-referenced maps by using this button or menu entry.

Select the button and adjust the position and scale of the base map using the cursor keys or the + and – keys.

If the position and scale is set, unselect this menu entry or the button and the new position is stored.

By pressing the “esc” key the original position is restored.

Automatically the top layer in the project is the one selected for this procedure, thus the map to be adjusted must be on top in the list of the “Object manager”.

The adjustment of base maps does not affect the corresponding world file.

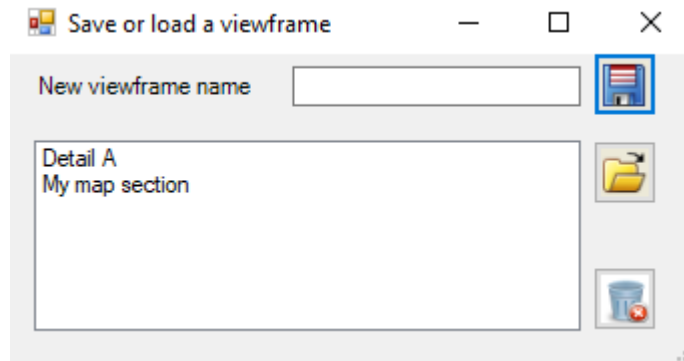
During this process, the navigation of the map is supported by the mouse-wheel and the middle mouse button.

11.9. Store and recall map details

Menu: “View – Save or load a viewframe”

Toolbox:  symbol

This option is used to save and recall map sections.

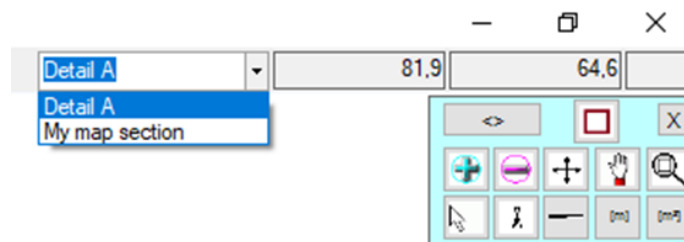


Specific map details can be stored by filling in any name in the text field “New viewframe name” and pressing the “Save” button.

Whenever the map detail is needed, it can be restored by selecting the desired name from the list box with a double click or by selecting the line and pressing the “Open” button.

Use the litter button to delete a selected entry from the list.

For a fast access the saved map sections are available in the menu bar of the GIS window. Simple select an entry to recall the map detail.



11.10. Definition of the GRAL model domain

Menu: "Edit – GRAL model domain"

Toolbox:  symbol

Press the button as indicated in the figure below, then left click on the map to mark one corner point of the domain. Move the mouse to the opposite corner point of the domain and release the mouse button to finish the procedure, which can be repeated at any time. If you press the shift key and the mouse button, it is possible to enter exact coordinates for the edge points.

It is recommended that no buildings are located nearby the borderlines of the GRAL model domain, otherwise they should be deleted (see chapter 11.12).

Use the layout manager to define the color and line width of the domain-frame, or to draw the GRAL concentration raster grid (see chapter 11.4).

It is not possible to define a GRAL domain area, if an invalid GRAMM model domain exists. In such cases close the GIS window and delete invalid GRAMM information by clicking the X buttons in the "Topography" tab (see chapter 16.3).

11.11. Editing emission sources and obstacles

11.11.1 Point sources

Menu: "Edit – Point sources"

Toolbar: ☐ Point sources checkbox

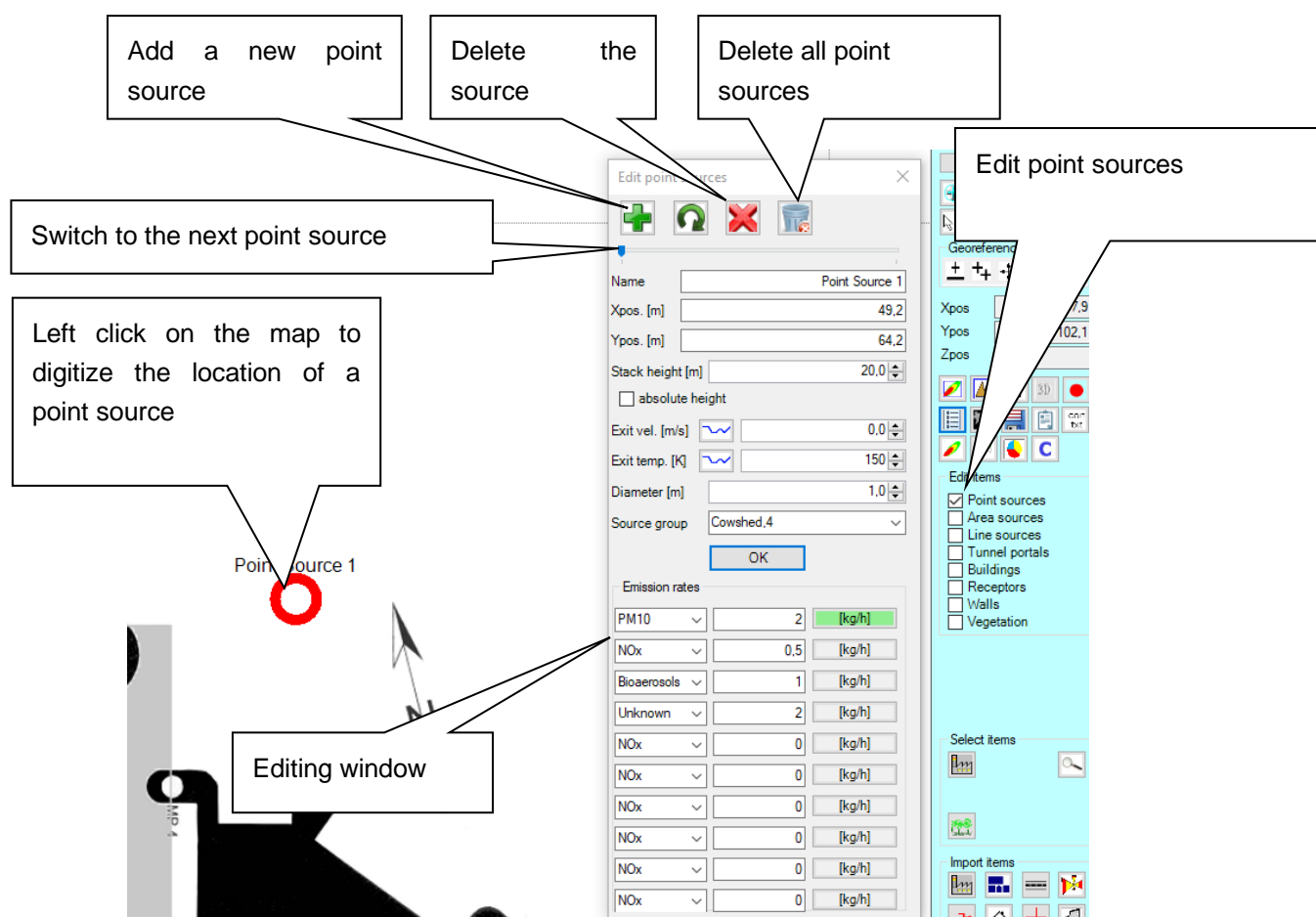
Use this dialog to digitize and define the parameters for point sources.

To save all data and finish the editing process, click the box again so that the checkmark disappears or press the OK button inside the point source dialog.

Data is stored only if a name is given to each source and if the location of the source has been digitized.


A point source location is digitized by left-clicking the location on the map or you can enter the coordinates into the textboxes Xpos and Ypos.


Up to ten different emission rates for different pollutants can be attributed.



The drawing options of point sources (diameter, color, show labels) can be set in the layout manager (see chapter 11.4).

Name	identifies the source and a name is needed, to store the data.
Stack height	height above ground.
Absolute height	used to set the point source to an absolute height above sea. Absolute heights force an error at the GRAL computation core without topography (see chapter 17.4).
Exit velocity	the velocity of exhaust gas in vertical direction.
Exit temperature	temperature of the exhaust gas above ambient temperature. Negative values are not supported.
Diameter	stack diameter at the exit section.
Source group	desired source group for this point source.

The  symbol is used to create a preset for an annual time series for the exit velocity and/or the exit temperature. This time series is used in transient GRAL simulations only.

The  button is used to save and reload all settings and the position of the source at the map is actualized.

11.11.2 Line sources and emissions from road traffic

Menu: "Edit – Line sources"

Toolbar: ☐ Line sources checkbox

Use this dialog to digitize and define the parameters for line sources.

To save all data and finish the editing process, click the box again so that the checkmark disappears or press the OK button inside the line source dialog.

Data is stored only if each line source is assigned a name, a section number, a source group and if edge points have been digitized.

Input data is only saved if the „Save SG“ button is pressed at the end!

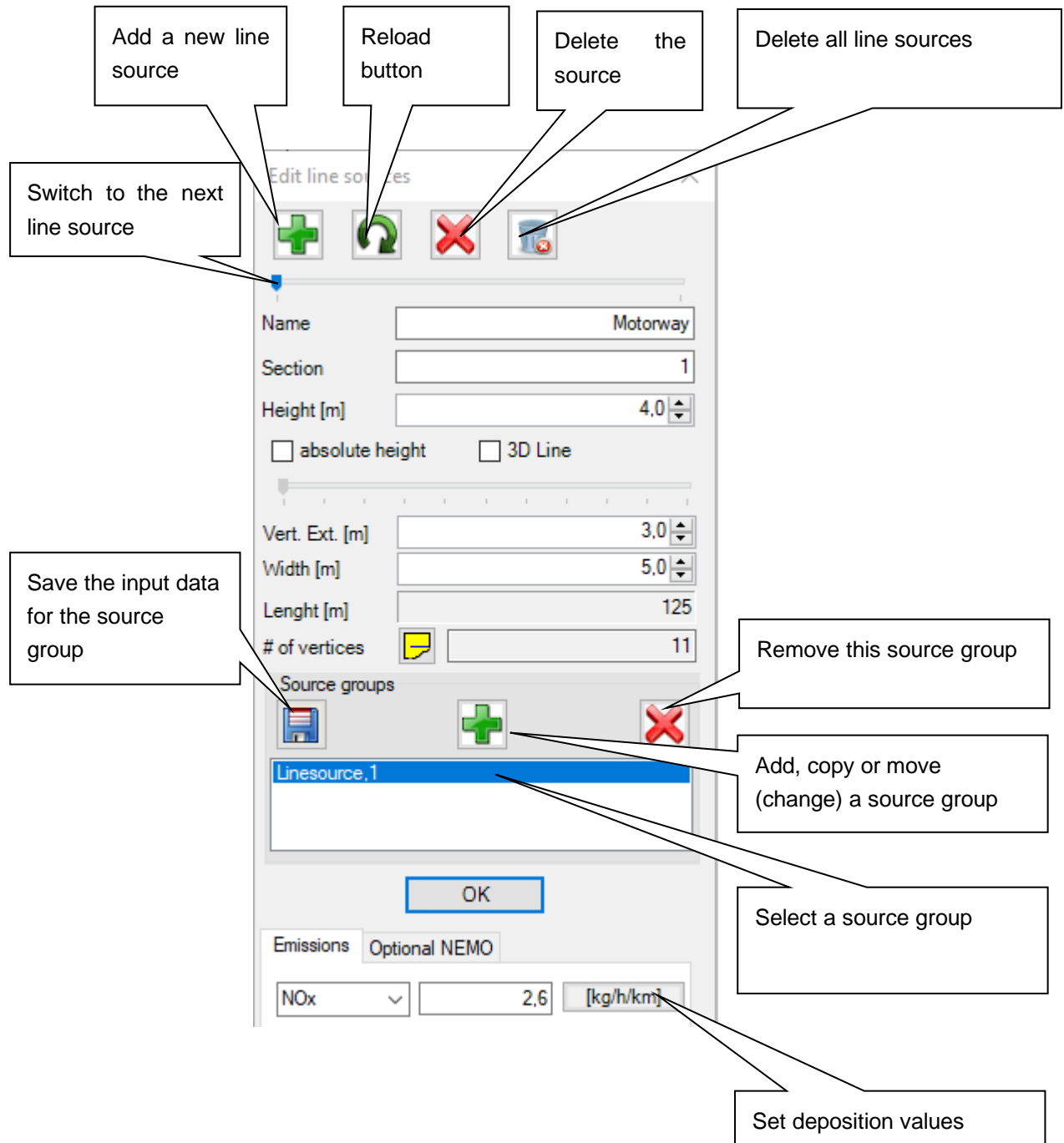
A line source polyline can be assigned to multiple source groups. This makes sense, because different pollutants might cause different diurnal or seasonal emission modulations.

Up to ten different emission rates for different pollutants can be attributed per source group.


If line sources are digitized manually, the following steps are recommended:

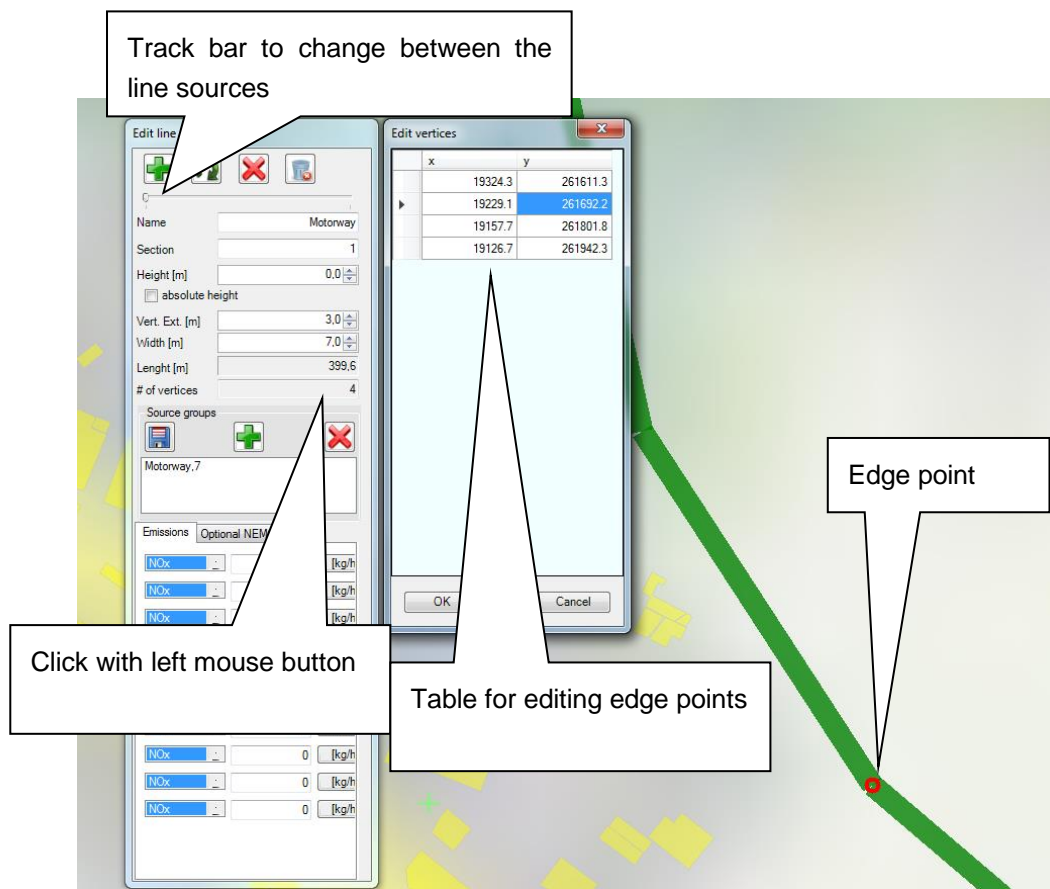
1. Fill in a name, the section, the height above ground or above sea, the vertical extension and the width
2. Add a source group
3. Digitize the edge point of the line source polygon by clicking to the map
4. Set the pollutant types and the emission rates
5. Press the button "Source groups – Save" (disk symbol) or the store&reload button (circular arrow)

The pollutant types can only be defined and changed in the topmost item of the source group list box.



The coordinates of an edge point can be changed by

- Use the edge point table by clicking the button 
- Use the "Select items" function and the right mouse button (context menu - see chapter 11.12.1).



The new position is shown by a little red circle.

Name	identifies the source and a name is needed, to store the data.
Section	a defined section for this road
Height	base height above ground (default 0 m)
Absolute height	used to set the base height to an absolute height above sea (e.g. for bridges). Absolute heights force an error at the GRAL computation core without topography (see chapter 17.4).
3D Line	Enable the creation of ascending or descending line sources
Vert. Ext [m]	vertical extension of the line source
Width [m]	width of the line source
Source group	desired source group for this line source.

The drawing options of line sources (linewidth, color) are described in chapter 11.4.

The height of line sources (i.e. lower boundary of the line source) can be defined as height above ground level (standard value) or optional as height above sea (checkbox “absolute height”). The option absolute height might be useful to digitize bridges.

If the absolute height is lower than the GRAL surface, the source is automatically set to the ground level by the GRAL computation core. If you would like to import line sources with absolute heights, set the height data at the shapefile to negative absolute values or use the checkbox “absolute height” at the import dialog.

Absolute heights force an error at the GRAL computation core without topography see chapter 17.4).

The option “3D Line” enables ascending or descending line sources. Use this option to define the height for each corner point. Switch between all edge points using the slider below the checkbox “3D Line”. It is possible to import shape files with 3D line sources.

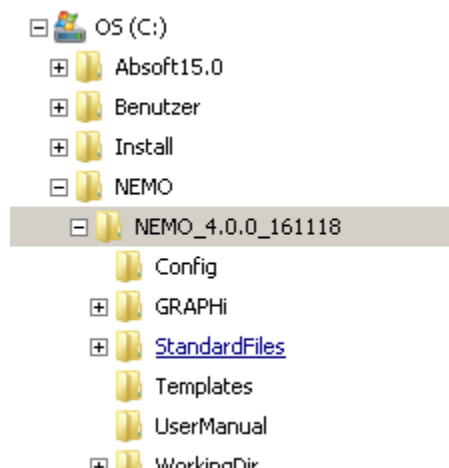
The field “Vert. Ext.” (vertical extension) when editing line sources has different meanings dependent on its value (because of the compatibility to old projects):

Positive values: the lower boundary of the line source is equal to the ground level or the value typed in the field “height”, respectively. The vertical extension of the line source is set to the value provided in the field “vert. ext.”.

Zero: the lower boundary of the line source is equal to the ground level or the value typed in the field “height”, respectively. The vertical extension of the line source is set automatically to 3 m.

Optional NEMO tool

In order to compute road traffic emissions, the Network Emission Model NEMO 5.0.0 developed by Graz University of Technology must be purchased (Contact: Dr. Martin Rexeis, Institute for Internal Combustion Engines and Thermodynamics). NEMO must be stored on the local C-disk as indicated in the next figure. Otherwise, all paths in the file GRAL.nemo in the working directory of the GUI must be adopted.



A line source can be digitized by left clicking on the map. The actual digitized line source is indicated during the process. Use a **right**-click to finish the procedure.

For calculating emissions from road traffic using NEMO, the following input parameters are required: Average daily traffic, share of heavy-duty vehicles, slope, traffic situation, and the base year.

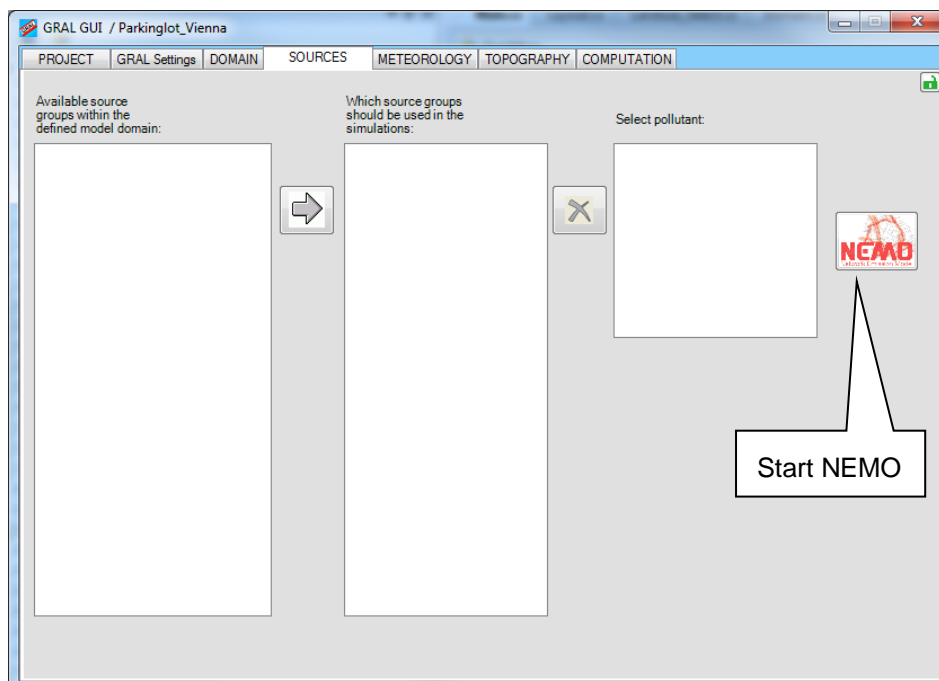
Finish editing and close the GIS Interface. Select the “Sources” tab in the main window and click the “NEMO” button. You’ll be asked for the location of the NEMO executable. Doing this starts the calculations, which can take some seconds up to some minutes depending on the number of line sources.

By default line source emissions are split into four different source categories:

- (i) passenger cars exhaust
- (ii) heavy duty vehicles exhaust
- (iii) passenger cars re-suspension
- (iv) heavy duty vehicles re-suspension.


The user can define the source group number for each of these categories (see also section 10). Another option is to sum up all sources for each line source, which can be done by clicking the box (such that the tick disappears) in the window appearing after the selection of the NEMO executable.

The result of the NEMO computation is not automatically updated if the GIS interface is still open. Close the GIS Interface first and open it again to control NEMO results.



11.11.3 Area sources

Menu: "Edit – Area sources"

Toolbar:  Area sources checkbox


Use this dialog to digitize and define the parameters for area sources.

To save all data and finish the editing process, click the box again so that the checkmark disappears or press the OK button inside the area source dialog.

Data is stored only if a name is given to each source and if the location of each source has been digitized.

Up to ten different emission rates for different pollutants can be assigned.

An area source can be digitized by left clicking on the map. The actual digitized area source is indicated during the process. Use a **right-click** at the **second to last** point of the polygon to finish the procedure.

Once an area source has been digitized, the vertices can be edited in a table by clicking on the button . It is also possible to use the "Select items" function and the right mouse button (context menu - see chapter 11.12.1).

Each area source is automatically split up into equally sized squares. By default, 26 squares are used to keep the form of the polygon. By changing the raster size, the number of squares used to raster the area source increases or decreases.

Edit area sources

Name: Parking place

Mean height [m]: 2.0

☐ absolute height

Vert. ext. [m]: 3.0

Source group: Parking lot.3

Raster size [m]: 2.2

of vertices: 5

Area [m²]: 115.8

OK

Emission rates

NOx	0.14	[kg/h]
NOx	0	[kg/h]
NOx	0	[kg/h]
NOx	0	[kg/h]
NOx	0	[kg/h]
NOx	0	[kg/h]
NOx	0	[kg/h]
NOx	0	[kg/h]
NOx	0	[kg/h]
NOx	0	[kg/h]

Name	identifies the source and a name is needed, to store the data
Mean height	mean height above ground
Absolute height	used to set the mean height to an absolute height above sea. Absolute heights force an error at the GRAL computation core without topography (see chapter 17.4).
Vert. Ext [m]	the vertical extension. Particles are released in a height of mean height \pm Vert.Ext. / 2
Source group	desired source group for this point source.
Raster size	size of the partial squares
Source group	desired source group for this area source.

Absolute source heights

The height of area sources can be defined as height above ground level (standard value) or optional as height above sea (checkbox “Absolute Height”). If the absolute height is lower than the surface, the source is automatically set to the ground level by the GRAL computation core. If you would like to


import area sources with absolute heights, set the height data at the shapefile to negative absolute values.

Absolute heights force an error at the GRAL computation core without topography (see chapter 17.4).

The drawing options of area sources (line width, color, fill options, show labels) are described in chapter 11.4.

11.11.4 Tunnel portals

Menu: "Edit – Tunnel portals"

Toolbar:  checkbox

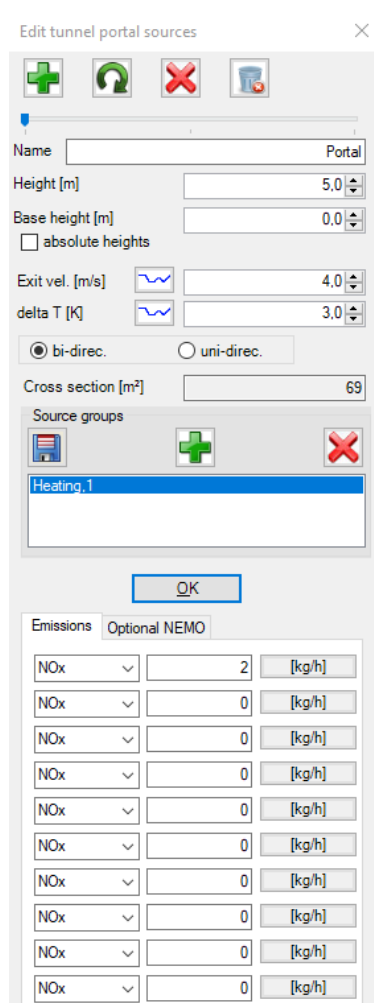
Use this dialog to digitize and define the parameters for portal sources.

To save all data and finish the editing process, click the box again so that the checkmark disappears or press the OK button inside the portal source dialog.





Data is stored only if a name is given to each source and if the location of each source has been digitized.

Imagine looking out of the tunnel: Left-click on the map for digitizing the right corner of the portal, then right-click to digitize the left corner of the tunnel portal. A symbol indicates the flow direction out of the portal.

Make sure that all source groups to be used for portal sources are defined before creating any portal source (source groups are assigned to tunnel portals as with line sources).



Edit tunnel portal sources








Name

Height [m]

Base height [m]

☐ absolute heights



Exit vel. [m/s] 

delta T [K] 

☒ bi-direc. ☐ uni-direc.

Cross section [m²]

Source groups


 

Heating,1

Emissions Optional NEMO

NOx	<input type="text" value="2"/>	[kg/h]
NOx	<input type="text" value="0"/>	[kg/h]
NOx	<input type="text" value="0"/>	[kg/h]
NOx	<input type="text" value="0"/>	[kg/h]
NOx	<input type="text" value="0"/>	[kg/h]
NOx	<input type="text" value="0"/>	[kg/h]
NOx	<input type="text" value="0"/>	[kg/h]
NOx	<input type="text" value="0"/>	[kg/h]
NOx	<input type="text" value="0"/>	[kg/h]
NOx	<input type="text" value="0"/>	[kg/h]

Name	identifies the source and a name is needed, to store the data
Height [m]	height above ground
Base height [m]	height of the tunnel base
Absolute height	used to set the portal source to an absolute height above sea. Absolute heights force an error at the GRAL computation core without topography (see chapter 17.4).
Exit vel. [m/s]	the velocity of exhaust gas in horizontal direction.
Delta T [K]	temperature difference between the temperature inside the tunnel and the ambient temperature. Positive and negative values are supported.
Bi-direc./uni-direc.	defines, if the portal is driven on bi directional or uni directional
Source group	desired source group for this point source.

The  symbol is used to create a preset for an annual time series for the exit velocity and/or the temperature difference. This time series is used in transient GRAL simulations only.

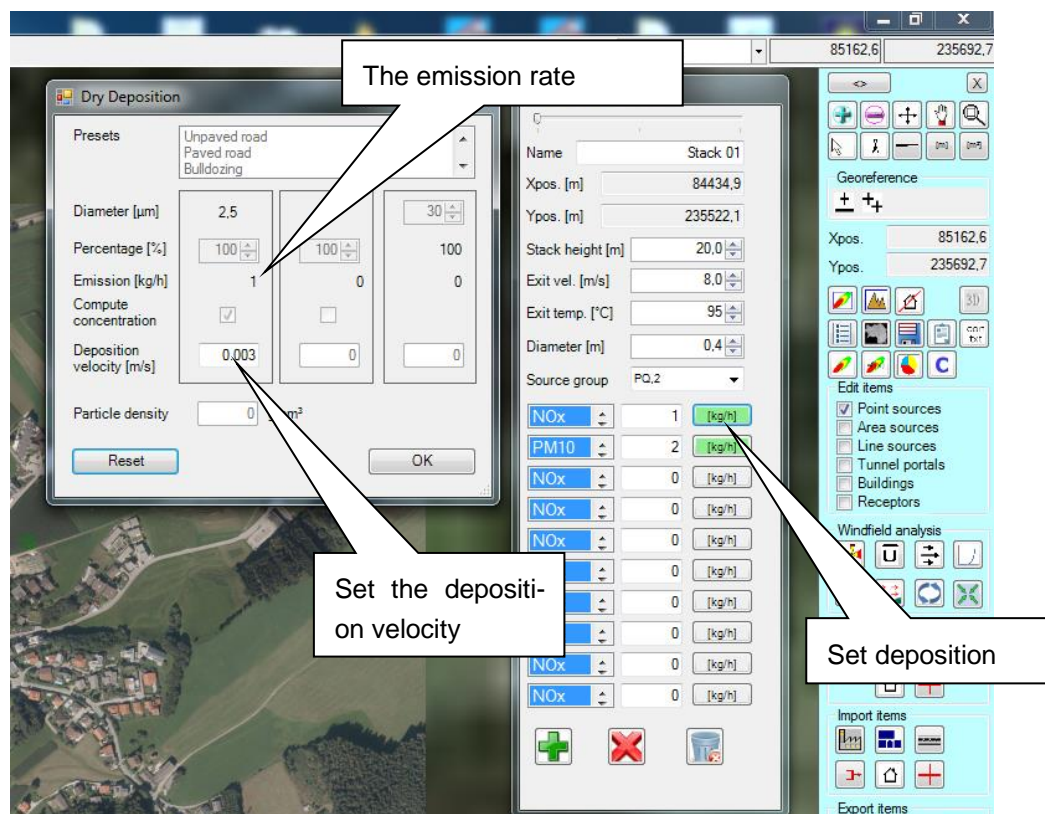
An emission computation using NEMO is not foreseen currently, thus the selected traffic situation is not used at the moment. The selection “bi-directional” or “uni-directional” has no effect on the dispersion calculations.

The drawing options of portal sources (linewidth, color, show labels) are described in chapter 11.4.

11.11.5 Edit deposition settings

It is possible to set the deposition settings for each source and pollutant individually. The deposition is set by clicking on the unit-button nearby the emission rate textbox.

If a deposition is defined, the background color of this button changes to green.



At gaseous pollutants the deposition dialog allows to edit the deposition velocity. All other options are greyed and locked.

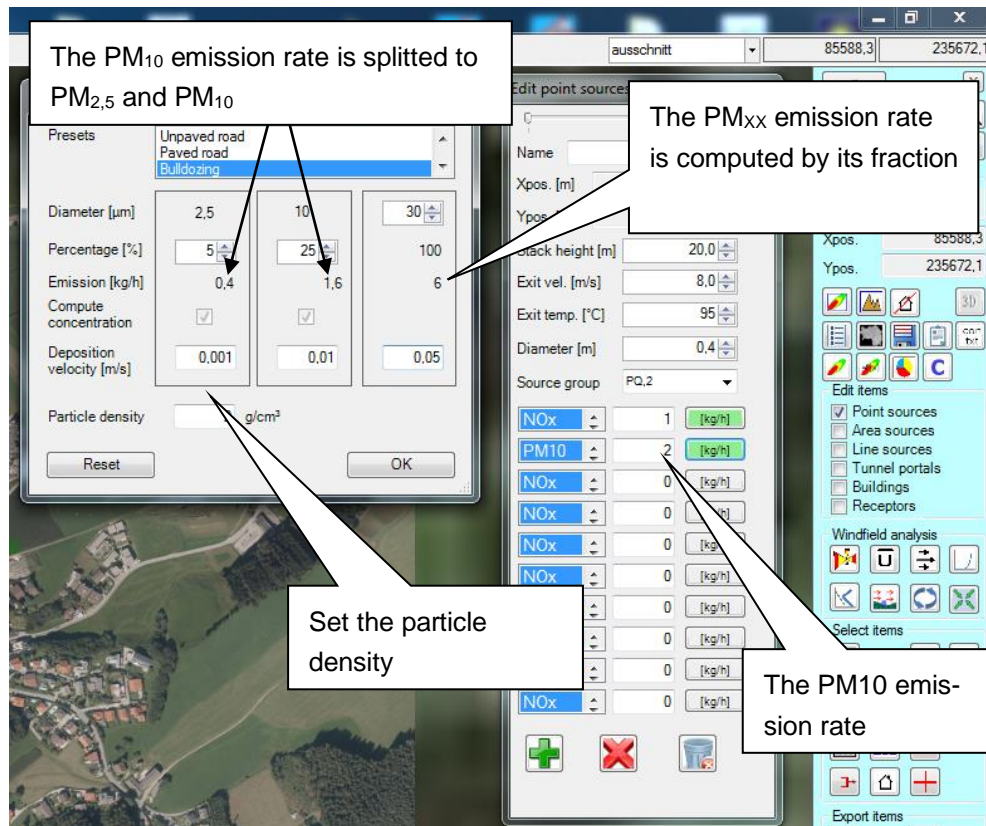
Computing dust particles (pollutants $PM_{2,5}$ or PM_{10}) allows the definition of the particle density. This value and the particle diameter are used by the GRAL computation core to compute the sedimentation velocity.

In this case it is possible to define fractions for PM_{10} or PM_{xx} particles. XX stands for the diameter of the largest particle size. The user can set this parameter.

Pressing the “Reset” button will delete all deposition information, no deposition will be computed.

GIS Interface

Here you see an example for PM₁₀.




The preset is stored in a file called "DepositionSettings.txt". This file is stored in the GUI application folder. This text file is tab separated and can be edited and appended using a text editor or a spreadsheet application.

In the case of dust (PM₁₀ or PM_{2.5}) emissions, GRAL calculates the three grain sizes PM_{2.5}, PM₁₀ and PM_{xx} simultaneously. The largest grain class (PM_{xx}) is only taken into account for the mass deposited on the ground. If PM₁₀ is calculated, the concentration of PM₁₀ in the atmosphere is the sum of the concentrations from the grain sizes PM_{2.5} and PM₁₀.

The evaluated "Deposition" files contain the calculated deposition value of all particle sizes when calculating PM₁₀ or PM_{2.5}. The file name part "Pollutant" "PM₁₀" or "PM₂₅" solely indicates the calculated pollutant for the dispersion (concentration) calculation.

11.11.6 Buildings and obstacles

Menu: "Edit - buildings"

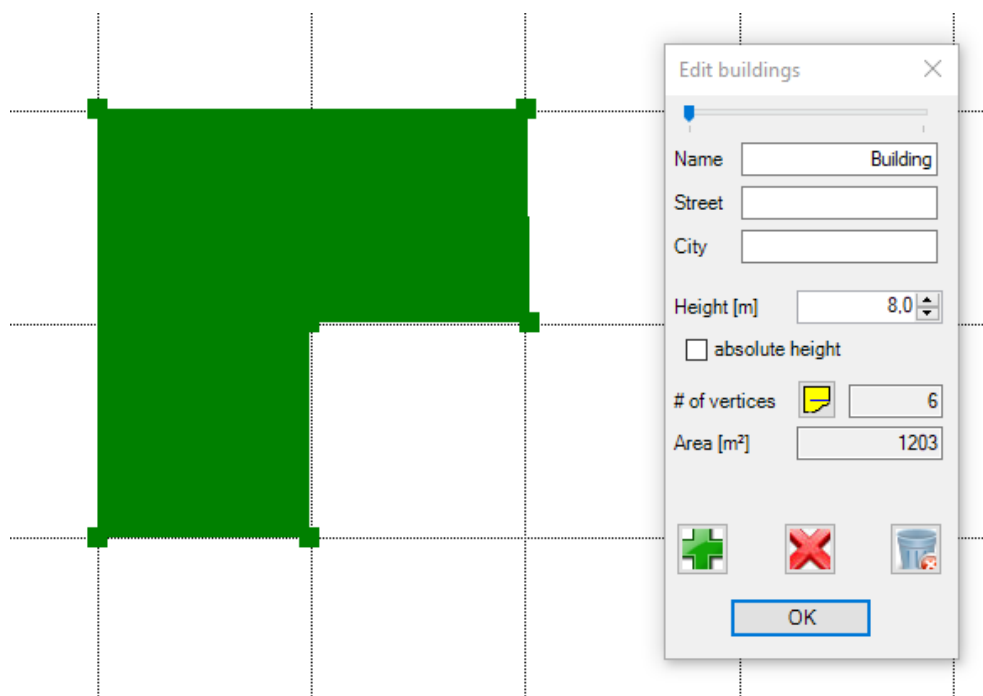
Toolbar:  Buildings checkbox

Use this dialog to digitize and define the parameters for buildings.


To save all data and finish the editing process, click the box again so that the checkmark disappears.

Data is stored only if a name has been assigned to the obstacle and if the edge points have been digitized.

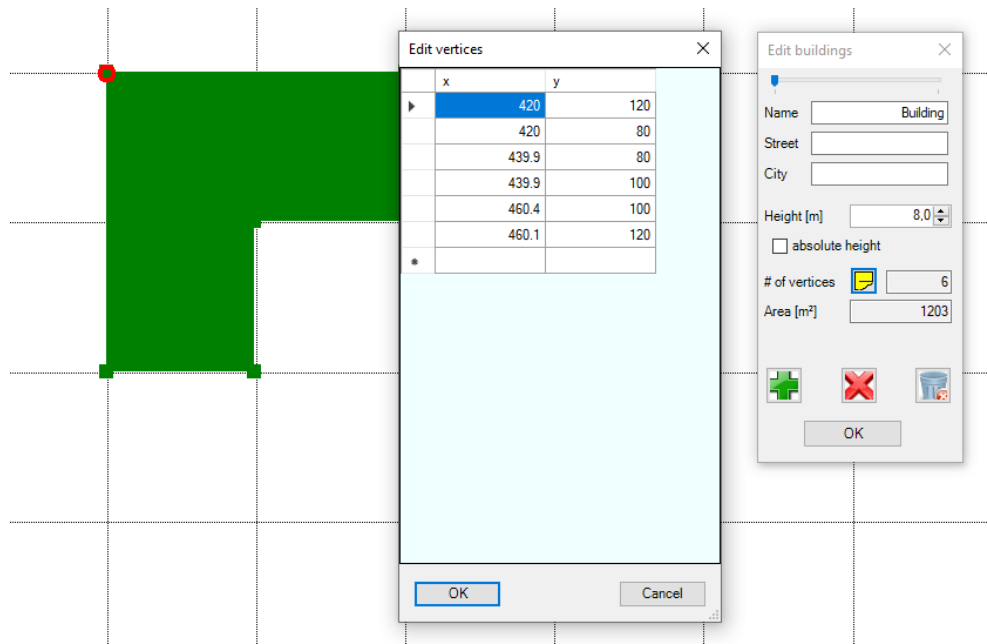
An obstacle can be digitized by left clicking on the map. The actual digitized building shape is indicated during the process. Use a **right-click** at the **second to last** point of the polygon to finish the procedure.



The coordinates of edge points can be changed by

- Use the "Select item" function and the right mouse button (see chapter 11.12.1).
- Edit the coordinates in a table by clicking on the button .


The absolute height is used to set the top of the building to an absolute height above sea. Absolute heights force an error at the GRAL computation core without topography (see chapter 17.4).



The drawing options of buildings (linewidth, color, fill options, show labels) are described in chapter 11.4.

11.11.7 Removing obstacles at the edge of a GRAL domain

Menu: "Edit - Delete buildings outside the GRAL domain"

Toolbar:  button

Buildings and obstacles near the lateral boundaries of the GRAL domain can sometimes cause numerical problems. Use this button to remove these buildings.

11.11.8 Receptor points

Menu: "Edit - Receptor points"

Toolbar: ☐ Receptors checkbox

Use this dialog to digitize and define the parameters for receptor points.

To save all data and finish the editing process, click the box again so that the checkmark disappears.

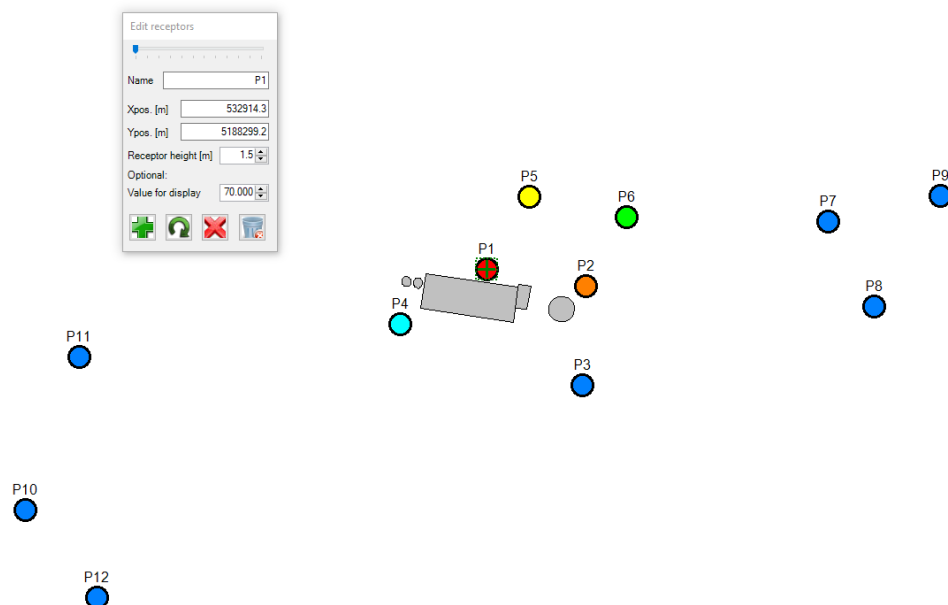
Data is stored only if a name is given to each receptor point and if each receptor has been digitized.

A receptor can be digitized by left clicking on the map.

The minimum height of any receptor needs to be larger than half the height of the vertical dimension of the concentration layer (see "GRAL settings" tab, chap. 8).

Each receptor can be assigned a value, which can be visualized on the map afterwards. The following figure shows an example of several receptor points, with different values visible by different colors.

The general drawing options of receptor points (diameter, color, labels, display values) are described in chapter 11.4.



The concentration at the receptor point is determined in GRAL Version 20.01 at the real position of the receptor with the following volume:

dx	Horizontal grid resolution of the concentration grid (see chapt. 8.2)
dy	Horizontal grid resolution of the concentration grid (see chapt. 8.2)
dz	Vertical dimension of the concentration grid (see chapt. 8.2)
xr	Receptor x position
yr	Receptor y position
zr	Receptor z position

The dimension of the receptor concentration volume is defined by:

$$xr \pm dx / 2, yr \pm dy / 2 \text{ and } zr \pm dz/2$$

The position of a receptor is gridded if a receptor is located at the boundary cell of the GRAL domain or near a building.

In the Steady State mode, a time series for the receptor concentrations and the meteorology at the receptor point can be created using the evaluation buttons in the Computation tab (see chapter 15.5.1.).


In Transient mode, the GRAL kernel 20.01 or higher writes a time series with the receptor concentrations and the meteorology at the receptor point. This file is called "Receptor_Timeseries_Transient.txt". In addition, the estimated statistical error for each receptor point is written to this file, also in the Steady State mode, as soon as the calculation is finished.

Example output for one receptor point and one source group in the Transient mode:

Rec/SourceGroup	Rez/SG: 2	Meteo	Rez	
X	9,4		9,4	
Y	43,1		43,1	
Z	10		10	
Date/time	[µg/m³]	U [m/s]	V [m/s]	SC
01.01.2020 01:00:00	1,4696e+002	-1,36	0,22	4
01.01.2020 02:00:00	1,2402e+002	-1,36	0,22	4
01.01.2020 03:00:00	1,2020e+002	-1,36	0,22	4
Est. statistical error [%]	0,3%			

11.11.9 Walls

Menu: "Edit - Walls"

Toolbar:  Walls checkbox

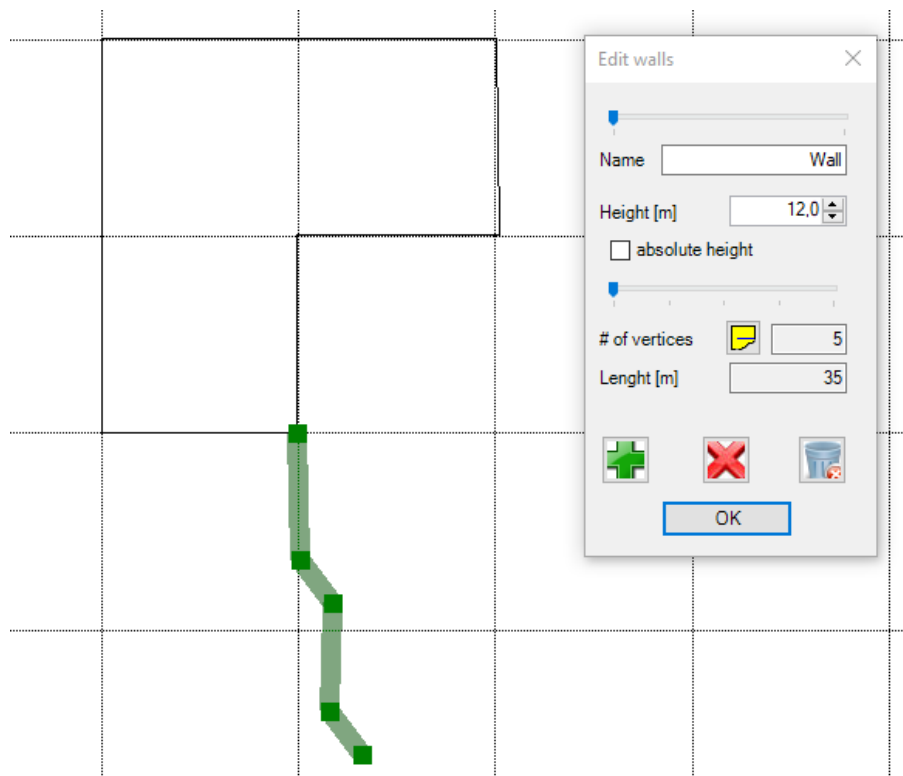
A wall is a special type of building. The width of a wall is set to the horizontal raster size of the microscale flow-field model.


The height of each edge point can be set by the user, ascending or descending walls are allowed.

To save all data and finish the editing process, click the box again so that the checkmark disappears.

Data is stored only if a name is given to each wall and if each wall has been digitized.

A wall can be digitized by left clicking on the map. The actual digitized wall shape is indicated during the process. Use a **right-click** at the **second to last** point of the polyline to finish the procedure.



If all edge points of a wall have been defined, the corner points can be edited in a table by clicking on the button  or by using the "Select item" function and the context menu (see chapter 11.12.1).

The option absolute height is used to set the top of the wall edge points to absolute heights above sea. Absolute heights force an error at the GRAL computation core without topography (see chapter 17.4). A mixture between absolute and relative heights along one wall is not possible.

11.11.10 Vegetation

Menu: "Edit - vegetation"

Toolbar: ☐ Vegetation checkbox

Use this dialog to digitize and define the parameters for vegetation areas.

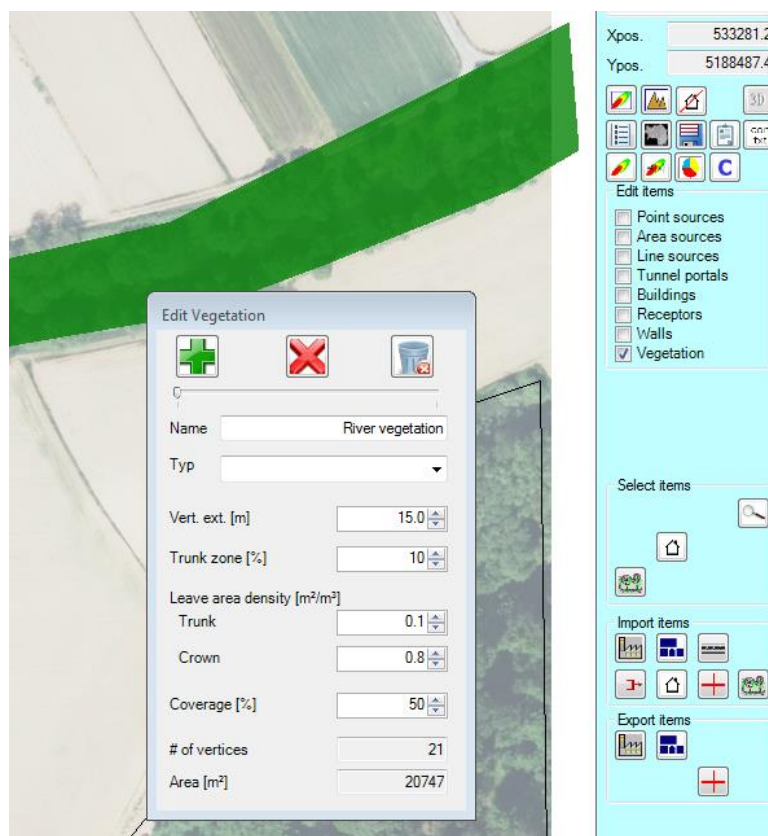
To save all data and finish the editing process, click the box again so that the checkmark disappears.


Data is stored only if a name is given to each vegetation layer and if each vegetation layer has been digitized.

Vegetation can be digitized by left clicking on the map. The actual digitized vegetation shape is indicated during the process. Use a **right-click** at the **second to last** point of the polygon to finish the procedure.

The following properties can be set for vegetation (note that there is a drop down menu with some default values for different types of vegetation to facilitate selecting proper input values):

Vert. ext. [m]:	total height of the vegetation
Trunk zone [%]:	height of trunks in percentage of the total height, e.g. if the total height is 10 m, and the trunk zone is set to 20 %, then the trunk is assigned a height of 2 m.
Leave area densities [m ² /m ³]	values separated for the trunk and the crown zones.
Coverage [%]:	defines the area occupied of the specified vegetation type within the digitized area in percentage of the total area.




Once vegetation has been digitized, the vertices can be edited by clicking on the button  or by using the “Select item” function and the context menu (see chapter 11.12.1).

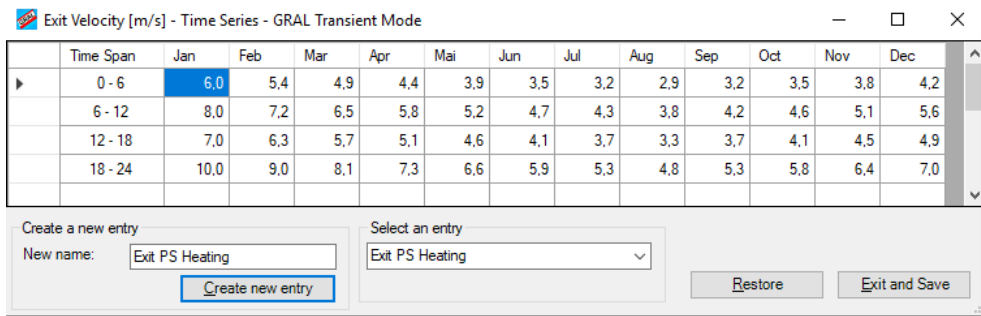
The drawing options of vegetation (line width, color, fill options, show labels) are described in chapter 11.4.

11.11.11 Time series for exit temperature and exit velocity

Point sources and portal sources support a time series for the exit temperature and the exit velocity in the transient GRAL mode starting with GRAL version 20.01.

Therefore, there is the button  in the point source and portal source dialog.

A time series is digitized in the following dialog.



The dialog shows a table with months as columns and time spans as rows. The first row (0-6) is selected. Below the table are fields for creating a new entry (New name: Exit PS Heating) and selecting an existing entry (Exit PS Heating). Buttons for 'Create new entry', 'Restore', and 'Exit and Save' are at the bottom.

	Time Span	Jan	Feb	Mar	Apr	Mai	Jun	Jul	Aug	Sep	Oct	Nov	Dec
▶	0 - 6	6,0	5,4	4,9	4,4	3,9	3,5	3,2	2,9	3,2	3,5	3,8	4,2
	6 - 12	8,0	7,2	6,5	5,8	5,2	4,7	4,3	3,8	4,2	4,6	5,1	5,6
	12 - 18	7,0	6,3	5,7	5,1	4,6	4,1	3,7	3,3	3,7	4,1	4,5	4,9
	18 - 24	10,0	9,0	8,1	7,3	6,6	5,9	5,3	4,8	5,3	5,8	6,4	7,0



It is possible to paste and copy values from this table.

To create a new entry, simply add a new name and press the button “Create new entry”.

If you like to assign an existing entry to the source, use the list box “Select an entry” and the existing values are displayed.

The button “Exit and save” writes the values to a file to the project folder “Emissions” and assigns the time series to the source.

The button “Restore” removes the time series from the recently edited source.

If a time series is assigned to a source, the symbol  changes to yellow color. 

When you create the emission files (tab computation in the main window) and time series are assigned to sources, time series files are written inside the folder “Computation”. The files are called

- TimeSeriesPointSourceVel.txt
- TimeSeriesPointSourceTemp.txt
- TimeSeriesPortalSourceVel.txt
- TimeSeriesPortalSourceTemp.txt

These files are simple text files. When needed, it is possible to edit the values for each hour of the time series within these files.

11.12. Selection of items (sources, buildings, receptors)

Menu: "Select"

Toolbox:



Click on the corresponding selection button or menu entry in the GIS Interface to select a desired item.

Left mouse button

If you select an item with the left mouse button, this item will be highlighted, and an info-box will be displayed with relevant information about the selected source. The info box is geo referenced, the design can be adjusted within the layout manager of the layer "ITEM INFO". You can show, hide or delete this layer using the object manager (chapter 11.3).

A selected item can be edited quickly by setting the tick in the editing box, or it can be easily deleted by pressing the "delete" key.

Right mouse button

Select an item with the right mouse button to open a context menu (see chapter 11.12.1).

11.12.1 Item Context Menu (edit, copy and paste an item)

Depending on the item type, there are various options in the context menu (right mouse button) when using the "Select items" function, such as

Edit	call the item dialog
Move	move the item, replace it using the left mouse key
Copy	create a copy, replace the copy using the right mouse key and use the context menu entry "Paste"
Delete	delete the item
Flip exit surface	flips the direction of a portal source
Move edge point	move an edge point of the item

Add edge point	add an edge point
Delete edge point	delete an edge point

During copying, objects can be rotated left or right in 10 degree steps using the R and L keys while the object is floating.

11.13. Select – Search items

Menu “Select – Search items”

Toolbox: 

Use the menu entry “Select –Search items” or the “magnifier glass” button at the “Select items” section of the toolbox to open this menu.

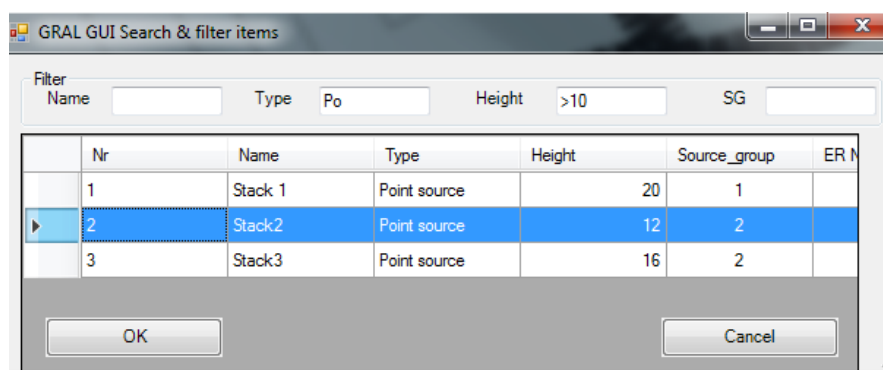
This menu is designed to display all items (sources, buildings, receptor points) in a data grid. It is possible to sort and to filter the items, as well as copy the information and to select an item.

You can sort the items by clicking to the top of the data column.

There are filters for the name, the type, the height and the source group (SG).

The filter for the height is a decimal filter, therefore you must use logical operators, like “=”, “<”, “>”, “<=” and “>”.

Example: Point sources (Po), height >10m, all source groups, all names



If you select an item (use the left column with the triangle to select a line) and double click the triangle or the "OK" button, the map will display the desired item and open the corresponding "Edit item" dialog.

If sources of a type appear with the same "Nr.", take a look to the column "Source_group", because this source will be assigned to multiple source groups.

The data of selected lines can be copied to the clipboard.

You can use a Right click in the header of the data table to hide or show additional columns.

From version 20.06 it is possible to delete and edit item data within the search dialog.

GRAL GUI Search, filter and edit items

Filter
Name Type Height SG

	Nr	Name	Type	Height	Vert. ext.	Source_group	ER Nr. 1	Poll. 1
	1	Motorway	Line source	4	3	1	2.5	NOx
▶	1	Area1	Area source	0	2	2	1.97	NOx
	1	Gebaeude	Building	8				
	1	Receptor	Receptor point	1				
	2	Receptor2	Receptor point	22				
	3	Receptor3	Receptor point	5				
	4	Receptor	Receptor point	1				
	5	Receptor	Receptor point	1				
	6	Receptor	Receptor point	1				
	7	Receptor	Receptor point	1				
	8	Receptor	Receptor point	1				

OK Save the changes Delete Items Cancel

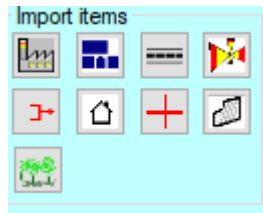
Select one or multiple lines and use the button “Delete item” to permanently delete items.

To change data, select the cell, change the value and leave the cell. The background color of the cell turns to brown and the text to a bold font. You can change several values, copy and paste is allowed. To save all changed values press the button “Save the changes”. In the example above the names and heights of the receptors 2 and 3 and the emission rate of the line source have been changed.

11.14. Import of sources, receptor points, buildings, wind data

Menu “File – Import”

Toolbox:



Shape files (.shp) containing information about sources, reference points, buildings or walls can be imported using the buttons in the GIS Interface or the menu entry.

Tunnel portals cannot be imported from shape files but from other GRAL projects.

Meteorological data can be imported to visualize wind roses within the GIS window.

11.14.1 Import of sources, receptor points, buildings

The GRAL domain must be defined prior to any import. Solely elements inside the model domain are imported.

Importing sources, reference points, or buildings from already existing GRAL projects is quite easy: After clicking the corresponding import button in the GIS Interface select the path and file to import. Note that the GUI stores all sources in the subdirectory “Emissions”, while buildings and receptor points are stored in the subdirectory “Computation”.

Subsequently the import of line sources is described briefly. Importing other features is done in a similar way.

After clicking on the corresponding button to import line sources, an additional window is opened, where the attributes of the shape file (stored in a .dbf file) can be selected.

It is imperative to define a source group.

As in many cases there'll be no such an attribute, columns can easily be added by pressing the button “Add column”. The user will be asked for the desired source group number. However, columns can be added for any kind of attribute, which is missing but needed.

Such can be for instance building heights, though only uniform heights can be defined for all buildings to be imported. Individual building heights can be manually adopted in the list view.

Emission rates are required in [kg/h]. In case that those are only available e.g. in [kg/a], a common conversion factor can be applied when importing sources (in this example to convert from [kg/h] to

[kg/a] the conversion factor would be 0.00011. Alternatively it is possible to check the checkbox $\wedge(-1)$ and use the conversion denominator, in the example 8760).

When adding columns, simple mathematical expressions including numerical values from another existing column can be applied. To do so, the first character in the input field must be the “=” sign, followed by the mathematical expression. The column to be used for the operation must be inserted with its full name considering capital letters, e.g. “=Breite*0.01”.

Only red indicated attributes must be defined in any case when importing features. All other (marked blue) attributes are thus optional.

The screenshot shows the 'Import line sources' dialog box. It contains several sections for defining import parameters. Callouts provide additional context:

- Define the deposition settings:** Points to the 'Gral. Deposition' section, which includes dropdowns for 'Gral. Deposition 1' through 'Gral. Deposition 10'.
- Add columns. Simple mathematical operations with any other existing column are possible to derive new values for an attribute.** Points to the 'Add column' button and the 'Common conversion factor' section.
- Conversion factor for emission rates:** Points to the 'Common conversion factor for transforming emissions to [kg/h/km]:' section, which includes a text input field (8760.00000) and a checkbox for $\wedge(-1)$.

The dialog also includes a table of imported data at the bottom:

	OBJECTID	GID	LINEID	NAME	SO2kgkm	NOxkgkm	COkgkm	CO2gestkm
	88	11701177238 ...	11701177238 ...	B178 - Loferer Str...	6.31143658601	4165.13302748	2730.41761766	994.998058
	89	11701177240 ...	11701177240 ...	B178 - Loferer Str...	9.02605784748	7069.98135905	4578.13445621	1422.95811
	90	11701177242 ...	11701177242 ...	L264 - Hombach...	0.244685559215	126.280240972	159.789153907	38.5746811

If you define deposition settings (see chapter 11.11.5), they are assigned to all imported sources.

A Douglas-Peucker filter can be applied (example 0.8 m) to reduce the number of edge points.

If the absolute height checkbox is checked, all imported heights are interpreted as absolute heights above sea. Absolute heights force an error at the GRAL computation core without topography (see chapter 17.4).

The following shape types are supported by the GUI

Value	Shape Type
1	Point

3	PolyLine
5	Polygon
11	PointZ
13	PolyLineZ
15	PolygonZ

When using the "3D line" option, the height of each edge point is taken from the shape type PolyLineZ for walls and line sources or the shape type PointZ for point sources and receptors. If the "Height" listbox within the shape import dialog is set to a column of the *.dbf file, the height from the *.dbf file is used. If you like to use the shape types 11 or 13, the listbox "Height" must be set to "none".

The *.dbf Reader was developed by Ahmed Lacevic and is licensed with the BSD 2 Clause "Simplified" license.

11.14.2 Import of wind roses

Menu "File – Import – Wind roses"

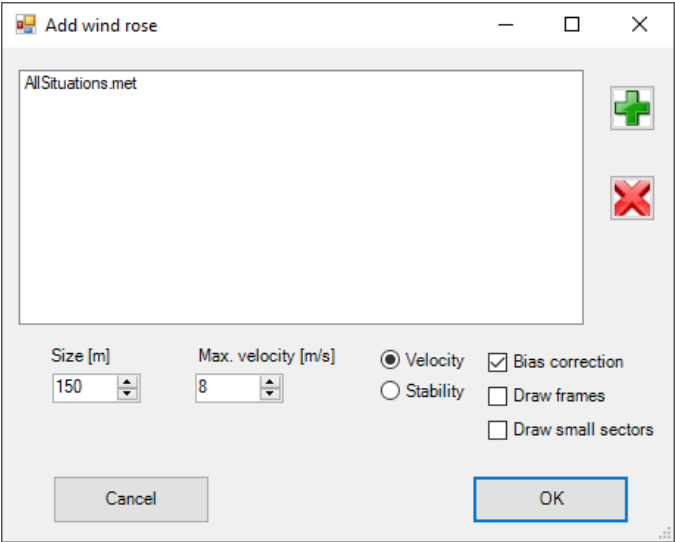
Toolbox: 



It is possible to visualize wind roses within the GIS window. The imported *.met files need an information about the measurement coordinates. This is an extension to the *.met file (compatible to the GUI 19.01 and older).

It is needed to add the coordinates and possible to add further information using the characters "//" without any separator characters in the meteo file "*.met" header, e.g.

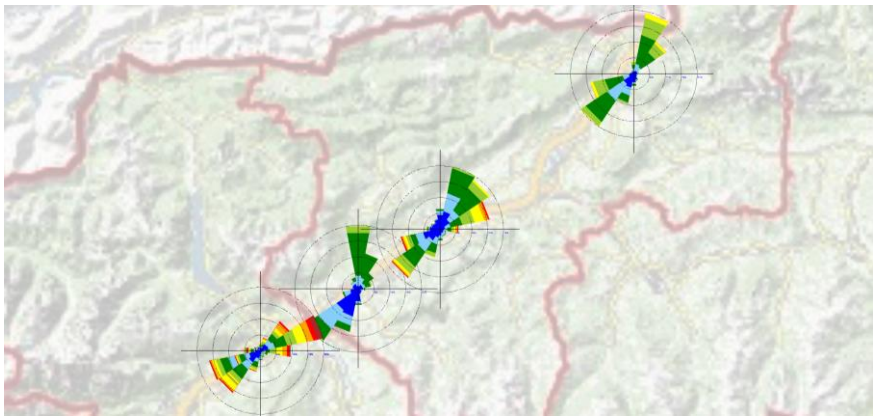
```
//X=80079
//Y=233949
//Z=10
01.01.2013,00:00, 2.1,218,6
01.01.2013,01:00, 3.2,204,4
.....
```

New wind statistics (from GRAMM wind fields) are created with this header automatically.



	Add a meteorological file (*.met)
	Remove the selected meteorological file.
Size [m]	Diameter of the wind rose in m (the wind roses are zoomed)
Max. velocity	Define the max. velocity class
Velocity/Stability	Show the wind speed or the stability class
Bias correction	Apply a bias correction for binned meteorological files
Draw frames	Draw a small frame around each wind sector
Draw small sectors	Draw small sectors with little gaps between the wind sectors

Example:



11.15. Export sources, rec. points, buildings or contour lines

Menu “Export – “

Toolbox:



All emission sources (except for tunnel portals), receptor points, or buildings can be exported as shape file, and in this way can be used in other software tools or subsequent GRAL projects.

Simply click the corresponding export button for the feature to be exported and define a file name and directory.

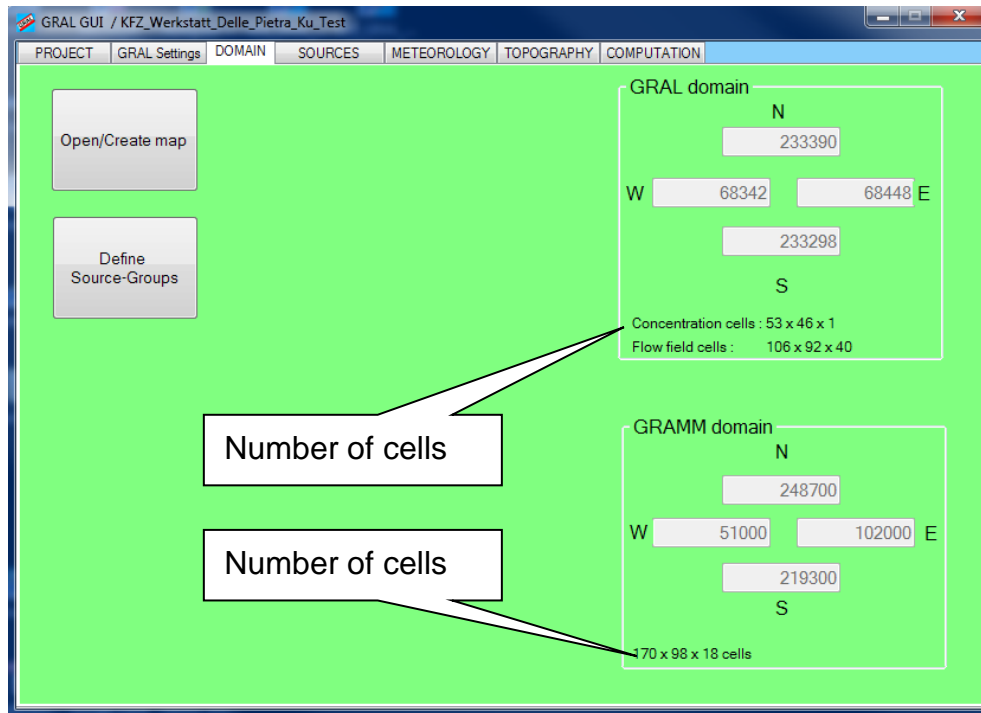
Not all attributes will be exported, but only those that are required for further evaluations or dispersion calculations.

Export of contour lines:

The topmost contour map within the object manager (see chapter 11.3) is exported. The export of contour lines is limited to contour lines with drawing option “Spline Line” (see chapter 11.4).

12. Number of concentration and flow field cells

If the GRAL domain area or the GRAMM domain area is defined, the domain borders and the numbers of concentration and flow field cells (if buildings are considered) are displayed at the “Domain” tab.



Please note that high cell counts increase the calculation time. If the calculation time is very high, it is recommended to reduce number of cells by increasing the horizontal grid resolution or setting smaller domain areas. Note that a larger horizontal flow field resolution may result in a poor building and surface resolution.

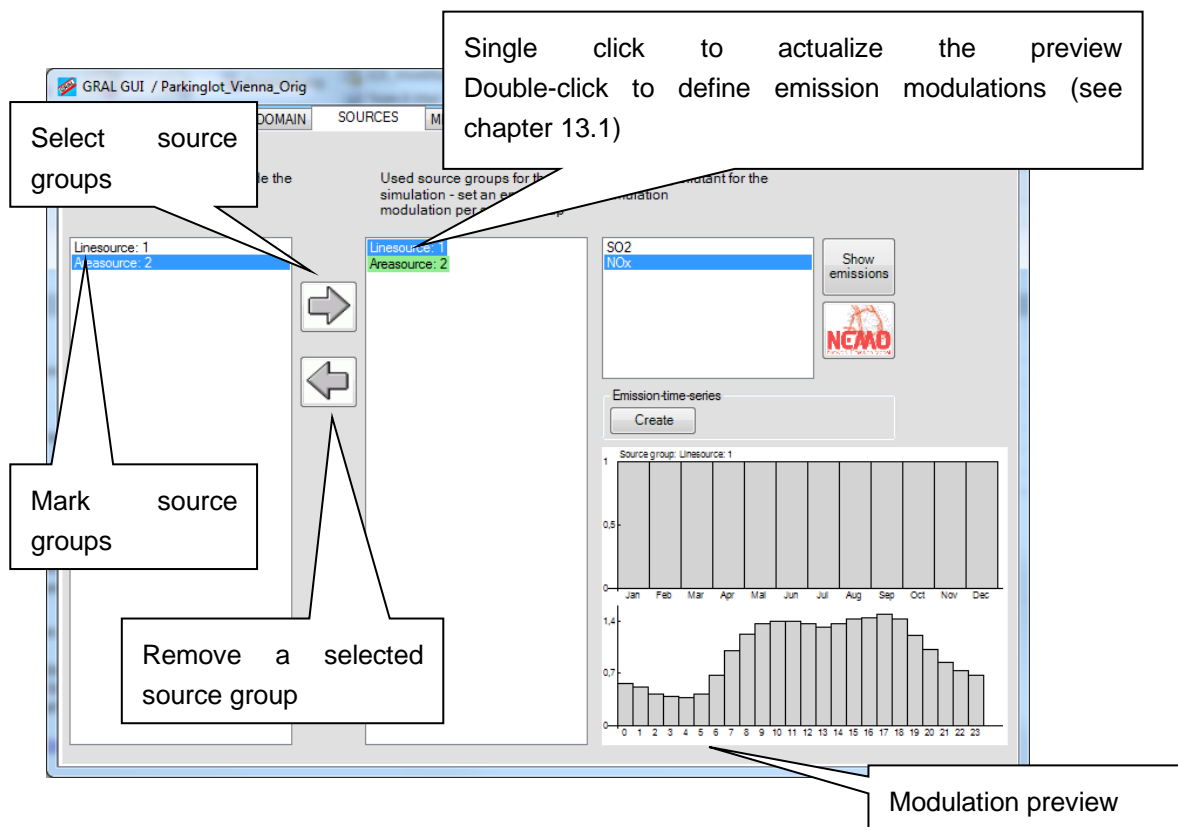
To check the GRAMM geometry, see chapter 16.6.

To check the GRAL geometry, see chapter 17.7.

To check the buildings geometry, see chapter 15.4.

13. Select source groups and a pollutant for dispersion calculations

After the editing process of all sources is completed and the GRAL model domain has been defined, close the GIS Interface. Select the “Sources” tab in the main menu, in which all defined source groups can be selected for subsequent dispersion calculations.



If you cannot see available source-groups: have you already defined a GRAL-model domain area and sources within the model domain?

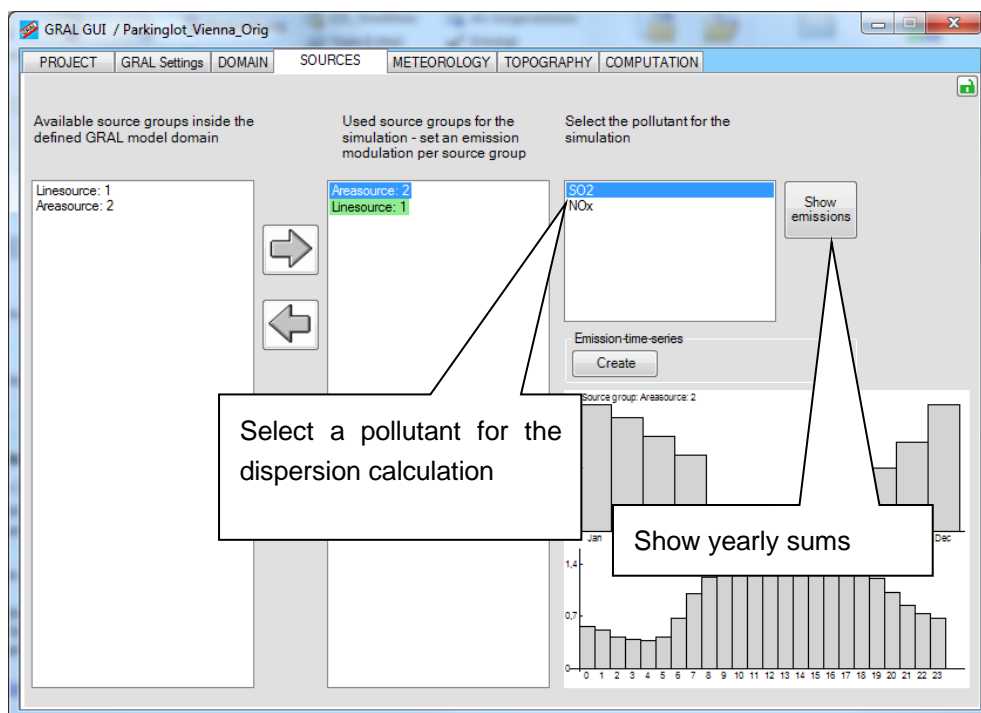
Select the desired source group entries in the list box “Available source groups”. Multiple selections are possible using the shift or ctrl button. Use the arrow button to move the selected items to the list box “Used source groups”.

A simple click on an entry in the list of used source groups updates the modulation preview.

Double-clicking on a selected source group item opens the edit form for the emission modulations.

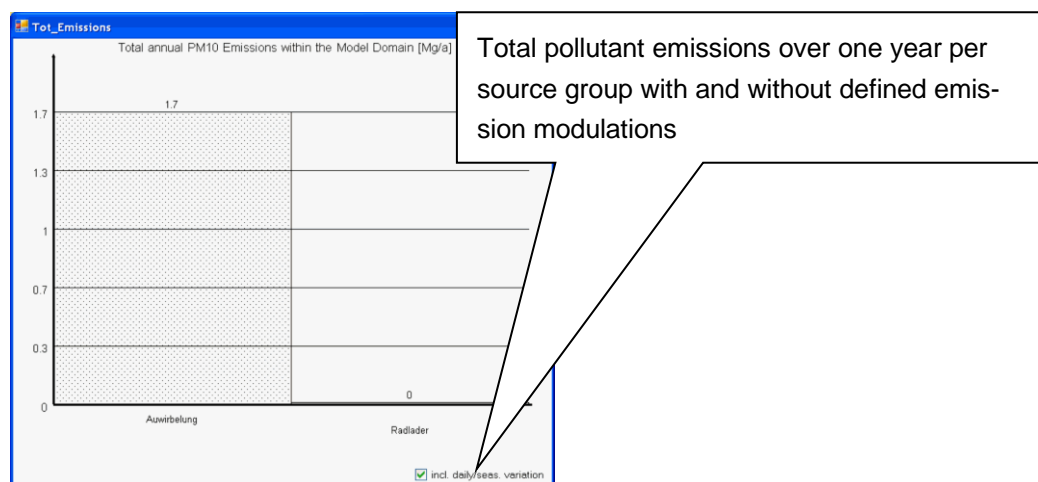
It is possible to set monthly and hourly emission variations. There are some pre-defined modulation functions, although custom modulations can also be added.

Select source groups and a pollutant for dispersion calculations



By pressing the „Show emissions“ button, annual totals for the selected pollutant are calculated and displayed in a diagram.

In the lower left corner of the diagram a click box allows for calculating annual sums with or without the defined emission modulations (assuming a constant emission modulation of 1 for each hour of the year). In this way, the effect of the defined emission modulations on the average emission rate for one year can be checked quickly.



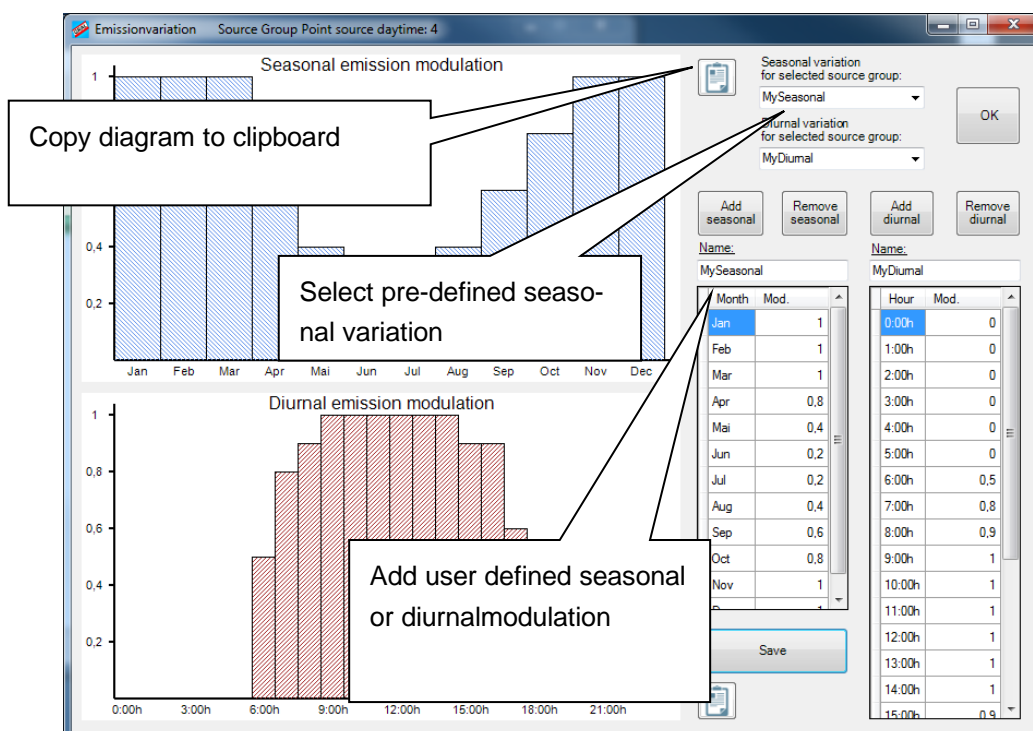
13.1. Emission modulation

13.1.1 Diurnal and seasonal emission modulation

This dialog is used to define emission modulations on an hourly and monthly basis for the selected source group. The selected source group is displayed in the title bar of the dialog.

Select source groups and a pollutant for dispersion calculations

You can use predefined emission variations or define own modulations.



To add user defined seasonal and diurnal emissions, you must click to the corresponding button, define a name and edit the values in the data grid. Only numerous values are accepted, you must use the standard decimal separator.

If a new modulation name already exists, the character “_” is automatically appended to the new name.

The values are temporarily stored and visualized if you press the “Save” button. The values are updated in the model when you confirm all settings with the “OK” button.

You paste values from a spreadsheet application by clicking in the data grid view (selecting the grid, do not switching to the edit mode) and pressing the “Control - V” keys on the keyboard.

It is possible to change the modulation for each source group after the computation is complete, since the evaluation of the results uses the current modulation data (see the flow charts in chapter 5). This is an easy and fast way to evaluate different emission modulations after a computation has been finished.

Presets of the emission modulation are stored in the application folder in the files "Emission_Mod_Diurnal.txt" and "Emission_Mod_Seasonal.txt". The data inside these files is tab separated and can easily be edited by any text editor or spreadsheet application. It is possible to edit or add own presets in these files. For more information about the file formats look to chapter 19.3.

13.1.2 Emission modulation on an hourly basis for a time series or the transient GRAL mode

An emission modulation on an hourly basis needed, if you compute a transient GRAL project, otherwise it is optional, but overrules the diurnal and seasonal emission modulation settings.

When utilizing one of the routines „**Multiple Sources**“, „**Mean, Max, Daily Max**“, „**Percentiles**“ or „**Receptor Concentrations**“ emissions modulations for every source group can be defined for each hour of a year. When doing so, the pre-defined emissions modulations for monthly and daily variations are **not considered**.

Hourly emissions modulation factors for each source group must be stored in a separate file called „emissions_timeseries.txt“, which needs to be stored in the sub-directory „Computation“ of the current project.

This file can be created by clicking the button „Emission time series - Create“ in the emission tab. This function uses the preset values of the diurnal and seasonal emission modulation and writes the file „emissions_timeseries.txt“ to the computation folder.

The generated file can also be visualized, using the button „Emission time series – Show“.

There is an indication that the file „emissions_timeseries.txt“ is being used in the modulation preview on the „Sources“ tab.

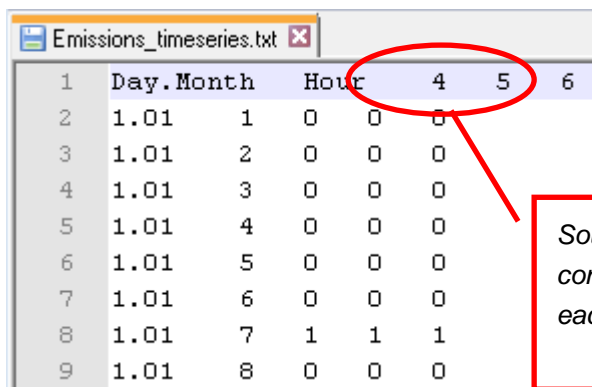
How do I manually create the emissions_timeseries.txt file?

It is imperative to use the date and time information as stored in the file „mettimeseries.dat“, which can be found in the sub-directory „Computation“ of the current project. It is recommended to copy the file in an application such as Excel. In a next step all columns, except the first two ones containing the date and time information, must be deleted.

It is important to define a correct header line for the file „emissions_timeseries.txt“:

The first column is the date, the second the hour, followed by the numbers of **each used** source group. It is not important in which order the source groups are aligned.

Select source groups and a pollutant for dispersion calculations



1	Day.Month	Hour	4	5	6
2	1.01	1	0	0	0
3	1.01	2	0	0	0
4	1.01	3	0	0	0
5	1.01	4	0	0	0
6	1.01	5	0	0	0
7	1.01	6	0	0	0
8	1.01	7	1	1	1
9	1.01	8	0	0	0

Source group numbers indicating the columns containing the emission modulation factors for each hour of the time-series.

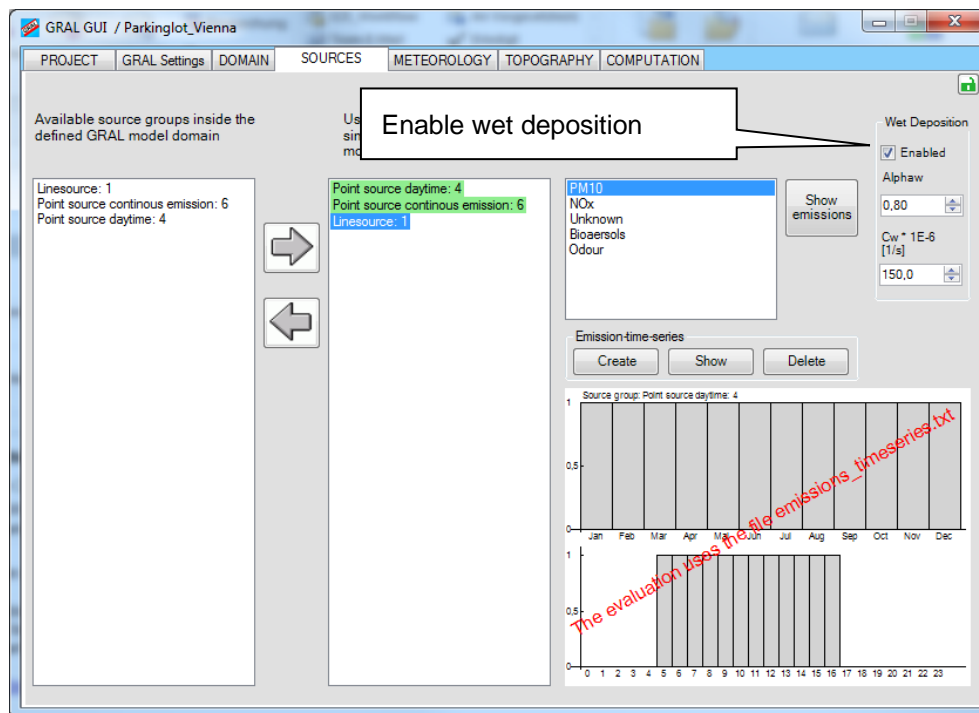
Tabulator (default), semi-colon, hyphen, blank or colon characters are accepted column separators

For more information about the file formats look to chapter 19.3.

13.2. Wet deposition settings

Using the GRAL transient mode, it is possible to consider the wet deposition.

In the 1st step select the pollutant and select the checkbox “Enabled” in the frame “Wet deposition”.



These settings are a property of the pollutant and apply to all source groups.

The washout rate is computed in analogy to VDI 3945/Part 3. If the wet deposition is enabled, a file `Precipitation.txt` will be written into the computation folder. This file contains precipitation data for the model domain for each entry of the time series. By default, all values are set to a precipitation of 0.0. You can replace these values by given precipitation data in mm/h.

The file is tab-separated. Example:

Day.Month	Hour	Precipitation[mm/h]
1.1 1	0.0	
1.1 2	2.0	
1.1 3	2.0	
1.1 4	1.0	
1.1 5	0.0	
1.1 6	0.0	

Select source groups and a pollutant for dispersion calculations

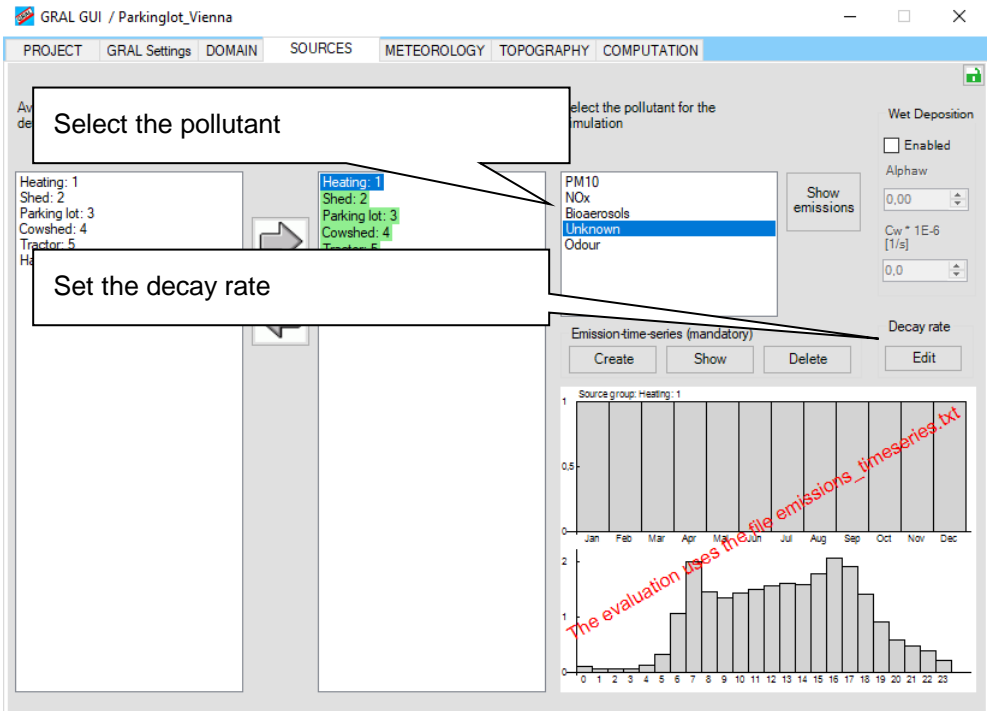
13.3. Decay rate

For the pollutant types "Bioaerosols" and "Unknown", a decay rate[s⁻¹] depending on the source group can be defined.

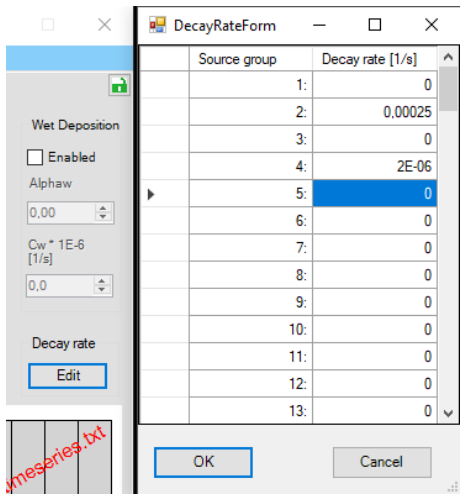
User-defined decay rates λ in s⁻¹ reduce the particle mass every time step by applying an exponential function:

$$m_{t+1} = m_t e^{-\lambda t}$$

Decay rates might be used to simulate inactivation rates of e.g. bacteria or radioactive decay.



The decay rate is a property of the pollutant and applies to the defined source groups. A custom decay rate per source group can be defined.



14. Meteorological data

14.1. Import data

Click the „Open Met-File“ button to import meteorological data in the „Meteorology“ tab of the main window. Two different input formats are provided:

*.met-files with following file format:

1st column: date (day.month.year)

2nd column: time (hour:minute) – half hourly or hourly values

3rd column: wind speed [m/s]

4th column: wind direction in deg.

5th column: stability class – 1=strong convective (A) to 7=strong stable (G)

Decimal separator: point (preferred) or comma

Column separator: comma (preferred), blank, semicolon or tab stop

For best support of all features, it is recommended to add a 3-line header with the coordinates of the measurement station (X, Y: coordinate, Z: relative height above ground).

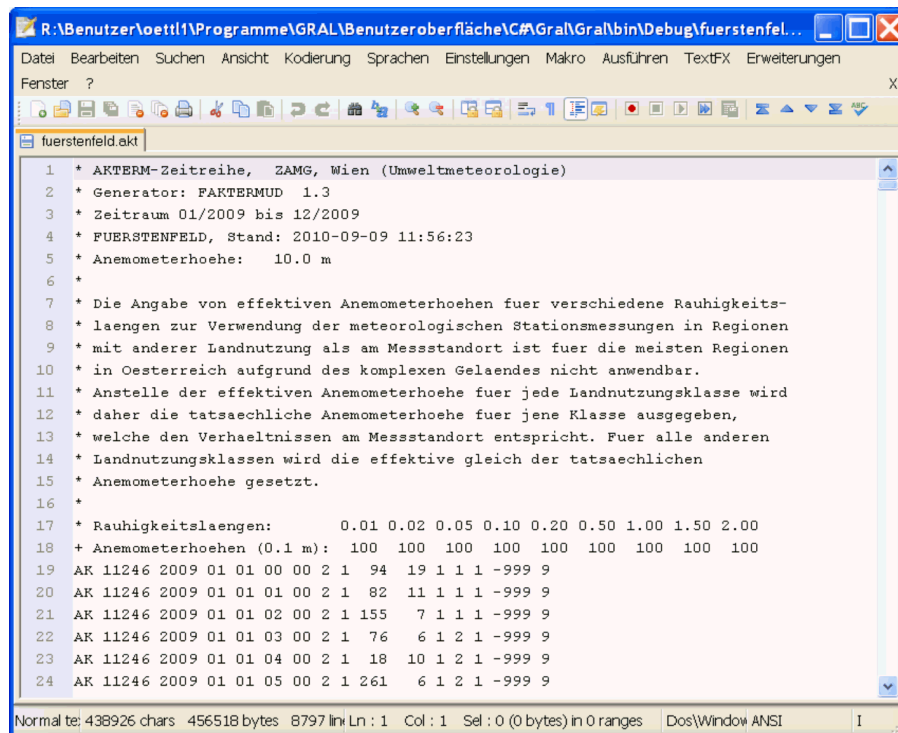
Example file

```
//X=116093
//Y=255564
//Z=10
01.08.2017,00:00,0.7,227,7
01.08.2017,01:00,1,210,7
01.08.2017,02:00,1.5,217,7
01.08.2017,03:00,1.1,215,7
```

For more information about file formats look to chapter 19.3.

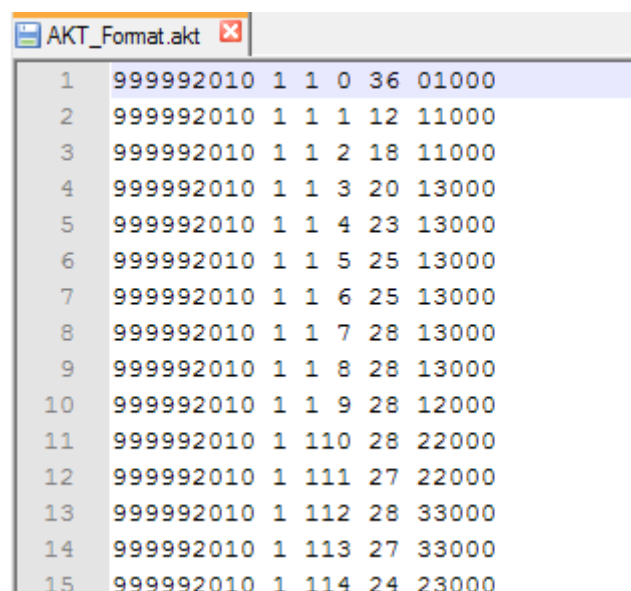
Meteorological data

AKTERM file format of the German TA-Luft:

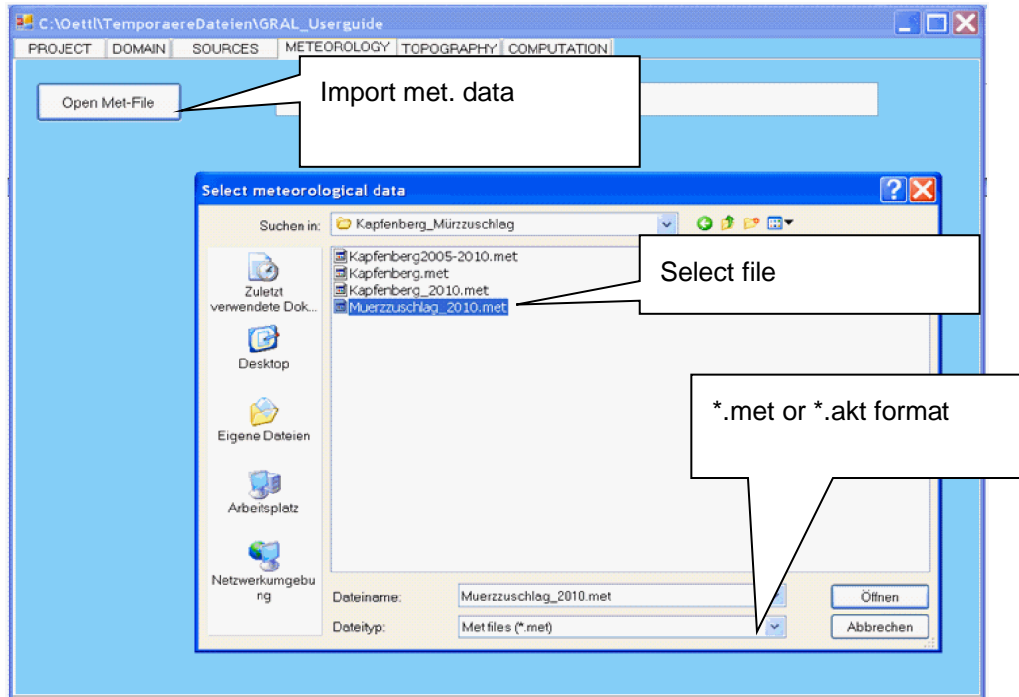


```
1 * AKTERM-Zeitreihe, ZAMG, Wien (Umweltmeteorologie)
2 * Generator: FAKTERMUD 1.3
3 * Zeitraum 01/2009 bis 12/2009
4 * FUERSTENFELD, Stand: 2010-09-09 11:56:23
5 * Anemometerhoehe: 10.0 m
6 *
7 * Die Angabe von effektiven Anemometerhoehen fuer verschiedene Rauigkeits-
8 * laengen zur Verwendung der meteorologischen Stationsmessungen in Regionen
9 * mit anderer Landnutzung als am Messstandort ist fuer die meisten Regionen
10 * in Oesterreich aufgrund des komplexen Gelaendes nicht anwendbar.
11 * Anstelle der effektiven Anemometerhoehe fuer jede Landnutzungs-klasse wird
12 * daher die tatsaechliche Anemometerhoehe fuer jene Klasse ausgegeben,
13 * welche den Verhaeltnissen am Messstandort entspricht. Fuer alle anderen
14 * Landnutzungs-klassen wird die effektive gleich der tatsaechlichen
15 * Anemometerhoehe gesetzt.
16 *
17 * Rauigkeitslaengen: 0.01 0.02 0.05 0.10 0.20 0.50 1.00 1.50 2.00
18 + Anemometerhoehen (0.1 m): 100 100 100 100 100 100 100 100 100
19 AK 11246 2009 01 01 00 00 2 1 94 19 1 1 1 -999 9
20 AK 11246 2009 01 01 01 00 2 1 82 11 1 1 1 -999 9
21 AK 11246 2009 01 01 02 00 2 1 155 7 1 1 1 -999 9
22 AK 11246 2009 01 01 03 00 2 1 76 6 1 2 1 -999 9
23 AK 11246 2009 01 01 04 00 2 1 18 10 1 2 1 -999 9
24 AK 11246 2009 01 01 05 00 2 1 261 6 1 2 1 -999 9
```

AKT file format

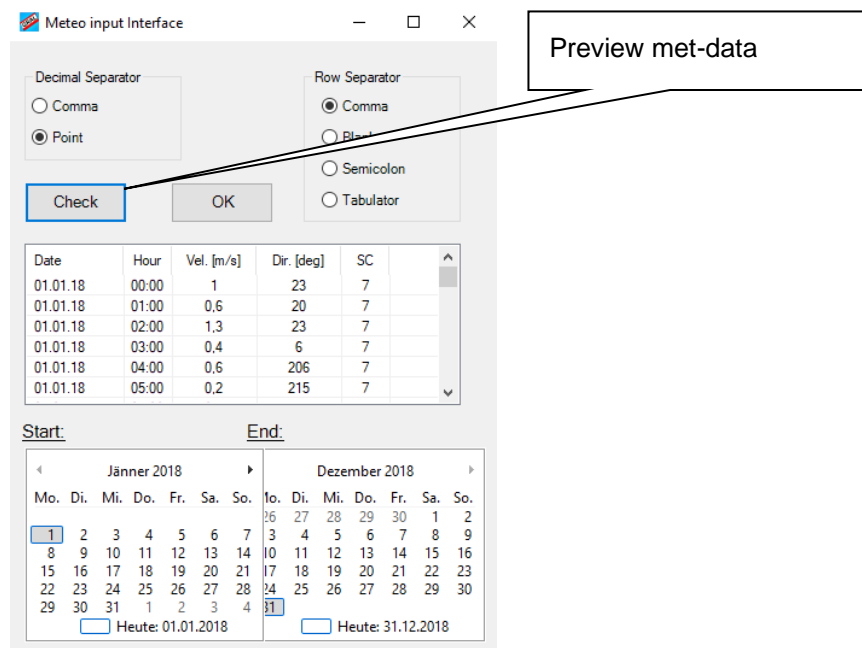


1	999992010	1	1	0	36	01000
2	999992010	1	1	1	12	11000
3	999992010	1	1	2	18	11000
4	999992010	1	1	3	20	13000
5	999992010	1	1	4	23	13000
6	999992010	1	1	5	25	13000
7	999992010	1	1	6	25	13000
8	999992010	1	1	7	28	13000
9	999992010	1	1	8	28	13000
10	999992010	1	1	9	28	12000
11	999992010	1	110	28	22000	
12	999992010	1	111	27	22000	
13	999992010	1	112	28	33000	
14	999992010	1	113	27	33000	
15	999992010	1	114	24	23000	



Make sure meteorological data has been imported correctly by selecting the correct separators for the decimal places and columns. Press the „Check“ button to visualize how the meteorological data is aligned.

Press the „OK“ button if the data preview is displayed correctly and the calendar functions are displayed at the bottom.



Use the calendar to select a time span for the dispersion simulations.

Meteorological data

If you want to choose a different meteorological time series, just repeat the procedure.

Data can be analyzed by using the buttons on the left side of the window.

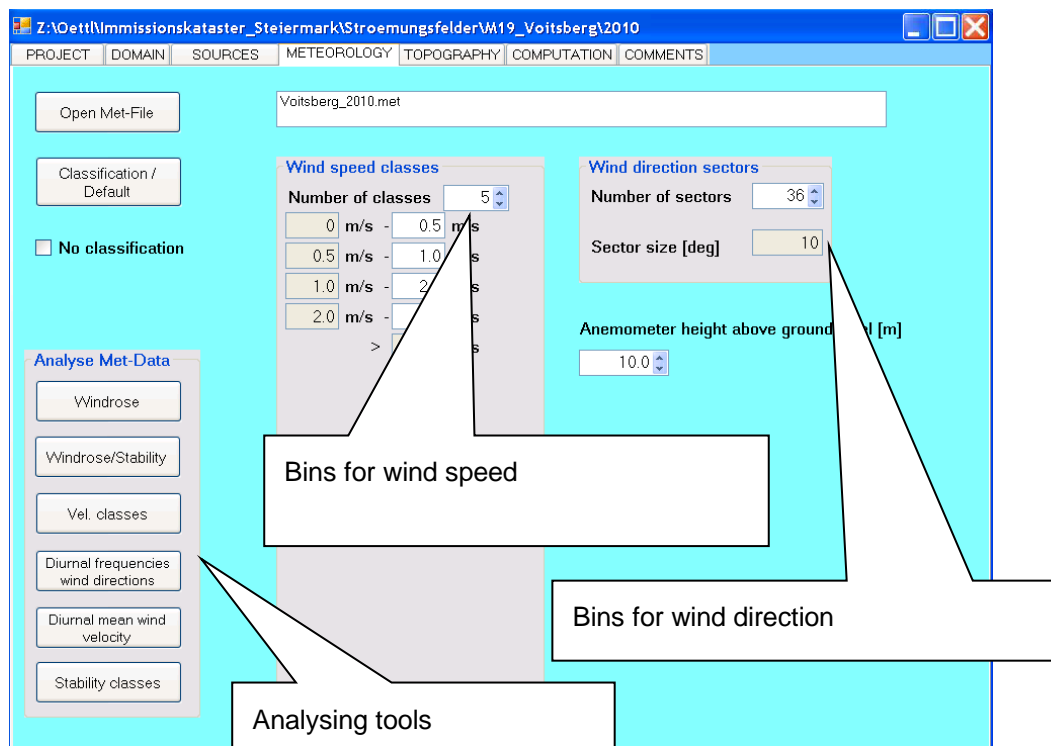
14.2. Classify meteorological data

In most applications, it makes sense to classify the meteorological data. The classification reduces the number of *.gff files to be calculated and in steady state GRAL mode the number of dispersion situations to be calculated.

Press the „Classification/Default“ button to define the number of wind speed and wind direction sectors. The standard wind speed bins are designed for regions with frequent low wind speed situations. These default values can be reloaded at any time by pressing the „Classification/Default“ button again.

If you check the “No classification” box, no classification will be made. Instead, every hour (half-hour) of the loaded time series is calculated.

Don't forget to set or check the anemometer height after importing a *.met file!



The meteorological data import function can be used to analyze such data at any time, even after a user has already selected a specific meteorological file for use in the dispersion simulations.

A new meteorological input for the dispersion calculation will only be created, if you press the „Classification/Default“ button.

It is possible to analyze other meteorological files, but the prior selected meteorological file for the dispersion calculations is still used.

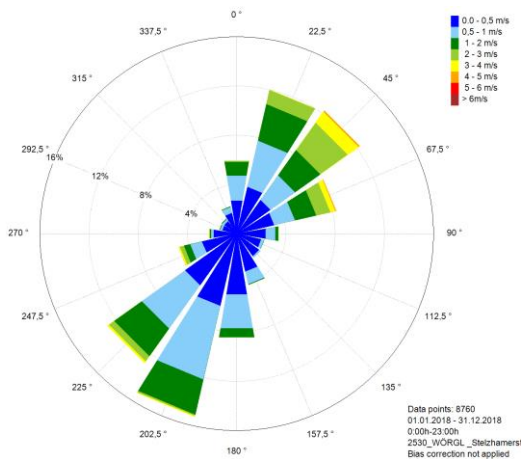
If you are not sure which meteorological file is used for the simulations, just close the project and open it again or look at the bottom line of the “Computation” tab – “Meteorological input”.

14.3. Analyze meteorological data

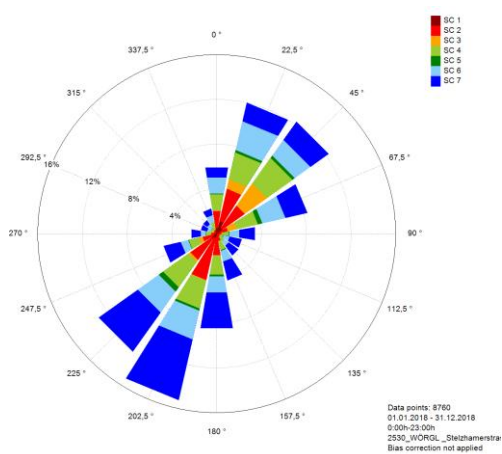
It's possible to generate the following graphs:

- Wind rose – if needed for parts of the day
- Wind rose for stability classes
- Wind velocity class diagram
- Wind direction – diurnal frequency diagram
- Wind speed – diurnal wind speed diagram
- Stability classes – frequency diagram

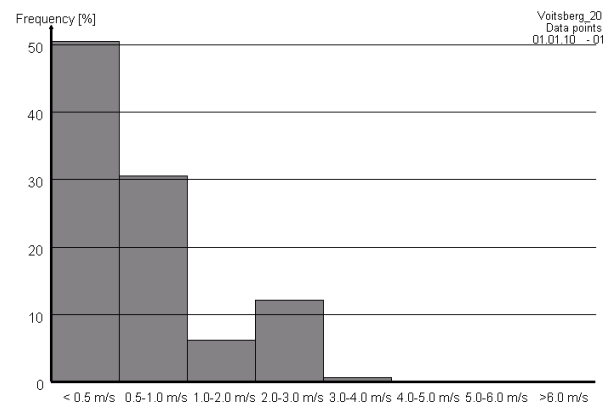
You can perform the following evaluations:



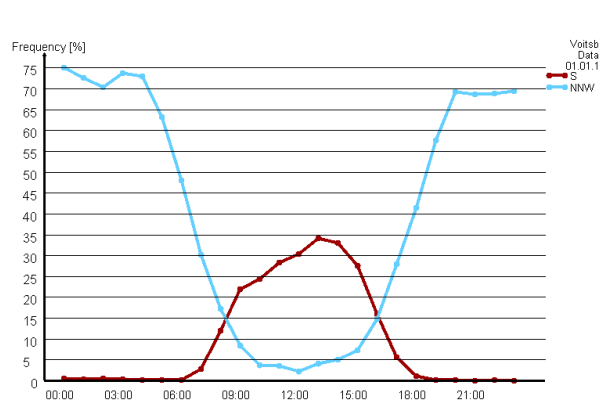
Windrose



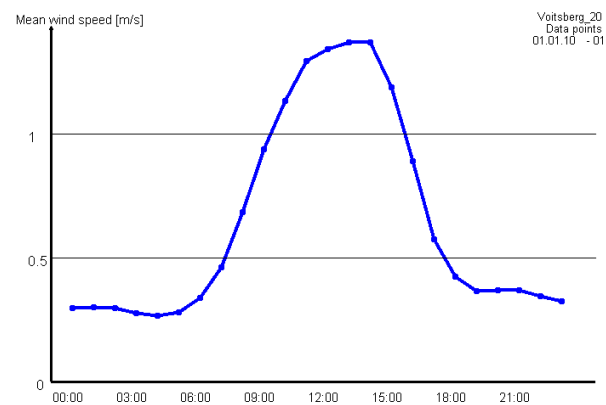
Windrose/Stability



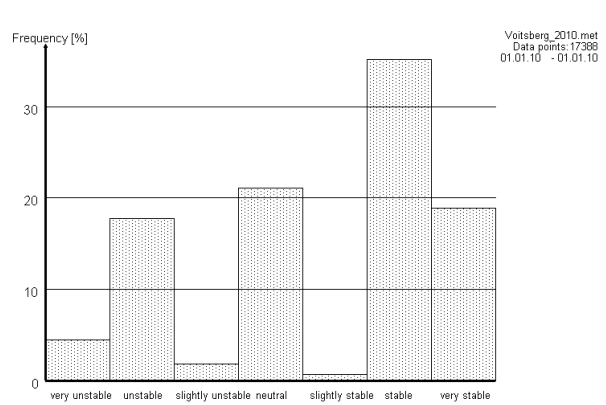
Velocity classes



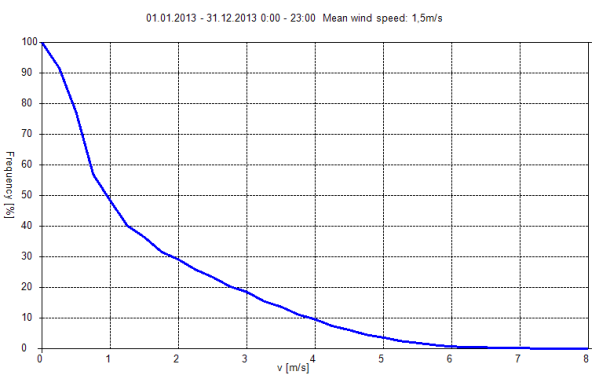
Diurnal frequency wind directions



Diurnal mean wind velocity

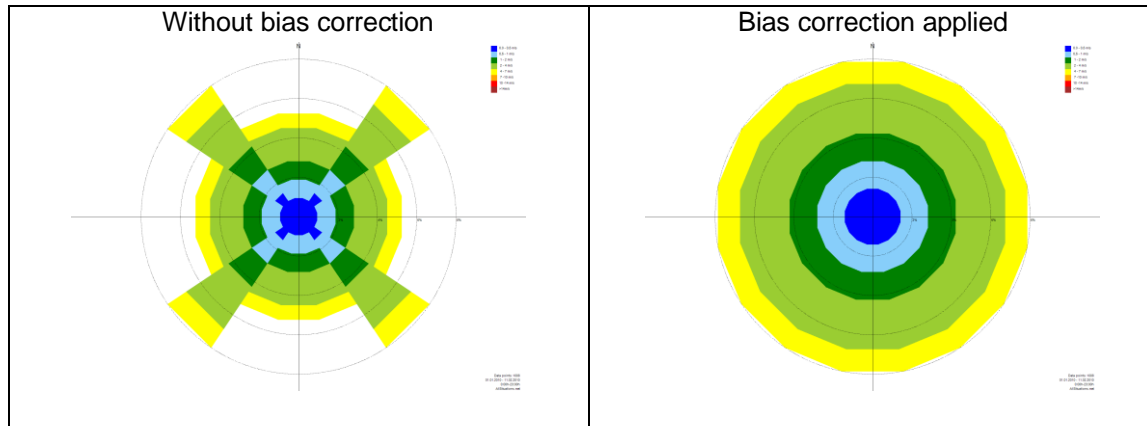


Stability classes



Cumulative velocity distribution

The bias correction is useful for binned meteorological files, e.g. for an artificial equally distributed wind rose.



You can set the position of the legend and the info box. Move the mouse pointer to the legend or the info box, press the left mouse button and move the item to the desired position. The position is saved and reused when a new wind rose window is opened, but it is reset to the default value when the wind rose window is resized.

15. Starting GRAL and analyzing results

15.1. Starting GRAL

Before a GRAL simulation can be started, all input data must be confirmed and saved.

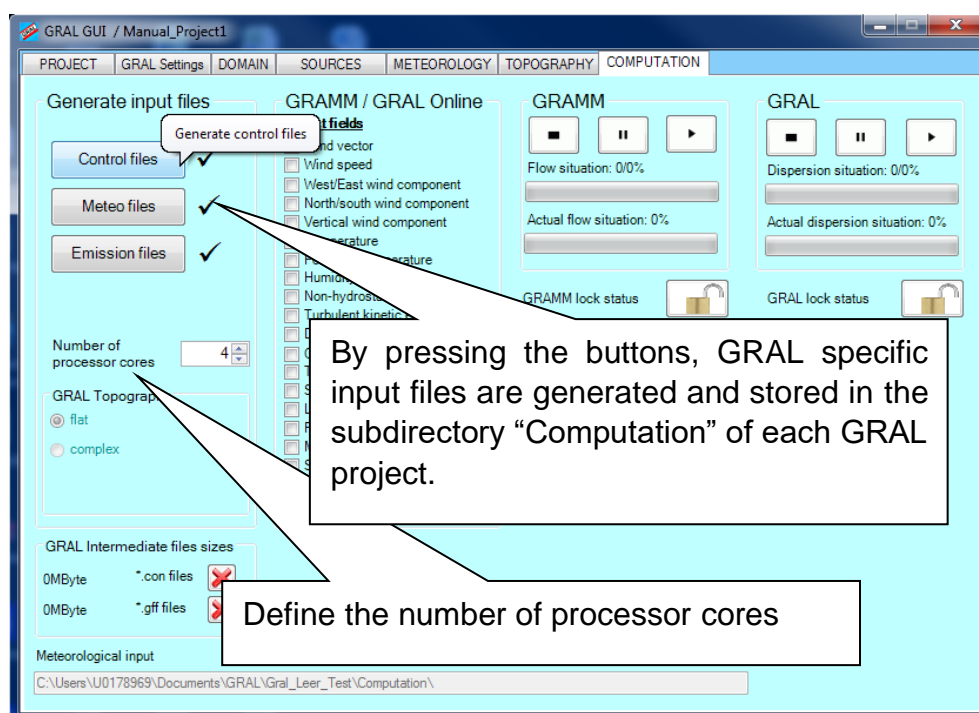
Select the “Computation” tab and press all buttons to the right with a red circle.

It is generally recommended to confirm the input data immediately after completing the appropriate fields. For example, after completing all the main control parameters in the “GRAL Settings” tab, it makes sense to go to the “Computation” tab and press the Control Files button to save all settings.

The “Meteo files” button generates GRAL specific input files from the selected meteorological input file.

Source files and building data in GRAL specific formats are generated by pressing the corresponding buttons. However, this data defined with the GIS Interface are stored anyway and aren’t lost, if the buttons are not pressed before the project is closed.

You can also define the number of processor cores to be used in the simulations. By default, the maximum available processor cores of the computer in use are suggested.



After each button has been pressed, a new group box appears at the top right of the window. The simulation can be started by pressing the right button (“Play”). In the next step the user is asked for a GRAL executable.

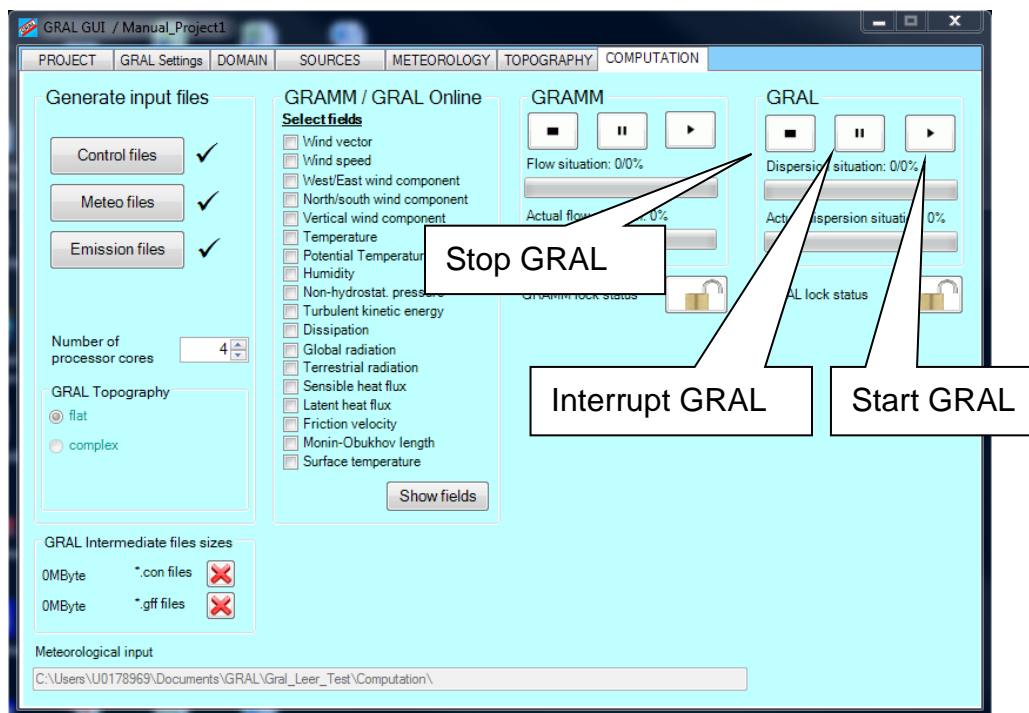
When you start a simulation for the first time, the GUI suggests an executable from the parent directory, where the GUI and all related files are stored. Subsequently, whenever GRAL is started again, the

Starting GRAL and analyzing results

GUI suggests the executable, which has been copied from the parent directory of the GUI to the current GRAL project in the “Computation” subdirectory. In this way, it is possible to re-start the simulation with the same executable even after years.

If you want to run a simulation again, but with a newer version of GRAL, either manually copy the newer version to the “Computation” subdirectory of the current GRAL project or delete the old executable file before starting GRAL with the GUI.

A user can press the “Pause” button at any time, which immediately interrupts the simulation and saves the current number of dispersion situation in the corresponding input field in the “Project” tab. If you press the “Play” button again, the simulation will start with the previously calculated dispersion situation.



Use the GRAMM/GRAL-Online checkboxes to select a parameter and the “Show fields” button to get a live view of the computed parameters. This option is sensefull if prognostic GRAL wind fields are computed. See also chapter 16.8.

15.2. The “Project locked” function

When a simulation is started, the actual project is locked by default to prevent the project from becoming invalid by changing the project parameters. After the project is locked, it's not possible to change emission and sources data, dispersion or flow field parameters.

Parameters for the post-processing, like daily and diurnal modulations, can be changed in a locked project (see chapter 13.1).

If you need to restart a project simulation at a higher dispersion situation, you can do this in a locked project. Simply set the situation on the “GRAL settings” tab – “Start with dispersion situation” - and switch to the “Computation” tab.

Now you must unlock the project manually by pressing the lock symbol and confirm the warning message.



In the next step you can click on the “Control files” button and to restart the calculation at the selected situation.

15.3. GRAL simulations in complex terrain

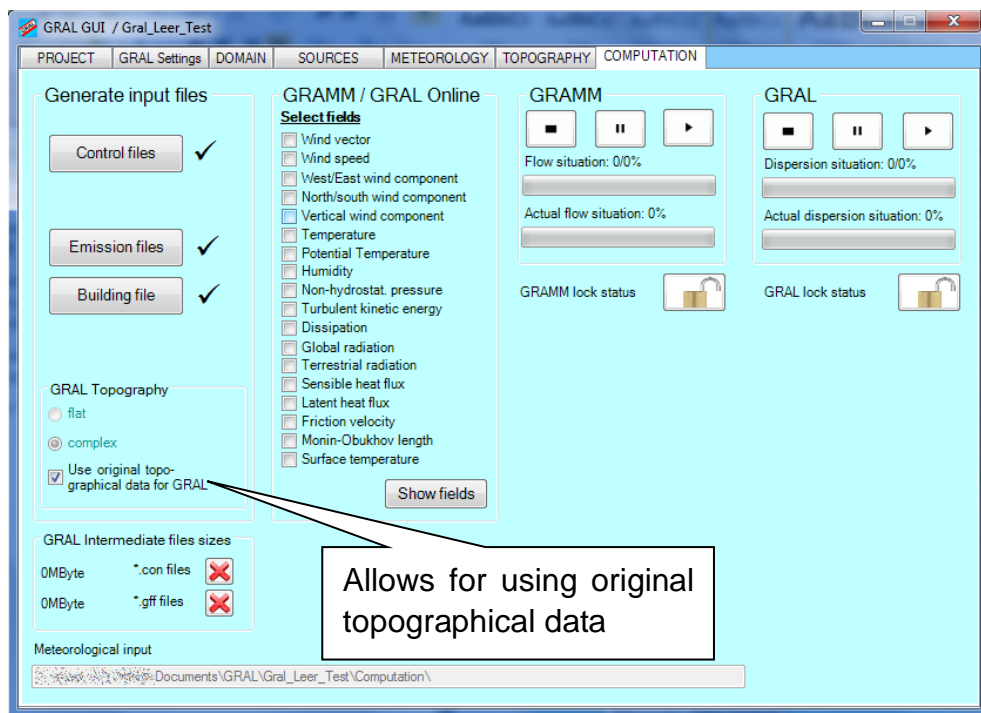
If the influence of complex terrain on the dispersion of pollutants is to be considered, GRAMM simulations must be carried out in a first step. Visit chapter 16 to learn about starting GRAMM.

Typically, GRAMM simulations are performed with a rather coarse horizontal resolution, e.g. a few hundred meters. In some applications it might be desirable to dissolve the terrain as fine as possible.

To do so, activate the check box in the 'Computation' tab as shown in the next figure or use the menu item "File – Import – Original GRAL topography" in the GIS window (see chapter 17.6). When you use the menu entry, the GRAL cell height is displayed in the GIS-window and you can edit and visualize the GRAL topography (chapter 17.7).


If you select the check box, you must select a (high resolution) terrain file at the start of the GRAL computation. The GUI automatically generates a subset of this terrain file with the exact dimensions and resolution as defined for the GRAL simulations. This new terrain file will then be used by GRAL as lower boundary.

If you restart GRAL, e.g. to calculate the dispersion for another pollutant, the already existing topography file will be used. Any change to the GRAL domain or resolution will delete the high-resolution topography file and it must be generated again to match with new dimensions and/or resolution.



15.4. Check the rasterized GRAL buildings and walls

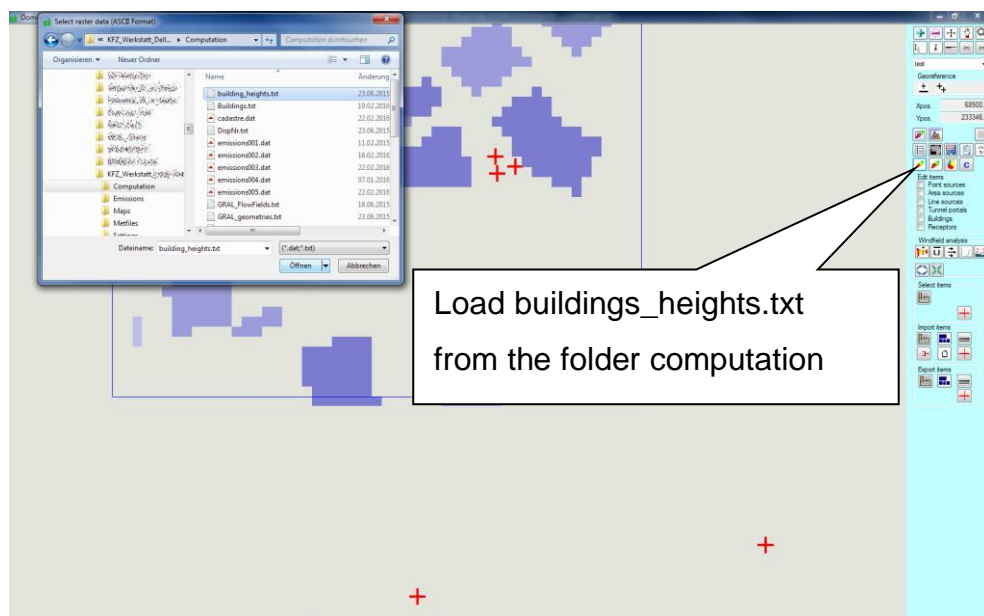
Menu: “Analyze results – Load contour map”

Toolbar: 

If your simulation considers buildings and/or walls, it is recommended to check the “Write file building_heights.txt” checkbox (see chapter 8).

If this checkbox has been checked, it is possible, to visualize the original rasterized GRAL buildings and walls at the GIS interface.

To do this, the GRAL computation must be started. After the 1st meteorological situation has been finished, the file “buildings_heights.txt” will be written. At this moment break the computation and open the GIS interface (chapter 11).



Just load the file “buildings_heights.txt” from the computation folder, like a contour map. The buildings are drawn rasterized as they are used in the model. The color depends on the building height and can be defined at the “Layout manager” (see chapter 11.4).

15.5. Analyzing GRAL results

If result files from the GRAL computation core are available, new buttons at the bottom right corner of the “Computation” tab allow the analysis of the calculated GRAL simulations. It may be necessary to switch to another tab to force a redraw of the “Computation” tab to display these options.

It is possible to analyze the already calculated results during the computation. It is strongly recommended to analyze the results after the first dispersion situation has been simulated (you can interrupt the computation by using the “Pause” button or analyzing the first results while the calculation is in progress). In this way a quick check of the first computed results can be carried out.

There are several methods to analyze the results after the computation has been finished.

15.5.1 Evaluation for any pollutants except odour

„Mean“

Calculate the average concentrations for the imported meteorological time series. Concentrations are calculated for each source group and additionally for the sum for all source groups. Note that this function is much faster than the one below. This function uses the diurnal and seasonal modulation for each source group, the “emissions_timeseries.txt” is not used by this function.

„Mean, Max, Daily Max“

Compute in addition the maximum daily mean concentration. It takes much longer than the function above and should therefore only be used if a maximum daily mean concentration is needed. This function uses the diurnal and seasonal or the “emissions_timeseries.txt” file modulation for each source group.

„Receptor Concentrations “

In case of existing receptor points (these must be defined by the user prior to any simulation) this procedure can be used to get time series of concentrations for each receptor point and source group. The result is stored in the file “receptor_timeseries.txt” in the subdirectory “Computation” of the GRAL project. This file can be further processed with any suitable software, e.g. Excel. This function uses the diurnal and seasonal or the “emissions_timeseries.txt” file modulation for each source group.

Starting GRAL and analyzing results

In addition, wind speed and –direction as simulated with the microscale flow field model of GRAL are exported to files named ‘GRAL_*name*.met’ and stored in the subdirectory ‘Metfiles’, in case that buildings have been considered in the GRAL simulations. These files can be read and visualized as outlined in chapter 14.

In GRAL transient mode, the GRAL computation kernel writes a file called “Receptor_Timeseries_Transient.txt” with a time series of concentrations for each receptor point and source group. You do not need to start this evaluation in this case.

„Percentiles“

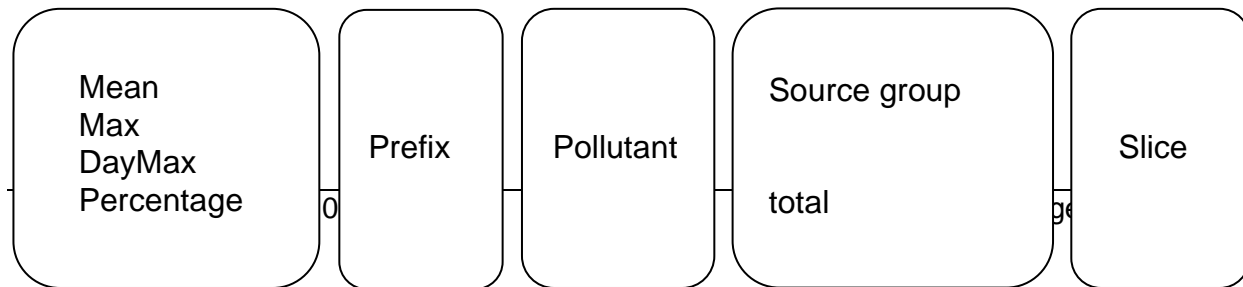
Compute the user defined percentile. Percentiles can be selected from the interval between 90 and 100. The function requires a lot of memory, so it might happen that an error message occurs when the available memory on the computer is exceeded.

This function uses the diurnal and seasonal or the “emissions_timeseries.txt” file modulation for each source group.

The results of the above described procedures are stored (except those for “Receptor Concentrations”) in the subdirectory “Maps” of the GRAL project.

The results are stored as modified Esri ASCII Files (see section 19.3.1 for more details), using automatically generated file names. You can specify an individual file – prefix at the “Select Source Groups” dialog. This prefix can be used to separate the results of several evaluations (e.g. different diurnal or seasonal emission modulations - chapter 13.1). Use the GIS Interface to visualize the results files as contour maps.

Parts of the automatically generated file name, separated by an underscore “_”



Examples

Mean_NOx_Linesource_8m.txt

Mean value, pollutant "NOx", source group "Linesource", slice "8 m"

98_Bioaersols_total_3m.txt

98 Percentil, pollutant "Bioaerosol", total = sum of all source groups, slice 3m

Deposition_Mean_NOx_total.txt

Deposition, mean value, pollutant "NOx", total = sum of all source groups

The evaluated "Deposition" files contain the calculated deposition value of all particle sizes when calculating PM10 or PM2,5. The file name part "Pollutant" "PM10" or "PM25" indicates the calculated pollutant for the dispersion.

15.5.2 Evaluation of odour simulations

„Multiple Sources“

Compute odour hours in [%] with regards to the meteorological time series used. Odour hours are computed for each defined source group as well as sum for all source groups.

Typically, the sum for all source groups gives not the same figure as when simply adding the figures at a receptor point for non-linear relationships resulting from the definition of an odour hour.

This function uses the diurnal and seasonal or the “emissions_timeseries.txt” file modulation for each source group.

„Compost“

This procedure has been developed to compute odour hours in [%] in the vicinity of compost works. Up to three different processes, each owing certain emission rate [MOU/h] and frequency for a year [%] can be defined by the user.

It is highly recommended to use 1 MOU/h as initial emission rate for compost works (to be defined via the GIS Interface) in order to facilitate the usage of this procedure. For further information, see also section 19.2.

„All in all out stables“

Compute odour hours [%] in the vicinity of animal farms applying the so-called all-in-all-out system, where all breeding animals are getting in the stable and finally put out of the stable together at once. The function considers linearly increasing odour emissions for a breeding cycle. The user will be asked for the number of days of an average breeding cycle.

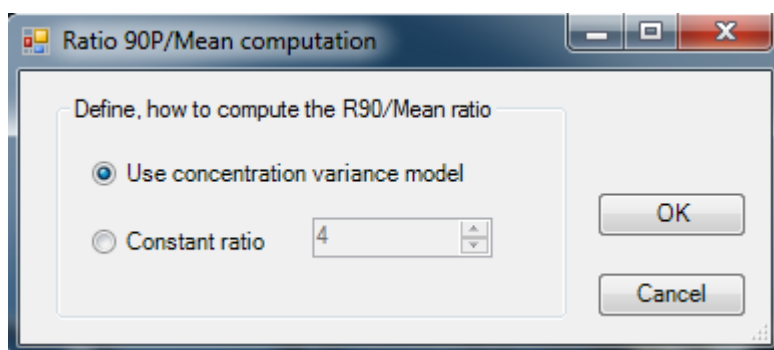
Odour emissions must be defined as maximum emissions at the end of a breeding cycle in the GIS Interface. Deadlocks in the livestock breeding during a year can be accounted for in the definition of the emission modulations. For further information, see also section 19.1.

“Receptor met files”

There is no evaluation of receptor concentrations in odour calculations, but the user might want to check the local GRAL flow field at the receptor points. This function exports wind meteorological files as simulated with the microscale flow field model of GRAL to the subdirectory ‘Metfiles’. These files can be read and visualized as outlined in chapter 14.

In each of the procedures above the user is asked for an odour threshold (e.g. 1 OU/m³, 3 OU/m³) and the ratio of the 90 percentiles of the odour cumulative frequency distribution and the average odour concentration for an hour. By default, a factor of 4 is suggested by the GUI.

Since GRAL version 17.1 and newer, as an alternative a more elaborate model that computes the factor in a more realistic way is provided (concentration-variance model). It is generally recommended to use this model for assessing odour hours.




This option is available, if the lowest height above ground for the concentration grid is higher than 1.5 * vertical dimension of concentration layers (see chapt. 8.2.).

In countries, where odour is regulated on the bases of percentiles of mean hourly odour concentrations in combination with a certain odour intensity (e.g. 1, 3 or 5 OU/m³), instead of a factor 4, a factor of 1 must be used.

15.6. Visualizing concentration and other kinds of maps

Menu: “Analyze results – Load contour map”

Toolbar: 

Concentration maps and other kinds of contour maps (e.g. topography maps) can be visualized in the GIS Interface. Click the button as shown in the next figure or use the menu entry “Analyze results – Load contour map” and select the desired file, which needs to be in ESRI-ASCII format (see section 19.3 for more details).

By default, contour lines are drawn.




A variety of options are offered to design such contour maps. Just open the layout manager (see chapter 11.3 and 11.4) for the desired contour map.

Note that it can take rather long to draw large contour maps, while it is much faster to draw pixel maps. To achieve this, the line width must be set to zero. The GUI sets the line width initially to zero if the number of cells exceed a value of 0.2 million.

15.7. Further analysis tools

15.7.1 Convert a *.grz or a *.con file to an ASCII file

Menu: "Analyze results –Convert *.grz file to ASCII file"

Toolbar: 

GRAL stores concentration files for each dispersion situation in the subdirectory "Computation" of the GRAL project as *.con (concentration), *.dep (deposition), *.odr (odour) or the compressed *.grz file format.

Such a file for a single dispersion situation can be converted in a readable ASCII format by pressing the ".con .txt" button or the "Export – Convert *.con files to ASCII raster format" menu. The converted files are stored in the subdirectory "Maps" of the GRAL project and can be visualized immediately like any other contour map.

15.7.2 Mathematical raster operations (e.g.NO2 conversion)

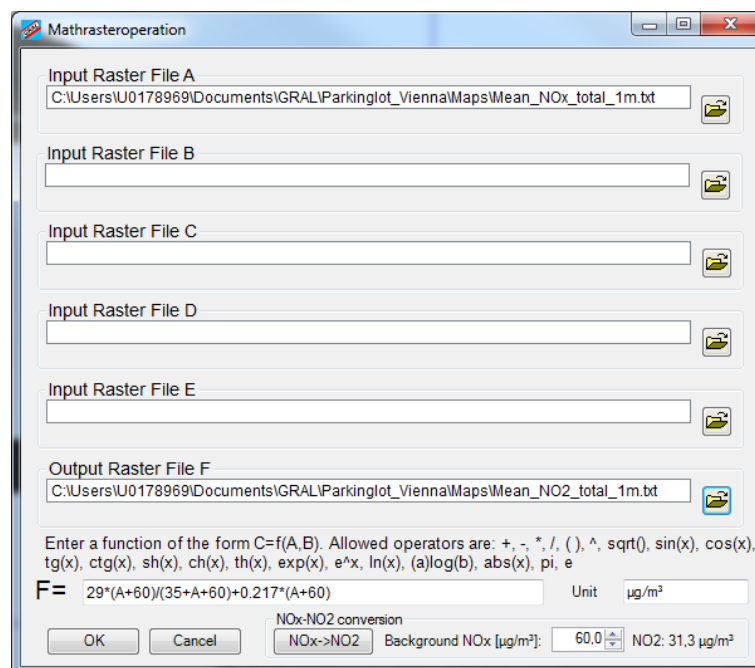
Menu: "Analyze results –Contour map math operations"

Toolbar: 

All maps stored in the ESRI-ASCII format (see section 19.3) can be further processed by applying user defined mathematical expressions. You can combine up to five maps. A simple empirical relationship to convert NO_x maps to NO₂ maps is provided.

Mathematical operations are provided by pressing the corresponding button in the GIS Interface (see figure above) or the menu entry "Analyze results – Contour map math operations". A maximum of five maps can be processed. The following example shows how to calculate NO₂ concentrations from an existing NO_x map using an empirical relationship (based on the Austrian guideline for dispersion modelling, BMWFJ 2010). Background NO_x can be considered. The empirical function is introduced when clicking the "NO_x->NO₂" button.

Starting GRAL and analyzing results



Following mathematical operators and functions are available:

Arithmetical operations () + - * / ^ sqrt cur

Trigonometric functions sin() cos() tan() arctan()

Hyperbolic functions sinh() cosh() tanh()

Other functions exp() e^x ln() log(b) abs() pow rcp

Constants pi e

Note: The mathematical parser “Mathos Project” has been developed by Artem Los. This part of the GRAL code underlies the BSD-3 license.

You must not use the scientific notation (e.g. 1E-3) as an input for the parser in your mathematical function.

The unit of the result is taken from the input raster file A, but can be changed manually by the user (e.g. conversion from µg/m³ to mg/m³ or mg/m².d to g/ha.Y).

15.7.3 Source apportionment at a location

Menu: “Analyze results –Source apportionment”

Toolbar: 

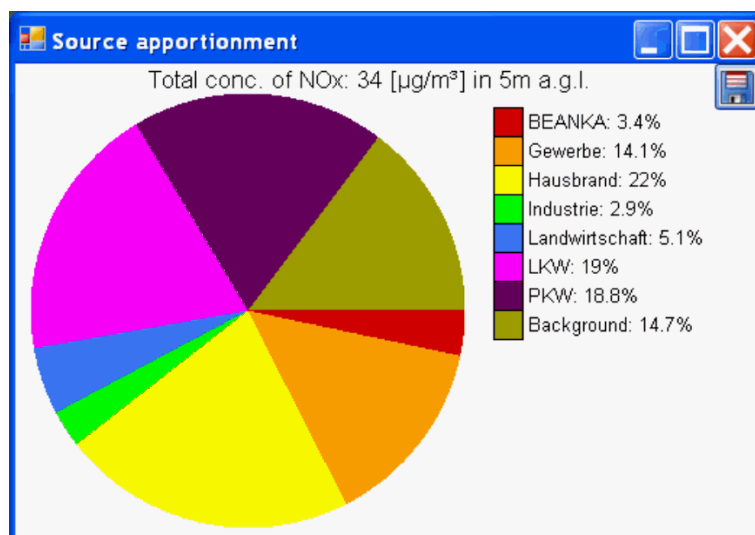
Calculation and visualizing of the source group assignment (pancake diagram) at any location. This option is useful if more than one source group was defined before simulations were started (chapter 15.7.3)

Click on the map to define the position of your analysis and select a "Mean*_total*.txt" file.

The function reads the concentration at the selected position from all available source group maps and displays the result.

The source group maps are searched by the selected file name, but the string part "total" is replaced by an asterisk "*" character. If you use this function, you should ensure that there are no "old" or invalid maps in the map folder.

Note that the procedure asks for the background concentration, which is added if desired.



15.7.4 Extract concentration values

Menu: "Analyze results – Extract concentration of a point"

Toolbar: 

Extracting values of any map at a certain position (chapter 15.7.4)

Using this function, you can extract computed concentration values for specific points to get a quick info of the simulated concentration at one raster point without low-pass filtering.


In a 1st step you can select the desired concentration file or dispersion situation. In a 2nd step, simply click on the GIS form to get the concentration values at the selected points.

Starting GRAL and analyzing results

When selecting this function, a new entry can be found in the object manager, called “Concentration values”. Use this entry to show and hide the values or to define the size of the symbol or the font.

15.7.5 Create animated GIF file

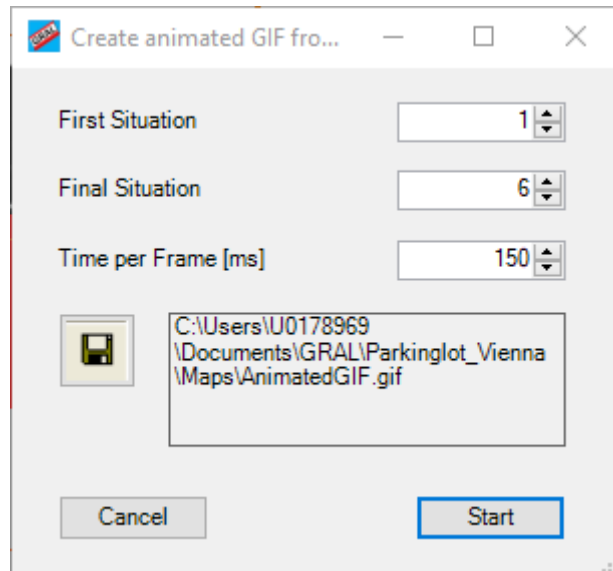
Menu: “Analyze results –Create animated GIF from *.grz files”

Toolbar: 


In the first step, you need to convert and load a single *.con file using the function from chapter 15.7.1.

In the second step set the visualization options using the layout manager (chapter 11.4).

After these preparations start the GIF recorder using the button above.



Set the 1st and final situations and the time per frame. Using the Save-Button you can define a folder and file name for the GIF file.

The press Start and the GUI loads and converts the *.grz files, displays the *.con file, makes a snapshot and stores the picture to the GIF file. The recording button turns to . You can stop the recording, using this button.

15.7.6 Show vertical profiles

Menu: “Analyze results – Vertical concentration profile at a point”

Toolbar: 

If the GRAL simulation has been performed in a transient mode and all dispersion situations within the “mettimeseries.dat” file are computed, it is possible to show vertical profiles at specific points.

Select the following icon in the toolbox

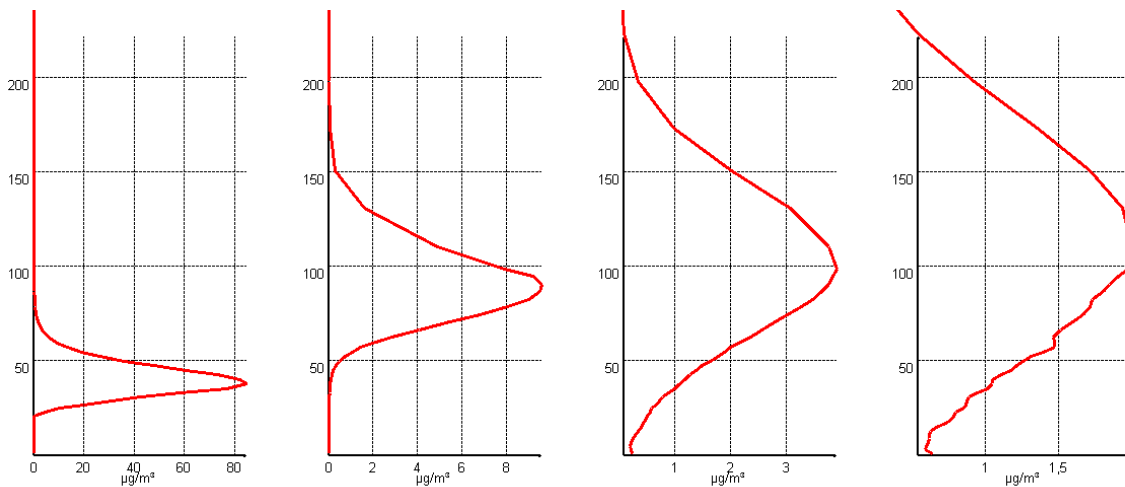


Vertical profile

Simply click on the GIS form to get the concentration profiles at the selected points. The z-axis can be zoomed, using the + and – button in the vertical concentration form.

The profile is refreshed at each new point if you select a new position.

Here are examples of vertical profiles for a single dispersion situation (stable, low wind speed) along the plume of a stack source



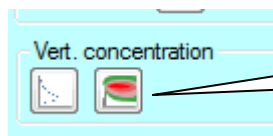
15.7.7 Show vertical profile along a section line

Menu: “Analyze results – Vertical concentration profile along a section”

Toolbar: 

If the GRAL simulation has been performed in a transient mode and all dispersion situations within the “mettimeseries.dat” file is computed, it is possible to show vertical profiles along a section line.

Select the following icon in the toolbox



Vertical profile - section

Start this function by creating a section line like in chapter 17.1.2

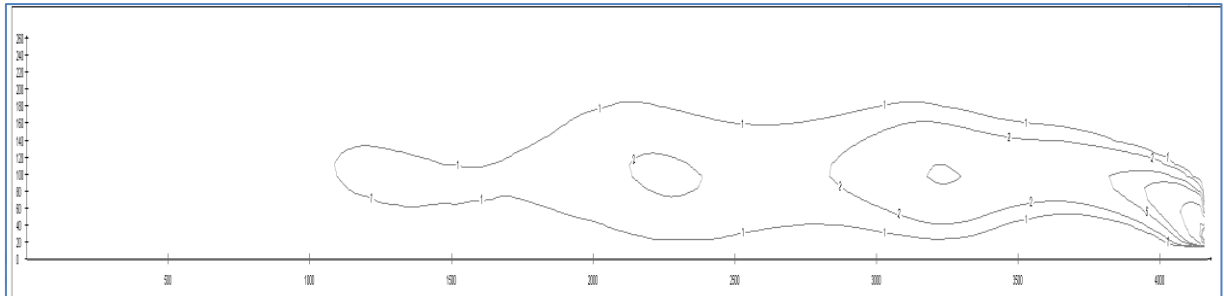
In the next step you can set a filename and the maximum height of the vertical profile.

Starting GRAL and analyzing results

This function creates an ESRI ASCII File along this section line with the given horizontal resolution of the concentration grid. The vertical resolution is set to the same value, the GRAL internal resolution will be much higher.

ESRI ASCII files can be visualized by the GUI and this file will be visualized as a concentration grid, except the additional x and y axis and the white background. All settings for concentration maps can be used (see chapter 15.6).

Here is an example of the vertical concentration along a section line for a single dispersion situation (stable, low wind speed) and the plume of a stack source (the picture is stretched in vertical direction).



Due to the interpolation for different raster widths, the concentration maps of the vertical section and horizontal section may differ. The vertical profile is much more accurate than the vertical section.

15.7.8 Generate a concentration time series for a raster cell

Menu: "Analyze Results - Generate time series for several evaluation points"

Using the menu item "Analyze Results - Generate time series for several evaluation points" you can perform a concentration time series evaluation for several grid cells.

After selecting the menu item, a dialog appears that allows you to define multiple points by manually entering coordinates or by clicking on the positions in the domain map. The entered data can also be edited (for example, the names).

The height value is not used in this case, the evaluation is done for all calculated heights above ground and all calculated source groups.

Using the button "Add receptor points" the defined receptors can be added to the evaluation table.

15.7.9 Generate a meteorological time series for a raster cell

Menu: "Windfield Analysis – Wind statistics at a point"

Toolbar: 

Using this function, you can create a wind field time series evaluation for several grid cells for GRAL or GRAMM wind fields.

GRAMM evaluations are available if *.wnd files are present, GRAL evaluations if the *.gff files have been saved.

After selecting the menu item, a dialog appears that allows you to define multiple points by manually entering coordinates or by clicking on the positions in the domain map. The entered data can also be edited.

Using the button "Add receptor points" the defined receptors can be added to the evaluation table.

16. Flow field simulations with GRAMM

16.1. Define grid

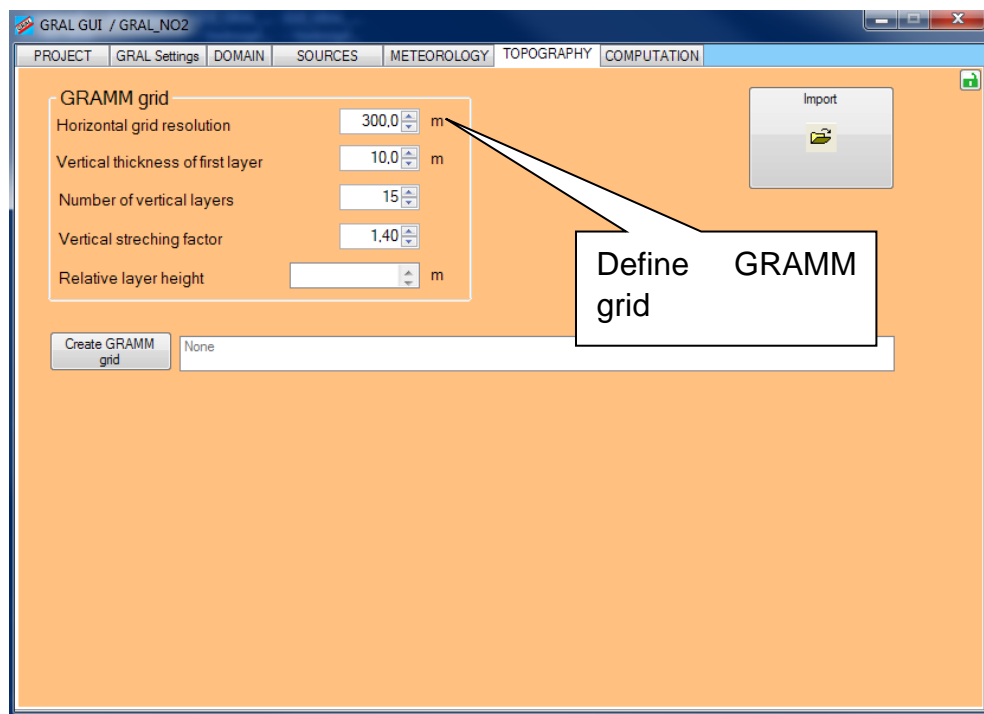
The GRAMM grid can be generated in the “Topography” tab.

The user must define the horizontal grid resolution, the height of the lowest cell (in most cases 10 m is useful), the number of cells in the vertical direction, and the vertical stretching factor.

The stretching factor must be chosen in a way, such that the top of the model domain is at three times as high as the highest elevation in the domain. The top of the model domain is indicated in the field “Relative top-level height”, which is the relative height from the lowest level in the domain.


The model domain itself should be defined after having fixed these parameters.

Every time the horizontal resolution of the GRAMM grid is changed, the GRAMM domain is deleted for numerical reasons.

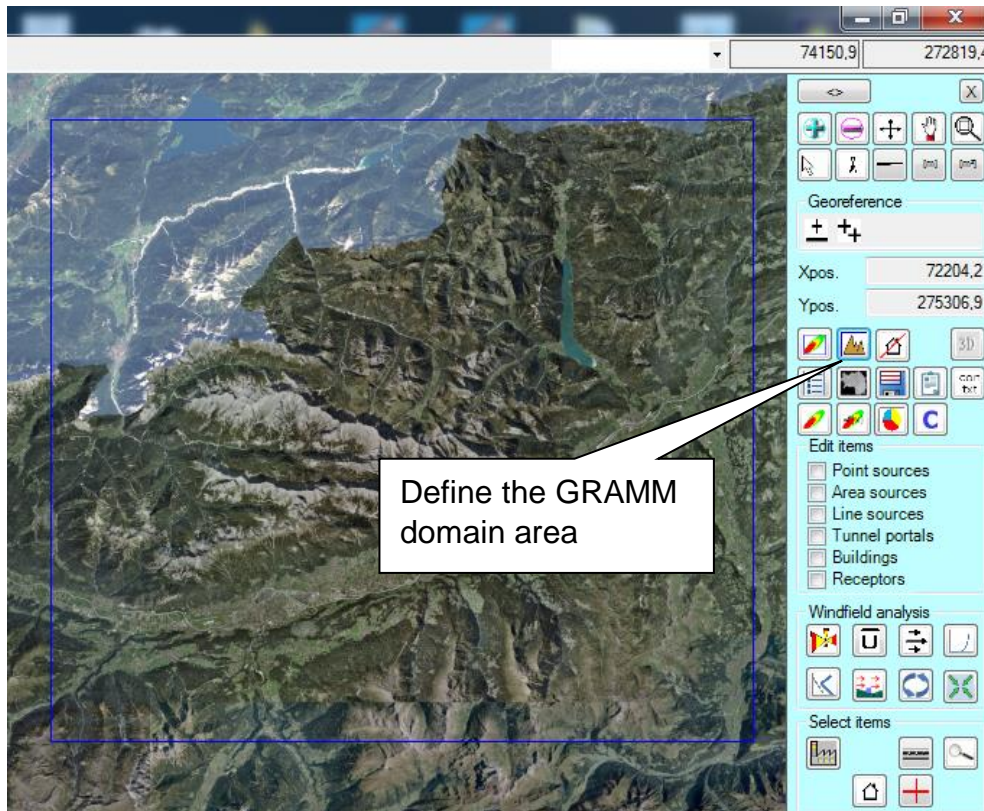


16.2. Define model domain

Menu: "Edit – GRAMM model domain"

Toolbar: 

Start the GIS Interface ("Domain" tab) and click the button as indicated in the next figure or use the menu entry "Edit – GRAMM model domain". The left mouse (left mouse button pressed) is used to drag a rectangle. This procedure can be repeated several times.



16.3. Generate GRAMM topography

After the GRAMM domain has been defined, a button named "Create GRAMM grid" appears in the "Topography" tab.

Clicking this button allows the user to first set the number of cells at the model boundaries for which the topography is smoothed. The number of cells multiplied with the chosen horizontal grid spacing indicates the spatial extent of this zone. The extension of the smoothed zone at the boundaries should be defined in dependence on the relative height between the highest and lowest elevation within the model domain. The higher the relative elevations within the model the larger should be the smoothed zone. It may vary between a few hundreds of meters and a few kilometers.

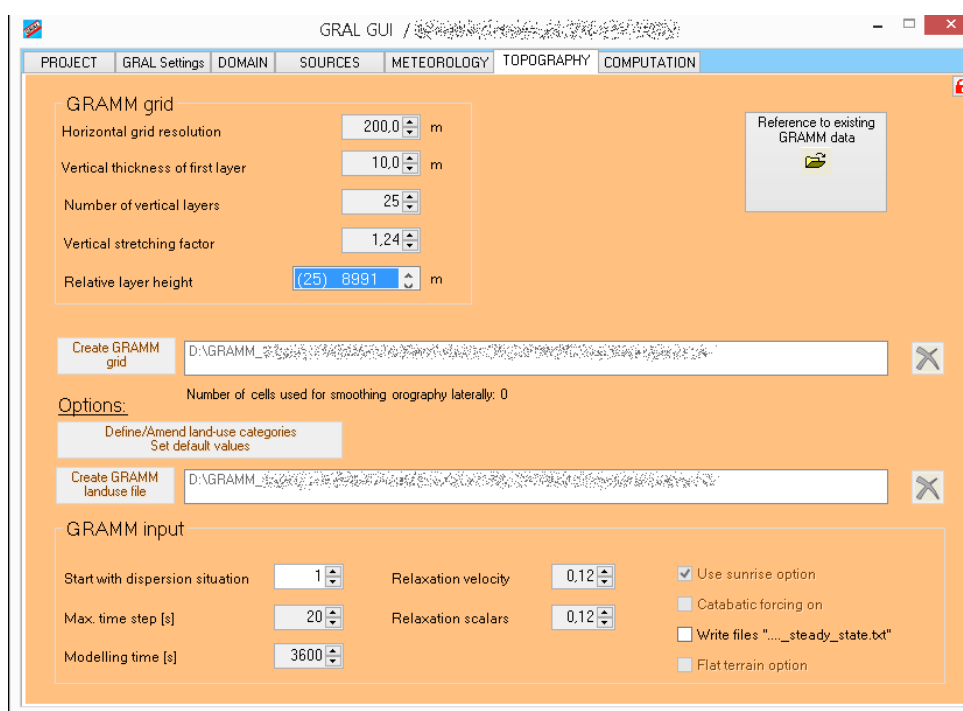
Afterwards the user can select the topographical data, which must be stored in ESRI-ASCII format (see section 19.3).

Flow field simulations with GRAMM

The horizontal resolution of this topography file needs to be less than or equal to the horizontal grid size of the GRAMM grid used.

If meteorological data has already been imported, GRAMM simulations are ready to be launched at this step.

The GUI generates automatically the file “ggeom.txt” in the subdirectory “Maps”, which can be used to visualize the model topography. It is recommended to generate a contour map prior to any GRAMM simulations to check if the topography has been generated in a right way. As an option use the 3D view (see chapter 16.6).



Use the “X” button to delete all GRAMM information (GRAMM area, GRAMM settings).

16.4. Consideration of land-use data

There are two options to take land-use data into account.

In case that no input file for land-use data is available, at least a homogenous land-use class can be defined for the model area. **Important: The code for the default land-use category is 0 (zero).** By pressing the button as illustrated in the following figure, a table is opened, where the characteristics of the default land-use category can be defined.

In addition, own land-use categories can be defined in this table. **Note: The codes are limited to the range 0 – 999.** In this way, the soil characteristics of own land-use categories with individual codes

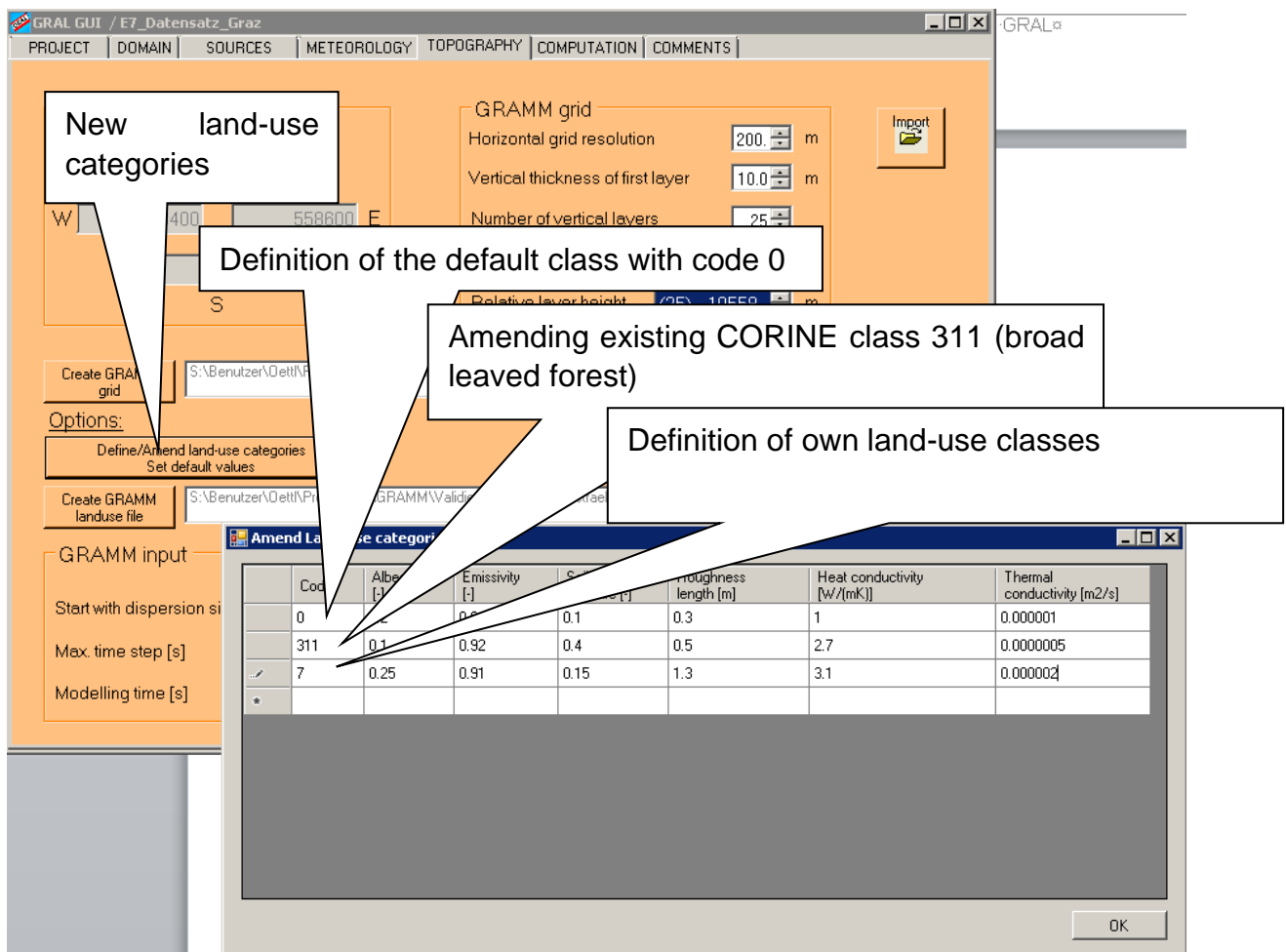
can be defined. These will be applied when importing a corresponding input file containing these codes.

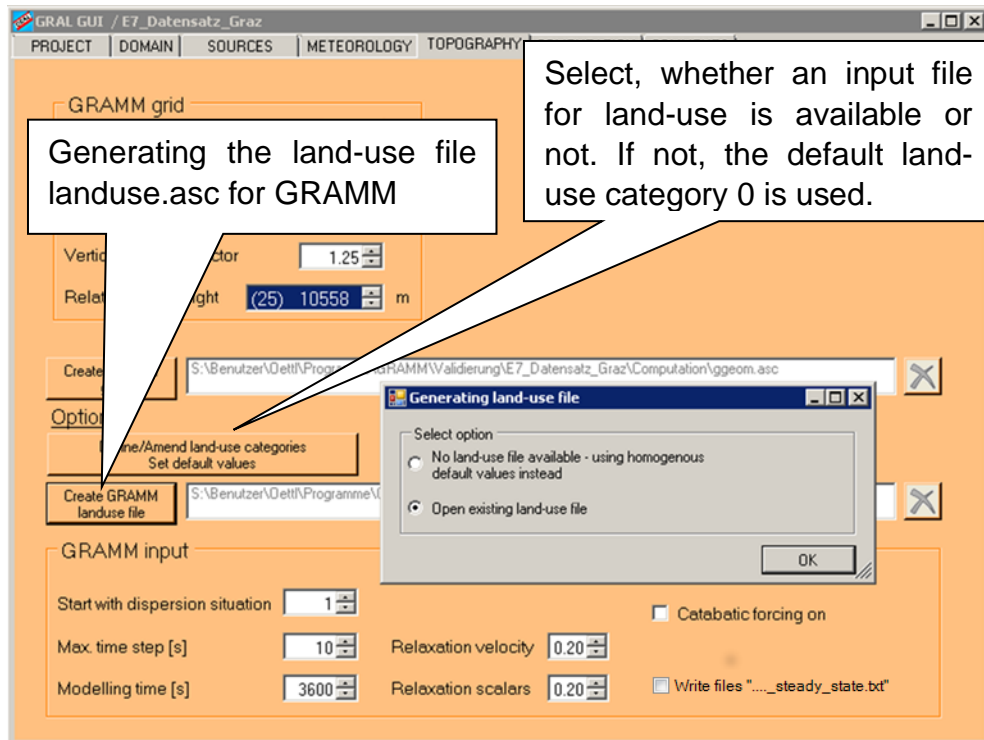
Basically, the CORINE classes (codes with 3 digits) are implemented. These pre-defined CORINE classes can also be changed, by providing new soil parameters in the table for each class.

All redefined land-use classes are stored in the file “Landuse_Default.txt”, which is saved in the sub-directory “Computation” of the project. The file can also be edited by using simple text-editing software.

The format for all files containing land-use data is ESRI-ASCII (see section 19.3.2). Such files can be loaded by pressing the „Create GRAMM land use file“ button as depicted in one of the figures below.

The horizontal resolution of this landuse file needs to be less than or equal to the horizontal grid size of the GRAMM grid used.





In case that land-use data has been imported, the following files are automatically generated in the subdirectory "Maps", which can be used to visualize surface parameters and check for validity:

- roughness.txt: Roughness lengths
- albedo.txt: Surface albedo
- density.txt: Soil density
- conductivity.txt: Soil conductivity
- emissivity.txt: Surface emissivity
- moisture.txt: Soil moisture content

16.5. Adjusting GRAMM control parameters

The following GRAMM control parameters are foreseen:

„Start with dispersion situation“

Number of dispersion situation with which the simulation starts.

„Max. time step [s]“

The numerical integration of the conservation equations for momentum, pot. temperature, and mass is done with discrete time steps, which are adopted by GRAMM automatically to accelerate simulations as much as possible. In order to keep the simulations numerically stable the maximum time step needs to be below a critical value. Unfortunately, there is no way to compute this critical value prior, but it is up to the experience of a user to set the upper limit for the time step. Complexity of topography and horizontal grid resolution are the most critical parameters influencing numerical stability of simulations.

„Relaxation velocity“ and „Relaxations scalars“

The smaller these relaxation factors are chosen, the better the numerical stability of the simulations. However, it might happen that momentum conservation is not fulfilled everywhere in the model domain, if these factors are set too small.

„Modelling time [s]“

Defines the integration time for each dispersion situation. This time is not necessarily the same as defined in the field “Dispersion time” for the GRAL simulations in the “GRAL settings” tab. It is recommended to use the default value of 3600 s, as GRAMM has been tested thoroughly for this integration time.

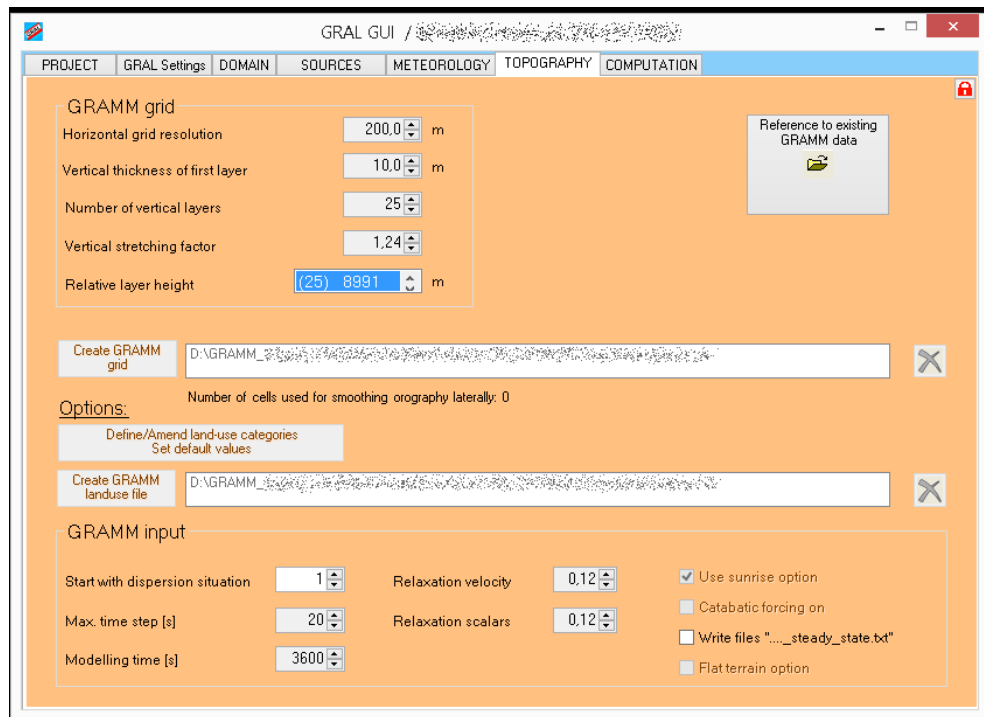
Since GRAMM 17.01 an improved time step adjustment is used in GRAMM making the choice of the maximum allowed time step less critical. Now it is possible to use a maximum time step of 10 to 30 s regardless of the horizontal grid size.

Up to GRAMM version 17.01 the following values for the maximum time step and relaxation factors were recommended to ensure numerical stability of GRAMM simulations:

Hor. grid size	Max. time step	Relaxation factors
200 – 500 m	10 s	0,2
50 – 200 m	5 s	0,1
25 – 50 m	2 s	0,05

Flow field simulations with GRAMM

All control parameters filled in for GRAMM simulations are stored automatically. There is no need for extra confirmation. Topographical data as used in GRAL simulations can be generated by ticking the checkbox “Write file ‘GRAL_Topography.txt’”. The corresponding file is stored during the computation of the first flow field situation in the sub-directory ‘Computation’. It can be viewed as any other contour map.



“Sunrise option”

If this option is checked, GRAMM will use an alternative approach to compute the flow fields using modelling times up to 6 hours to simulate sunrise or sunset effects. The calculation time and the number of resulting flow fields increases, because intermediate results are stored, too. This option is useful for computing large valley-wind systems by making use of artificial dispersion situations for the GRAMM initialization in combination with the match-to-observation function after finishing the simulations. GRAMM numbers automatically these intermediate flow fields and stores them correspondingly in separate files. To make these intermediate flow fields available for the match-to-observation function, additional “dispersion situations” are introduced in the meteorological input file “meteopgt.all”. Instead of wind direction, wind speed, stability class, and frequency, a different coding is used for the intermediate files in order to reference them quickly to the original dispersion situation. The coding is for example:

“65.15,207.1,6,0”

65 is the wind direction in degrees; 15 is the wind speed in cm/s, 207 is the corresponding original dispersion situation, 1 indicates that this is the first intermediate (after one hour simulation time) flow

field of this original dispersion situation, 6 is the stability class of the original dispersion situation, and the frequency is set to zero (for numerical reasons in subsequent GRAL dispersion simulations).

“Write files „..._steady_state.txt“

This option forces GRAMM to write a file “xxxxx_steady_state.txt” for every dispersion situation. This file contains a value for every cell at the lowest grid level. The value describes if the steady state criterion according to VDI 3783-7 is reached.

This file can be displayed at the GIS-interface like a concentration file.


Value	Steady-state criterion met		
	U	V	W
0	✗	✗	✗
1	✓	✗	✗
2	✗	✓	✗
3	✓	✓	✗
4	✗	✗	✓
5	✓	✗	✓
6	✗	✓	✓
7	✓	✓	✓

“Flat terrain option”

This option should be used only for flat or almost flat terrain, whenever GRAMM wind field simulations are performed to consider the effect of inhomogeneous land use (roughness) and vertical wind shifts (Ekman shift) on subsequent GRAL simulations. Both effects cannot be accounted for when applying GRAL without GRAMM.

16.6. Check the GRAMM geometry

Menu: “View – 3D view”

Toolbar: 

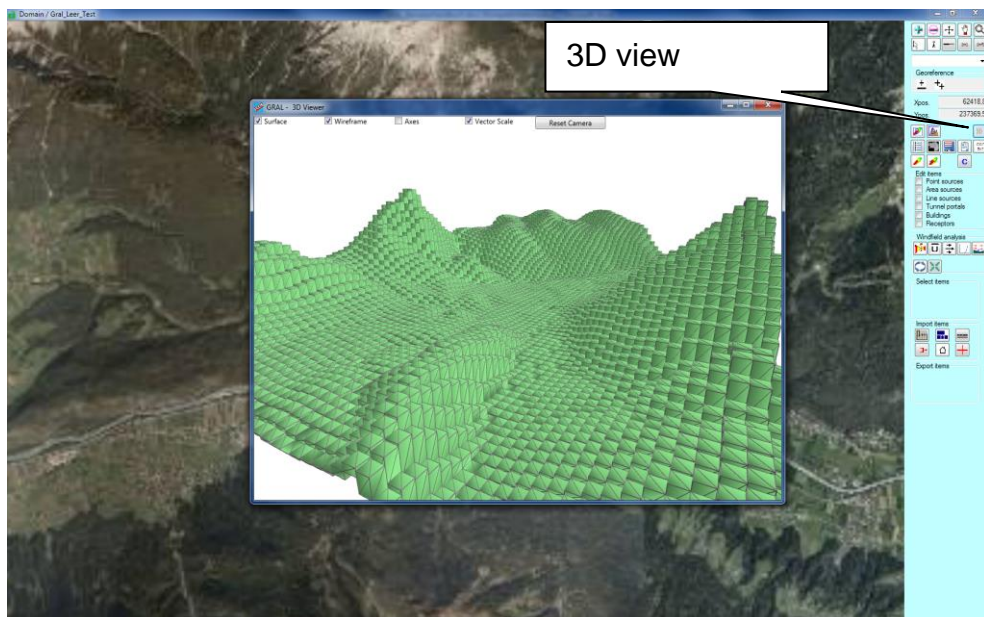
Back in the GIS form it is possible to check the GRAMM geometry by a 3 D module.

For a better orientation, also at later post processing like the match to observation function, the receptor points are displayed in the 3D drawing too.

The actual screen section is shown at the 3D module.

The checkbox “Smooth” creates a commonly used smooth view of the surface. For quality checks it’s recommended to use the non-smooth view. The vertical scaling factor should be set to 1 for realistic drawings.

The movement of the camera in the 3D view is done by the cursor-keys (see chapter 19.4).



The inn-valley nearby Telfs, view angle to northeast, no smooth, the GRAMM grid has a horizontal resolution of 300 m.

To get a feeling about the GRAMM grid frame, you can check the “Show raster” checkbox in the “Layout manager” (see chap. 11.4) by selecting “GRAMM DOMAIN” in the “Object manager” (see chap. 11.3).

16.7. Starting GRAMM

Before starting GRAMM, it might be necessary to generate “Control files” and “Meteo files” by pushing the corresponding button on the “Computation” tab.

Further you can define the number of processor cores to be used in the simulations. The used processor cores per GRAMM computation should not exceed a value of

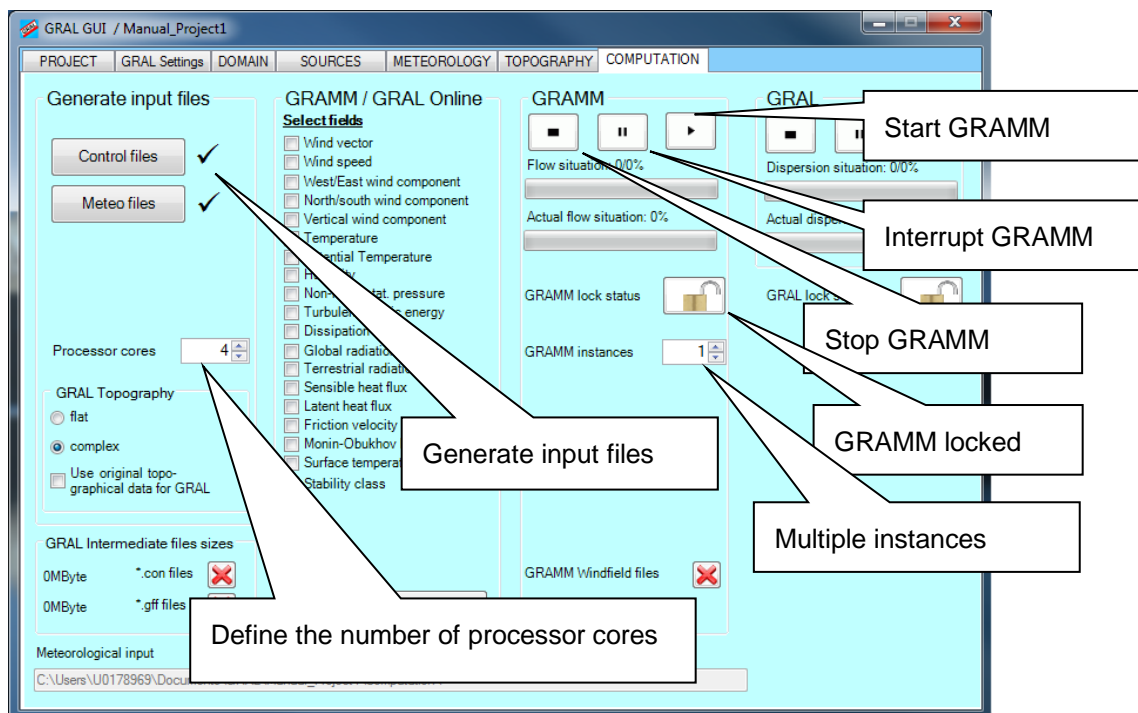
$$\frac{\text{minimum count of cells per x - or y direction}}{30}$$

If more cores are available on your machine, it is recommended to use multiple GRAMM instances to use the whole computation power. Each instance uses the defined number of processor cores.

If you use multiple instances, the stop and interrupt button and the GRAMM online feature will work for the first instance only.

Starting GRAMM is done like GRAL (see sec.15.1). After clicking the “Start” button, the GUI asks for the GRAMM executable. As for GRAL, the GRAMM simulation is launched with CMD window.

Pressing the “Pause” button interrupts GRAMM and the number of the actual computed dispersion situation is stored in the corresponding field in the “Topography” tab, while pressing the “Stop” button just interrupts GRAMM.



If GRAMM wind fields are computed, the GRAMM project is locked to prevent the project to become invalid. Manual unlocking is possible by clicking in the corresponding lock symbol.

16.8. Visualizing GRAMM computations online

Several parameters computed by GRAMM can be visualized online (updated with every time step) to help analyzing possible numerical instabilities. Some of these parameters can also be visualized online in GRAL simulations, when the prognostic microscale wind field model is activated. This can be done in a similar way.

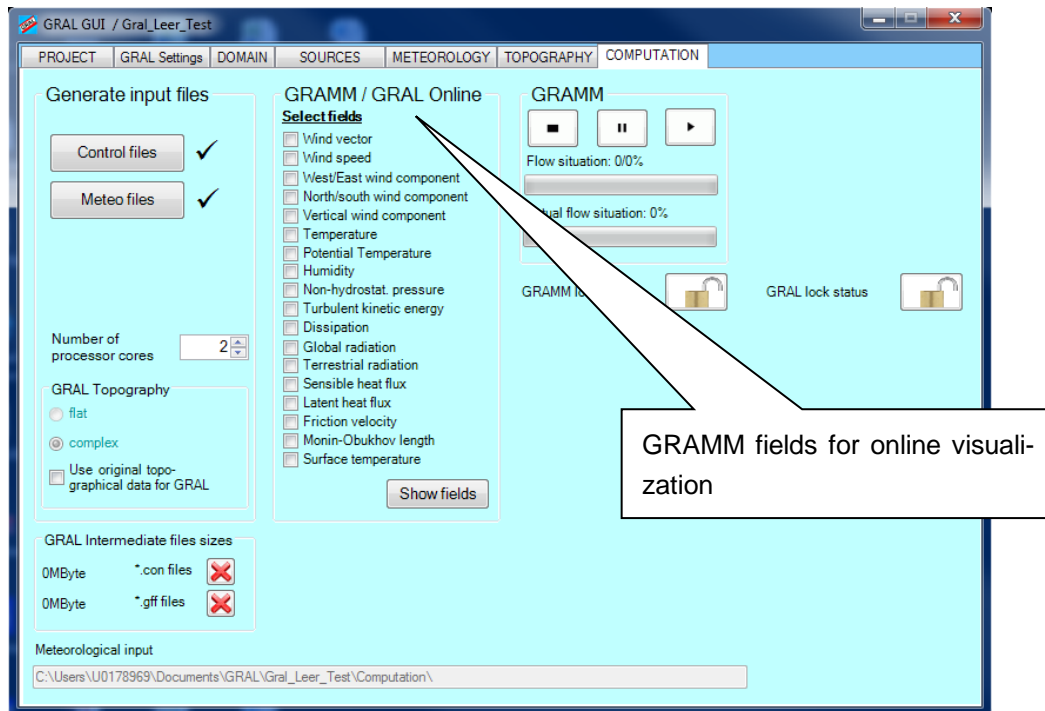
It is recommended to select the desired parameters, and to adjust the graphical design of the fields to be displayed in the GIS Interface prior to starting GRAMM. Once GRAMM is running, it is tedious to make changes, because CPUs will be operated at high load.

For instance, when aiming at visualizing the flow field online, click the box “Wind vector”. The user will be asked for the vertical layer to be displayed. Then click on the “Show fields” button to launch the GIS Interface.

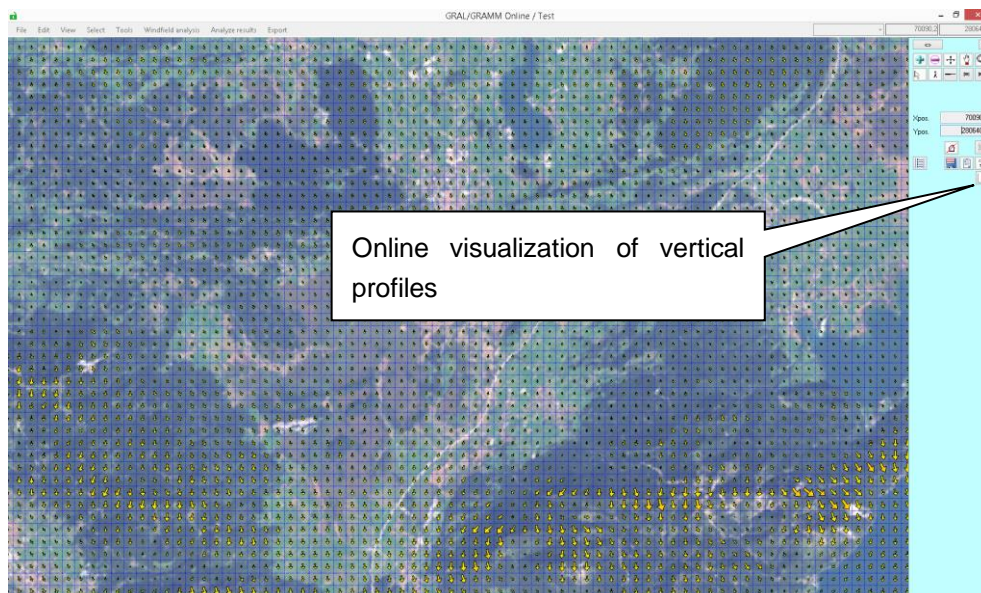
<i>All changes made in the GIS Interface during the online session will not be stored.</i>
--

Before starting GRAMM adjust all visualized fields as desired, because after GRAMM has been launched it will be tedious to make any change for CPUs will be at full load.

GRAMM updates the fields at every time step. It is possible to add or remove parameters for online visualizing during the simulation. To facilitate such actions, it is recommended to open the CMD window, where GRAMM is running, and to press the “Pause” key. Then make all changes and afterwards press the “Return” key to continue the GRAMM simulation.




Besides horizontal fields, vertical profiles can be visualized online, too. Use the corresponding button in the GIS Interface, select a field and click on the map, where the profile should be computed.



16.9. Post processing GRAMM wind fields: The Re-Order function

It is generally recommended to apply the Match-to-Observation (see chapter 16.12) function after GRAMM simulations.

Menu: "Windfield analysis – Re-order wind fields"

Toolbar: 

The re-order function can be used but is not further developed anymore.

The re-order function works as follows:

Simulated GRAMM flow fields are based on time series of wind speed, -direction, and stability class for a certain location in most cases inside the GRAMM model domain. The GUI breaks the time series into some hundred frequency bins of different dispersion situations. At the end of GRAMM simulations, for each of dispersion situations, a wind field has been simulated and stored. The Re-Order function searches within these flow fields, which fits best observed data at the location of the measurement site. It might happen, for instance, that flow field nr. 479 fits best for dispersion situation nr. 1 and so on. In this example, flow field 479 is renamed to 00001.wnd and stored in the subdirectory "Re-ordered". This procedure is repeated for all dispersion situations. In this way significant improvement between simulated and observed wind data at the monitoring site used as input to GRAMM can be achieved. However, model results should also be checked at other monitoring sites as well, if available.

The philosophy of the Re-Order function is based on the knowledge that initial- and boundary conditions of a mesoscale model are never known in all detail, especially in complex terrain. The function searches only for wind fields with similar stability categories, e.g. a flow field, with a stable dispersion category, is compared only within other flow fields owing stable classes, too.

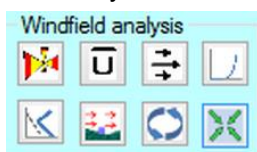
The more flow fields have been computed the better the results of the Re-Order function. Therefore, it is best to compute all flow field situations first, before launching the Re-Order function, although this is not imperative. It should be noted, that the Re-Order function can also be applied to old GRAMM projects.

Open the GIS-Interface a start the Re-Order function by pressing the button as indicated in the following figure. Then click on the location of the monitoring site in the map, which has been used as input data for the GRAMM simulations. The user will be asked for the anemometer height above ground level. The new, re-ordered flow fields are stored in a new subdirectory "Re-ordered". The original flow field files are not overwritten but remain in the subdirectory "Computation". In order to evaluate the new results after the Re-Order function has been carried out, it is recommended to create manually another sub-directory (e.g. "Original") and to copy the original flow field files (extension *.wnd) to that directory. In a next step, move all flow field files computed via the Re-Order function from the subdirectory "Re-ordered" to the subdirectory "Computation". Now the new flow fields can be analyzed (see sect. 16.10 for more details).

16.10. Evaluation of GRAMM results

Menu: "Windfield analysis"


Toolbar:



There exist three different types of evaluation procedures (simulations need not to be finished to use them). In a 1st step open the GIS interface to select an evaluation method, like in the picture below.

Compute wind statistic at a specific point

Menu: "Windfield analysis – Wind statistic at a point"

Toolbar: 

This function is used to produce a new time series at any user-defined location or at all receptor points set in the project. Afterwards this time series can be analyzed in the "Meteorology" tab.

Extracting meteorological time series at any location within the model domain

It is generally recommended to evaluate simulated wind fields at those sites, where subsequent dispersion simulations will be performed. These evaluations should also be documented in the report about the dispersion simulations. To get the statistics, click the left button in the group box "Wind field analysis" of the GIS-Interface or the menu entry "Wind field analysis – Wind statistic at a point". The user will be asked for a file name and the height above ground for the analysis and if local stability classes should be used (preferred option for GRAMM computations since version 17.1). Define a point at the GIS window (left click on the map). Usually the procedure takes several minutes; a pop-up window indicates when it is finished. Analyses can be carried out for several sites simultaneously.

At the end of the analysis, select the tab "Meteorology" in the main window of the GUI and use the corresponding buttons to get wind roses and so on. The file name is indicated in the list box on top, the file itself is stored for re-use in the subdirectory "Metfiles". Thus, it can be re-opened at any time the project is launched again and be analyzed.

Whenever meteorological time series are imported for reasons of analyzing the data only, without any intention to use this data for GRAMM or GRAL simulations, it is imperative not to click the “Classification/Default” button; otherwise the meteo-data for the simulation will be overwritten!

The “Classification/Default” button is blocked by the project-lock function at already simulated projects (see chap. 15.2)


Analyze stability classes

If you check this button or the menu entry “Wind field analysis – Analyze stability classes” you can select a flow field situation and display the local stability classes for this flow field situation (available since GRAMM 17.1).

You can analyze the meteorological data like described in chapter 14.3.

Mean wind speed at a height

Menu: “Windfield analysis – Mean wind speed at a height”

Toolbar: 

An average wind speed at a specific height above ground for the time period can be calculated and visualized by clicking the second button from the left in the group box or the menu “Wind field analysis – Mean wind speed at a height”.

The result (note that this procedure may also take several minutes) will immediately be visualized and can be further processed using the object- and layout managers. The low-pass filter should not be activated in this case.

Horizontal wind field at a given height

Menu: “Windfield analysis – Wind field at a height”

Toolbar: 


Single vector flow fields can be visualized by pressing the third button from the left in the group box “wind field analysis” or the menu entry “Wind field analysis – Wind field at a height”.

The user will be asked for the flow field to be shown and the height above ground. Also, this vector field can be further graphically processed using the object- and layout manager (see chapter 11.4).

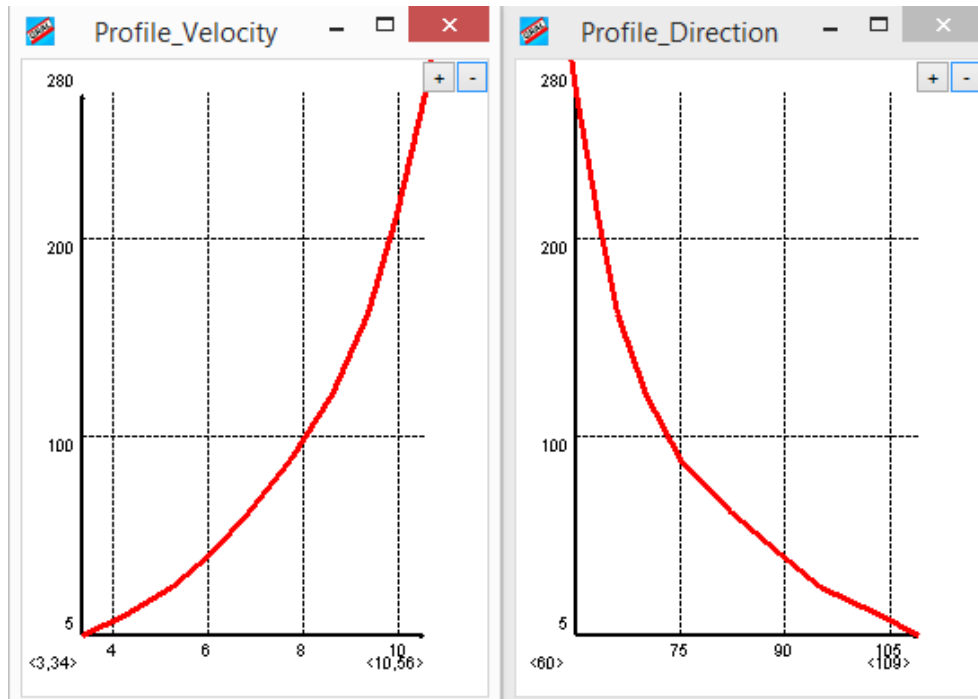
For further information can be found in chapter 17.1.1.

Vertical profiles of wind speed and –direction for a single flow field

Menu: “Windfield analysis – Vertical profile at a point”


Toolbar: 

Select a dispersion situation in the first step. Left mouse clicks at the map loads the vertical wind profile at this position and the forms with the vertical graphs are refreshed.



Section wind field drawing

Menu: “Windfield analysis – Section wind field drawing”

Toolbar: 

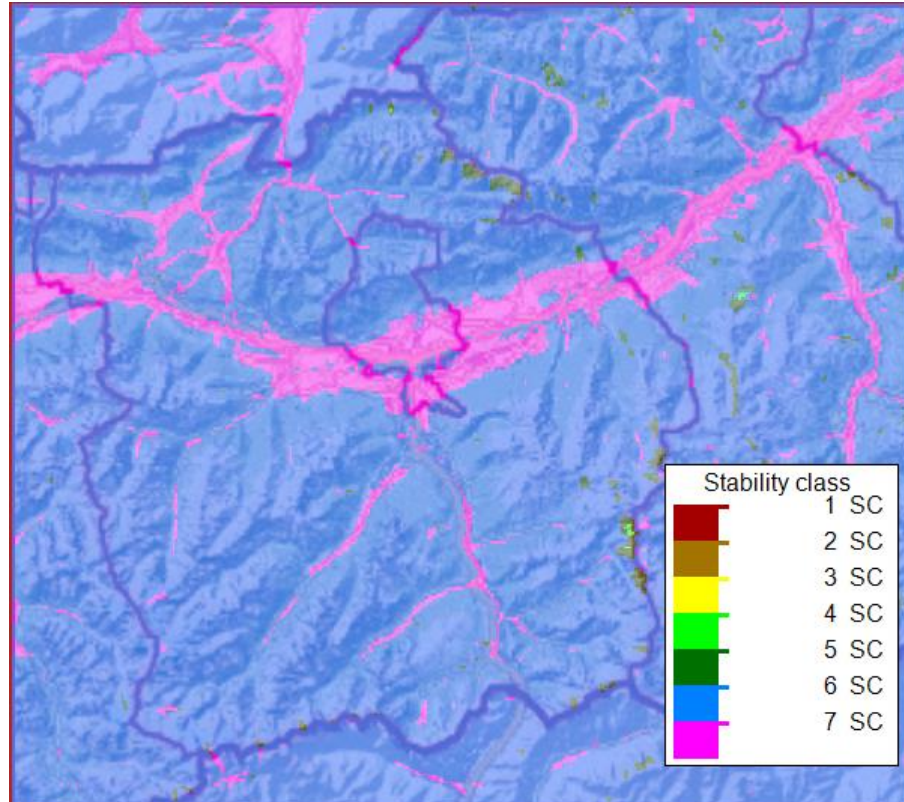
See chapter 17.1.2.

Local GRAMM stability classes for each raster cell

Menu: “Windfield analysis – Analyze stability classes”

Toolbar: 

Select a dispersion situation and the stability class is displayed as an overlay at the GIS window.



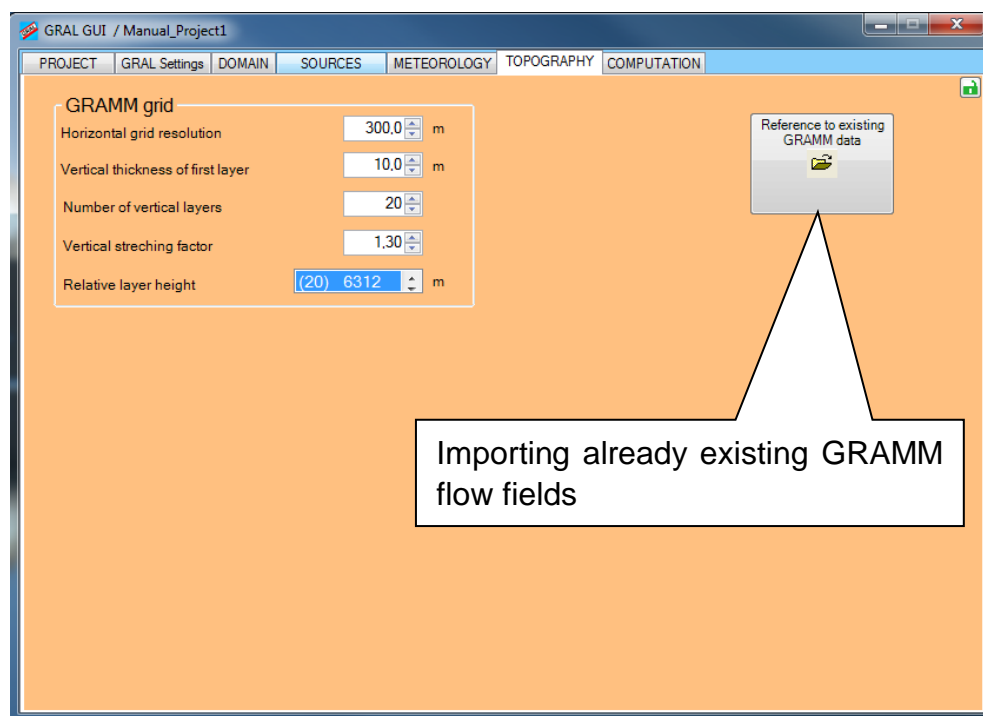
Picture: local stability classes for a stable situation at the Inn-valley

16.11. Import of existing wind field libraries

In case that dispersion simulations with GRAL are to be carried out in domain, where GRAMM wind fields are already existent from a different project, these wind fields (wind field library) can be easily imported in the new project, without additional costs of computer space (the application creates a reference link to the existing data) . Click on the “Reference to existing GRAMM data” button in the “Topography” tab and select the subdirectory, where the following GRAMM files are stored:


- ggeom.asc (topography)
- *.wnd-Files (flow field files)
- Gramm.geb (definition of model domain)
- mettimeseries.dat
- meteopgt.all
- IIN.dat
- optional: landuse.asc
- optional: reinit.dat

In case that GRAMM wind fields have already been computed by using the GUI, all these files can be found in the subdirectory “Computation” of the project.



16.12. The Match-to-Observation Function

Menu: "Windfield analysis – Match to observation"

Toolbar: 

Basic concept

The Match-to-Observation procedure allows for an additional and the recommended modeling strategy with GRAMM::

In a first step, the simulations are initialized with an artificial wind rose that includes a variation of classified situations. Although simulation times will increase primarily, these wind fields can later be used to match any new meteorological observations inside the domain. In the second step, the Match-to-Observation function searches for the best matching flow field of the defined meteorological stations for each weather situation (see chapter 19.5).

The philosophy of the Match-to-Observation function is based on the knowledge, that initial- and boundary conditions of a mesoscale model are never known in every detail, especially in complex terrain.

This function aims at matching already existing GRAMM wind fields to any meteorological observations inside a GRAMM domain, regardless the period of time when these measurements have been taken. The imported time series of meteorological data are synchronized automatically. Thus, it is not necessary to prepare the data in a way that each time series covers exactly the same time period. Synchronizing is also possible if one station uses half-hourly mean values and another one is based on hourly mean values.

The more flow fields are available for the matching process, the better the results of the Match-to-Observation function. In order to use this functionality optimally, a calculation of all flow field situations is recommended before starting the "Match to Observation" function, although this is not mandatory.

The implementation

It should be noted, that the Match-to-Observation function can be applied to every existing GRAMM project.

In order to use of the Match-to-Observation function, you need to create a new project. The procedure cannot be started within the original GRAMM wind field project!

So the original GRAMM flow fields are always preserved.

Hints: before starting a match to observation function, create a meteorological time series (“Compute wind statistic at a specific point” - see chapter 16.10) at the position of the observation station(s) inside the original GRAMM project and analyze the results.

Since the matching algorithm can only use existing cases, the measured wind direction, wind speed, and stability class combinations must exist in the original GRAMM project; otherwise, the GRAMM wind fields are not sufficiently well calculated

Start the Match-to Observation process

After a new project has been defined, import GRAMM wind fields as explained in sect. 16.11. Then activate the Match-to-Observation procedure (menu or toolbar button). A new window will open, which allows for importing as many meteo-stations as desired. Note, that the first (topmost) meteo-station will be used to bin the dispersion situations. The order of meteo-stations has no impact on the fitting process itself, but the stability class is matched for the first meteorological station only.

Basically, a new meteo-station is imported by clicking the button “Add station”. If this meteorological file is a *.met file with a coordinate header, the position of that measurement station is set automatically. If you like to set or change the position of the measurement station, select the meteo file in the list box and click on the map at the very location of the observation site.

In this way, multiple meteo-stations can be imported.

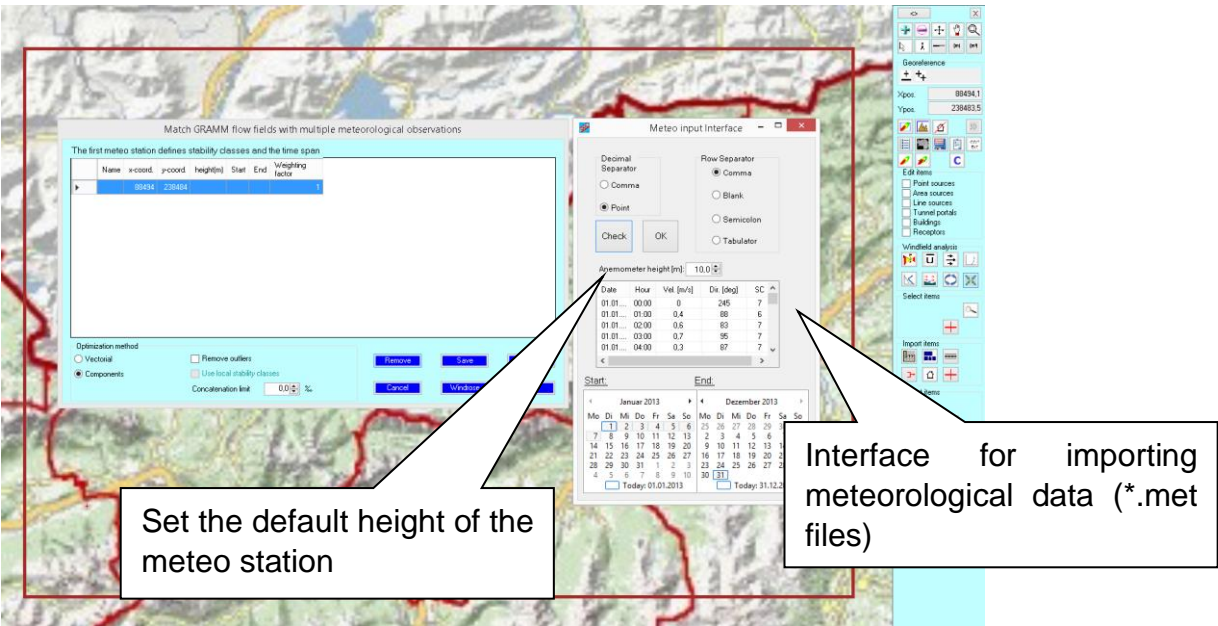
Already imported meteo-stations can be selected in the table by clicking on the station.

In doing so, a wind-rose can quickly be depicted (click “Windrose” button), the selected station can be removed from the list (click “Remove” button), or the coordinates of this station can be defined again by clicking on the location on the map or by entering the coordinates manually into the list. If you like to add a new station, use the add station button.

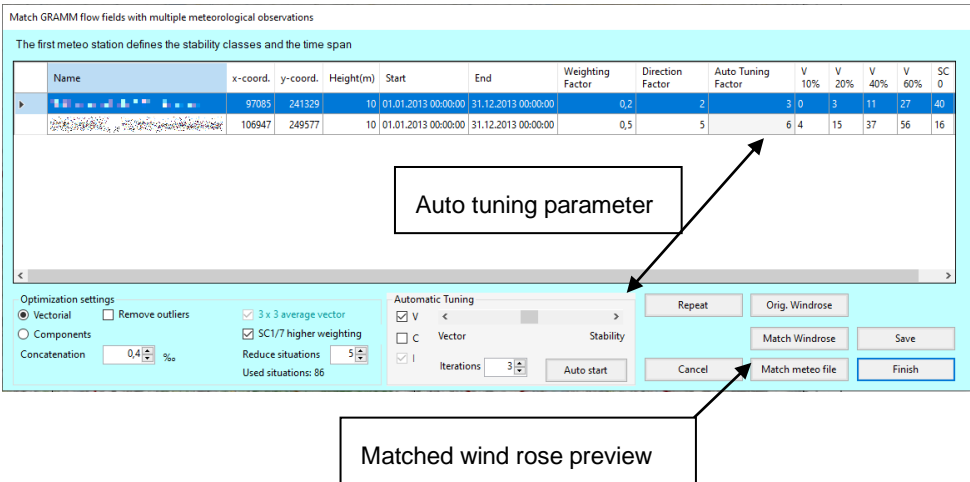
The height of the meteo station can be adjusted in the data grid after the meteo file has been defined.

If all meteo stations are defined, you can start the matching process. The function reads all computed GRAMM wind fields and stores the wind vectors for all measurement stations.

Flow field simulations with GRAMM



For the improvement of the matched wind data you can use numerous parameters:



Optimization settings

Vectorial	The best fitting weather situation to your observed station is searched by a method of least squares of the wind components u and w
Components	The best fitting weather situation is searched by a nonlinear algorithm, based on separated weighting of direction and wind speed (see chapter 19.5).
Remove outliers	For single hours of the time-series meteorological observations will not be considered, if they don't fit to the other ones (experimental)
Use local stability classes	Since the GRAMM version 17.1 information about the local stability class for each raster cell will be stored. The prognostic wind model GRAMM computes the wind vectors and the turbulence in dependence of several parameters,

	<p>e.g. the topography, the land use and the radiation. Therefore, the local wind vectors at one raster cell will be assigned to a stability class that differs from the initial wind field. It is recommended to use the local stability classes to get a reliable coherence between the local wind vectors and the corresponding stability class.</p>
3x3 average vector	<p>This option uses the average wind vector of a 3x3 raster around the measurement station</p>
SC1/7 weighted stronger	<p>This option increases the weighting of the stability classes 1 and 7</p>
Reduce situations	<p>You can set a value (0 to 90) to apply a bonus value to the calculated vectorial error for already used weather situations at the match tuning process. A value of 0 means no bonus (default behavior), 90 applies the maximum bonus (90 %) for the wind vector of already used situations.</p> <p>Whether and how much the number of situations can be reduced depends on the specific issue. In the case of odor dispersion, a reduction is not advisable; in the case of large-scale infrastructure projects, a reduction can certainly make sense.</p> <p>How to use this option?</p> <p>Define a value and press the "Repeat" or the auto tuning "Start" button. The number of used situations (the concatenation option is not considered here) will be displayed in the dialog. The preview for the matched situations shows the resulting wind rose for the selected meteorological station. Here you see an example showing the measured wind rose and matched wind roses using 1082, 485 and 111 situations.</p>

	<div> </div>
Used situations	Preview showing the number of used weather situations for the match to observation
Weighting factor	Inside the data grid you can set a weighting factor per observation station. The higher this value, the stronger the influence of this station to the optimization algorithm. There is an influence on the weighting of the stability class error too. The higher the weighting factor, the lower the influence of the stability class error values (see chapter 19.5).
Direction factor	Inside the data grid you can set a weighting factor for the wind direction per observation station. The higher this value is set, the stronger the influence of the wind direction of this station. This factor is used at the component's method solely (see chapter 19.5).
Concatenation limit	<p>This value allows to reduce the number of flow situations without losing many details of the flow structures. For all flow situations with a frequency lower the value "Concatenation limit" a corresponding situation is searched (see chapter 19.5). A reduced number of flow situations will speed up the GRAL calculation time and reduce the required disk space.</p> <p>Note that the concatenation limit does not work entirely in combination with the sunrise option.</p>

Match Windrose	Preview for the matched wind roses for the selected meteorological station during the match process (before you finish the match process). This makes it possible to check and improve the adjustment of the matching parameters, because the results at the meteorological stations can be visualized in advance.
Match meteo file	Creates a meteorological time series of the selected station in the Meteorology folder. Switching to the Meteorology tab, you can analyze this preview of the matched meteorology. The file name starts with the text "MatchPreview" followed by the name of the original meteorological file

Breaking changes (V2109)

The new Auto Tuning Factor affects the match tuning in such a way that stations with an Auto Tuning Factor of 0 (zero) are no longer considered in the tuning process. This allows to include additional meteorological stations in the match to observation process and deactivate/activate them during the match process. However, you can use the preview of the wind rose at all stations in the match process.

New created match settings files *.mmo are updated to include the new Automatic Match to Observation tuning options and therefore loading these files is not possible with GUI versions prior to 21.09.

Optional (from V2109) Automatic Match to Observation tuning function

After all GRAMM wind fields have been read and a first match tuning with the standard parameters has been processed, the auto tuning can be started.

Settings

Slider: you can use the slider to select whether you want to represent the wind vectors or the stability classes more accurately.

Checkboxes: use the checkboxes to select the optimization method: V for the vectorial and C for the component method. The cycles for the iteration can be selected using the numeric up-down item.

Auto Tuning Factors: a new column in the DataGridView enables the setting for a weighting factor for the automatic tuning process. This factor is applied to find the best weighting (and direction) factor for the automatic tuning process. A value of 0 means, the station is not used for the auto tuning optimization. The higher the factor, the stronger the weighting for the meteorological station.

Once the auto-match is finished, you can check the percentile results V10% to V60% and SC0/SC1 (if you like, you can also use the preview of the adjusted wind roses). Change some parameters (weighting factor or direction factor, the latter is not active in the vectorial mode) and then start either a default match ("Repeat" button) or a new auto-match (when changing the Auto Tuning factor or if you like to apply additional iterations).

Flow field simulations with GRAMM

The matching optimizer is partially parallelized from version 21.09 but is still quite time consuming, especially when several stations are considered.

For a detailed description about the matching algorithm look to chapter 19.5

It is possible to save and reload all settings using the respective buttons.

In the next step, the dialog shows how much percent of situations fit within a vectorial error of 10, 20, 40 and 60 percent and a stability class error of 0 or +-1 classes.

Match GRAMM flow fields with multiple meteorological observations

The first meteo station defines the stability classes and the time span

	Name	x-coord	y-coord	height(m)	Start	End	Weighting factor	Direction factor	V 10%	V 20%	V 40%	V 60%	SC 0	SC 1
▶	Meteo station 1	66508	238321	10	01.01.2013 00:00:00	31.12.2013 00:00:00	1		1.17	47	85	95	76	99

Optimization settings


☐ Vectorial
 ☐ Remove outliers
 ☐ 3 x 3 average vector
 ☒ SC1/7 weighted stronger

Concatenation limit %

Now it is your decision to finish the matching process, change the optimization parameters and repeat the matching procedure or the automatic tuning function (additional iterations). If you repeat the process, the wind vectors at the meteo positions are already available and the matching process is much faster than at the first step.

16.13. Exporting GRAMM flow fields for sub-domains

Menu: "Export – GRAMM Windfields"

Toolbar: 

This function exports a defined part of an existing GRAMM project (including already computed flow fields) into a new independent GRAMM project.

This new GRAMM project covering only a user-defined part of the original project and can be used for e.g. subsequent GRAL simulations. It has the advantage of lower disc-space requirements and faster evaluations and can therefore easily be stored on computers with low hardware capabilities.

When starting this function, the sub-domain must be defined by clicking and holding the left mouse button to draw up a rectangle. Thereafter, you need to define the name of the new GRAMM project and the transfer of data will start automatically.


17. Miscellaneous

17.1. Horizontal and vertical slices for GRAMM and GRAL flow fields

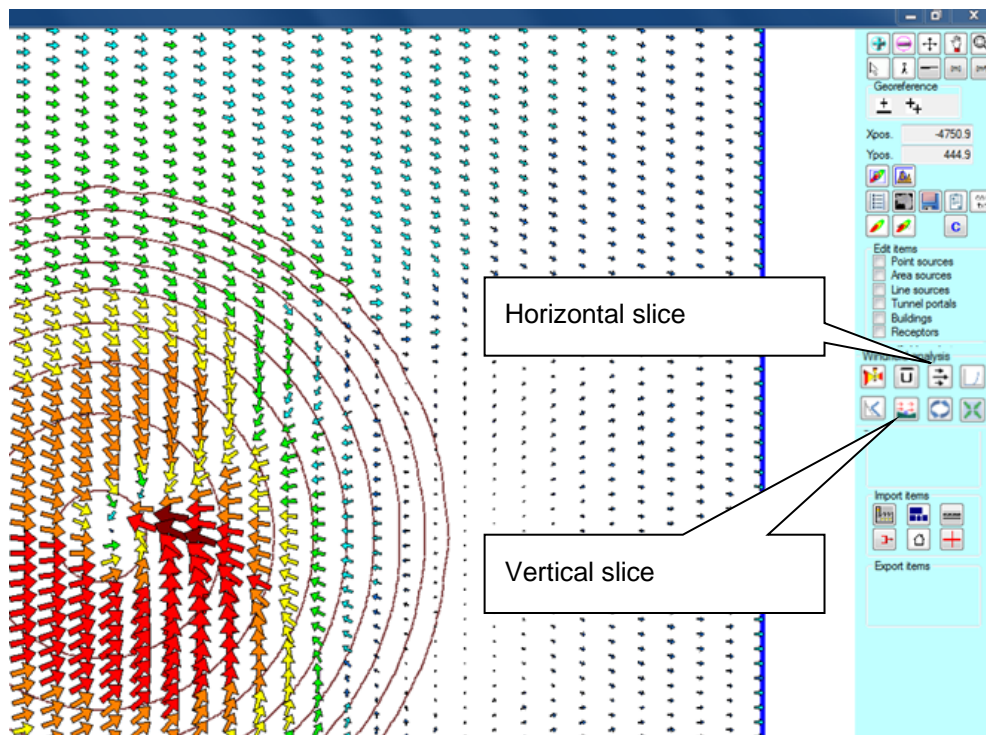
In the GIS-interface horizontal and vertical slices of GRAMM and GRAL flow fields can be created by clicking the corresponding buttons, as shown in the following figure.

17.1.1 Horizontal slice

Menu: "Windfield analysis – Wind field at a height"

Toolbar: 

In case of horizontal slices, the user is asked first to select either GRAMM or GRAL. Note, that only those models are provided for which wind fields have already been calculated and stored. Horizontal slices are terrain-following at the height specified by the user.



17.1.2 Vertical slice

Menu: "Windfield analysis – Section wind field drawing"

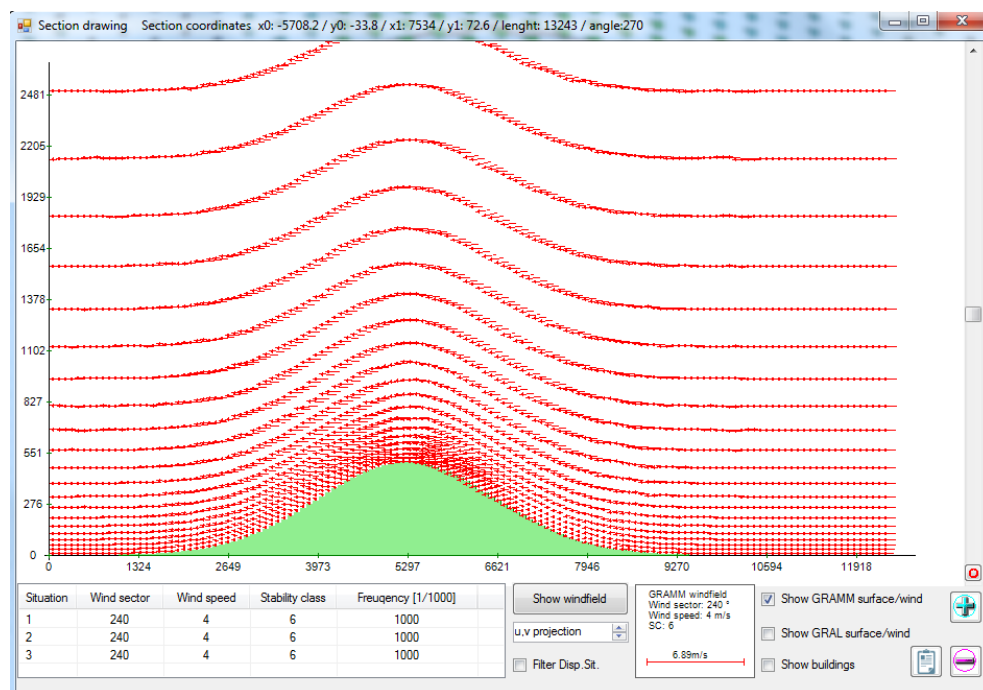
Toolbar: 

In a first step the position of the vertical slice to be performed must be determined by the user. This is simply accomplished by drawing a line in the map (start with left and end with right mouse button). Subsequently, a new window opens, where the categorized meteorological situation must be selected in the list in the lower left corner of the window. Pressing the "Show wind field" button displays the desired flow pattern.

There are "+" and "-" buttons for the vertical zoom, the horizontal zoom and the length of the wind vectors.

Vertical slices for GRAL surface and buildings need the file "GRAL_geometries.txt". GRAL writes that file, if *.gff files have been written (see chapter 8.3). At least one dispersion situation must be computed to use that option.

GRAL windfields can be displayed, if the corresponding *.gff files for the dispersion situation exist.

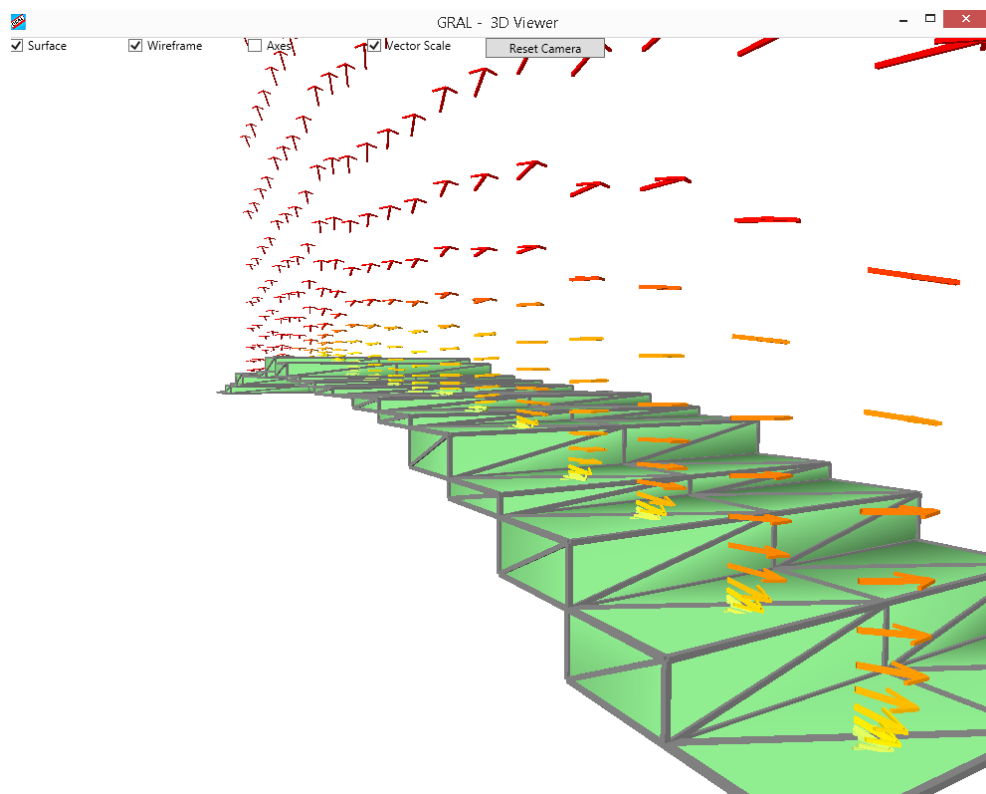


17.1.3 3D view of sections for GRAMM and GRAL surfaces and wind vectors

Use the “3D view” button to show the actual surface and wind vectors as shown in the “Section drawing” form. The length of the wind vectors is corresponding to the length in the 2D view of the vertical slice.

The checkbox “Smooth” creates a commonly used smooth view of the surface. For quality checks it’s recommended to use the non-smooth view.


The vertical scaling factor should be set to 1 for realistic drawings.



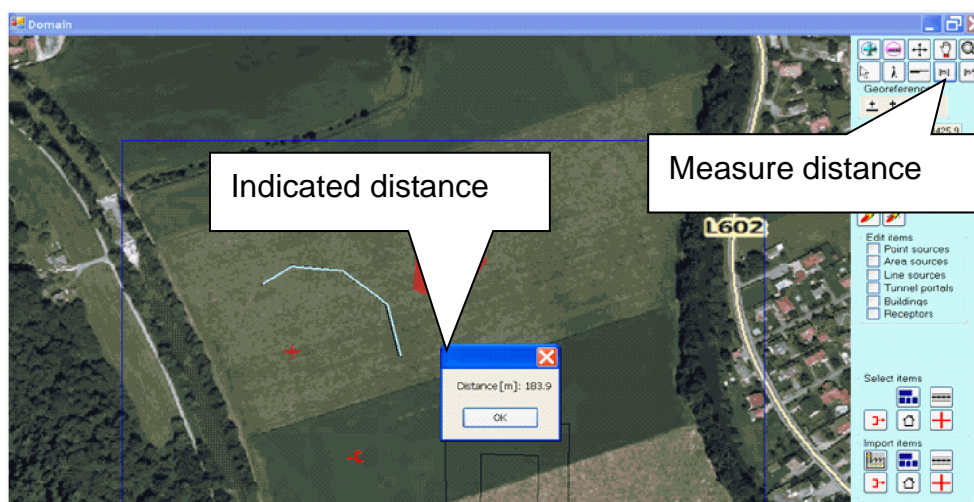
The 3D camera can be moved by the cursor-keys (see chapter 19.4).

17.2. Measure distances

Menu: "Tools – Length measurement"

Toolbar:  symbol

Corner points of the polygon to be measured can be set by left clicking on the map. Measurements are finished with a right-click on the final point or by pressing the ESC key.

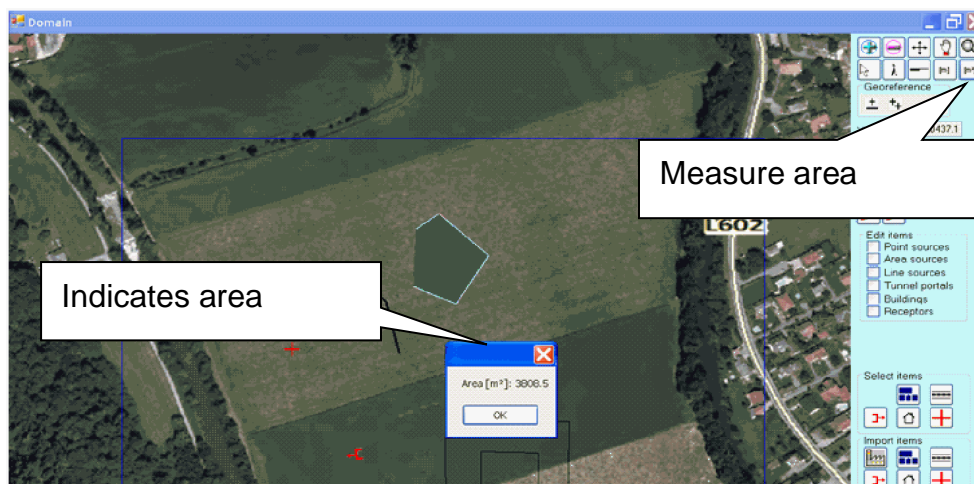


17.3. Measure areas

Menu: "Tools – Area measurement"

Toolbox: 

Corner points of the area-polygon to be measured can be set by left-clicking on the map. Measurements are finished with a right-click on the final corner point or by pressing the ESC key.



17.4. Absolute heights

The height of point sources, area sources, buildings and walls can be defined as height above ground level (standard value) or optional as height above sea (checkbox “Absolute Height”).

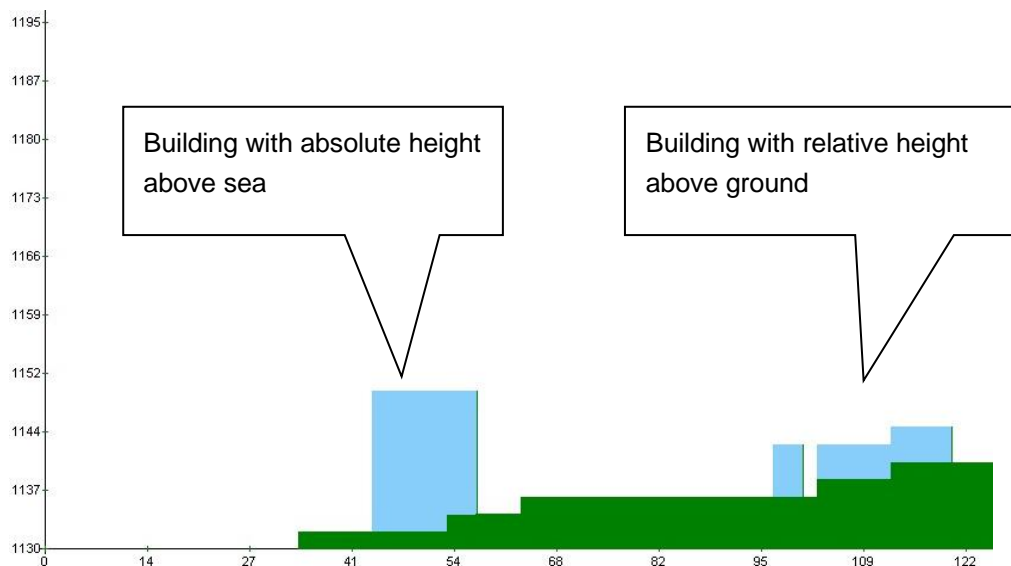
The option absolute height might be useful to digitize bridges.

If the absolute height is lower than the topography surface, the item is automatically set to the ground level by the GRAL computation core. If you would like to import items with absolute heights, set the height data at the shapefile to negative absolute values or use the corresponding checkbox at the import dialog.

Internal, the absolute heights are coded as negative values. Usually the absolute value is displayed. At the search, select function (see chapter 11.13) the negative value is shown. That allows the user to filter all items with absolute heights.

Absolute heights will force an error at the GRAL computation core if you try to compute flat terrain without topography (see chapter 17.4).

The difference between absolute and relative heights is shown in the following section view at an ascending slope.



17.5. Cell heights

While editing a project, the cell height of a GRAMM, or if available the GRAL topography is displayed in the right corner of the menu bar.

To visualize the GRAL topography while editing, it is possible to load the GRAL topography inside the GIS window. Use the menu entry “File – Import – Original GRAL topography”.

Although you see the GRAL cell heights, the section view (based on the file “GRAL_topography.txt”) is available, if the GRAL computation has been started and the 1st situation is computed.

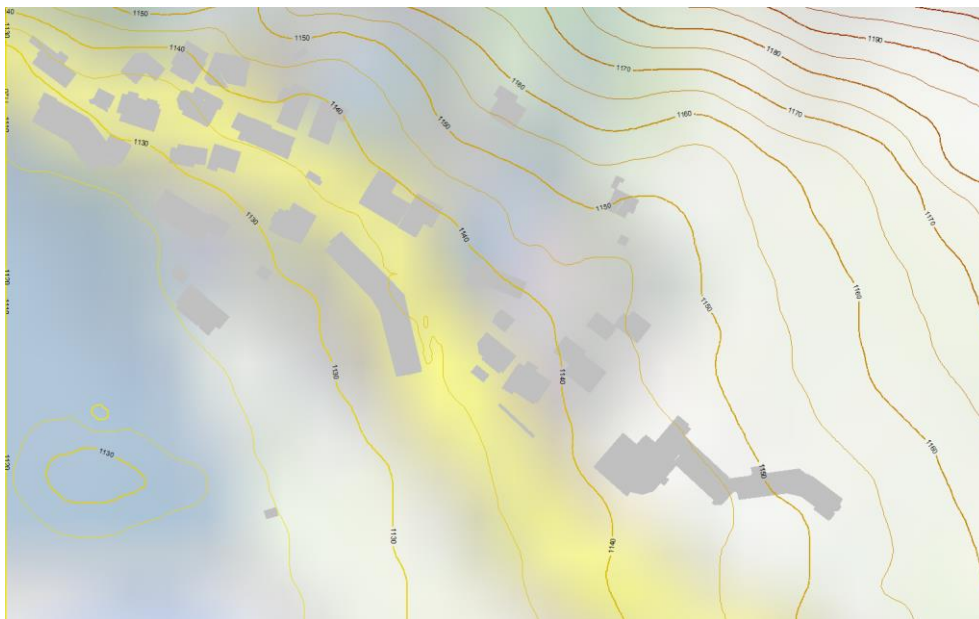
17.6. Import GRAL topography

Use the GIS-form menu entry “File – Import – Original GRAL topography” to import a detailed GRAL topography, corresponding to the GRAL flowfield raster.

The cell height is visualized in the right corner of the menu bar (see chapter 17.5).

17.7. Edit and visualize the GRAL topography

As soon as the 1st situation of the GRAL calculation is finished, it is possible to load the file “GRAL_Topography.txt” as a contour map (chapter 15.6) as indicated in the next figure.



If a GRAL topography is available, it is possible to visualize the topography in a three-dimensional view, using the 3D button or the menu entry “View – 3D view”.

Use the menu entry “Tools – Modify GRAL topography” to fill or strip parts of the topography. This function is useful to digitize e.g. landfills, gravel quarries or mounds.

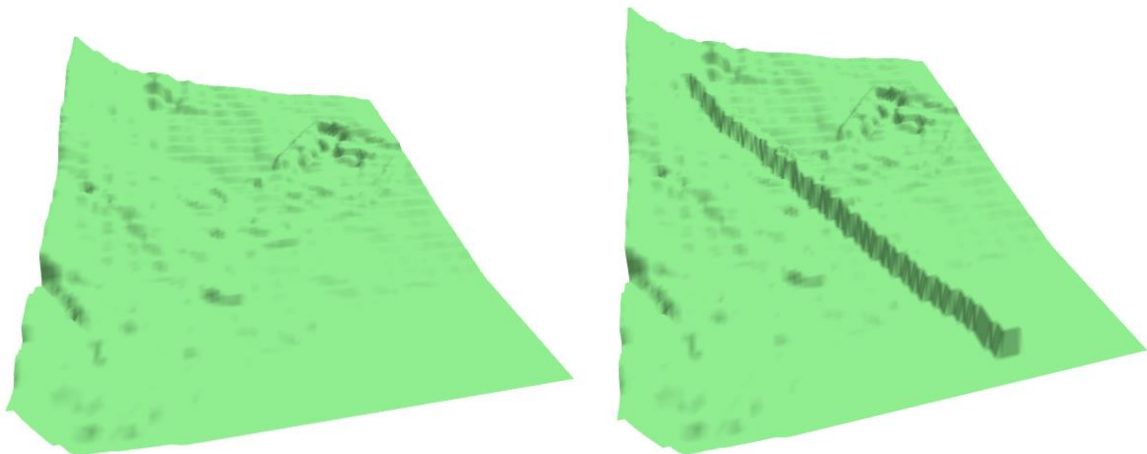
The function operates like a “drawing pen” for adding or ablating a relative height or set an absolute height at the selected cells. The pen-size corresponds to the number of cells around the mouse cursor affected by this tool.

The absolute height affects cells just once, if the left mouse button is pressed. That means, if you add +5 m and “paint” several times at the same cell – keeping the mouse button pressed – only 5 m are added. If you click on a cell multiple times (release the mouse button while painting) the height changes several times up to the Hmin/Hmax parameter value.

Using the “absolute height” setting changes the cell height to the specified value (multiple clicks do not add up).

The absolute height can be limited to a minimum or maximum value by the optional parameters Hmax and Hmin.

The following example shows the GRAL 3D view of the upper example (left figure) and a modified topography (right figure) including a soil wall with a relative height of + 10 m and a maximum height of 1180 m.



Smoothing topography is possible using the menu entry “Tools – Low pass filter” using a 5-point low pass filter. Discard all manual changes using the menu entry “Tools – Restore GRAL topography”.

Use the menu entry “Tools – Save GRAL topography” to save the modified topography. It is not possible to discard the changes, after the modified topography has been saved.

17.8. Moving (archiving) a project to a different directory

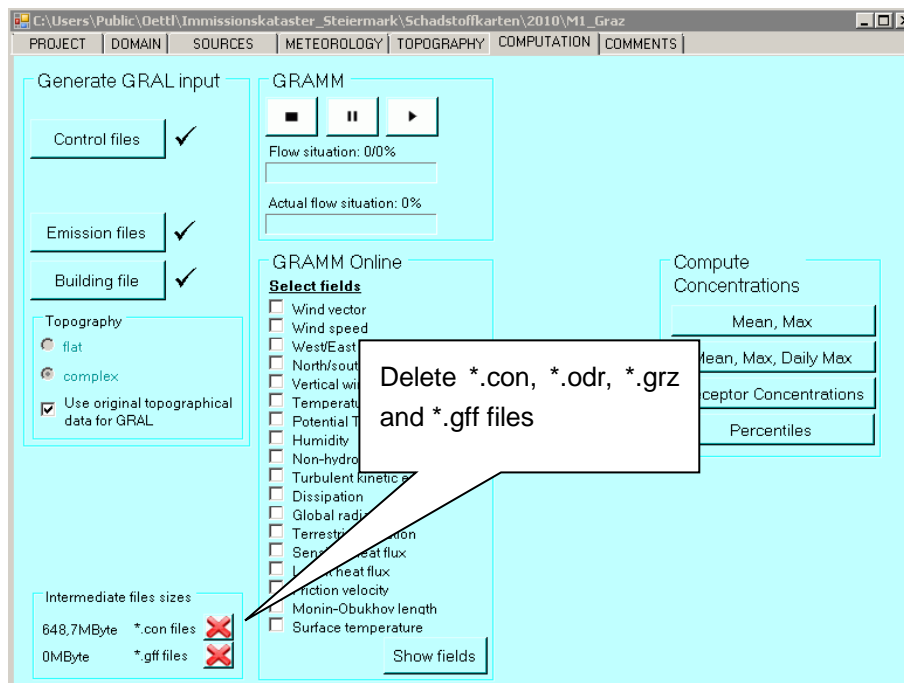
Often, projects are archived after a certain while and moved to a different directory. In case that such a project is opened again, data stored by the GUI by absolute paths, cannot be opened. The GUI will ask automatically for the new paths of such data. Just navigate with the automatically launched explorer to the place, where the required data is stored.

The following data might be concerned:

- All kinds of maps (Shape-Files, geo-referenced pixel graphics)
- Wind field libraries
- Sometimes concentration maps, if they are not stored in the default directory “Maps”

17.9. Archiving a project after finishing all work

GRAL saves concentration- and optionally flow field files as intermediate files in the subdirectory “Computation” (*.con and *.gff-files). These files occupy often an enormous amount of disk space. To free the disk readily, just press the buttons as indicated in the following figure.



A locked GRAL project will be unlocked immediately, if all *.con or *.grz files are deleted.

17.10. Invalid numerical inputs

Numerous number inputs are validated while the input process. If a number is not valid, the background color of the input field changes to yellow.

Since the GUI version 19.01, inputs are either “numerical up-downs” or text fields, using the installed local culture.

For example, in German culture, the “,” is the decimal separator, the “.” a digit group separator.

In German culture, a value like “25.5” is parsed as 255, but a “25,5” is parsed as 25,5. With German culture settings it is possible to write “1.526,6”, this will be parsed as 1526,6.

Miscellaneous

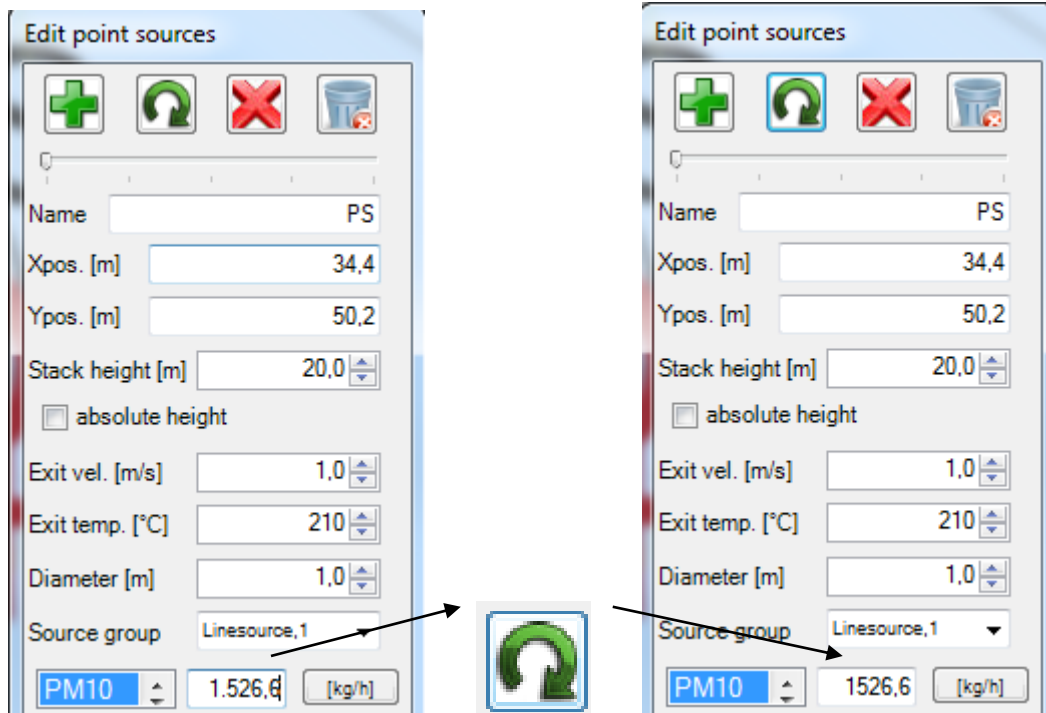
There are two strategies to avoid invalid inputs:

Validation during the input process

The screenshot shows the 'Edit point sources' dialog box. It contains various input fields for defining a point source. The 'Odour' field is highlighted in yellow and contains the text '5,,6', which is an invalid input due to the presence of two commas. A callout box with the text 'Invalid input (two commas)' points to this field. Other fields include Name (PS), Xpos. [m] (34,4), Ypos. [m] (50,2), Stack height [m] (20,0), Exit vel. [m/s] (1,0), Exit temp. [°C] (210), Diameter [m] (1,0), and Source group (Linesource,1). Below these are three rows for pollutant concentrations: Odour [MOU/h], Bioaers [kg/h], and NOx [kg/h].

Field	Value	Unit
Name	PS	
Xpos. [m]	34,4	m
Ypos. [m]	50,2	m
Stack height [m]	20,0	m
absolute height	<input type="checkbox"/>	
Exit vel. [m/s]	1,0	m/s
Exit temp. [°C]	210	°C
Diameter [m]	1,0	m
Source group	Linesource,1	
Odour	5,,6	[MOU/h]
Bioaers	0,5	[kg/h]
NOx	0	[kg/h]

Manual check



If you hit the „store&reload“ button, the inputs are parsed and reloaded. If the process was successful, you see the input value in your culture (without digit group separators), otherwise the input will be 0.

Because of these changes since version 19.01, it is possible to use exponential numbers, like „2,5E-3“.

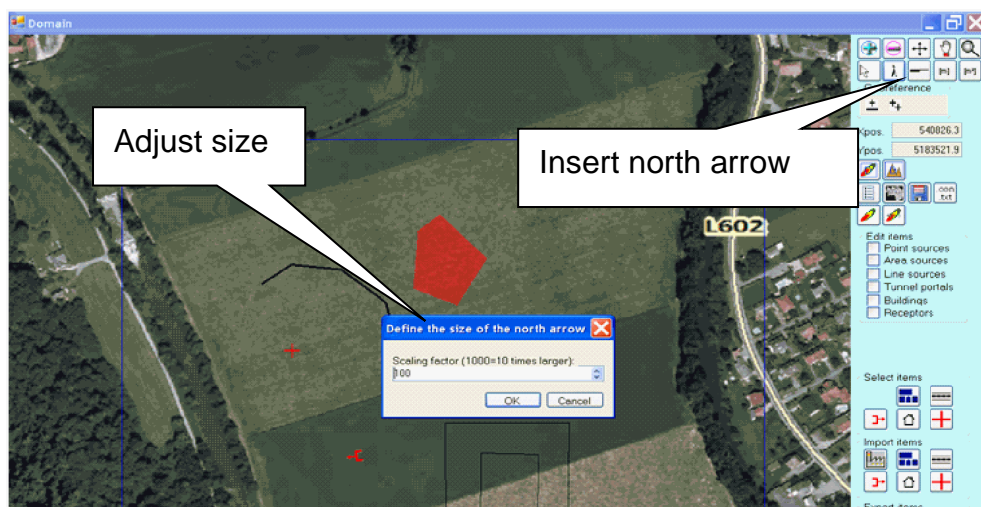
18. Further map features

18.1. North arrow

Menu: "Edit – North arrow"

Toolbox: 

After pressing the button for inserting a north arrow (see next figure), an input field opens, which asks for the size of the north arrow (default = 100 %). The position can be adjusted by left clicking on the map. If the position or the size should be changed, just click again on the button for inserting a north arrow. There is no way to change the type of arrow.



18.2. Scale bar

Menu: "Edit – Scale bar"

Toolbox: 

After pressing the button for inserting a scale bar, an input field opens, asking for the number of sections of the scale bar. The position of the scale bar can be adjusted by left clicking on the map. If the position of a scale should be changed, repeat this procedure. The length, font type, and color can be adjusted by opening the layout manager (see chapter 11.4). Just double click on the scale bar and the layout manager opens.

18.3. Saving a map as pixel graphics

Menu: "Export – Save map to disk"

Toolbox: 

It is possible to save the map as seen on the screen in various pixel graphic formats (e.g. gif, bmp, etc.). Click on the save button as indicated in the next figure and name the map to be saved. Afterwards a user will be asked for the desired resolution. Font sizes may vary according to the chosen resolution.



18.4. Copy a map to the clipboard

Menu: "Export - Copy map to the clipboard"

Toolbox: 

Using this menu entry or the keyboard sequence Control-C the actual map section is copied to the clipboard.

19. Appendices

19.1. Methodology „All in all out stables“

Animal farms applying the so-called “all in all out system” do not emit odour at a constant rate but odour emissions increase from the beginning of each breeding cycle from zero until they reach a maximum E_{\max} in the end. This is accounted for by assuming a linear increase in odour emissions for a breeding cycle. Indeed, emissions will rather follow an exponential function than a linear one. Assuming the latter relationship gives in the following frequency of a particular odour emission during a breeding cycle:

$P = \frac{1}{T_{\text{Zyklus}}}$ Normalized frequency of a particular odour emission E_{Geruch} at any day during the breeding cycle

T_{Zyklus} : Cycle duration in days

$$E_{\text{Geruch}} = E_{\max} \frac{\text{MAX}[t - T_{\text{leer}}, 0]}{T_{\text{Zyklus}}}$$

t : Actual time of the cycle in days (starting from zero to T_{Zyklus}).

T_{leer} : Time between two breeding cycles without animals in days.

The probability of an odour hour P_{Geruch} without taking into account other odour sources in the surroundings, is given by the integral of all frequencies, where the odour emission results in corresponding odour concentrations in the surroundings C_{geruch} above a pre-defined threshold $C_{\text{Geruchsschwelle}}$:

$$P_{\text{Geruch}} = \sum_{T_{\text{Zyklus}}} P \cdot \Delta t \quad \forall C_{\text{geruch}} \geq C_{\text{Geruchsschwelle}}$$

Δt is the time increment in days.

If there are other odour sources (e.g. further animal farms) in the surrounding, background odour concentrations must be considered in the following way:

$$P_{\text{Geruch}} = \sum_{T_{\text{gesamt}}} P \cdot \Delta t \quad \forall (C_{\text{geruch}} + C_{\text{Vorbelastung}}) \geq C_{\text{Geruchsschwelle}}$$

At last, the frequency of annual odour hours is given at any location in the model domain by summing up all probabilities of odour perceptions for each hour.

19.2. Methodology “compost works”

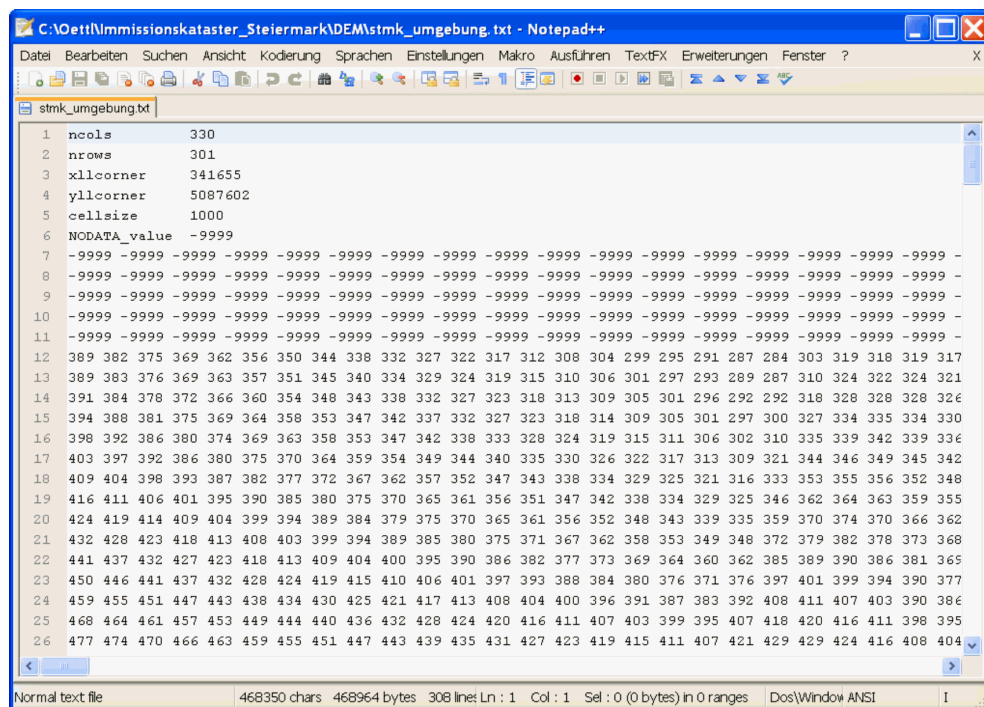
Odour emissions from open compost works without any artificial ventilation are governed by various processes with corresponding frequencies of occurrence. As an example, the following table illustrates

computed odour emissions using the German emission model GERDA. As can be seen there are three different frequencies for the individual processes that can be summed up giving three emission strengths as indicated by the different colors in the table below.

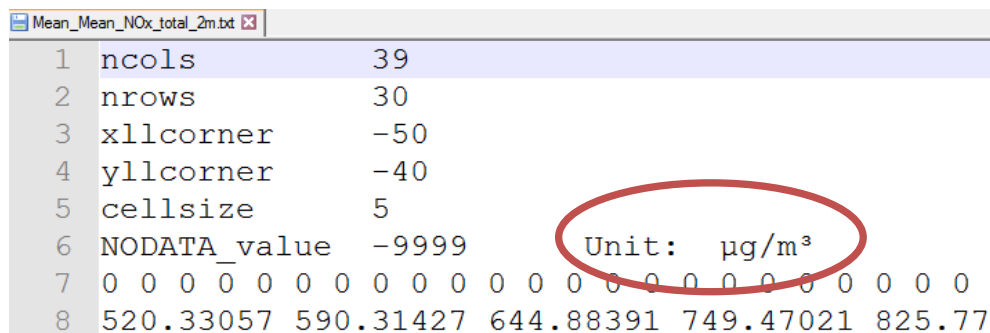
For every receptor point in the model domain the resulting odour concentration from all of these three different emissions strengths is calculated. There exist seven different ways to combine these three emission strengths and frequencies. The possibility of an odour hour for a certain hour of a year (as defined by the odour threshold concentration – e.g. 1 OU/m³) is set to the maximum frequency out of the seven combinations, each owing a different frequency.

Activity	Emission factor	Emission	Frequency
	[OU/(m ³ s)]	[MOU/h]	[%]
Delivery of input material	8,5	0,3	85
Pre-processing the rotting	10,0	0,3	85
Positioning the rotting	10,5	0,3	85
Main rotting	0,5	0,2	100
Tramming the rotting	1,7	6,8	13
Finalizing compost	1,0	0,0	85
Outside storage	0,17	0,2	100
Fugitive sources	--	0,0	85
Average		2,05	

The ECDL ASCII format is the most important input format for displaying QIC data with the QIC 5.0.



Mean_Mean_NOx_total_2m.bt



19.3.2 Topography and landuse data

The GUI supports the import of ESRI ASCII files with topography or landuse information.

Accepted row delimiters are the characters blank, tab, and semi-colon. The comma is allowed as delimiter at the landuse files and for GRAL topography files, if the comma is not used as decimal separator.

For best compatibility all data values should use the dot as decimal separator. Digit group separators (for thousands, millions) are not supported.

19.3.3 Meteorological files *.met

The format is described in section 14.1. Here we give some additional technical information.

Accepted row delimiter characters are the blank, comma, tabulator or semi-colon characters, allowed decimal separators are the dot or the comma. The user can set the delimiter and decimal character at the import dialog.

The date and time format must use the colon, hyphen or dot character to separate day, month and year or hour and minute.

Examples: "01:01:2010", "01-01-2010", "01.01.2010" and „12:00“, „12-00“, „12.00“

Accepted formats for the year are 4 and two digits.

For best support of all features, it is recommended to add a 3-line header with the coordinates of the measurement station (X, Y: coordinate, Z: relative height above ground).

Example header:

```
//X=80079
//Y=233949
//Z=10
01.01.2013,00:00, 2.1,218,6
01.01.2013,01:00, 3.2,204,4
.....
```

19.3.4 Emission_Mod_Diurnal.txt and Emission_Mod_Seasonal.txt

Accepted row separators are the tabulator or the comma character. All values must use the dot as decimal separator.

19.3.5 DepositionSettings.txt

The rows are tabulator separated. Empty entries are removed. All values must use the dot as decimal separator.

19.3.6 Emissions_timeseries.txt

Accepted separators are the blank, colon, hyphen, tabulator or semi-colon characters. Empty entries are removed.

For best compatibility all data values should use the dot as decimal separator. Digit group separators are not supported.

19.3.7 Precipitation.txt

The accepted separator is the tabulator character. All data values must use the dot as decimal separator. Digit group separators are not supported.

The number of entries in that file must match the number of entries of the file mettimeseries.dat.

19.4. Mouse- and key-references

GIS interface:

+	key	Zoom in
-	key	Zoom out
Cursor	keys	Move the map up/down/left/right
Delete	key	Deletes the selected item
Delete	key and shift key	Deletes all selected items
ESC	Key	Cancel a copy process, cancel a length or area measurement
Left	mouse button	Set a point
Right	mouse button	Set the endpoint of a polyline (e.g. line source), a polygon (e.g. area source, building) or the portal source
		Set a new coordinate of a point
Left	mouse button + shift key	Set a point and the coordinates manually
Right	mouse button + shift key	Set an endpoint and the coordinates manually
Left	Mouse button + ctrl key	Select a point of a line- or area source or a building
Middle	Mouse button	Keep the button pressed and move the mouse to move the map
Mouse	scroll-wheel	Zoom in or out
Ctrl 1		Edit point sources
Ctrl 2		Edit area sources
Ctrl 3		Edit line sources
Ctrl 4		Edit portal sources
Ctrl 5		Edit buildings
Ctrl 6		Edit receptor points
Ctrl 7		Edit walls
Ctrl 0		Open object manager
Ctrl L		Load a contour map

Appendices

Ctrl F1	Select point sources
Ctrl F2	Select area sources
Ctrl F3	Select line sources
Ctrl F4	Select portal sources
Ctrl F5	Select buildings
Ctrl F6	Select receptor points
Ctrl F7	Select walls
Ctrl C	Copy map
Ctrl L	Load contour map
Ctrl S	Search items

Section drawing

+	key	Vertical zoom in
-	key	Vertical zoom out
+	key numerical keypad	Wind-vector zoom in
-	key numerical keypad	Wind-vector zoom out
+	button and Shift	Wind-vector zoom in
-	button and Shift	Wind-vector zoom out
+	button and Ctrl	Horizontal zoom
-	button and Ctrl	Horizontal zoom

3D module

+	key	Zoom in
-	key	Zoom out
+	key and shift key	Higher Focal length
-	Key and shift key	Lower Focal length
Cursor up	key	Rotate about the X-axis
Cursor down	key	Rotate about the X-axis
Cursor left	key	Rotate about the Y axis
Cursor right	key	Rotate about the Y axis
Cursor up	Key and shift key	Shift up
Cursor down	Key and shift key	Shift down
Cursor left	Key and shift key	Shift along the X-axis
Cursor right	Key and shift key	Shift along the X-axis
Cursor left	Key and control key	Shift along the Z-axis
Cursor right	Key and control key	Shift along the Z-axis

19.5. The match to observation algorithm

The goal of the matching algorithm is to find the best fit between computed wind fields and measurement data for a given number n of observation stations.

Required input data to be provided by the user:

- Location of each observation station
- Corresponding time series of observed meteorological data
- Weighting factor f_n for each observation station
- Direction weighting factor fd_n for each observation station
- Optimization method: vector or component
- Usage of local stability classes (SC_{local})
- Remove outliers (RMO)
- Concatenation limit ($Conc$)

In a first step, the algorithm reads the computed wind velocity (w_{in}), wind directions (d_{in}) and the wind vectors (u_{in}, v_{in}) for each computed weather situation i and the position of each observation station n .

If the variable SC_{local} is set, the computed local stability class at the position of the observation station, otherwise the global stability class of the initial wind field is used for the value of S_{in} .

Secondly, the algorithm synchronizes the time series of all observation stations, as meteorological data doesn't necessarily need to be provided without data gaps for the time period in consideration.

Next, the best-fitting computed wind field is picked up from all available wind fields by the following algorithm:

The measured wind data for each time step M of the time series is read from the corresponding "*.met" files and the measured values of wind velocity (w_{Mn}), wind direction (d_{Mn}), wind vectors (u_{Mn}, v_{Mn}) for each observation station n and the stability class of the first observation station are extracted.

For each time step M an error value err_i is calculated for each computed weather situation i and observation station n by the following procedure:

Error value for the stability class:

If the mode "SC1/7 weighted stronger" is activated

$$\text{If } S_{M1} < 3 \vee S_{M1} > 5$$

$$err_{SCL} = |S_{M1} - S_{i1}| * 200$$

$$\text{If } S_{M1} > 2 \wedge S_{M1} < 6$$

$$err_{SCL} = \max\{0, (|S_{M1} - S_{i1}| - 1)\} * 200$$

If the mode "SC1/7 weighted stronger" is deactivated

$$err_{SCL} = \max\{0, (|S_{M1} - S_{i1}| - 1)\} * 200$$

If the optimization method is set to “Component”:

$$err_i = \sum_{o=1}^n \left(err_{SCL} + 100 * f_o * \left(fd_o * \max\{0 | |d_{io} - d_{MO}| - 12 \}^{1.8} + \frac{|w_{io} - w_{MO}|}{\max(0.35 | \min\{w_{io} | w_{MO}\})} \right) \right)$$

with $0 \leq fd_o \leq 10$

If the optimization method is set to “Vector”:

$$err_i = \sum_{o=1}^n \sqrt{[(u_{MO} - u_{io}) * 400]^2 + [(v_{MO} - v_{io}) * 400]^2} * f_o^2 + err_{SCL}^2$$

The lowest value err_i indicates the best-fitting computed weather situation for the time step M .

If the option “remove outliers” is activated (experimental), the inner part of the sum (here called err_{in} , err_{min} is the minimum error value for each observation station) is filtered before the sum

$$\sum_{o=1}^n$$

will be computed, in the following way:

$$if (err_{in} < err_{min} * 2) \rightarrow err_i += err_{in}$$

and the error value for each computed situation i is normalized by the number of summarized observation stations.

As soon as the best-fitting flow field has been found, the frequency of this situation is incremented by the fraction of time represented by one-time step of the complete time series. The best fitting situation is written to the file “mettimeseries.dat” using the date and time stamp from the measurement data and the wind direction, wind speed and stability class from the initial wind field.

After the matching procedure has been finished for the complete time series, the computed situations are sorted by their frequency and a new file “meteopgt.all” is generated.

If the postprocessing - option “Concatenation limit” is set, the algorithm searches for a well-fitting situation for each wind field with a frequency lower than the value *Conc* in permille (at a time series of one year, 1 ‰ represents 9 hours). A situation is considered “well-fitting”, when it has the same initial wind speed, same initial stability class, and an initial wind direction within an allowed range of $\pm 25^\circ$. The underlying idea is that initial wind fields, with almost the same initial conditions, produce quite similar wind fields within the model domain and, therefore, it is possible to reduce the number of weather situations without losing too much accuracy. This option leads to an updating of the “mettimeseries.dat” and “meteopgt.all” files again.

19.6. GUI Error messages

Here you can find an excerpt of the GRAL GUI error messages and small descriptions of their meanings. Most of them are straightforward.

Sometimes the application seems to have no response without a “no response” message. In such cases it might happen, that a message box or a notification message is waiting for a response, but the box is hidden by another (topmost) window. In such cases try to switch between the windows using the “Alt” + “Tab” keys or try to quit the message using “Alt” + “O” keys or “Alt” + “F4” keys.

Modulation

"Error when reading modulation factor for ..."

One or more values at the modulation grid are not valid

"There must remain at least five modulations."

You tried to delete a modulation

Line sources / Portal sources / Point sources

"Only integer numbers 1-99 are allowed"

You may not define source group numbers > 99

"No source group defined - data not imported"

Select a source group before importing data

"Problems when reading temperature value - set to zero"

Invalid temperature

Source groups

"This number has already been assigned"

You tried to save a source group-number that already exists

"There must remain at least one source group."

You tried to delete the last remaining source group

Match function

"Error when synchronizing times of meteo stations"

The used meteo station have different timestamps

"Point is outside GRAMM domain"

The point for the meteorological reference station must be within the GRAMM domain

"Problems when reading selected met-file."

The selected meteorological station has an invalid dataset

"No station selected - click again on the line to be removed"

To add stations no line of the data grid must be selected, to remove a station, exactly this line must be selected

"No data set selected - click on a line in the table"

You tried to view a wind rose, but no line of the data grid was selected

Georeferencing

"Invalid Input values. Georeferencing failed."

The stretch factor for the bitmap is unrealistic high or low

Load contour map / Section view

"Blank raster dataset"

The computed contour map has only null values

"Unable to open, read or process the data"

Raster data are not valid

"Error reading wind fields"

The GRAMM or GRAL wind fields *wnd or *gff does not fit to the project or they are not available

Dispersion selection

"No situation selected"

Select a situation

Topography tab

"Unable to generate land use file"

Can't write the land use file I/O error

"Unable to read file Landuse_Default.txt"

I/O Error

"Could not copy GRAMM grid 'ggeom.asc'."

"Could not copy GRAMM control file 'GRAMM.geb'."

"Could not copy GRAMM control file 'IIN.dat'."

"Could not copy GRAMM meteo file 'meteopgt.all'."

"Could not copy GRAMM meteo file 'mettimeseries.dat'."

"Could not copy GRAMM landuse file 'landuse.asc'."

"Could not copy GRAMM landuse file 'windfeld.txt'."

This files can't be copied from the original import folder I/O error or the original folder is not valid

Generating GRAMM configuration files

"Number of cells in x-direction to small"

"Number of cells in y-direction to small"

The GRAMM Domain is to small or the GRAMM grid to large

"Selected area is to far in the ..."

The GRAMM Domain area doesn't fit to the surface or landuse datasets

"Topography file is too large. Exceeding available memory space of this computer"

The grids are to large, the needed memory can't be accessed

"Height of the model domain is too low. Increase vertical stretching factor or increase the number of vertical grid points or increase the height of the first layer"

The top of the model domain needs to be larger than 3 times the maximum elevation

"Unable to generate GRAMM grid"

Error reading or writing ggeom.asc

Meteorology tab - read and show wind data

"Wind speed implausible - check line number"

"Wind direction implausible - check line number"

"Stability class implausible - check line number"

Check your wind data

"Start- and end time must be different values"

Set different start and end values of the time span for the wind plot

"Do you really want to overwrite the existing meteorological files?"

If classified wind data already exists, this message asks, if these data should be overwritten.

Be careful, overwriting existing wind data may cause in serious problems with your project, if simulations have already been done

Appendices

"Wind speed needs to be larger"

The (or some) classification values are too low – increase the low values

"Input needs to be a valid number"

The classification data can't be interpreted as a valid number – check the values

"Could not copy meteorological input file."

The original meteorological input (used for the classification) file can't be copied to the project

GRAL Settings tab

"Roughness length needs to be larger than zero"

"Roughness length is extremely large!"

"Roughness length needs to be a valid number"

"Latitude needs to be larger than -90 deg."

"Latitude needs to be smaller than 90 deg."

"Latitude needs to be a valid number"

"Horizontal slice needs to be larger than zero plus half of the chosen vertical layer thickness"

"Horizontal slice is quite high above the ground level!"

"Horizontal slices need to be valid numbers"

Check the values, the messages give the hints

Computation tab

"GRAL simulation stopped or interrupted"

The simulation is finished or has been interrupted by the user or an error

"GRAMM simulation stopped or interrupted"

The simulation is finished or has been interrupted by the user or an error

Generating GRAL input files

"GRAL Domain is outside the borders of the selected topography file"

Check your GRAL Domain area or the topography file

"Error when reading topography file."

The topography file is corrupt, too small or doesn't fit to the GRAL domain

"Unable to write file 'GRAL_topofile.txt'"

I/O Error

"Topography file is too large. Exceeding available memory space of this computer"

The grids are too large, the needed memory can't be accessed

Receptor concentrations

"File zeitreihe.dat not found"

This file might be needed for the computation of time series

"File GRAL_Meteozeitreihe.dat not found"

This file should exist after a finished GRAL-computation with receptors

High Percentiles

"Can't read meteopgt.all"

This file is needed for GRAL computations and should be in your project computation -folder

"Can't read mettimeseries.dat"

This file sets the basis data for high percentile computations

Some I/O Errors - Read Images and Files

"Can't open this project"

Does this path exist? Do you have read and write access to this path? Is there enough free space on your disk?

"Could not read image"

"Error when reading file"

"Error when reading modulation factor for ..."

"Error when reading modulation factor in line ..."

"Error when exporting line source data"
"Error when exporting area source data"
"Error when exporting point source data"
"Error when exporting receptors"
"Unable to read wind field nr. ..."

"Error when reading point source data in line number "
"Error when reading shape file"
"Error when reading area source data in line number "
"Error when reading buildings in line number "
"Error when reading .shp file"
"Error when reading or filtering line source data"
"Error when reading or filtering portal source data"

"Unable to read vector file."

"Error when importing .tif file"
"Error when reading world file or when loading Map"

"Problems when reading selected met-file."