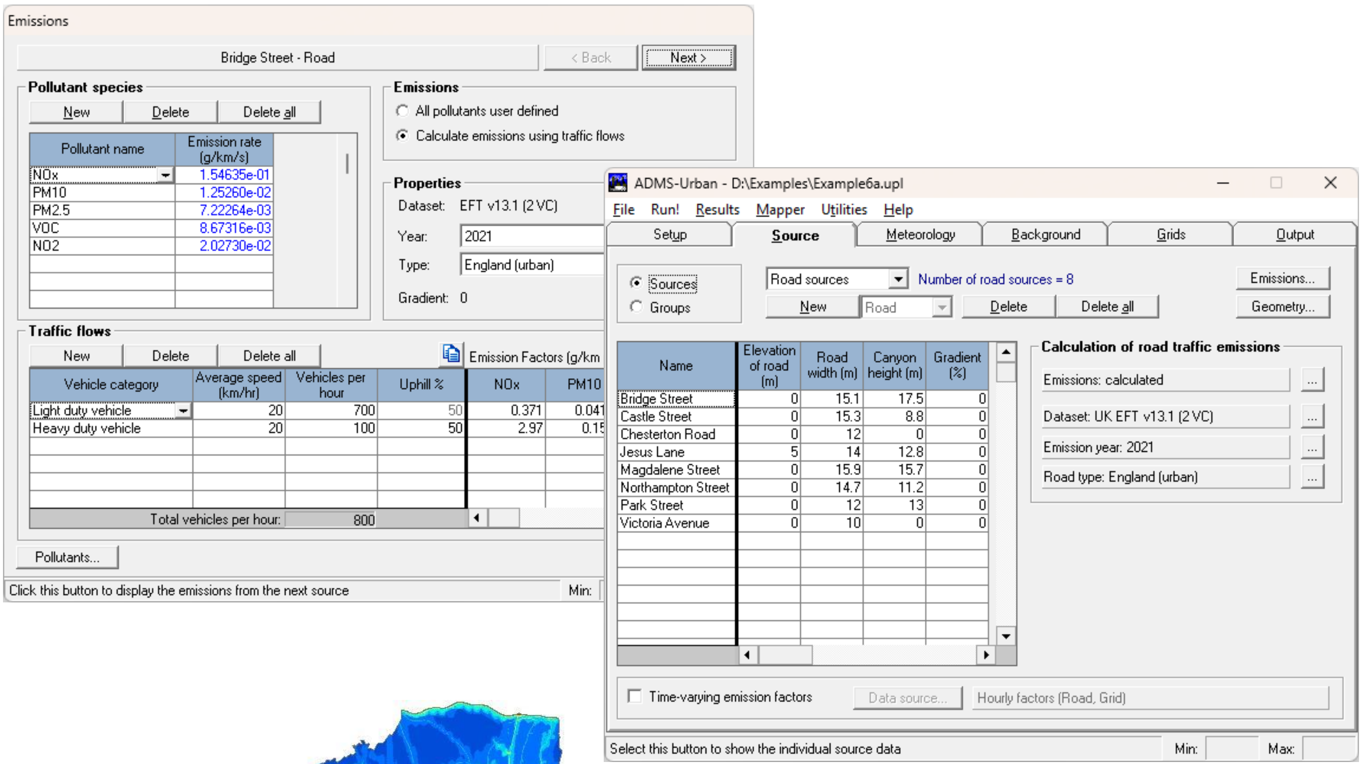
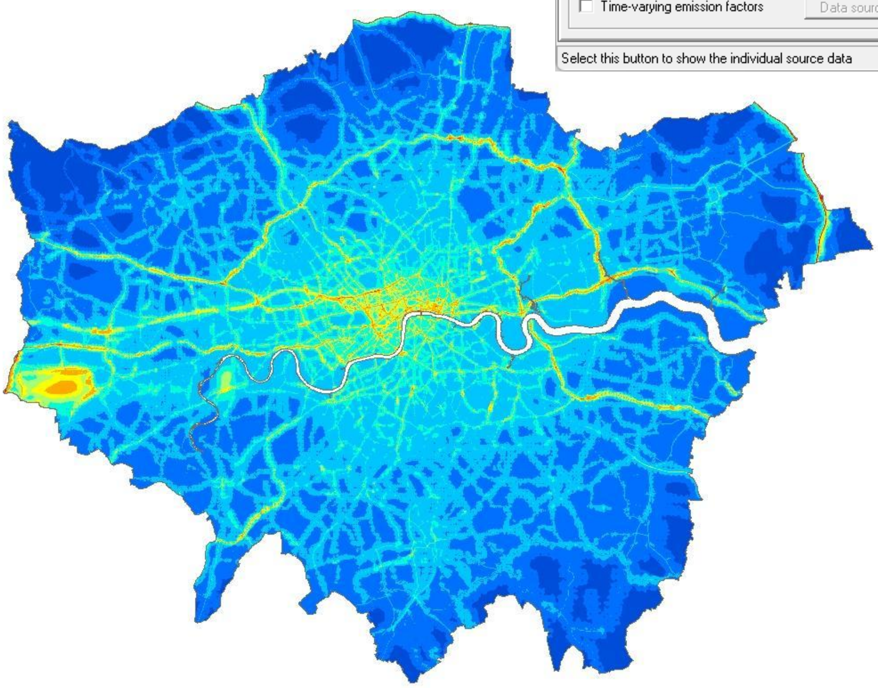
ADMS-Urban

Urban Air Quality Management System Version 5.1





User Guide CERC

ADMS-Urban

Urban Air Quality Management System

User Guide

Version 5.1

July 2025

Cambridge Environmental Research Consultants Ltd.
3, King's Parade
Cambridge
CB2 1SJ
UK

Telephone: +44 (0)1223 357 773
Email: help@cerc.co.uk
Website: www.cerc.co.uk

Contents

SECTION 1 Introduction

1.1 About ADMS-Urban

ADMS-Urban, the most comprehensive version of the Atmospheric Dispersion Modelling System (ADMS), is a PC-based model of dispersion in the atmosphere of pollutants released from industrial, domestic and road traffic sources in urban areas. ADMS-Urban models these using point, line, area, volume and grid source models. It is designed to allow consideration of dispersion ranging from the simplest scenarios (e.g. a single isolated point source or a single road) to the most complex urban scenarios (e.g. multiple industrial, domestic and road traffic emissions over a large urban area).

ADMS-Urban is being used across the world for air quality management and assessment studies of complex situations in urban areas, cities, towns and close to motorways, roads and large industrial areas. The model is distinctive in its ability to describe in detail what happens on a range of scales, from the street scale to the city-wide scale, taking into account the whole range of relevant emission sources. Typical applications of the model include the following: developing and testing policy on air quality; the development of air quality action plans; investigation of air quality management and planning options for a wide range of sources including transport sources; source apportionment studies; air quality and health impact assessments of proposed developments and use of the model for the provision of detailed street-level air quality forecasts.

ADMS-Urban is supplied with a Mapper that can be used to visualise, add and edit sources, buildings and output points and to view modelled concentrations. ADMS-Urban also links to the third-party software package Surfer, a contour plotting package, for easy and effective display of results. A separate User Guide is provided for using the Mapper. More information about using Surfer with ADMS-Urban is provided in Sections 6.2 and 7.3.

1.2 Model features

ADMS-Urban has a number of distinctive features that are summarised below and described in detail in the subsequent sections of the User Guide. These are:

- Versatility of applications such as: comparisons with National Air Quality Standards (NAQS), EU and/or WHO limits and guidelines; air quality action plans; traffic management planning; Low Emission Zones (LEZs); environmental impact assessments; ‘What if?’ scenarios; and future projections;
- Advanced dispersion model in which the boundary layer structure is characterised by the height of the boundary layer and the Monin-Obukhov length, a length scale dependent on the friction velocity and the heat flux at the surface. The ‘local’ Gaussian type model is nested within a trajectory model so that significant areas (e.g. greater than 50 km by 50 km) may be considered;

- A full range of explicit source types – road sources (each with up to 51 vertices) and industrial point, line, area and volume sources, which can be modelled simultaneously. With the aggregation of smaller sources into a grid source, this allows consideration of very large numbers of sources in model runs;
- Integrated basic and advanced street canyon models;
- An integrated model for dispersion of emissions from road tunnels;
- An integrated model for urban canopy flow field effects;
- The modelling of chemical reactions involving NO, NO_2 and Ozone, and generation of sulphate particles from SO_2;
- The calculation of emissions from traffic count data, using a database of up-to-date UK emission factors;
- Import/export from comma-separated value files, and import of data from EMIT, CERC's Emissions Inventory Toolkit software;
- An easy-to-use interactive graphical interface;
- The Mapper, a utility for visualising model input and output;
- Integration with the contouring package Surfer;
- A meteorological processor that calculates the boundary layer parameters from a variety of input data: e.g. wind speed, day, time and cloud cover, or wind speed, surface heat flux and boundary layer height. Meteorological data may be raw hourly values or statistically analysed;
- A non-Gaussian vertical profile of concentration in convective conditions, which improves accuracy by allowing for the skewed nature of turbulence within the atmospheric boundary layer that can lead to high surface concentrations near the source;
- The realistic calculation of flow and dispersion over complex terrain and around buildings;
- The modelling of concentrations in units of ou_E/m^3 for odour studies;
- The modelling of coastline effects.

ADMS-Urban can produce contour plots of pollutant concentrations that may be overlaid onto digital map data within the Mapper, a GIS package or in the plotting package Surfer. An X-Y plotting facility for concentration, deposition and plume parameters for a single point source is also included.

ADMS-Urban produces numerical output in comma-separated variable text file format, which may be viewed using a spreadsheet package such as Microsoft Excel, or using a text editor such as Windows Notepad.

1.3 About this User Guide

This *ADMS-Urban User Guide* is both a manual and a technical summary of the model.

Conventions

To make this User Guide simpler to use, certain conventions have been followed with regard to layout and style.

- ADMS-Urban interface controls are shown in bold **Arial** font, e.g. the **Grids** screen, click on the **Plot** button.
- Keyboard keys are shown in **bold**, e.g. press **Enter**.
- Directory and file names are shown in *italics*, e.g. *adms.exe*, *<install_path>\Data*.
- Tips and other notes are shown thus:

Think about the area you want to include in the calculation before specifying the output grid.

- Table and figure references are shown in **bold**, e.g. refer to **Table 3.2**, **Figure 2.1**.

SECTION 2 Getting Started

2.1 System requirements

This version of ADMS-Urban is supported on Windows 11 environment. Please visit our website[1] for the latest information regarding computer specifications, supported operating systems and third-party software.

2.2 Installation

The installation of ADMS-Urban is straightforward. It uses an Installation Wizard, which guides you through a short series of screens, collecting information on the user and installation parameters, before installing the software.

Please check first with your own IT personnel for company procedures for installing software.

There is a single installation process for ADMS-Urban, ADMS-Roads and ADMS-Airport. Therefore, if you already have version 5.1 of ADMS-Roads or ADMS-Airport installed on your PC, you will also have ADMS-Urban installed, and only need to install the ADMS-Urban model licence and create a shortcut to ADMS-Urban. Proceed as explained in Section 2.2.1.

If you have earlier versions of ADMS-Urban, ADMS-Roads or ADMS-Airport installed on your PC, you should uninstall these models before installing ADMS-Urban 5.1. Proceed as explained in Section 2.2.2 and then follow instructions given in Section 2.2.3.

If you do not have any of ADMS-Urban, ADMS-Roads or ADMS-Airport installed on your PC, you should follow the instructions in Section 2.2.3.

2.2.1 Using ADMS-Urban if ADMS-Roads or ADMS-Airport is installed

The abbreviation <install_path> *will be used in the rest of the User Guide to denote the directory in which ADMS-Urban, ADMS-Roads and ADMS-Airport is installed, for example* C:\Program Files (x86)\CERC\ADMS-Urban.

If the current version of ADMS-Roads or ADMS-Airport is already installed on your PC, please follow these steps:

Step 1 You have been provided with a unique licence file, *ADMS-Urban.lic*, by email, which is required in order to run the model. It is important that you install this licence file by copying the file *ADMS-Urban.lic* to the directory *<install_path>*. Alternatively, drag and drop the licence file into the ADMS-Roads/ADMS-Airport interface.

[1] www.cerc.co.uk/systemrequirements

Launching the interface and checking the licence details (through **Help, Licence Details***) will give the location of the licence currently being used.*

Step 2 To set up a shortcut to ADMS-Urban on your Windows desktop, browse to the *<install_path>\Support\Shortcuts* directory in Explorer, copy the ADMS-Urban shortcut and paste it on to your desktop.

You are now ready to use ADMS-Urban.

2.2.2 Earlier versions of ADMS-Urban, ADMS-Roads or ADMS-Airport

If you have earlier versions of ADMS-Urban, ADMS-Roads or ADMS-Airport installed on your computer, these should be uninstalled before installing ADMS-Urban 5.1.

To uninstall a previous version, log on as Local Administrator for the PC and right click on the Windows **Start** button. From the menu options, select **System** and then **Apps**. Select **Installed apps** option and search for the program name (ADMS-Urban, ADMS-Roads or ADMS-Airport) from the list of installed applications. Click on the three dots corresponding to the program name and select **Uninstall**.

After the uninstall is complete, it is advised to delete the installation directory and its remaining contents if it still exists, archiving any licence file(s) first if desired.

2.2.3 Installing ADMS-Urban 5.1

The following steps lead you through the ADMS-Urban installation process.

Step 1 Log on as Local Administrator for the PC.

Step 2 ADMS-Urban will have been supplied by download link. Unzip the downloaded *.zip* file to a local directory.

Try not to choose a local directory with an excessively long pathname, as this may cause issues with file pathnames being longer than the allowed maximum of 256 characters.

In Explorer, browse to this directory, right click on the file '*setup.exe*' and select **Run as administrator**.

The screen shown in **Figure 2.1** will be launched.

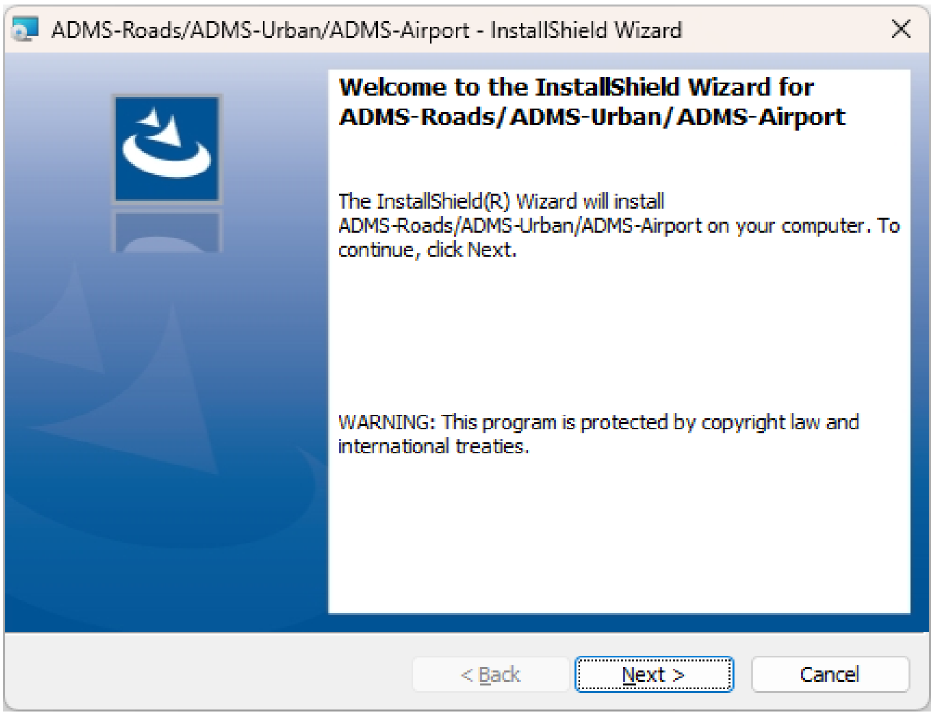


Figure 2.1 – The **Welcome** screen.

Step 3 Click **Next >** on the welcome screen. Select **I accept the terms in the licence agreement**, and click **Next >** in the **Licence Agreement** screen, if you accept the licence terms. The Customer Information screen is then displayed, as shown in **Figure 2.2**. If you do not accept the licence terms select **I do not accept the terms in the licence agreement** and click **Next >** to finish the install.

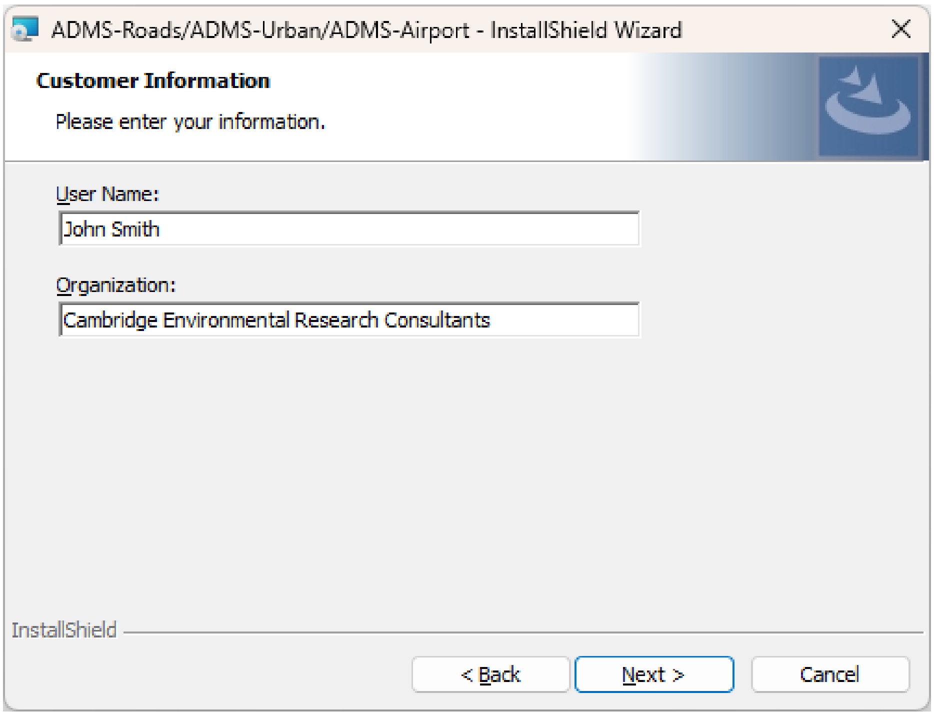


Figure 2.2 – The **Customer Information** screen.

Step 4 Enter your user name and organisation in the designated places. Click **Next >** to go to the Destination Folder screen, as shown in **Figure 2.3**.

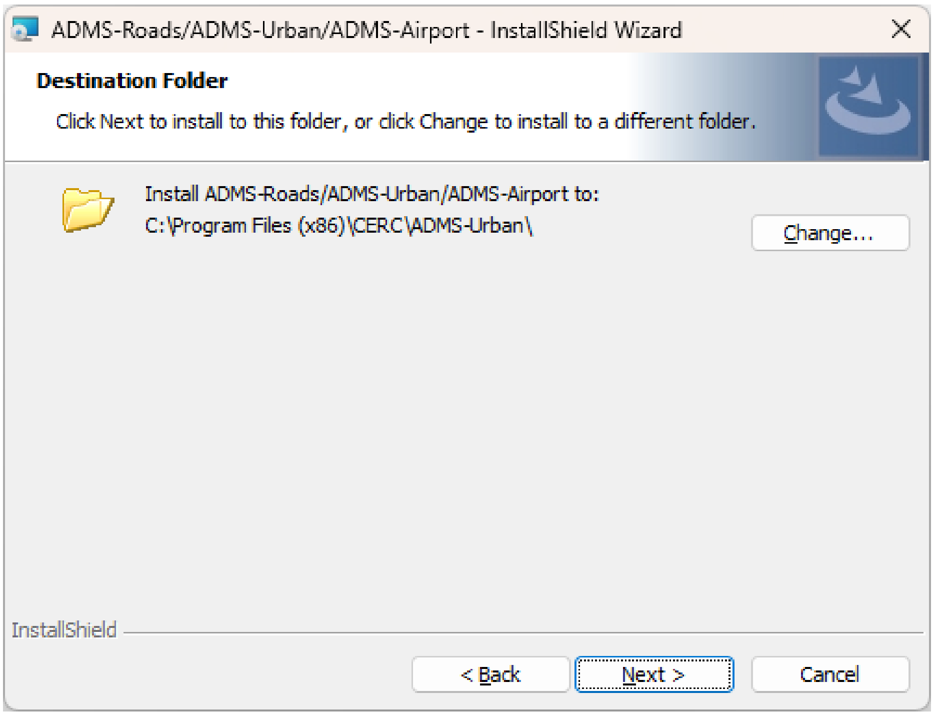


Figure 2.3 – The **Destination Folder** screen.

Step 5 You should select a drive with at least 3 GB of available disk space. If required, use the **Change...** button to select your own installation directory (**Figure 2.4**). Click **OK** to return to the Destination Folder screen.

Try not to choose an installation directory with an excessively long pathname, as this may cause issues with file pathnames being longer than the allowed maximum of 256 characters.

It is advised to always choose a new installation directory rather than, for example, the directory of a previously uninstalled version of the same software.

The abbreviation <install_path> *will be used in the rest of the User Guide to denote the installation directory you have chosen, for example* C:\Program Files (x86)\CERC\ADMS-Urban.

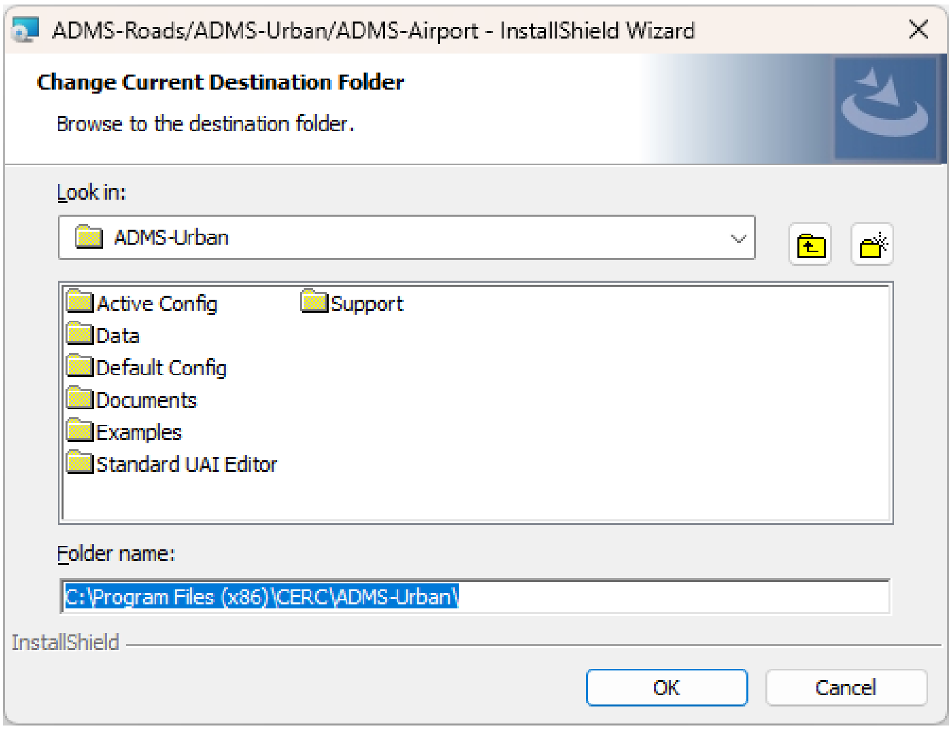


Figure 2.4 – The **Change Current Destination Folder** screen.

Click **Next >** to move to the **Ready to Install the Program** screen, as shown in **Figure 2.5**.

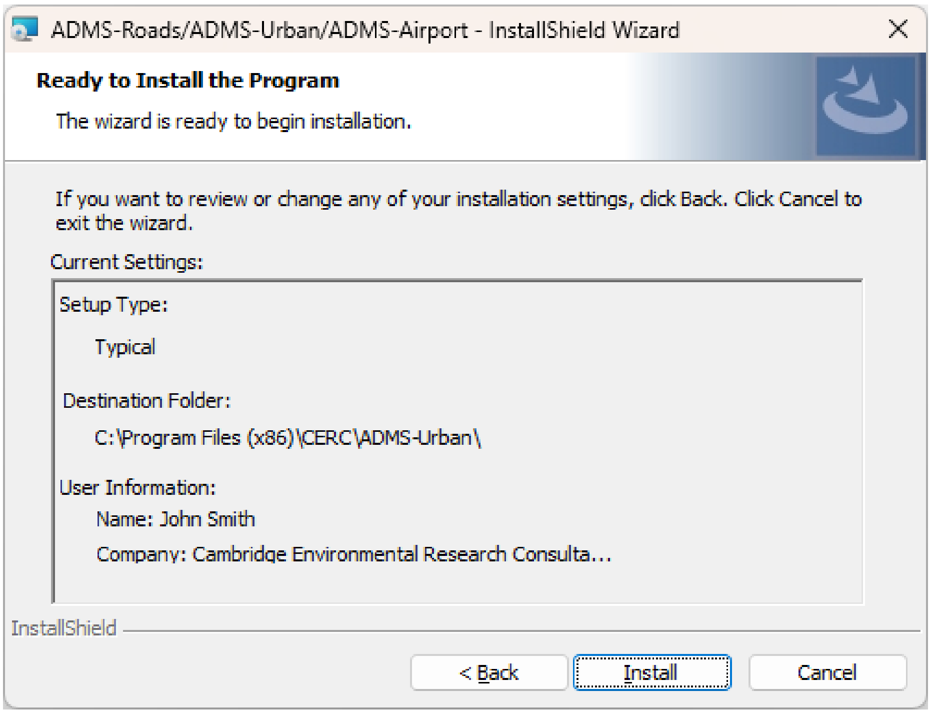


Figure 2.5 – The **Ready to Install the Program** screen.

Step 6 Click **Install** to proceed with the installation of ADMS-Urban files. Once the process has been completed, the final screen will appear, as shown in **Figure 2.6**.

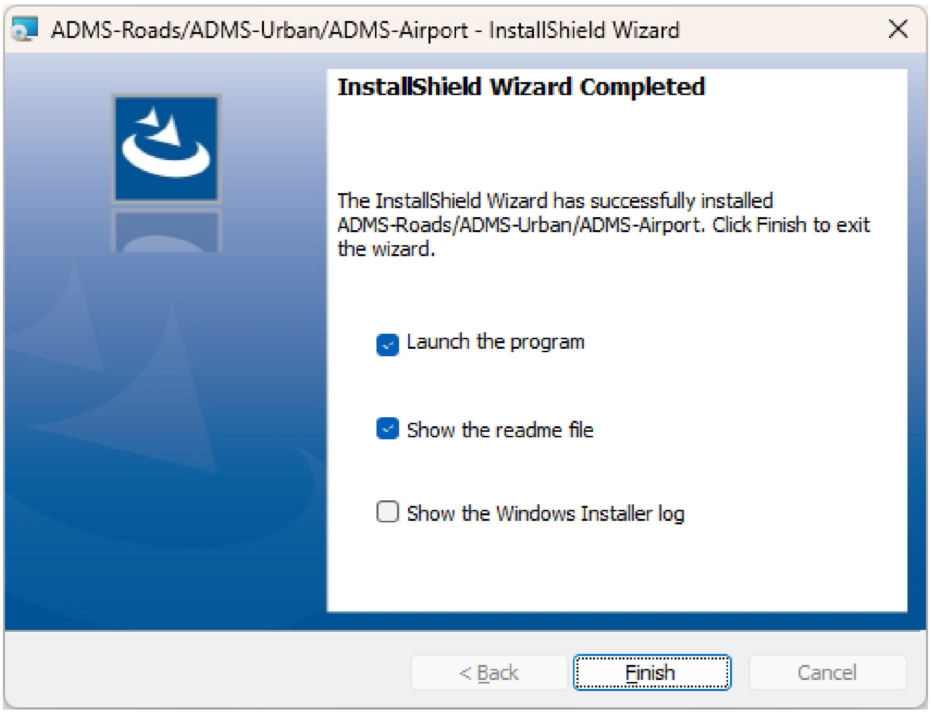


Figure 2.6 – The **Installation Wizard Completed** screen.

Step 7 Click **Finish** to complete the installation. If the **Show the readme file** box is checked, a *What's New* document will be opened automatically once you click on **Finish**. If the **Show the Windows Installer log** is checked then the install log will be opened automatically once you click on **Finish**. If the **Launch the program** is checked then the screen shown in **Figure 2.7** will be displayed.

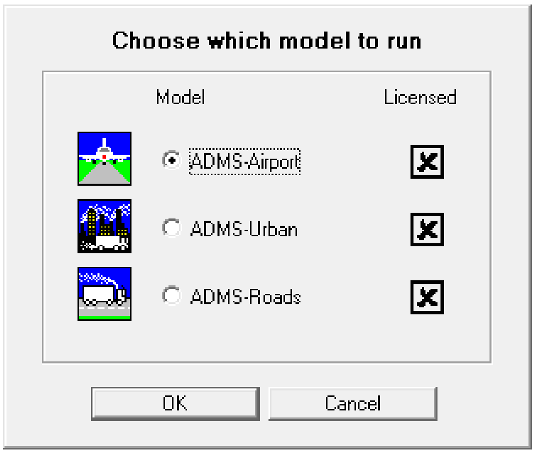


Figure 2.7 – Model selection screen.

You will have been provided with a unique licence file, by email, which is required in order to run the model. It is important that you install this licence file as instructed below.

Step 8 To install the ADMS-Urban licence, copy the file *ADMS-Urban.lic* to the *<install_path>* directory.

Step 9 Depending on your licence type, the first time that you launch ADMS-Urban after installation, it may be necessary to be connected to the Internet so that your licence can be registered. If there is no Internet connection, the message shown in **Figure 2.8** will appear.

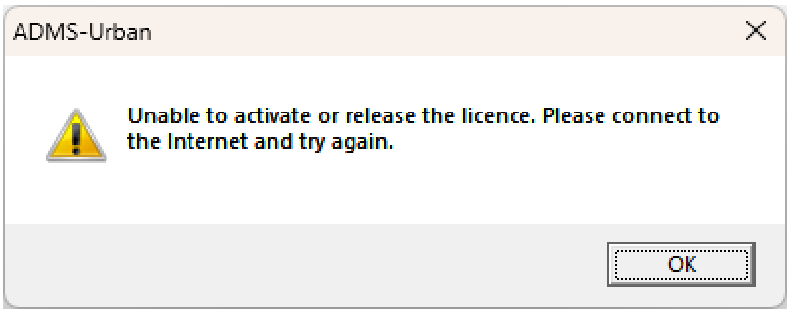


Figure 2.8 – Licence registration failure.

Step 10 Your licence will allow up to a certain number of users. Depending on your licence type, if the maximum number of users has already been reached, the message shown in **Figure 2.9** will appear. If you are issued this message, it is first necessary to release another user's licence (via the **Help, Return licence** menu item of the ADMS-Urban interface on that user's PC) before launching ADMS-Urban again on the current PC.

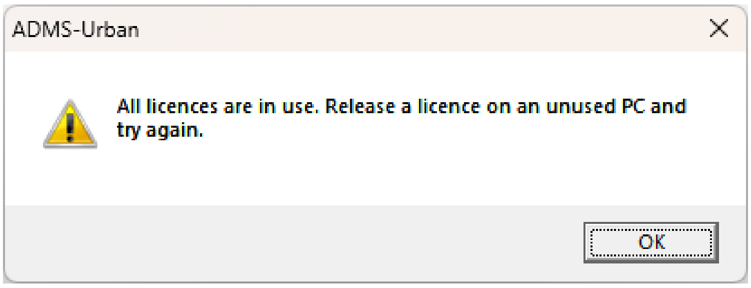


Figure 2.9 – All licences in use.

Step 11 The installation procedure automatically puts a generic shortcut to ADMS-Urban, ADMS-Roads and ADMS-Airport on your desktop. The first time you double-click on this shortcut, the screen shown in **Figure 2.10** will be displayed, unless the **Launch the program** option was checked at the end of the installation process.

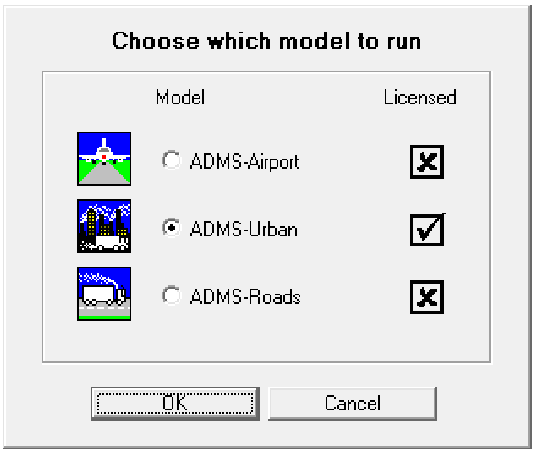


Figure 2.10 – Model selection screen.

Select ADMS-Urban and click **OK**.

If you also use ADMS-Roads and ADMS-Airport, then this generic shortcut will launch the most recently used model. You can change the start up preference in the interface under the **File** menu.

To set up a dedicated shortcut to ADMS-Urban on your Windows desktop, browse to the *<install_path>\Support\Shortcuts* directory in Explorer, copy the ADMS-Urban shortcut and paste it on to your desktop.

Launching ADMS-Urban and checking the licence details (through ***Help, Licence Details****) will give the location of the licence currently being used.*

If ADMS-Urban is already installed and you are simply renewing an expired licence, it is also possible to drag and drop the new licence file into the ADMS-Urban interface in order to automatically install the new licence.

2.3 Getting around the interface

2.3.1 Mouse buttons

Unless otherwise stated, mouse instructions refer to the left button. If the mouse options have been used to reverse the mapping (e.g. because you are left-handed), the right mouse button should be used instead.

2.3.2 Keyboard access

All mouse instructions in this User Guide can be reproduced using keystrokes. A brief guide to these keystrokes is given in **Table 2.1**.

Also known as shortcut keys, these are combinations of keys that perform some of the main commands. For example, menu commands that have one letter underlined are accessible by holding down the **ALT** key and then typing the underlined letter. For example, the menu command **Open...** located on the **File** menu, may be executed by typing **ALT + F** and then **ALT + O**.

Key	Description
Moving the cursor between data entry boxes	
TAB	Move the cursor forward through data entry boxes or buttons
SHIFT + TAB	Move the cursor backwards through data entry boxes and buttons
RETURN	'Enter' or accept the current data page or execute the action of a highlighted button
SPACEBAR	Select or deselect the highlighted option
Entering data in a box	
DELETE	Delete the character immediately to the right of the cursor
BACKSPACE	Delete the character immediately to the left of the cursor
← arrow	Move the cursor one space to the left in the current box
→ arrow	Move the cursor one space to the right in the current box
SHIFT + arrow	Begin highlighting characters in the direction of the arrow (see above)
Highlighted text	
DELETE	Delete all highlighted characters
(Type)	Typing text replaces the highlighted text with new text
Radio buttons	
← arrow	Move the cursor up through the radio buttons for the current item
→ arrow	Move the cursor down through the radio buttons for the current item

Table 2.1 − Keystrokes to enable you to move through the ADMS-Urban interface.

2.4 Main menu options

The menu bar has six headings: **File**, **Run!**, **Results**, **Mapper**, **Utilities** and **Help**. All menu headings have drop-down lists of options apart from the **Run!** and **Mapper** (see for example the **File** menu options listed in **Figure 2.11**). **Table 2.2** gives the list of options and roles of each menu item.

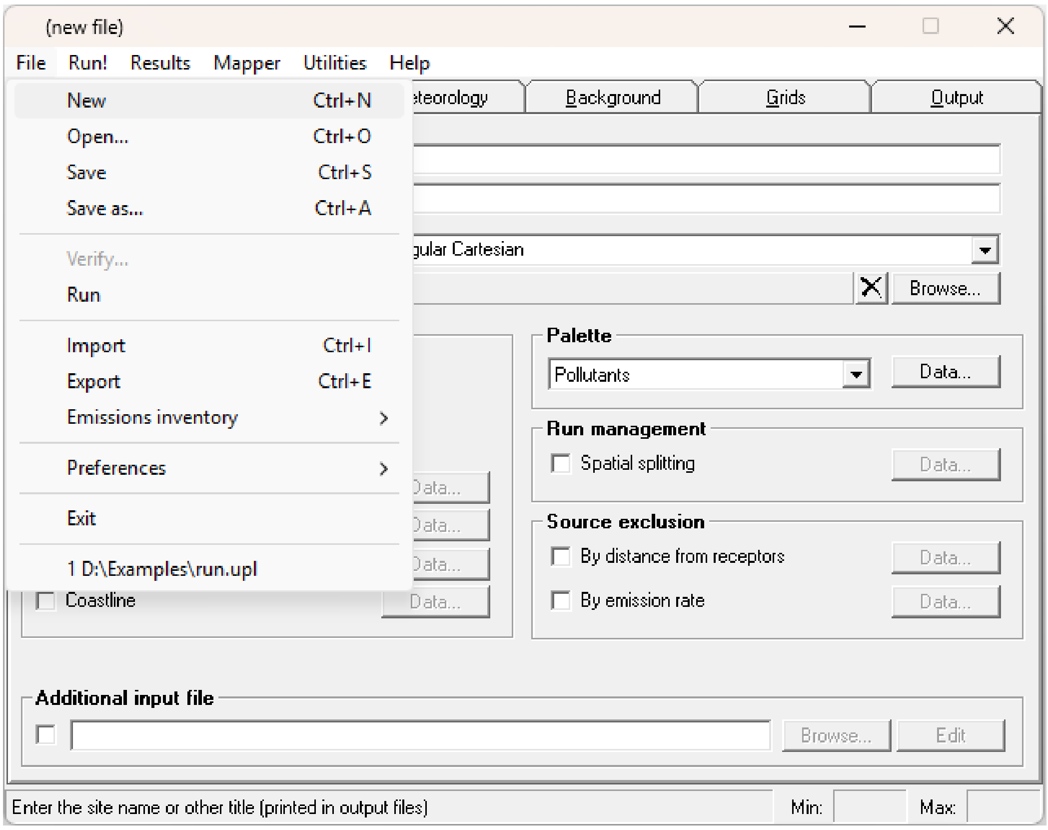


Figure 2.11 – The **File** menu from the menu bar in ADMS-Urban.

Menu	Item		Role	Ref.
File	New		Reset parameters in model file to their defaults	
	Open…		Open a previously saved parameter file	
	Save		Save current parameters under current file name	
	Save as…		Save current parameters under a new file name	
	Verify…		Run verification checks on the *.upl* file	2.5.4
	Run		Run the model using the current parameters	2.5.5
	Import		Import sources, groups, pollutants and/or buildings from a set of comma-separated files	5.2
	Export		Export sources, groups, pollutants and/or buildings to a set of comma-separated files	5.3
	Emissions Inventory	Import	Import pollutant and/or source data from an Emissions Inventory Database	C.1
		Export	Export source data to an Emissions Inventory Database	
		Inventory database	Specify path name of the Emission Inventory Database to be linked to ADMS-Urban	
	Preferences	Percentiles method	Force the model to use one of the available percentiles methods (normally, the model chooses the method automatically)	
		UPL verification	Change the settings that affect when UPL verification should run and how the results of UPL verification should be presented	2.5.4
		Model execution	Choose options for the run time window state and exit mode	2.5.5
		Viewing options	Choose the application for viewing numerical results files in ADMS-Urban (the default application is Microsoft Excel)	6.6
		Coordinate system	Specify coordinate system to be used by default	3.1.2
		Restrict source counts	Specify whether the interface should enforce licence limits on number of sources	
		Licence management	Specify whether the online licence should be released automatically when ADMS-Urban is closed. (Only visible for some licence types).	
		Start As	Specify the mode (ADMS-Urban, ADMS-Roads or ADMS-Airport) in which the interface should start. Also indicates unlicensed/expired modes	
	Exit		Quit ADMS-Urban	
	1, …		Names of the most recent model files opened in ADMS-Urban (click on the required file name to open the selected file in ADMS-Urban)	
Run!			Run the model using the current parameters	2.5.5
Results	Contour plot	Mapper	Launch the 2-D Output Plotter for plotting contours in the Mapper	6.2
		Surfer	Launch the 2-D Output Plotter for plotting contours in Surfer (if Surfer is installed)	
	Flow field plot	Mapper	Launch the 2-D Output Plotter for plotting flow fields in the Mapper	6.2
		Surfer	Launch the 2-D Output Plotter for plotting flow fields in Surfer (if Surfer is installed)	
	Line plot		Launch the line plotting facility	6.3

Table 2.2 – Options and roles of menu items. The last column (Ref.) indicates the section of the user guide where the item is further described.

Menu	Item	Role	Ref.
Results	Process comprehensive output	Launch the Comprehensive Output File Processor	3.6.4
	Display footprint	Launch a form that can be used to display the results from an 'output per source' run	6.4
	Log files	Visualise error, warning and log files	
	Numerical output	Open the numerical output of the current model file in the preferred viewing software	6.6
	Results folder	Launch an explorer window with the results folder for the run open.	
	Automate Surfer contour plots	Launch an Excel template containing macros for the automation of contour plotting in Surfer (if Surfer and Excel are installed)	*
	Make Slideshow	Launch an Excel template containing macros for creating a slideshow (if Excel is installed)	*
Mapper		Launch the Mapper	7.1
Utilities	View a wind rose	Launch the Wind Rose viewer	7.2
	Visualise input in Surfer…	Select features in the current model file, such as location of sources, for visualisation in a Surfer plot (if Surfer is installed)	7.3
	Create/Run batch file	Create and run batch files	7.4
	Convert met. data	Convert US format met. data to ADMS-Urban format	7.5
	Create terrain file	Prepare a terrain file from digital terrain data	7.6
	Create ASP grid	Create an *.asp* file containing grid(s) of output points	7.7
	Create import templates	Create templates for the SPT file import facility	5.1.8
	Create advanced canyon template	Create a template input file for the advanced street canyon modelling option based on the current road source data	4.2.2
	Download terrain data	Link to a website from which SRTM data can be downloaded	7
	Start Excel	Launch Microsoft Excel (if Excel is installed)	
	Start Surfer	Launch Surfer plotting package (if Surfer is installed)	

Table 2.2 − Options and roles of menu items. The last column (Ref.) indicates the section of the user guide where the item is further described *(continued)* *Please refer to the *Surfer Automation and Slideshow Creator User Guide* for more details.

Menu	Item	Role	Ref.
Help	**User Guide**	Open the User Guide in a PDF viewer	
	Comprehensive Output File Processor User Guide	Open the Comprehensive Output File Processor User Guide in a PDF viewer	
	What's New	Open the *What's New* in a PDF Viewer	
	Contact helpdesk	Auto-addresses a new email to the ADMS-Urban helpdesk in the user's default email client	
	CERC website	Open the CERC homepage in the user's default internet browser	
	Licence details	Show licensee and licence details including expiry date and licence number	
	Return licence	Releases the licence from the current PC so it can be registered by another PC. (Only visible for some licence types).	
	About model	Show model version number along with contact information for CERC and the ADMS-Urban Helpdesk	

Table 2.2 − Options and roles of menu items. The last column (Ref.) indicates the section of the user guide where the item is further described *(continued)*

2.5 Creating a model file and running the model

To generate results using ADMS-Urban, there are several steps to complete:

- create a new model file,
- enter data to define the problem,
- save the model file,
- run the model,
- display the output.

The first four of these steps are described in Sections 2.5.1 through to 2.5.5. Entering model data is described in general terms here, and in full detail in Sections 3 and 4. Displaying model output is covered in detail in Section 6.

It is also possible to open and run model files from CERC's industrial air pollution model ADMS 6 directly in ADMS-Urban. This is described in Section 2.5.7.

2.5.1 Creating a model file

When the ADMS-Urban model interface is loaded or when you select the **New** command from the **File** menu, a new model file, or scenario, is created and default values are loaded into the screens for you to edit.

An existing model file can be opened for editing or running either in the ADMS-Urban model interface or from Explorer.

- To open a model file in the ADMS-Urban interface, choose **Open...** from the **File** menu. (By default, ADMS-Urban will display only files with the *.upl* extension.)
- To open a model file from Explorer, either
  - right-click on the file and select **Open with ADMS-Urban**, or
  - drag and drop the file into the ADMS-Urban interface, or
  - double-click on the file, which will open the file in the most recently used of ADMS-Urban, ADMS-Roads and ADMS-Airport.

Files prepared in previous versions of ADMS-Urban (ADMS-Urban 2.0 or later) are converted into the current ADMS-Urban format as they are loaded into the interface. Simply open the file you wish to convert and follow the instructions given on screen.

It is possible to have multiple instances of the ADMS-Urban interface open simultaneously. This is particularly useful for comparing two model files side by side.

2.5.2 Entering information

Changing values in the input screens

To change a parameter value in an input screen, move the pointer until it is over the appropriate text box and click. Alternatively, use the **TAB** or **arrow keys** to move systematically through the sections contained in each screen. The selected area will be highlighted. Now type the new value, which will automatically replace what was highlighted. Alternatively, use **DELETE** and/or **BACKSPACE** to remove unwanted characters before typing in the new value.

Note that a blank cell does not denote a value of zero.

The helpline

This is a single line of text that appears at the bottom of the active screen. The information in the helpline changes when different controls on the screen are selected. It gives a brief description of the selected control's function. Where you are prompted for a numerical value, the helpline will give the maximum and minimum values allowed.

For example, when the **Enter on screen** option is selected in the **Meteorology** screen and the **Julian day number** column is highlighted, the help bar will display the line:

Julian day number eg. January 1st = 1, Min: 1, Max: 366

Data validity and integrity checking

As you enter data, the ADMS-Urban model interface performs checks to ensure that all user-entered data are consistent with the model's logic and that minimum and maximum values are satisfied.

Importing data

If you have a large number of sources, you may prefer to import the data from files. This option is described in Section 5.2. These files may be user-created, exported from EMIT or generated using the Mapper's **Export layer for SPT** tool.

2.5.3 Saving input data to a model file

When you are ready to run the model, choose **Save** from the **File** menu. If the current scenario has not been saved before, you will be prompted to choose a directory and file name. ADMS-Urban model files are always saved with the extension *.upl*.

2.5.4 Verifying the model file

An option exists to verify the contents of an *.upl* file. This carries out all of the model data checks, runs the meteorological pre-processor, and produces a report file, but stops before calculating concentrations. It is therefore quick to run, providing a useful check of the modelling input data before the main run.

While running UPL verification, the model will carry on past as many errors as it can so as to produce as comprehensive a list of errors as possible. Select **File**, **Verify…** to run UPL verification.

Additionally, the ADMS-Urban interface can be set so that the *.upl* file is verified each time it is saved and also before the model is run. To alter these settings select **File, Preferences**, **UPL verification**. This will bring up the **Data file verification preferences** screen, shown in **Figure 2.12**.

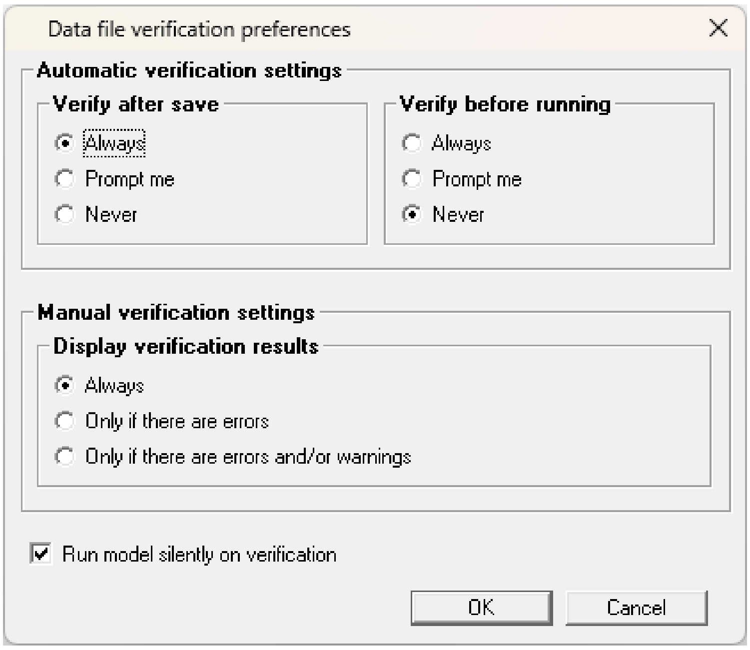


Figure 2.12 – The **Data file verification preferences** screen.

From this screen there are four options that can be set:

- **Verify data after save**: determines whether UPL verification should be carried out each time the *.upl* file is saved. Verification results are only displayed if any errors are found.
- **Verify data before run**: determines whether UPL verification should be carried out each time before the model is run. Verification results are only displayed if any errors are found. If no errors are found, the model then runs as normal.
- **Display verification results**: if UPL verification is selected manually from the **File** menu, these settings select when the results should be shown.
- **Run model silently on verification**: if checked, the UPL verification will run the model minimised and the run window will automatically close when verification has finished (default option).

2.5.5 Running ADMS-Urban

Save the current scenario as a model file, go to the menu bar and click on **Run!** to run the model (or select **Run** from the **File** menu). If you have never saved the current scenario or if you have changed anything in the interface since the scenario was last saved, then you will be prompted to save the modifications.

Alternatively, you can run the model with a batch file. These files allow you to run several files consecutively without opening and running each model file separately. Batch files can be created with the **Create/Run Batch File** utility (see Section 7.4).

While the model is running, information is displayed in a progress window (see **Figure 2.13**) to show the status of the model run in terms of the number of lines of meteorological data completed. When all the calculations have successfully completed, the progress window displays "`***RUN SUCCESSFULLY COMPLETED***`".

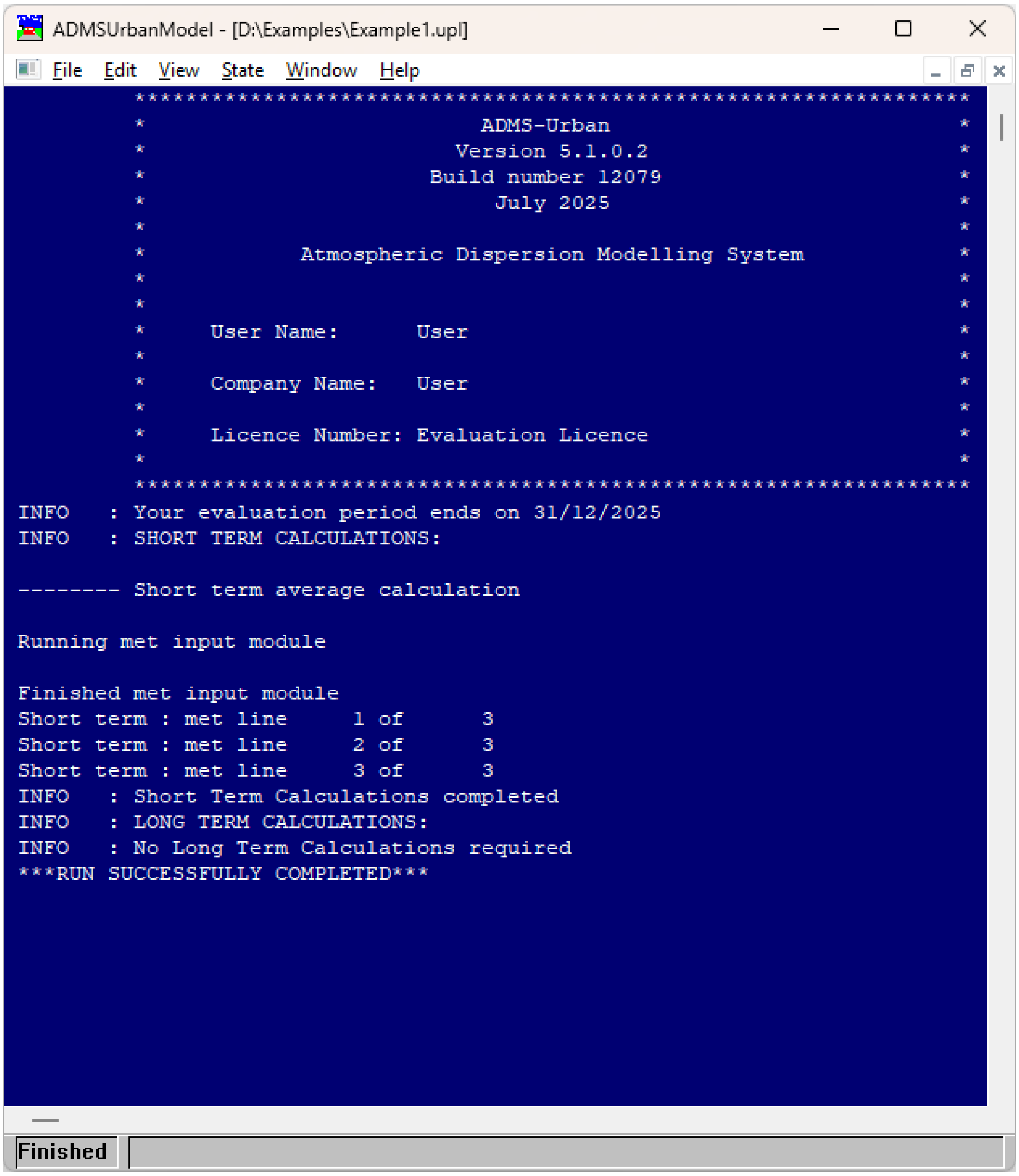


Figure 2.13 – Progress window of an ADMS-Urban run.

Runtime preferences

The user can edit the runtime options from the **File**, **Preferences**, **Model Execution** menu. This will bring up the **Runtime Preferences** dialogue box shown in **Figure 2.14**, divided into two sections:

- The **Window State Options** control the size and state of the run window: normal, minimized or maximised, with focus (active window) or without focus.
- The **Exit Mode Options** determine if the run window closes after the run has completed. If **Normal termination box** is selected, a dialogue box appears at the end of the run asking if you want the window to close or not. If **No termination box, window open** is selected, no dialogue box is displayed and the progress window is kept open at the end of the run. If **No termination box, window closes** is selected, no dialogue box is displayed and the progress window is closed at the end of the run.

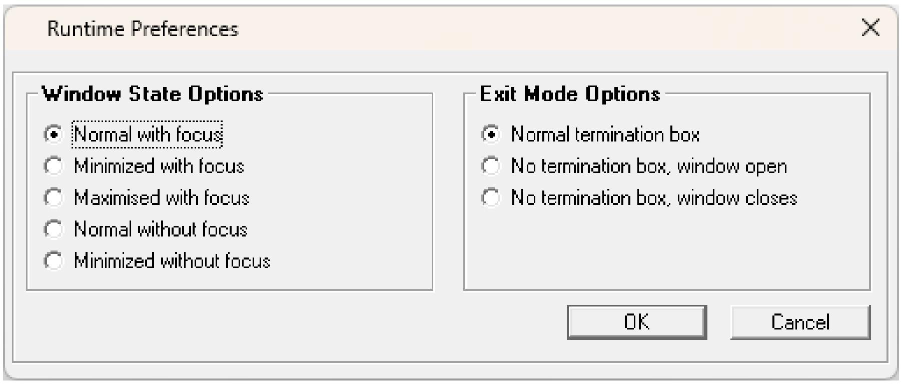


Figure 2.14 – Runtime Preferences dialogue box.

2.5.6 Displaying model output

Please refer to Section 6 for details about displaying output from ADMS-Urban model runs.

2.5.7 Opening and running ADMS 6 model files

It is also possible to open model files from CERC's industrial air pollution model ADMS 6, which have the extension *.apl*. Once an *.apl* file has been loaded into the ADMS-Urban model interface, it can then be edited and saved as a regular ADMS-Urban model file (*.upl*) and run in the standard way, as described above. The original *.apl* file will remain unaffected. This can be useful if, for example, you wish to add a local road network to a model setup of an industrial site originally created with ADMS 6.

To open an *.apl* file using the **File\Open..**. menu option, make sure that you use the drop-down list to display *.apl* (or all) files rather than just *.upl* files. Alternatively, drag and drop the *.apl* file into the ADMS-Urban interface.

Note that there are several modelling options in ADMS 6 that are not available in ADMS-Urban, for example the fluctuations module. If any of these options are present in the *.apl* file being opened, ADMS-Urban will issue an appropriate warning message(s), e.g. see **Figure 2.15**.

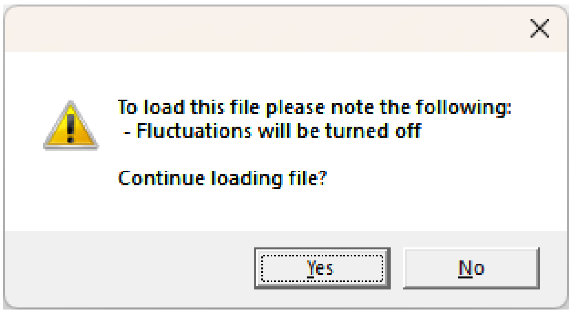


Figure 2.15 – Example warning message when opening an *.apl* file.

SECTION 3 Model Input

Setting up a modelling problem requires the user to input information specifying the release conditions, meteorological conditions and required output. This section provides an overall guide to the model interface. It describes the minimum input data required to run the model and briefly presents the additional modelling options available (full details on these options can be found in Section 4). Sections 3.1 to 3.6 give details of each type of input data. Practice in use of the model can be obtained by following the six detailed worked examples in Section 8.

There are six basic input screens associated with an ADMS-Urban model run, as shown in **Figure 3.1**. Data must be entered into each of these basic screens for every model run. These screens are:

- **Setup**: general site details and modelling options to be used;
- **Source**: source dimensions and locations, release conditions, emissions;
- **Meteorology**: meteorological conditions;
- **Background**: background concentration data;
- **Grids**: type and size of grid for output;
- **Output**: output required and sources/groups to include in the calculations.

It is advisable (though not necessary) to enter data in the screen order from left to right on the interface, namely ***Setup****,* ***Source****,* ***Meteorology****,* ***Background****,* ***Grids*** *and* ***Output****.*

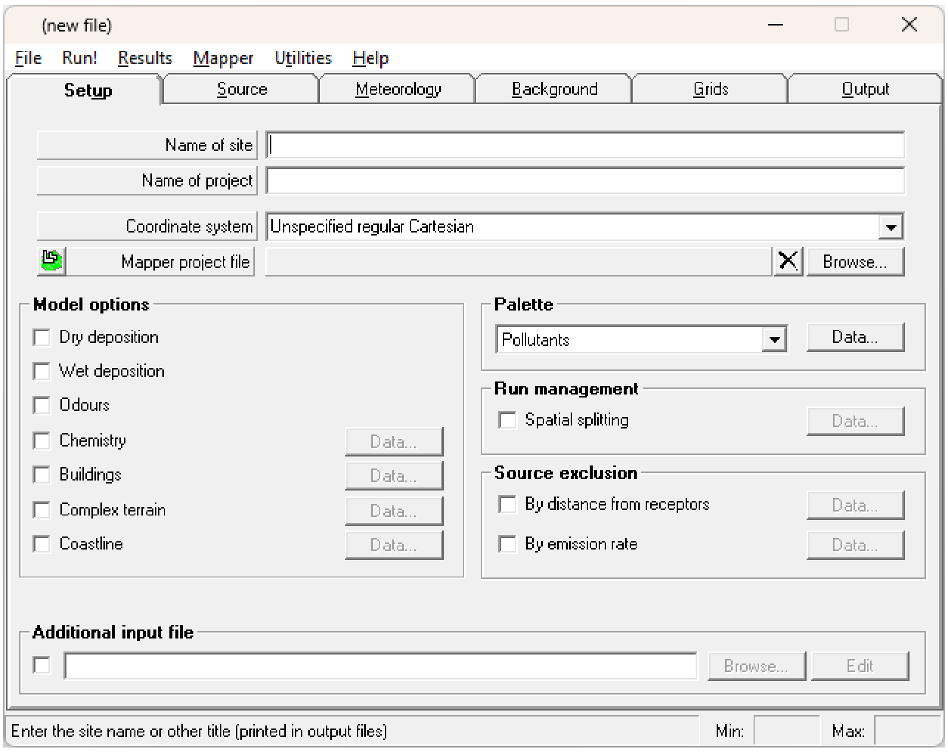


Figure 3.1 – The **Setup** screen of the ADMS-Urban interface.

3.1 Setup screen

The **Setup** screen is shown in **Figure 3.1**. This is the screen that appears when ADMS-Urban is first opened. The general site details and modelling options are entered here.

3.1.1 Name of site, Name of project

Up to 80 characters may be entered for each of these, or the boxes may be left empty.

3.1.2 Coordinate system

Spatial data entered into ADMS-Urban should be in a Cartesian coordinate system, measured in metres. The coordinate system may either be a standard coordinate system, e.g. British National Grid, or site-specific, e.g. centred on the source. The **Coordinate system** should be selected from the list, if a site-specific coordinate system is being used then select **Unspecified regular Cartesian**. To match a custom coordinate system specified in the Mapper select **Use Map coordinate system**. The coordinate system is used by the Mapper and for some of the Mapper export options a recognised coordinate system must be used. For more information refer to the *Mapper User Guide*.

The default coordinate system used for a new file can be specified from the ***File, Preferences****,* ***Coordinate system*** *menu option.*

When using a projected coordinate system, it is important to choose one that is well suited to geographic location of the study site, i.e. one that does not result in significant distortion of distances and/or orientation. For example, a global Mercator projection (e.g. 'Sphere Mercator (epsg:53004)') is only well suited to regions close to the equator; using this projection for a site in, say, central England would result in an inflation of distances of around 65%.

3.1.3 Mapper project file

The **Mapper project file** describes how to display information in the Mapper. Refer to Section 2.1 in the Mapper User Guide for more details. The Mapper project file can be changed either by using the **Browse** button or from the Mapper. The  button can be used to launch the Mapper.

3.1.4 Model options

The box presents a choice of modelling options to be used in the ADMS-Urban run. These options are fully described in Section 4. Permitted combinations of model options are given in Appendix A.

If no model options are selected, then ADMS-Urban will calculate concentrations for a pollutant release in flat terrain with no chemistry, buildings or deposition.

3.1.5 Palette

The **Palette** box provides access to the **Palette of Pollutants**. The **Data...** button brings up the **Palette of Pollutants** screen which is shown in **Figure 3.2**. The **Palette of Pollutants** screen can also be accessed by clicking on **Pollutants...** in the **Emissions** screen, shown in **Figure 3.5**.

The **New** and **Delete** buttons are used to add new pollutants to the palette and remove existing ones, respectively. The **Rename** button is used to change the name of existing pollutants. The **Copy** button can be used to create a copy of an existing pollutant. When pollutants are deleted they remain in the palette marked in red until the **Apply** or **OK** buttons are clicked. While the deleted pollutant remains in the palette the pollutant is no longer editable but can be restored using the **Restore** button.

The **Reset** button restores the palette to its default contents (including the removal of user-defined pollutants although they can then be restored).

The purpose of the palette of pollutants is to define properties of pollutants for a given model set up. Up to 80 pollutants may be defined: 12 pollutants are pre-defined, namely NO_X, NO_2, NO, VOC (volatile organic compounds), O_3, SO_2, CO, benzene and butadiene (1,3 Butadiene) together with the particulates PM_{10}, $PM_{2.5}$ and TSP. NO_X, NO_2, NO, VOC, O_3, SO_2, PM_{10} and $PM_{2.5}$ may not be deleted from the palette.

The ***Import*** *and* ***Export*** *facilities can be used to easily transfer a* ***Palette*** *between modelling scenarios. Refer to Section 5 for more details.*

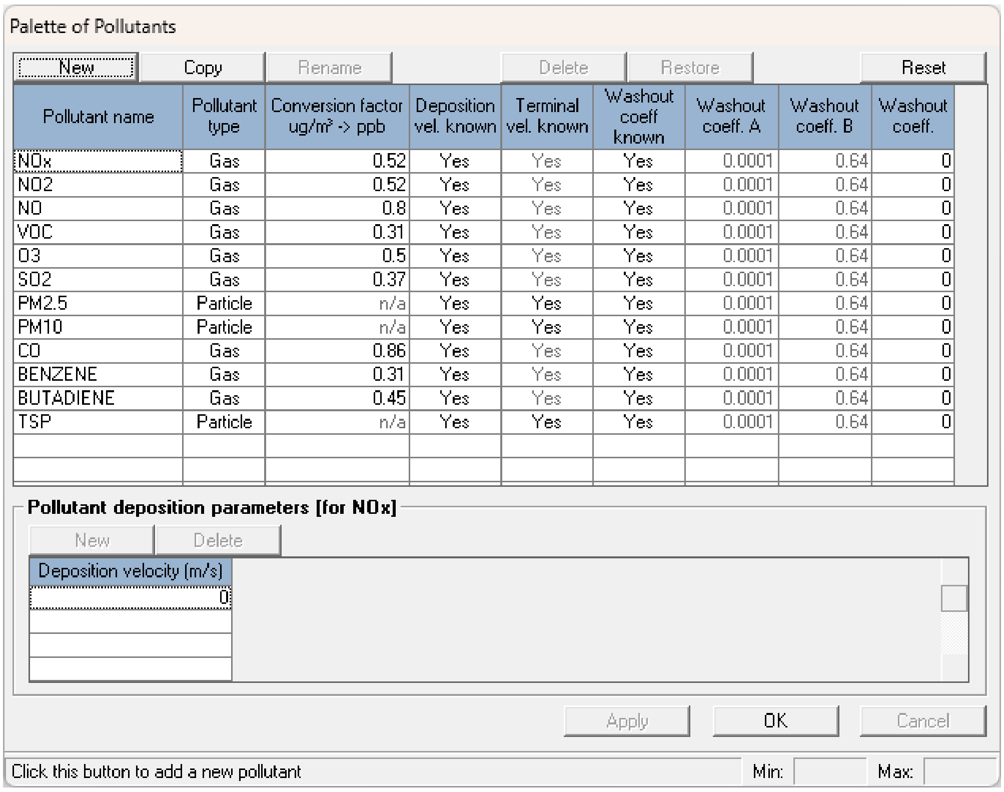


Figure 3.2 – Palette of pollutants in ADMS-Urban, showing default values for the 12 pre-defined pollutants.

The palette requires definition of a number of parameters related to each pollutant:

1. **Pollutant name**

   Each pollutant must have a unique name which can be up to 30 characters long. The pollutant name must not contain commas.

2. **Pollutant type**: **Gas** (for gaseous emissions) or **Particle** (for particulates)

   Double-click in the **Pollutant type** column to change between **Gas** and **Particle**. The number of columns in the lower table changes according to the different options in the upper table (refer to item 4 below).

3. **Conversion factor (ug/m^3 → ppb)**
   For gaseous pollutants, the user should specify a factor to convert output concentrations from μg/m^3 to parts per billion by volume (ppb). The conversion factor from μg/m^3 to ppb is 24.06/M (derived from the Ideal Gas Equation at standard temperature and pressure, 20 ^{o}C and 1013mb), where M is the molecular mass of the pollutant in grams. Default values:

   User-defined pollutant = 1 (Gas) or n/a (Particle)

   The default value for the conversion factor of VOC (0.31) is the value appropriate for benzene. This value should be edited if the VOC emissions you are modelling are not predominantly benzene.

4. **Dry deposition parameters**

   The lower table in the palette gives parameters that are related to dry deposition processes. As different pollutants are selected in the upper table the pollutant name in the title of the pollutant deposition parameters box changes. Different variables must be entered for different pollutant types: deposition velocity or gas type for gaseous emissions; and deposition and terminal velocities or particle size and particle density, and mass fraction, for particulate emissions. Dry deposition parameters are described in detail in Section 4.11.

5. **Wet deposition parameters**

   The parameters in the last three columns of the upper table of the palette (**Washout coeff. A**, etc.) are related to wet deposition processes (i.e. precipitation). Wet deposition is described further in Section 4.12.

3.1.6 Run management

Users with large model files may wish to model only a limited spatial region within their full modelling domain, or split their run into a number of smaller sub-regions in order to keep run times down, make better use of available computational resources and/or stay within source limits imposed by their model licence. This can be achieved via the **Spatial splitting** option, which is described in detail in Section 4.24.

3.1.7 Source exclusion

Another way of reducing run times for large model files is to exclude sources that are unlikely to contribute significantly to modelled concentrations. Two options for excluding such sources are available in ADMS-Urban; **By distance from receptors** and **By emission rate**. These options are described in detail in Section 4.25.

3.1.8 Additional input file (*.uai*)

There are a number of additional model options that are accessible via an additional input file (*.uai*). These options are fully described in other sections of this User Guide. Permitted combinations of model options are defined in Appendix A.

To use an additional input file, check the box in the **Additional input file** section and then enter the path to the file, or click on **Browse...** to browse to find the file, or drag and drop the file into the interface from Explorer. The **Edit** button launches the **Additional Input file editor** (shown in **Figure 3.3**) for the editing of existing additional input files or creation of new ones.

---

*If the **Additional Input file editor** is used to create a new additional input file, the path to that file will still need to be entered into the **Setup** screen of the interface.*

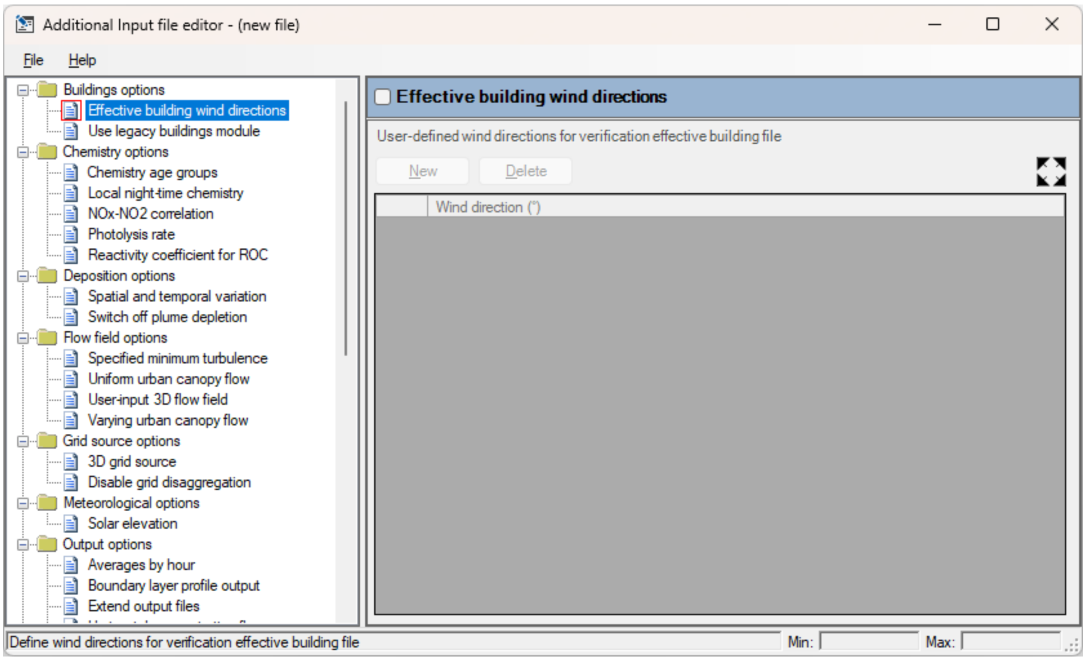


Figure 3.3 – Additional Input file editor with no options selected.

The **Additional Input file editor** contains a series of folders on the left hand side of the window. Each folder contains one or more options and the options selected for inclusion in this file are highlighted in bold.

Clicking on an option in the left hand side takes you to the screen for that option; check the box at the top of the screen to enable the option. Most of the options then require further details to be entered regarding that option, which are described in more detail in the corresponding model option sections of this user guide. Each of the sections in the **Additional Input file editor** is identified by a keyword, which will appear as the first line of that section in the additional input file itself.

Double clicking on an option in the left hand side will take you to that option and automatically enable/disable it. Double clicking on a folder will contract/expand the folder; this can be useful to hide options which are not being used.

Two preferences for the behaviour of the **Additional Input file editor** can be defined using the **File\Preferences...** menu option. On the **Viewing** tab the default application for viewing input text or comma-separated variable files can be chosen. The **Template** tab can be used to define a template data file. This data file is an additional input file, which may be complete or incomplete, with the default settings you would like to use every time you create a new additional input file.

3.2 Source screen

The **Source** screen is where individual ADMS-Urban sources are defined. A source can be one of six different types, as shown in **Table 3.1**. Point, line, area and volume sources are referred to collectively as 'industrial' sources. The number of each source type allowed depends on the licence type; details are given in Appendix A.

Source type		Standard emission units	Emission units when modelling odours
Road		g/km/s	ou_e/km/s
Point	Industrial	g/s	ou_e /s
Line	Industrial	g/m/s	ou_e /m/s
Area	Industrial	$g/m^2/s$	ou_e $/m^2/s$
Volume	Industrial	$g/m^3/s$	ou_e $/m^3/s$
Grid		$g/m^2/s$	ou_e $/m^2/s$

Table 3.1 – Source types available in ADMS-Urban

At any one time, only the road, industrial or grid sources are displayed in the **Source** screen. To change the source type displayed, select the source type from the list on the left hand side of the **Source** screen (**Figure 3.4**).

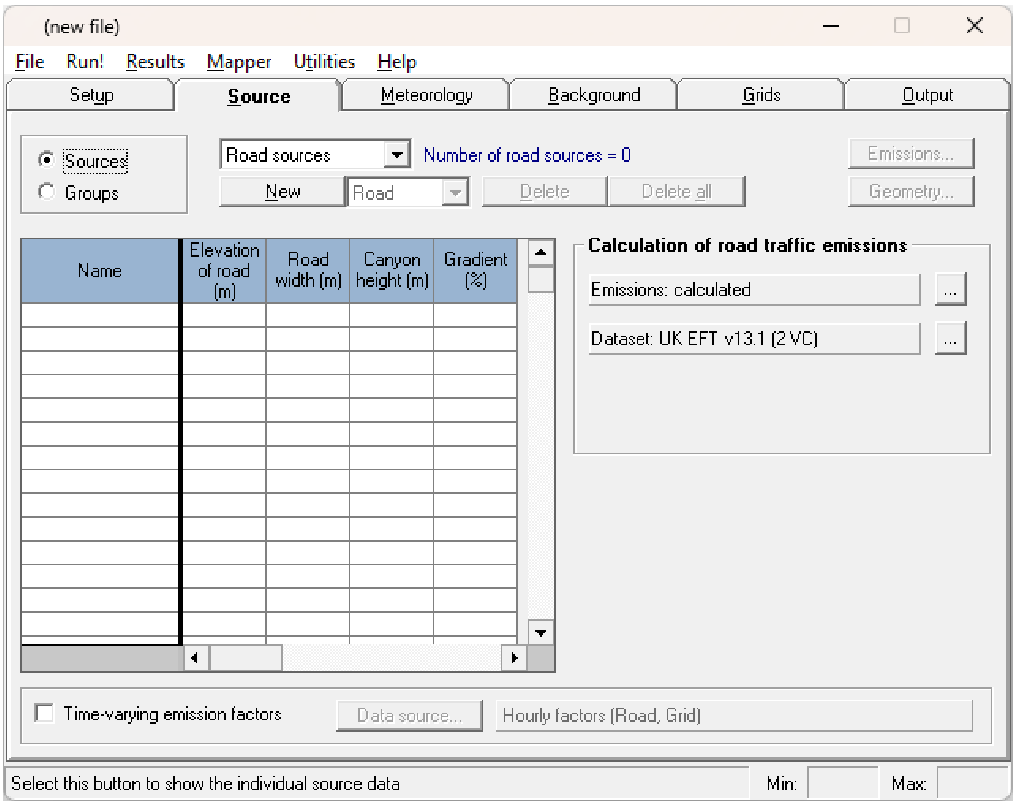


Figure 3.4 – The **Source** screen for road sources

This section contains instructions and advice for defining road, industrial and grid sources. It is strongly recommended that you read all the advice about each of these types of source before you begin modelling using ADMS-Urban. The information about the grid source is particularly important.

The maximum number of each source type that can be defined in the ADMS-Urban interface is very large; however when the model file is run, limits are imposed by the model licence. The ADMS-Urban interface can also be set to enforce the same limits using the **Restrict source counts** option. The spatial splitting option, described in Section 4.24, allows the sources defined in the model file to be divided up into one or more spatial subsets (regions); each region can then be processed in a separate model run, making it possible to stay within these source limits.

There are four ways you can enter new sources into ADMS-Urban:

- select the appropriate source table and click the **New** button above the table (or right-click on an existing source and click **Copy <source name>** to make a new copy of that source with a different name);
- using the Mapper, select the appropriate **Add Source** tool and click on the source position(s) on a map;
- import sources from a set of comma-separated variable files; or
- import sources from an ADMS-Urban Emissions Inventory.

This section gives information about the first of these methods. The use of the Mapper is covered in the Mapper User Guide, importing source information from comma-separated variable files is covered in Section 5 and emissions inventories are discussed in detail in Appendix C.

The **Source** screen is also where groups of sources are defined. In ADMS-Urban, sources can be modelled one at a time, or as part of a 'group' of sources. Up to 20 user-defined groups can be modelled at one time; refer to Section 3.2.5 for details on defining these groups. The groups to be modelled are selected in the **Output** screen, where it is also possible to select a group containing all of the sources for output; refer to Section 3.6.3 for details. Results are given separately for each group modelled.

Select the **Sources** option in order to add source data into the table. Details of the data required for each source type are given in Sections 3.2.1 through to 3.2.3 below.

3.2.1 Road Sources

This section describes the basic set-up for modelling a road source in ADMS-Urban. For details on the **Advanced street canyon** module refer to Section 4.2, for details on modelling noise barriers refer to Section 4.3 and for details on converting a road source into a tunnel refer to Section 4.4.

Creating a road source using the ADMS-Urban interface

Step 1 Select **Sources**, **Road sources** from the list and click on the **New** button. The model will input a default road source name, e.g. 'Road00001', with a set of associated default data. The source name will appear highlighted.

Step 2 Change the default name to one you want by clicking on the default name and editing the text.

Step 3 Click on the **Geometry...** button to enter the road coordinates. Each road source may have up to 51 vertices (i.e. 50 segments) which must be at least 1 m apart. The number of vertices currently defined is shown in brackets next to the word **Vertices**. To add a new vertex, click on the **New** button and edit the X and Y coordinates as appropriate. To remove a vertex select it and click the **Delete** button. To insert a vertex between two existing vertices (or before/after the current start/end vertex), right-click on one of the existing vertices and select **Insert above** or **Insert below** as appropriate. The initial coordinates of a new vertex inserted between two existing vertices will take the mid-point between those two vertices, whereas a new start/end vertex will take the same coordinates as the start/end vertex it is replacing; the user can then further edit these values as appropriate.

Step 4 Change the road elevation, road width and canyon height parameters of the default source to the actual values for your road by clicking on the chosen cell until it is highlighted by a dashed box. If using an emission dataset that supports it, also change the gradient of the road (i.e. the uphill slope as a percentage) to the actual value for your road. Type your value and press **ENTER**. Alternatively, click on the cell once more and edit the value via the text cursor. The road width cannot be less than 2 m. If a road width of less than 2 m is entered the model will automatically reset the width to be exactly 2 m. Refer to the paragraph *The basic street canyon model* below for advice about road widths and canyon heights, then proceed to the section on defining road source emissions.

The columns of the road source table are resizable.

To delete a road source, highlight the source **Name** in the table by clicking on it, then click on the **Delete** button. There is also a right-click option to **Delete all but this** road source from the table, as well as a button to **Delete all** road sources simultaneously.

The basic street canyon model

The basic street canyon model takes account of the additional turbulent flow patterns occurring inside a street with relatively tall buildings on both sides, known as a “street canyon”. It only affects results at output points inside the street canyon, at heights below the height of the canyon. For such roads, the appropriate road width to enter is the building edge to building edge distance. Only roads with a canyon height greater than 0.5 m are modelled as basic street canyons, so for roads with canyon height less than 0.5 m, you should enter the actual road (carriageway) width. As mentioned above, the minimum road width allowed is 2 m.

The **Advanced street canyon** *module can be used if a more detailed model of the street canyon is required. Refer to Section 4.2 for details. In particular note the advice about road width and position.*

Defining road source emissions

Road source emission rates can be entered directly, or calculated from traffic flow data using the in-built database of traffic emission factors.

To define emissions for a road source, make sure that you are in **Sources** mode. If you wish to calculate emission rates from traffic flows then ensure that you have selected the appropriate emission factors dataset (more information about these datasets is given below).

Highlight the source in the list of sources, and click on the **Emissions...** button. This displays the **Emissions** screen for the selected road source, as shown in **Figure 3.5**.

To enter your own pre-calculated emission rates directly, select **All pollutants user defined** in the **Emissions** box.

Select **Calculate emissions using traffic flows** in the **Emissions** box to calculate emission rates from traffic flow data using the selected emission factors dataset.

Note that you can use the '...' button next to ***Emissions:*** *on the* ***Source*** *screen to change the emissions mode between* ***User defined emissions*** *and* ***Calculate emissions using traffic flows*** *for all roads simultaneously, as an alternative to changing it for individual roads via the* ***Emissions*** *screen. However, this should be used with care – when switching from* ***User defined emissions*** *to* ***Calculate emissions using traffic flows****, any user-defined emissions previously entered for pollutants included in the chosen emission factor dataset will be lost.*

In both cases, emission rates for road sources are expressed in g/km/s.

All NO_X emissions are assumed to be "NO_X as NO_2". Please refer to Appendix B for further information.

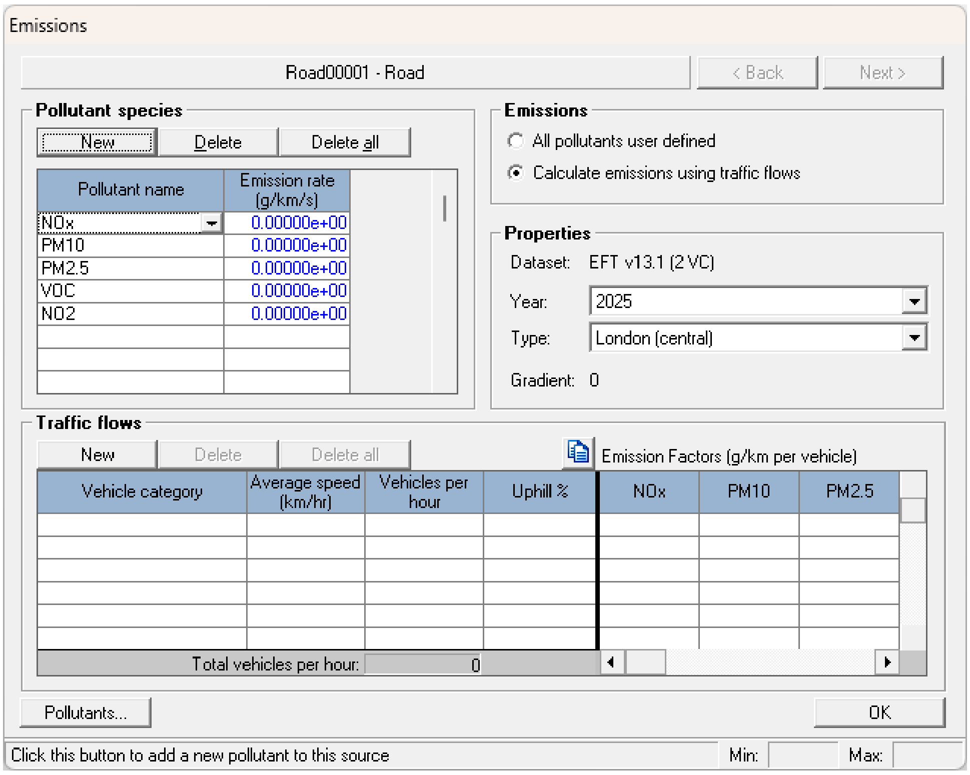


Figure 3.5 – The **Emissions** screen for road sources.

To enter emission details for other sources, click on the **< Back** and **Next >** buttons in the top right-hand corner of the screen. The name and type of the source you are currently considering is shown above the **Pollutant species** table, and the appearance of the **Emissions** screen will vary according to the type of source you have chosen.

If you are working with a large number of sources, you may prefer to import source information from a set of comma-separated variable files; please refer to Section 5.

The ADMS-Urban database of traffic emission factors

ADMS-Urban contains a range of emission factor datasets. These are summarised in **Tables 3.2** to **3.5**.

For studies within the UK, the most up-to-date emission factors to use are the UK EFT v13.1 datasets and the UK DMRB IAN 185/15 dataset.

The Chinese MoT 2006 emission factor dataset is only appropriate for estimating emissions from roads in China. Any further details of this dataset will therefore be omitted from the discussions below.

In order to use the emission factor datasets contained within ADMS-Urban to estimate emissions from roads, activity data in terms of vehicle counts are required. In addition, the emissions vary for different types of roads. Details are given below.

All emission factor datasets included within the model contain emissions for the following pollutants:

- NO_x Total oxides of Nitrogen (given as "NO_X as NO_2" – please refer to Appendix B for further information)
- PM_{10} Particulates with a maximum diameter of 10 µm
- $PM_{2.5}$ Particulates with a maximum diameter of 2.5 µm

The PM_{10} and $PM_{2.5}$ emissions within the EFT v13.1, 12.1, 12.0, 11.0, 10.1, 9.0, 8.0 and 7.0 datasets are total emission values, i.e. they are the sum of exhaust, brake and tyre wear and road abrasion.

Only the EFT v13.1, 12.1, 12.0, 11.0, 10.1, 9.0 and 8.0 datasets contain emissions for NO_2. This is because the source data for other datasets do not include these emissions. For details of how to include NO_2 emissions in a particular model run, please refer to Section 4.13.2 .

All emission factor datasets apart from UK DMRB IAN 185/15 contain emissions for VOC (Volatile Organic Compounds (hydrocarbons, excluding methane)). For EFT v13.1 the VOC emissions have been derived from COPERT v5.8. For EFT v12.1 and v12.0 the VOC emissions have been derived from COPERT v5.7. For EFT v11.0, 10.1, 9.0, 8.0 and 7.0 the VOC emissions are taken from EFT v.6.0.1.

CO emissions are not included with the EFT v13.1, 12.1, v12.0, v11.0, v10.1, v9.0 and v8.0 datasets. The CO emissions for older EFT datasets have been derived from the Highways Agency (DMRB 2003) dataset, as the EFT does not contain CO emissions.

The most recent EFT v13.1, 12.1, v12.0, 11.0, 10.1 datasets and the UK DMRB IAN 185/15 dataset include emissions for a range of years up to 2030. EFT v13.1, 12.1 and 12.0 also include data for years up to 2050. All datasets include emissions for a range of speeds. The EFT v13.1, 12.1, v12.0, 11.0, 10.1 and 9.0 datasets include emissions for a range of road gradients for heavy goods vehicles, from -6% to 6% in steps of 2%; emissions will be interpolated for gradients between these values, but not extrapolated (i.e. emissions are capped at the ±6% values). A summary is given in **Tables 3.2** and **3.5**.

The source data for the UK DMRB IAN 185/15 dataset gives emissions for a small number of speed-band categories, each associated with a different level of congestion. For consistency with other datasets, the emission dataset in the model includes a range of speeds; the emissions for each speed are taken from the appropriate speed-band in the source data. Refer to Highways Agency (2015) for more details.

Traffic count data can be categorised in a number of different ways. The simplest categorisation is in terms of light duty and heavy duty vehicles only. More detailed categorisations, such as including the number of buses and taxis, may be available. **Table 3.6** and **3.7** gives the vehicle categories for each emission factor dataset included in the model.

In some cases it is more appropriate to use an emission factor dataset with a more detailed vehicle categorisation (e.g. UK EFT v13.1 (6 VC)), for example, when considering bus lanes separately, or doing source apportionment studies. In other cases, the simple light and heavy duty vehicle categorisation is sufficient. However, it is useful to remember that changing the classification of the traffic counts will change emissions and this may have a significant effect on the modelled concentrations. For detailed emissions modelling studies, such as the investigation of a Low Emission Zone, the use of an external emissions tool such as EMIT may be appropriate.

Dataset name (as displayed in the interface)	Suggested usage of dataset	Reference	Dataset name (to be used when importing)	Years	Speeds (km/hr)			Gradient (%)			Number of vehicle categories (Tables 3.6 and 3.7)	Number of road types (Table 3.9)
					Minimum	Maximum	Increment	Minimum	Maximum	Increment		
UK EFT v13.1 (2 VC)	For all new projects where the regulator requires the use of the EFT, and the traffic data are simply categorised into light and heavy duty vehicles.	EFT v13.1 (2025), COPERT 5.8*	EFT v13.1 (2 VC)	2021-2050	5	140	1	-6	6	2	2	16
UK EFT v13.1 (6 VC)	For all new projects where the regulator requires the use of the EFT, and the traffic data is binned into the six vehicle counts specified in **Table 3.6**.		EFT v13.1 (6 VC)	2021-2050	5	140	1	-6	6	2	6	16
UK EFT v13.1 (7 VC)	For all new projects where the regulator requires the use of the EFT, and the traffic data is binned into the seven vehicle counts specified in **Table 3.6**.		EFT v13.1 (7 VC)	2021-2050	5	140	1	-6	6	2	7	16
UK EFT v13.1 (9 VC)	For all new projects where the regulator requires the use of the EFT, and the traffic data is binned into the nine vehicle counts specified in **Table 3.6**.		EFT v13.1 (9 VC)	2021-2050	5	140	1	-6	6	2	9	16
UK EFT v13.1 (11 VC)	For all new projects where the regulator requires the use of the EFT, and the traffic data is binned into the eleven vehicle counts specified in **Table 3.6**.		EFT v13.1 (11 VC)	2021-2050	5	140	1	-6	6	2	11	16
UK DMRB IAN 185/15 (2 VC)	For all new projects where the regulator requires the use of the DMRB.	Highways Agency (2015)	UK DMRB IAN 185/15	2011-2030	5	140	1**	N/A			2	7

Table 3.2 – Summary of newer emission factor datasets for the UK. * The VOC emission factors for the EFT v13.1 dataset are derived from COPERT 5.8. ** The source data gives emissions for a small number of speed-band categories, see Highways Agency (2015).

Dataset name (as displayed in the interface)	Suggested usage of dataset	Reference	Dataset name (to be used when importing)	Years	Speeds (km/hr)			Gradient (%)			Number of vehicle categories (Tables 3.6 and 3.7)	Number of road types (Table 3.9)
					Minimum	Maximum	Increment	Minimum	Maximum	Increment		
UK EFT v12.1 (2 VC)	These datasets should only be used for old projects that require consistency between emission factors. Any new modelling studies should use a newer dataset.	EFT v12.1 (2024), COPERT 5.7*	EFT v12.1 (2 VC)	2018-2050	5	140	1	-6	6	2	2	16
UK EFT v12.1 (6 VC)			EFT v12.1 (6 VC)	2018-2050	5	140	1	-6	6	2	6	16
UK EFT v12.1 (7 VC)			EFT v12.1 (7 VC)	2018-2050	5	140	1	-6	6	2	7	16
UK EFT v12.1 (9 VC)			EFT v12.1 (9 VC)	2018-2050	5	140	1	-6	6	2	9	16
UK EFT v12.1 (11 VC)			EFT v12.1 (11 VC)	2018-2050	5	140	1	-6	6	2	11	16
UK EFT v12.0 (2 VC)		EFT v12.0 (2023), COPERT 5.7*	EFT v12.0 (2 VC)	2018-2050	5	140	1	-6	6	2	2	16
UK EFT v12.0 (6 VC)			EFT v12.0 (6 VC)	2018-2050	5	140	1	-6	6	2	6	16
UK EFT v12.0 (7 VC)			EFT v12.0 (7 VC)	2018-2050	5	140	1	-6	6	2	7	16
UK EFT v12.0 (9 VC)			EFT v12.0 (9 VC)	2018-2050	5	140	1	-6	6	2	9	16
UK EFT v12.0 (11 VC)			EFT v12.0 (11 VC)	2018-2050	5	140	1	-6	6	2	11	16
UK EFT v11.0 (2 VC)		EFT v11.0 (2021), EFT v6.0.1 (2014) **	EFT v11.0 (2 VC)	2018-2030	5	140	1	-6	6	2	2	16
UK EFT v11.0 (6 VC)			EFT v11.0 (6 VC)	2018-2030	5	140	1	-6	6	2	6	16
UK EFT v11.0 (7 VC)			EFT v11.0 (7 VC)	2018-2030	5	140	1	-6	6	2	7	16
UK EFT v11.0 (8 VC)			EFT v11.0 (8 VC)	2018-2030	5	140	1	-6	6	2	8	16
UK EFT v10.1 (2 VC)		EFT v10.1 (2020), EFT v6.0.1 (2014) **	EFT v10.1 (2 VC)	2018-2030	5	140	1	-6	6	2	2	16
UK EFT v10.1 (6 VC)			EFT v10.1 (6 VC)	2018-2030	5	140	1	-6	6	2	6	16
UK EFT v10.1 (7 VC)			EFT v10.1 (7 VC)	2018-2030	5	140	1	-6	6	2	7	16
UK EFT v10.1 (8 VC)			EFT v10.1 (8 VC)	2018-2030	5	140	1	-6	6	2	8	16
UK EFT v9.0 (2 VC)		EFT v9.0 (2019), EFT v6.0.1 (2014) **	EFT v9.0 (2 VC)	2017-2030	5	140	1	-6	6	2	2	16
UK EFT v9.0 (6 VC)			EFT v9.0 (6 VC)	2017-2030	5	140	1	-6	6	2	6	16
UK EFT v9.0 (7 VC)			EFT v9.0 (7 VC)	2017-2030	5	140	1	-6	6	2	7	16
UK EFT v9.0 (8 VC)			EFT v9.0 (8 VC)	2017-2030	5	140	1	-6	6	2	8	16

Table 3.3 – Summary of older emission factor datasets (part 1). * The VOC emission factors for the EFT v12.0 dataset are derived from COPERT 5.7. ** The VOC emission factors are derived from the UK EFT v6.0.1 dataset.

Dataset name (as displayed in the interface)	Suggested usage of dataset	Reference	Dataset name (to be used when importing)	Years	Speeds (km/hr)			Number of vehicle categories (Tables 3.6 and 3.7)	Number of road types (Table 3.9)
					Minimum	Maximum	Increment		
UK EFT v8.0 (2 VC)	These datasets should only be used for old projects that require consistency between emission factors. Any new modelling studies should use a newer dataset.	EFT v8.0 (2017), EFT v6.0.1 (2014) +	EFT v8.0 (2 VC)	2015-2030	5	140	1	2	16
UK EFT v8.0 (6 VC)			EFT v8.0 (6 VC)	2015-2030	5	140	1	6	16
UK EFT v8.0 (7 VC)			EFT v8.0 (7 VC)	2015-2030	5	140	1	7	16
UK EFT v8.0 (8 VC)			EFT v8.0 (8 VC)	2015-2030	5	140	1	8	16
UK EFT v7.0 (6 VC)		EFT v7.0 (2016), EFT v6.0.1 (2014) and Highways Agency (2003)*	EFT v7.0 (6 VC)	2013-2030	5	140	1	6	16
UK EFT v7.0 (7 VC)			EFT v7.0 (7 VC)	2013-2030	5	140	1	7	16
UK EFT v7.0 (8 VC)			EFT v7.0 (8 VC)	2013-2030	5	140	1	8	16
UK EFT v7.0 (8 VC)			EFT v7.0 (8 VC)	2013-2030	5	140	1	8	16
UK EFT v6.0.1 (2 VC)		EFT v6.0.1 (2014) and Highways Agency (2003)**	EFT v6.0.1 (2 VC)	2008-2030	5	140	1	2	16
UK EFT v6.0.1 (6 VC)			EFT v6.0.1 (6 VC)	2008-2030	5	140	1	6	16
UK EFT v6.0.1 (7 VC)			EFT v6.0.1 (7 VC)	2008-2030	5	140	1	7	16
UK EFT v6.0.1 (8 VC)			EFT v6.0.1 (8 VC)	2008-2030	5	140	1	8	16
UK EFT v5.1 (2 VC)		EFT v5.1 (2012) and Highways Agency (2003)**	EFT v5.1 (2 VC)	2008-2030	5	140	1	2	16
UK EFT v5.1 (6 VC)			EFT v5.1 (6 VC)	2008-2030	5	140	1	6	16
UK EFT v5.1 (7 VC)			EFT v5.1 (7 VC)	2008-2030	5	140	1	7	16
UK EFT v5.1 (8 VC)			EFT v5.1 (8 VC)	2008-2030	5	140	1	8	16

Table 3.4 – Summary of older emission factor datasets (part 2). + The VOC emission factors are derived from the UK EFT v6.0.1 dataset. * The VOC emission factors are derived from the UK EFT v6.0.1 and emission factors for CO are derived from the Highways Agency (2003) dataset. ** Emission factors for CO are derived from the Highways Agency (2003) dataset.

Dataset name (as displayed in the interface)	Suggested usage of dataset	Reference	Dataset name (to be used when importing)	Years	Speeds (km/hr)			Number of vehicle categories (Tables 3.6 and 3.7)	Number of road types (Table 3.9)
					Minimum	Maximum	Increment		
UK EFT v4.2 (2 VC)	These datasets should only be used for old projects that require consistency between emission factors. Any new modelling studies should use a newer dataset.	EFT v4.2 (2010) and Highways Agency (2003)*	EFT v4.2 (2 VC)	2006-2025	5	120	1	2	6
UK EFT v4.2 (6 VC)			EFT v4.2 (6 VC)	2006-2025	5	120	1	6	6
UK DMRB 1999	These datasets have been superseded by the DMRB IAN 185/15 dataset and should not be used for new modelling studies.	Highways Agency (1999)	DMRB 1999	1996-2025	5	130	5	2	1
UK DMRB 2003		Highways Agency (2003)	DMRB 2003	1996-2025	5	130	5	2	3
Chinese MoT 2006	This dataset should only be used for modelling studies within China.	ADMS-EIA (2010)	Chinese MoT 2006	n/a**	5	100	1	3	10

Table 3.5 – Summary of older emission factor datasets (part 3). * Emission factors for CO are derived from the Highways Agency (2003) dataset. ** The Chinese MoT 2006 dataset is not categorised by year, but by road type.

Dataset	Vehicle categories	Vehicle category name (in the interface)	Vehicle category description	Vehicle category name (to be used when importing)	Vehicle cross sectional area (m²)
UK EFT v13.1 (2 VC) **UK EFT v12.1 (2 VC)** **UK EFT v12.0 (2 VC)** **UK EFT v11.0 (2 VC)** **UK EFT v10.1 (2 VC)** **UK EFT v9.0 (2 VC)** **UK EFT v8.0 (2 VC)** **UK EFT v7.0 (2 VC)** **UK EFT v6.0.1 (2 VC)** **UK EFT v5.1 (2 VC)** **UK EFT v4.2 (2 VC)** **UK DMRB IAN 185/15**	2	Light duty vehicle	Any vehicle < 3.5t	light duty vehicle	4
		Heavy duty vehicle	Any vehicle ≥ 3.5t	heavy duty vehicle	16
UK EFT v13.1 (6 VC) **UK EFT v12.1 (6 VC)** **UK EFT v12.0 (6 VC)** **UK EFT v11.0 (6 VC)** **UK EFT v10.1 (6 VC)** **UK EFT v9.0 (6 VC)** **UK EFT v8.0 (6 VC)** **UK EFT v7.0 (6 VC)** **UK EFT v6.0.1 (6 VC)** **UK EFT v5.1 (6 VC)** **UK EFT v4.2 (6 VC)**	6	Car	Passenger car < 3.5t	car	4
		Taxi (‘black cab’)	London taxi < 3.5t	taxi	4
		LGV	Light goods vehicle < 3.5t	LGV	8
		Motorcycle	Motorcycle / moped < 3.5t	motorcycle	2
		HGV	Heavy goods vehicle ≥ 3.5t	HGV	16
		Bus	Bus / coach ≥ 3.5t	bus	12
UK EFT v13.1 (7 VC) **UK EFT v12.1 (7 VC)** **UK EFT v12.0 (7 VC)** **UK EFT v11.0 (7 VC)** **UK EFT v10.1 (7 VC)** **UK EFT v9.0 (7 VC)** **UK EFT v8.0 (7 VC)** **UK EFT v7.0 (7 VC)** **UK EFT v6.0.1 (7 VC)** **UK EFT v5.1 (7 VC)**	7	Car	Passenger car < 3.5t	car	4
		Taxi (‘black cab’)	London taxi < 3.5t	taxi	4
		LGV	Light goods vehicle < 3.5t	LGV	8
		Motorcycle	Motorcycle / moped < 3.5t	motorcycle	2
		Rigid HGV	Rigid heavy goods vehicle ≥ 3.5t	rigid HGV	16
		Articulated HGV	Artic heavy goods vehicle ≥ 3.5t	artic HGV	16
		Bus	Bus / coach ≥ 3.5t	bus	12

Table 3.6 – Summary of ADMS-Urban emission factor datasets by vehicle category (part 1).

Dataset	Vehicle categories	Vehicle category name (in the interface)	Vehicle category description	Vehicle category name (to be used when importing)	Vehicle cross sectional area (m^2)
UK EFT v11.0 (8 VC) **UK EFT v10.1 (8 VC)** **UK EFT v9.0 (8 VC)** **UK EFT v8.0 (8 VC)** **UK EFT v7.0 (8 VC)** **UK EFT v6.0.1 (8 VC)** **UK EFT v5.1 (8 VC)**	8	Petrol car	Petrol passenger car < 3.5t	petrol car	4
		Diesel car	Diesel passenger car < 3.5t	diesel car	4
		Taxi (‘black cab’)	London taxi < 3.5t	taxi	4
		LGV	Light goods vehicle < 3.5t	LGV	8
		Motorcycle	Motorcycle / moped < 3.5t	motorcycle	2
		Rigid HGV	Rigid heavy goods vehicle ≥ 3.5t	rigid HGV	16
		Articulated HGV	Artic heavy goods vehicle ≥ 3.5t	artic HGV	16
		Bus	Bus / coach ≥ 3.5t	bus	12
UK EFT v13.1 (9 VC) **UK EFT v12.1 (9 VC)** **UK EFT v12.0 (9 VC)**	9	Petrol car	Petrol passenger car < 3.5t	petrol car	4
		Diesel car	Diesel passenger car < 3.5t	diesel car	4
		Electric car	Electric passenger car < 3.5t	electric car	4
		Taxi (‘black cab’)	London taxi < 3.5t	taxi	4
		LGV	Light goods vehicle < 3.5t	LGV	8
		Motorcycle	Motorcycle / moped < 3.5t	motorcycle	2
		Rigid HGV	Rigid heavy goods vehicle ≥ 3.5t	rigid HGV	16
		Articulated HGV	Artic heavy goods vehicle ≥ 3.5t	artic HGV	16
		Bus	Bus / coach ≥ 3.5t	bus	12
UK EFT v13.1 (11 VC) **UK EFT v12.1 (11 VC)** **UK EFT v12.0 (11 VC)**	11	Petrol car	Petrol passenger car < 3.5t	petrol car	4
		Diesel car	Diesel passenger car < 3.5t	diesel car	4
		Electric car	Electric passenger car < 3.5t	electric car	4
		Taxi (‘black cab’)	London taxi < 3.5t	taxi	4
		Petrol LGV	Petrol light goods vehicle < 3.5t	petrol LGV	8
		Diesel LGV	Diesel light goods vehicle < 3.5t	diesel LGV	8
		Electric LGV	Electric light goods vehicle < 3.5t	electric LGV	8
		Motorcycle	Motorcycle / moped < 3.5t	motorcycle	2
		Rigid HGV	Rigid heavy goods vehicle ≥ 3.5t	rigid HGV	16
		Articulated HGV	Artic heavy goods vehicle ≥ 3.5t	artic HGV	16
		Bus	Bus / coach ≥ 3.5t	bus	12
Chinese MoT 2006	3	Light duty vehicle	Any vehicle < 3.5t	Chinese LDV	4
		Medium duty vehicle	Any vehicle ≥ 3.5t and < 12t	Chinese MDV	8
		Heavy duty vehicle	Any vehicle ≥ 12t	Chinese HDV	16

Table 3.7 – Summary of ADMS-Urban emission factor datasets by vehicle category (part 2).

Dataset	No. of road types	Road type name (in the interface)	Road type description	Road type name (to be used when importing)
UK EFT v13.1 **UK EFT v12.1** **UK EFT v12.0** **UK EFT v11.0** **UK EFT v10.1** **UK EFT v9.0** **UK EFT v8.0** **UK EFT v7.0** **UK EFT v6.0.1** **UK EFT v5.1**	16	London (central)	Roads in central London (excluding motorways)	London (central)
		London (inner)	Roads in inner London (excluding motorways)	London (inner)
		London (outer)	Roads in outer London (excluding motorways)	London (outer)
		London (motorway)	Motorways in London	London (motorway)
		England (urban)	Roads in urban areas in England outside London (excluding motorways)	England (urban)
		England (rural)	Roads in rural areas in England outside London (excluding motorways)	England (rural)
		England (motorway)	Motorways in England	England (motorway)
		Scotland (urban)	Roads in urban areas in Scotland (excluding motorways)	Scotland (urban)
		Scotland (rural)	Roads in rural areas in Scotland (excluding motorways)	Scotland (rural)
		Scotland (motorway)	Motorways in Scotland	Scotland (motorway)
		Wales (urban)	Roads in urban areas in Wales (excluding motorways)	Wales (urban)
		Wales (rural)	Roads in rural areas in Wales (excluding motorways)	Wales (rural)
		Wales (motorway)	Motorways in Wales	Wales (motorway)
		Northern Ireland (urban)	Roads in urban areas in Northern Ireland (excluding motorways)	NI (urban)
		Northern Ireland (rural)	Roads in rural areas in Northern Ireland (excluding motorways)	NI (rural)
		Northern Ireland (motorway)	Motorways in Northern Ireland	NI (motorway)
UK DMRB IAN 185/15	7	London (central)	Roads in central London (excluding motorways)	London (central)
		London (inner)	Roads in inner London (excluding motorways)	London (inner)
		London (outer)	Roads in outer London (excluding motorways)	London (outer)
		London (motorway)	Motorways in London	London (motorway)
		Urban (not London)	Roads in UK urban areas outside London (excluding motorways)	Urban (not London)
		Rural (not London)	Roads in UK rural areas outside London (excluding motorways)	Rural (not London)
		Motorway (not London)	UK motorways outside London	Mway (not London)

Table 3.8 – Summary of EFT v13.1, v12.1, v12.0, v11.0, v10.1, v9.0, v8.0, v7.0, v6.0.1 and v5.1 and DMRB IAN/185/15 emission factor datasets by Road type.

Dataset	Number of road types	Road type name (in the interface)	Road type description	Road type name (to be used when importing)
UK EFT v4.2	6	Urban (not London)	Roads in UK urban areas outside London (excluding motorways)	Urban (not London)
		Rural (not London)	Roads in UK rural areas outside London (excluding motorways)	Rural (not London)
		Motorway (not London)	UK motorways outside London	Mway (not London)
		Urban (London)	Roads in London	Urban (London)
		Rural (London)	*	Rural (London)
		Motorway (London)	Motorways in London	Mway (London)
UK DMRB 2003	3	Urban	Roads in UK urban areas (excluding motorways)	Urban
		Rural	Roads in UK rural areas (excluding motorways)	Rural
		Motorway	UK motorways	Motorway
UK DMRB 1999	1	All roads	All roads	All roads
Chinese MoT 2006	10	100% Pre	100% Pre-Euro Chinese vehicles	C MoT 2006_0
		100% EU 1	100% Euro 1/I Chinese vehicles	C MoT 2006_1
		100% EU 2	100% Euro 2/II Chinese vehicles	C MoT 2006_2
		80% Pre, 20% Euro	80% Pre-Euro and 20% Euro 1/I Chinese vehicles	C MoT 2006_3
		60% Pre, 40% Euro	60% Pre-Euro and 40% Euro 1/I Chinese vehicles	C MoT 2006_4
		40% Pre, 60% Euro	40% Pre-Euro and 60% Euro 1/I Chinese vehicles	C MoT 2006_5
		15% Pre, 70% Euro 1, 15% Euro 2	15% Pre-Euro, 70% Euro 1/I and 15% Euro 2/II Chinese vehicles	C MoT 2006_6
		60% Euro 1, 40%	60% Euro 1/I and 40% Euro 2/II Chinese vehicles	C MoT 2006_7
		40% Euro 1, 60%	40% Euro 1/I and 60% Euro 2/II Chinese vehicles	C MoT 2006_8
		20% Euro 1, 80%	20% Euro 1/I and 80% Euro 2/II Chinese vehicles	C MoT 2006_9

Table 3.9 – Summary of emission factor datasets by Road type (pre-EFT v5.1). * The 'Rural (London)' category is specified in the source data, so the emissions have been included, but it is not clear what road type this is appropriate for.

The traffic fleet on different roads varies. For example, the fleet in an urban area is likely to have a lower proportion of HGVs than on a motorway. Further, the fleet may vary geographically. For example, the London fleet of vehicles differs from that in other urban areas.

In order to represent the variation of the fleet correctly on different roads, the model includes a **Road type** classification. The different classifications are summarised in **Table 3.8** for the EFT v13.1, v12.1, v12.0, v11.0, v10.1, v9.0, v8.0, v7.0, v6.0.1 and v5.1 datasets and the UK DMRB IAN 185/15 dataset and in **Table 3.9** for all other datasets.

The "Taxi ('black cab')" emissions for non-London regions for the six, seven and eight vehicle category EFT v8.0, v7.0, v6.0.1 and v5.1 datasets have been taken to be the same as the London emissions for these vehicles, as these EFT versions do not contain taxi emissions for non-London regions[1]. As London does not have any rural route types, non-London rural route types have the same taxi emissions as urban route types. Note that the latest versions of EFT, EFT v13.1, v12.1, v12.0, v11.0, v10.1 and v9.0, do contain taxi emissions for all regions.

Non-London rural taxi emissions are the same as non-London urban taxi emissions for all pollutants, despite taxi CO emissions from DMRB 2003 having separate factors for urban and rural route types. This is to ensure all pollutants are consistent between the two route types.

Changing between traffic emissions datasets

It is possible to change between emission factor datasets. This may be appropriate, for example, when updating a model file to use the most up-to-date traffic emission factors available.

---

*It is only possible to change between emission factor datasets that use the same number of vehicle categories if road sources have already been defined (for details of vehicle categories, refer to **Tables 3.6** and **3.7**).*

In order to update a model file to use a different emission factor dataset, select the '...' button next to **Dataset:** in the **Calculation of road traffic emissions** section of the **Source** screen. The screen shown in **Figure 3.6** will be displayed.

[1] This assumption may be overly optimistic for taxis outside London; within London, taxis must comply with TfL's taxi emissions strategy.

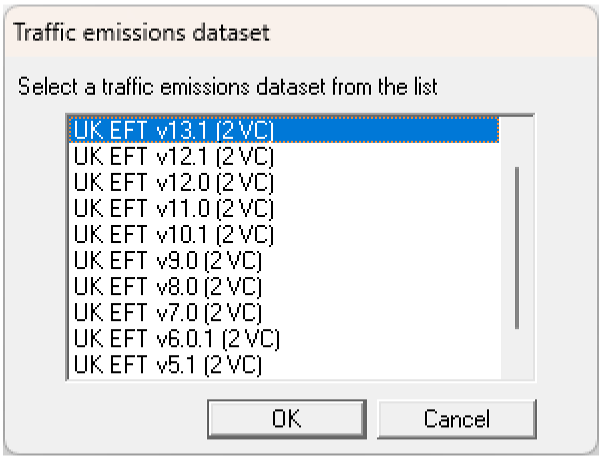


Figure 3.6 – The **Traffic emissions dataset** screen

Select a new emission factor dataset and then click **OK**. This will open the screen shown in **Figure 3.7**.

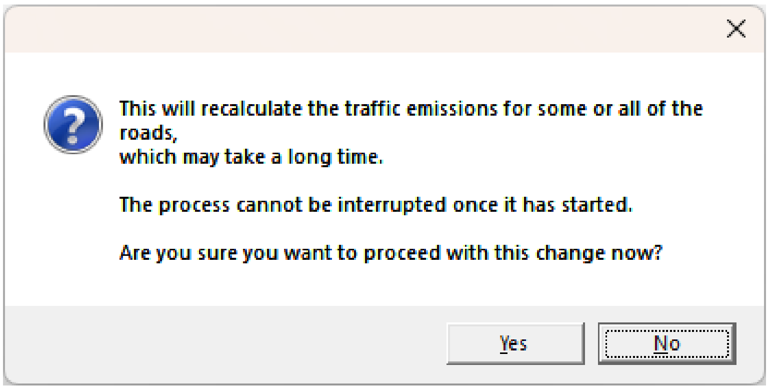


Figure 3.7 – Warning message when the traffic dataset is changed.

Click **Yes** to proceed.

Some datasets use different road types (for details of the road types associated with each dataset, please refer to **Tables 3.8** and **3.9**). Therefore, when changing between datasets, you may be prompted to select an alternative **Road type**, as shown in **Figure 3.8**.

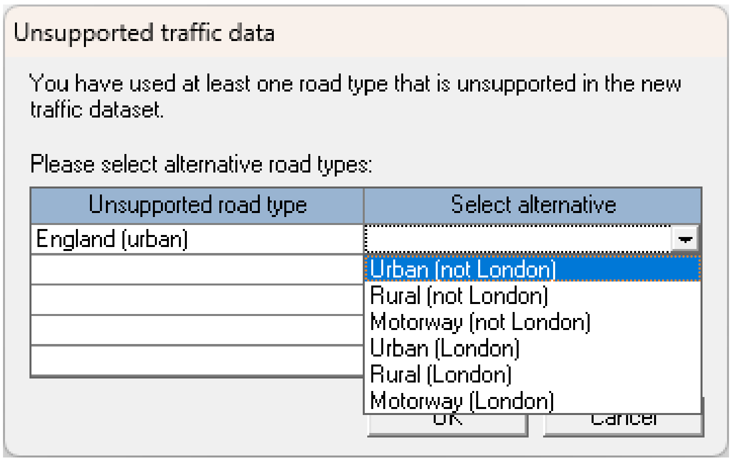


Figure 3.8 – Dialogue box displayed when the dataset is changed

Once a new **Road type** has been selected, click **OK**. The emission factor dataset and **Road type** will be changed for all roads included in the model file. Any new roads will use the current emission factor dataset displayed in the **Source** screen.

When changing between datasets which contain different pollutants you may wish to add or remove pollutants to ensure consistency with the current dataset. Refer to the paragraph on *Entering traffic flow data* for more details.

The EFT v13.1, v12.1, v12.0, v11.0, v10.1, v9.0 and 8.0 datasets are the only datasets containing emission factors for NO_2. When changing to the EFT v13.1, v12.1, v12.0, v11.0, v10.1, v9.0 or v8.0 dataset for existing road sources, any user-defined emissions of NO_2 will be overwritten with values calculated using the EFT emission factors. If there are no user-defined emissions of NO_2, you may wish to add NO_2 as an emission pollutant when changing to the EFT v13.1, v12.1, v12.0, v11.0, v10.1, v9.0 or v8.0 dataset.

Entering traffic flow data

The emission factors on the right hand side of the **Traffic flows** table in the **Emissions** screen will be automatically calculated based on the traffic flow data provided in the left hand side of the table (and the information in the **Properties** box).

In the **Properties** box, select the emission factor **Year** for the current road source from the drop-down list. In the same way, select the road **Type** for the current road source.

Note that you can use the '...' buttons next to ***Emission year:*** *and* ***Road type:*** *on the* ***Source*** *screen to change the scenario year and the road type respectively for all roads simultaneously, as an alternative to changing them for individual roads via the* ***Emissions*** *screen.*

Data relating to the traffic on the road being considered are entered in the **Traffic flows** box. To add a new traffic flow, click on the **New** button; up to 40 traffic flows can be defined. For each traffic flow:

- Select a vehicle category (as summarised in **Tables 3.6** and **3.7**) from the list given in the **Vehicle category** column.
- Enter the speed in the **Average speed km/hr** column. Allowed ranges are summarised in **Tables 3.2** to **3.5**.
- Enter the average hourly traffic flow in the **Vehicles per hour** column; the minimum value is 0, the maximum is 100 000.
- If using an EFT v9.0 dataset and a heavy duty vehicle category is selected, enter the percentage of vehicles in this category that are travelling uphill in the **Uphill %** column. The default is 50 %.

The emission factors in g/km for all pollutants are displayed in the **Emission Factors g/km per vehicle** area of the table. These factors are not editable.

The total emission rate for the road in g/km/s is displayed in the **Pollutant species** table. It is possible to add emissions of extra pollutants to this table (for example, SO_2) by clicking on **New** and selecting a pollutant from the list. If the required pollutant is not included in the list, it must first be added to the table in the **Pollutants** screen (please refer to Section 3.1.5 for more details).

In the **Pollutant species** table, emission rates calculated directly from traffic flows are displayed in blue, and those entered by the user are displayed in black.

Right-clicking on a pollutant in the Pollutant species table allows that pollutant to be added to or removed from all sources of the current type, e.g. all road sources. This can be useful when switching between traffic datasets with different calculated pollutants.

If time-varying emission factors are being modelled, as described in Section 4.1, the emission rate displayed here will be multiplied by the factor that has been specified for each hour of the day when the model is run.

Traffic-produced turbulence

When road sources are modelled, the effects of the additional turbulence caused by the traffic are included, as described in Section 9.6.1. For road sources for which traffic flow data have been entered in the **Traffic flows** table, these traffic flows are used in the calculation of the traffic produced turbulence.

It is still possible to add traffic flow data to road sources whose emissions are entered directly, for the purposes of improving the estimate of traffic produced turbulence. For these sources, the emission factors on the right hand side of the ***Traffic flows*** *table can be ignored as they are not used.*

For road sources where the emissions have been entered directly and no traffic flow data have been provided, a back calculation from emissions of NO_x, PM_{10} or VOC is used to determine the number of vehicles. This calculation is only performed for road sources that emit one of the standard pollutants NO_x, PM_{10} or VOC. The calculation used to estimate the number of vehicles uses a speed of 30 km/hr and a default traffic split of 95% light duty vehicles and 5% heavy duty vehicles. If this vehicle split is inappropriate, it is possible to alter the vehicle split used in estimating the number of vehicles using an additional input file. Create a new additional input file, or edit the existing one (see Section 3.1.8), enable the **Vehicle Split** option (keyword VEHICLESPLIT) and then enter the **Percentage of light vehicles**, as shown in **Figure 3.9**.

The time varying emissions factor used in the calculation of traffic induced turbulence is calculated based on the time varying emissions factors for all emitted pollutants. If pollutant-specific time varying profiles are used it is important to ensure all emitted pollutants are covered by appropriate profiles.

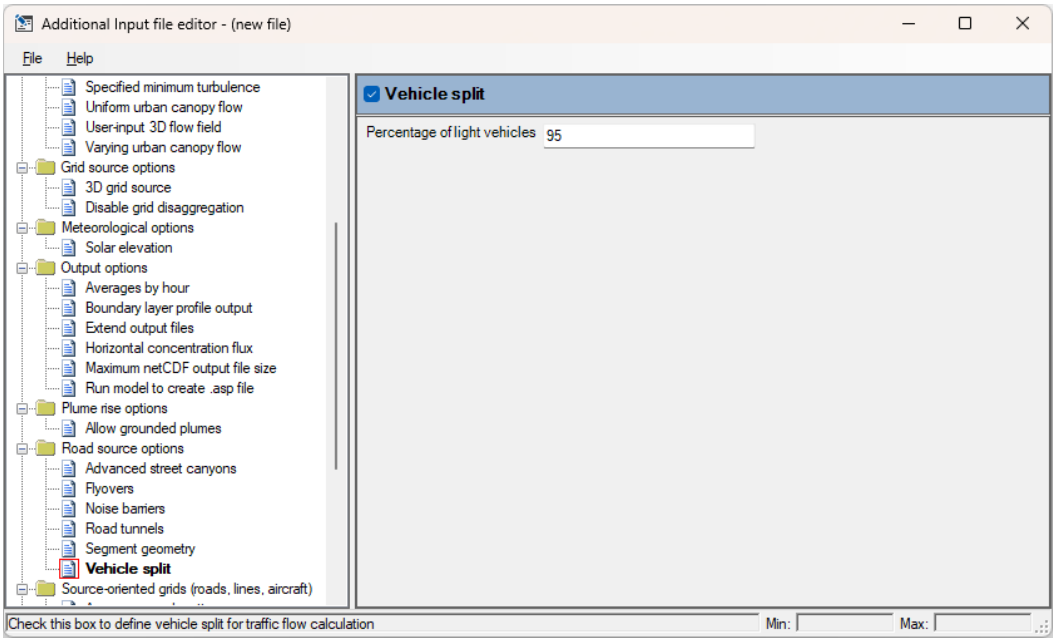


Figure 3.9 – The **Vehicle Split** option in the **Additional Input file editor**.

3.2.2 Industrial Sources

Creating an industrial source using the ADMS-Urban interface

Step 1 Select **Sources**, then **Industrial Sources** from the drop-down list. Select the required type (**Point**, **Area**, **Volume** or **Line**) from the list and click the **New** button to create a source of that type (**Figure 3.10**).

Step 2 Change the default parameters to those of your particular scenario by clicking on a cell in the source table so that it is highlighted and editing the value. See below for a description of each parameter.

The columns of the industrial source table are resizable.

Step 3 For area, volume or line sources, click the **Geometry** button to enter the coordinates of the vertices. Area and volume sources must have at least three, and up to 50, vertices defined in either clockwise or anti-clockwise order around the source. The shape defined must be convex. Line sources may have up to 51 vertices (i.e. 50 segments). Adjacent vertices must be at least 1m apart.

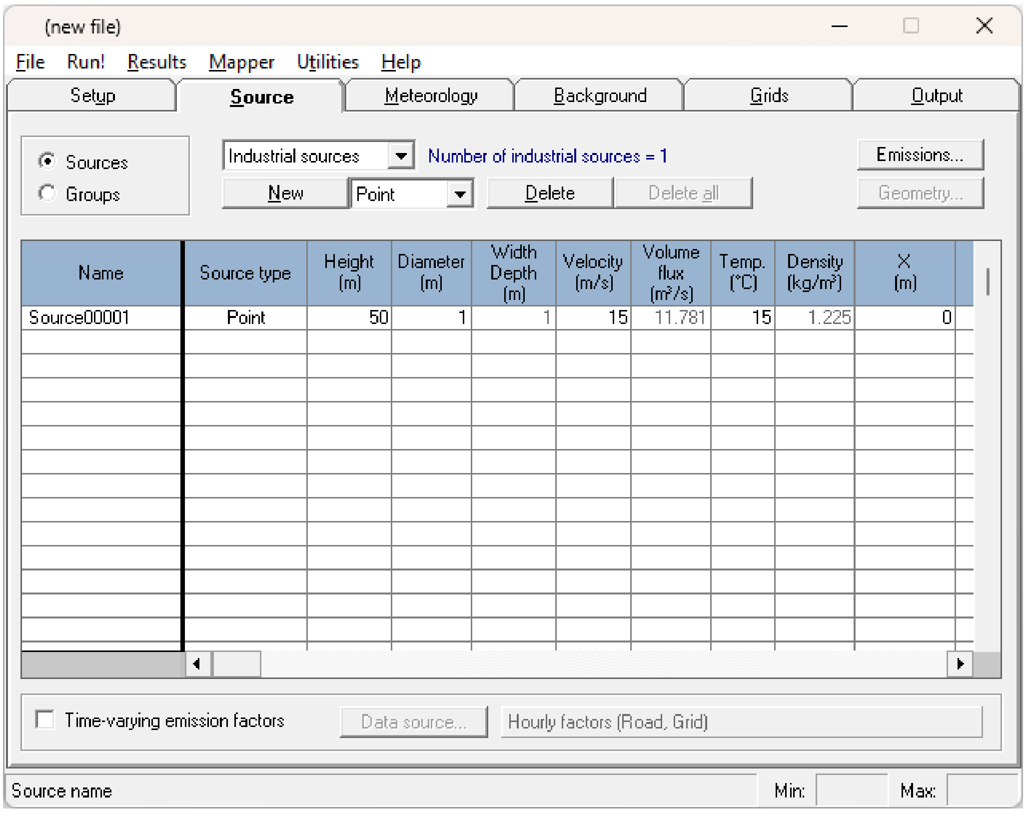


Figure 3.10 – The **Source** screen for industrial sources.

You can also edit values or names in the source table using the arrow keys to navigate to a cell that will be highlighted by a dashed box. A new value may be typed in directly without further clicking on the cell.

To delete an industrial source, highlight the source **Name** in the table by clicking on it, then click on the **Delete** button. There is also a right-click option to **Delete all but this** industrial source from the table. The **Delete all** button brings up the **Delete sources by type** screen, as shown in **Figure 3.11**, which allows for all sources of the chosen type(s) to be deleted.

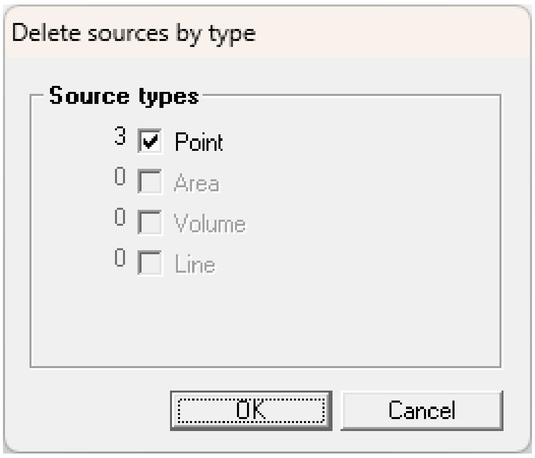


Figure 3.11 – The **Delete sources by type** screen for industrial sources.

Industrial source parameters

The following list gives a description of each input parameter in the industrial source table, and the maximum and minimum values that may be entered. Note that not all these parameters are visible at the same time in the source table. They can all be displayed by scrolling to the left and right.

1. **Name**: it is advised to give a meaningful source name for the study, for example “incinerator”. By default, a name such as Source00001 is used. The source name can be up to 30 characters long and must not contain commas.
2. **Source type**: type of the source.

   Point point source
   Area area source
   Volume volume source
   Line line source

   The default source type depends on the choice in the menu to the right of the **New** button.

3. **Main building**: if buildings are being modelled, then any of the buildings defined can be selected as the main building for each source. The default **Main building** is **“(Main)”** indicating the building ticked on the **Buildings** screen. For further details, please refer to Section 4.15.3. This option is only available for point sources.
4. **Height (m)**: height of source above the ground. For a volume source, it is the mid-height of the volume above the ground.

   Minimum = 0 m
   Maximum = 15 000 m
   Default = 50 m

5. **Diameter (m)**: internal diameter of a point source.

   Minimum = 0.001 m
   Maximum = 100 m
   Default = 1 m

6. **Width Depth (m)**: width of a line source or vertical depth of a volume source.

   Minimum = 0.001 m
   Maximum = 1 000 m
   Default = 1 m

7. **Velocity (m/s):** vertical velocity of release at source exit (in m/s if **Actual** is selected in 14 or in Nm/s if **NTP** is selected where N means normal pressure and temperature).

   Minimum = 0 m/s
   Maximum = 1 000 m/s
   Default = 15 m/s

8. **Volume flux (m^3/s)**: volume flow rate of the release (in m^3/s if **Actual** is selected in 14 or in Nm^3/s if **NTP** is selected).

Minimum = 0 m^3/s
Maximum = 10^9 m^3/s
Default = 11.781 m^3/s

9. **Temp. (°C)**: temperature of the release.

   Minimum = -100°C
   Maximum = 5 000°C
   Default = 15°C

10. **Density (kg/m³)**: density of the whole release.

    Minimum = 0.01 kg/m^3
    Maximum = 2 kg/m^3
    Default = 1.225 kg/m^3

11. **X (m)**, **Y (m)**: X and Y coordinates of the centre of a point source.

    Minimum = -9 999 999 m
    Maximum = 9 999 999 m
    Default = 0 m

    The large magnitude of the minimum and maximum values allows the user to input UK National Grid coordinates or the worldwide Universal Transverse Mercator coordinates (UTM).

12. **Density, Temp. or Ambient**: temperature or density of the release. Type **T** to enter a (constant) temperature (**T**), **R** to enter a density (**RHO**), or **A** if the release is at ambient temperature and density (**Ambient**).

    Default = **T**

13. **Vel. or Vol.**: exit velocity or volume flow rate. Enter **Exit V.** for exit velocity or **Vol.** for volumetric flow rate.

    Default = **Exit V.**

14. **Actual or NTP**: emission parameters are given at normal temperature and pressure (NTP = 1 atm and 273.15 K) or at the actual release temperature and pressure.

    Default = **Actual**

15. **Cp (J/K/kg)**: specific heat capacity of the whole release.

    Minimum = 1 J/K/kg
    Maximum = 10^5 J/K/kg
    Default = 1 012 J/K/kg

16. **Mol. mass (g)**: molecular mass of the whole release (mass of one mole of the material).

    Minimum = 1 g
    Maximum = 300 g
    Default = 28.966 g (typical value for air)

The source name (1), source type (2) and source height (4) should always be entered. The efflux type (13) should be entered for all source types other than volume sources. **Table 3.10** summarises the required source dimension/orientation data for each source type. The source geometry parameters 5, 6 and 11 should be defined according to the source type. Efflux parameters 7 to 8 should be specified as summarised in **Table 3.11**. An entry for parameter 3 is only required if buildings are included in the model run.

Default values (those for air) would normally be used for **Cp** (15) and **Mol. Mass** (16) unless a very pure gas is being emitted. If **Cp** and **Mol. Mass** are being altered it must be ensured that these two parameters are set to consistent values.

Source type	Height	Diameter	Width Depth	Xp	Yp
Point	✓	✓	✗	✓	✓
Area	✓	✗	✗	✗	✗
Volume	✓	✗	✓	✗	✗
Line	✓	✗	✓	✗	✗

Table 3.10 − Source dimensions that must be specified for each source type.

Efflux type	Actual or NTP	Velocity	Volume flux	Density, Temp. or Ambient
Exit V.	✓	✓	✗	✓
Vol.	✓	✗	✓	✓

Table 3.11 − Efflux parameters that must be specified for each efflux type.

Source geometry

The source type of an individual source may be selected as point, area, volume or line. For each source type, the position and size are defined differently.

Recall that the source height (Zs) has already been defined (list element 4 above).

The geometry of a **Point** source should be given as follows:

Point source

A point source is a release at the specified height, located at (**X**,**Y**). It is assumed to be horizontal and circular in cross-section, with diameter as specified by the user. Sources of large diameter should be modelled as area sources.

For an **Area**, **Volume** or **Line** source, the **Geometry...** button becomes available to allow the definition of the source vertices (see **Figure 3.12**). Adjacent vertices for such sources should be more than 1 m apart. The geometry parameters for volume and line sources are shown in **Figure 3.13**. **< Back** and **Next >** buttons allow the user to change between sources.

Area source

An area source is a release at the specified height over a horizontal *convex* polygon with between 3 and 50 vertices, e.g. emissions from a sewage tank.

The X and Y coordinates of the vertices of the area source must be specified by the user in order around the source. The number of vertices currently defined is shown in brackets next to the word **Vertices**. **Figure 3.12** (left) shows the geometry screen for an area source. To add a new vertex click on the **New** button and edit the X and Y coordinates as appropriate. To remove a vertex select it and press the **Delete** button. To insert a vertex between two existing vertices (or before/after the current start/end vertex), right-click on one of the existing vertices and select **Insert above** or **Insert below** as appropriate. The initial coordinates of a new vertex inserted between two existing vertices will take the mid-point between those two vertices, whereas a new start/end vertex will take the same coordinates as the start/end vertex it is replacing; the user can then further edit these values as appropriate.

Volume source

A volume source is a release from an area source with vertical extent but no plume rise, e.g. fugitive emissions around a building. The vertical extent of the source, ΔZ, as specified in the **Width Depth** column of the source table, is the height over which the source is initially well-mixed, and the source height (Zs) is the height of the *middle* of the vertical extent (see **Figure 3.13**). The source must be wholly above ground level. The horizontal cross-section of the volume is defined as for an area source, i.e. it must be a *convex* polygon with between 3 and 50 vertices, defined by the X and Y coordinates of the vertices specified in order around the polygon. The number of vertices currently defined is shown in brackets next to the word **Vertices**. Note that since no plume rise is modelled, the efflux parameters are automatically greyed out. **Figure 3.12** (middle) shows the geometry screen for the horizontal area of a volume source. Vertices are added and deleted in the same way as for area sources, described above.

Line source

A line source is a release emitted uniformly along a multi-segment line defined by between 2 and 51 vertices at the source height. The number of vertices currently defined is shown in brackets next to the word **Vertices**. The line is defined by the X and Y coordinates of the centre points of the end of each of its segments, and its width (L), as specified in the **Width Depth** column of the source table, distributed equally about its centreline (see **Figure 3.13**). Vertices are added and deleted in the same way as for area sources, described above.

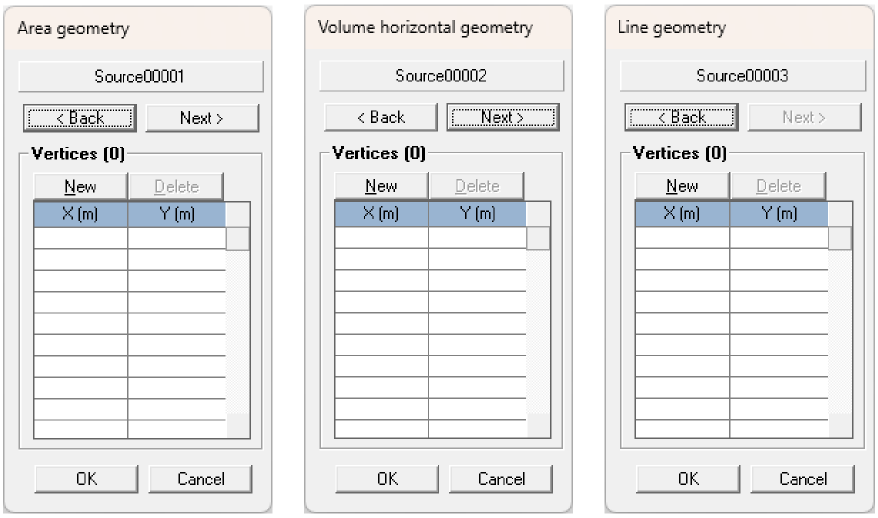


Figure 3.12 − Screen for the definition of the source geometry: area source (left), volume source (middle), line source (right).

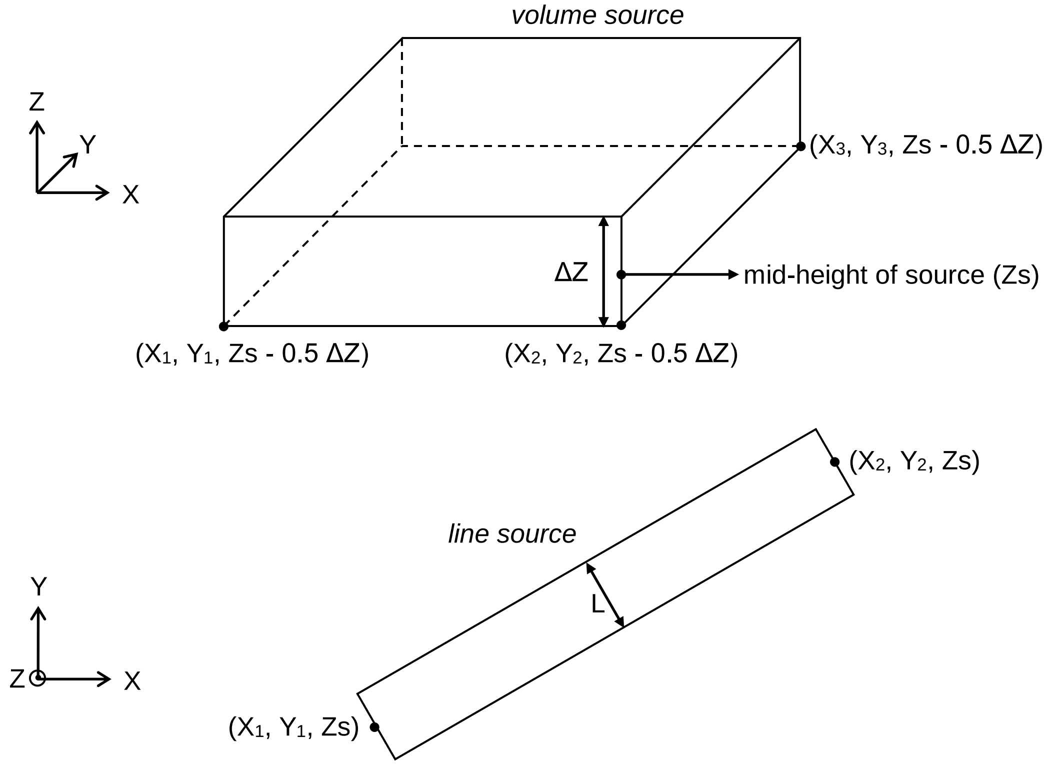


Figure 3.13 – Definition of geometry parameters for a volume source (top) and a single-segment line source (bottom). X_1, Y_1, etc., refer to the vertex coordinates entered, while Zs is the source height.

Source emissions

To enter emissions data for individual sources, select **Sources** on the **Source** screen, then click on the **Emissions...** button. The **Emissions** screen is then displayed (see **Figure 3.14**). Up to 80 pollutants may be emitted by each source; these emissions are defined using the **New** and **Delete** buttons. Properties of pollutants are defined in ADMS-Urban by means of the **Palette of Pollutants** screen, which can be accessed by selecting **Pollutants...** from the **Emissions** screen or from the **Data...** button in the **Palette** section of the **Setup** screen of the ADMS-Urban interface.

The **< Back** and **Next >** buttons in the top right-hand corner of the **Emissions** screen allow the user to change between sources. The current source name and type is displayed at the top of the screen.

The units used to define the emission rate depend upon the source type. Unless odorous releases are being modelled, the units are as follows:

g/s	for a point source,
g/m^2/s	for an area source,
g/m^3/s	for a volume source,
g/m/s	for a line source.

If the **Odours** option has been selected, the units are:

ou_e /s	for a point source,
ou_e /m^2/s	for an area source,
ou_e /m^3/s	for a volume source,
ou_e /m/s	for a line source.

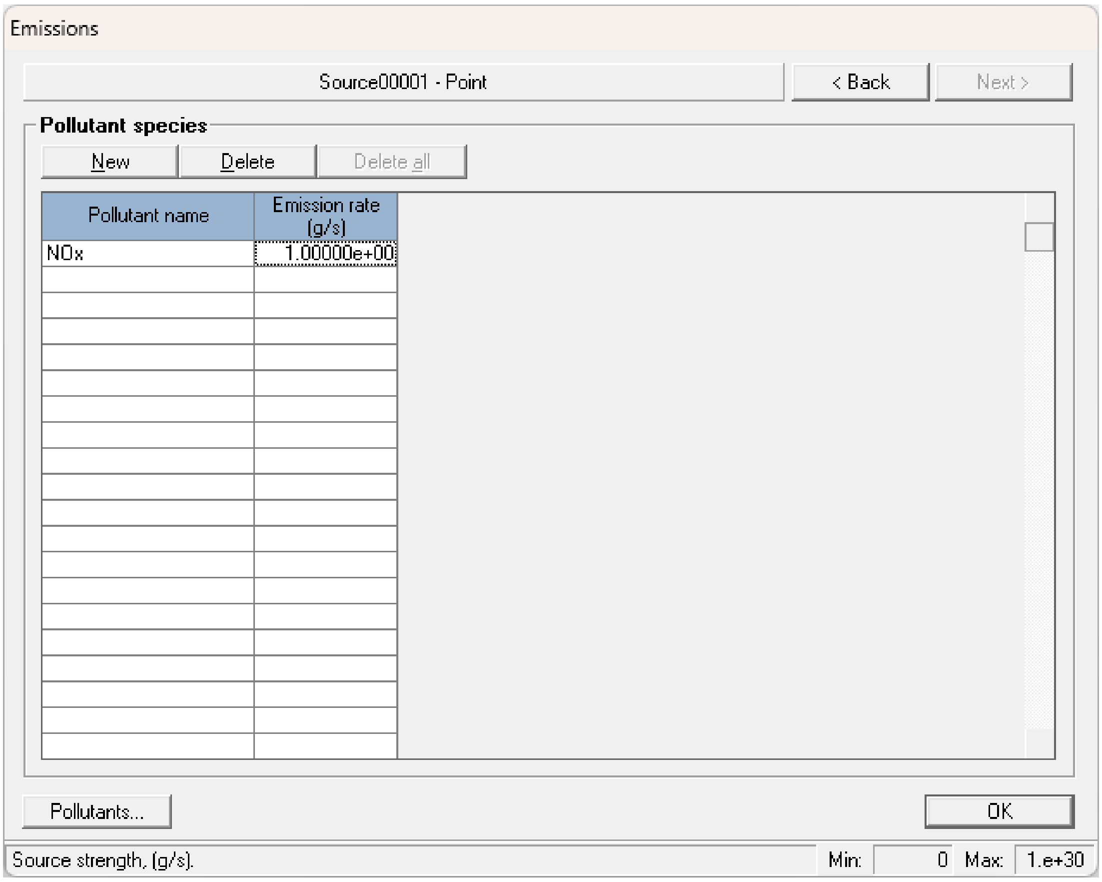


Figure 3.14 – **Emissions** screen for specifying pollutant emission rates.

Note that successive clicks on the **New** button will cause pollutants to be added to the list for this source in the order they appear in the **Palette of Pollutants** (see Section 3.1.5).

3.2.3 Grid Source

Introduction to grid sources

What is the purpose of a grid source?

ADMS-Urban uses a grid source as a fast way to model residual, poorly-defined or diffuse emissions in urban areas, such as those from domestic heating sources, and minor roads. This allows ADMS-Urban to model emissions from a greater number of sources than would be possible if each source were to be modelled explicitly.

What does a grid source consist of?

A grid source consists of a matrix of cells, where each can have a different emission rate. By default each grid source cell should contain the *total* emissions of *all pollutants* from *all sources* within that cell, such as industrial sources, road sources and residual emissions. It therefore follows that the extent of the grid source should be defined in order to cover all sources included in a model run.

Figure 3.15 shows an incorrect model set-up where a grid source has been defined with 3 cells in the horizontal plane (shaded squares) covering only some of the road sources being modelled explicitly (thick blue lines). **Figure 3.16** shows a correct alternative, where grid source cells cover the entire area within which sources are defined.

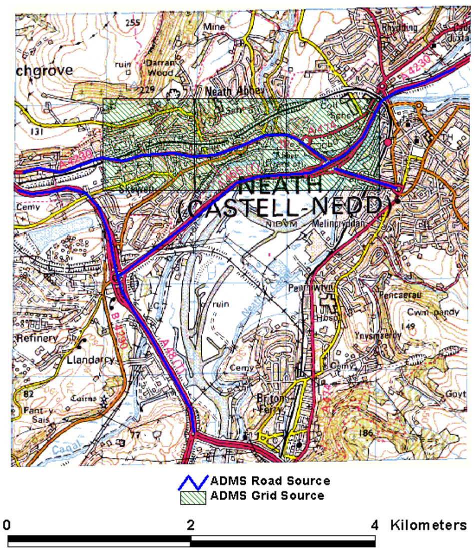


Figure 3.15 – Incorrect grid source definition, because grid source cells do not cover all other defined sources.

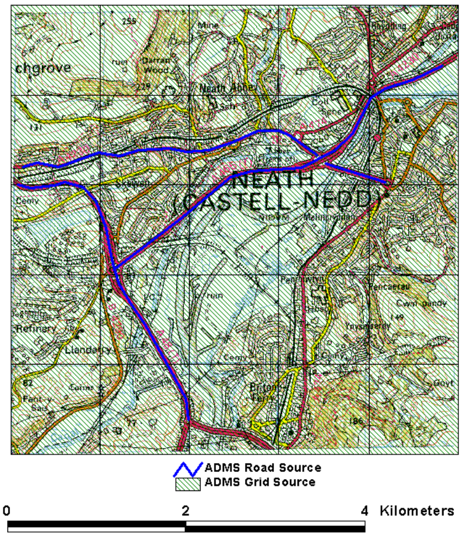


Figure 3.16 – Correct grid source definition, as grid source cells extend to cover the area of all defined sources.

How does the model treat a grid source?

Before calculating the dispersion of grid source emissions, the model disaggregates the explicitly defined source emissions from the grid source emission rates, leaving only the residual emissions. In other words, the emissions of all the industrial and road sources defined in the model file are subtracted from the grid source totals.

It is also possible to disable grid disaggregation for specific - or all - sources using the additional input file as detailed in Section 4.8.

Concentrations calculated using a grid source are modelled as if each cell in the grid source was modelled as an individual volume source, but run-times are significantly reduced due to some simplifications used in the dispersion calculations. A grid source cannot have any plume rise.

Standard grid source vs. 3D grid source

A grid source can be modelled in one of two ways:

- *Standard grid source*: This is the grid source that would be displayed in the **Source** screen, and is the typical way of defining a grid source in ADMS-Urban. A standard grid source consists of one vertical layer of identically sized grid cells, located on the ground, with a constant depth that is specified by the user. Emission rates are defined per grid cell per pollutant; any time variation in the emissions is represented via the use of time-varying emission factors, as described in Section 4.1, which are applied to the grid source as a whole. If a grid cell contains no emissions, it is not necessary for that grid cell to be included in the run, i.e. the standard grid source does not have to cover a rectangular region with no gaps.

- *3D grid source*: A 3D grid source allows for multiple vertical layers of varying depths. Emission rates are defined per grid cell per pollutant per met line, allowing for full independence of grid cell emissions in both space and time.

3D grid source information is provided via a netCDF (*.nc*) data file, with the path to the *.nc* file specified via the additional input file. A typical user of 3D grid sources is likely to be someone who is also running a regional air quality model such as CMAQ or WRF-Chem to take into account both regional and local pollutant transport and chemistry effects.

It is not possible to define both a standard and a 3D grid source in the same run.

The rest of this sub-section deals with standard grid sources. 3D grid sources are described in detail in Section 4.7.

Adding a standard grid source

A standard grid source can be added to a model file in one of three ways:

- created by hand in the ADMS-Urban model interface,
- imported from a set of comma-separated variable files, for instance from EMIT, or
- imported from an emissions inventory.

Only one standard grid source is allowed per model run. The grid source can have up to 100000 cells when defined in the interface, but only a smaller number are permitted in each model run (400 with package A, 3000 with a packages B and C of ADMS-Urban licence).

Creating a standard grid source using the ADMS-Urban interface

Step 1 To create a new standard grid source, select **Sources,** then **Grid Source** from the list and click on the **New** button. This opens the **Create Grid Source** screen, shown in **Figure 3.17** below.

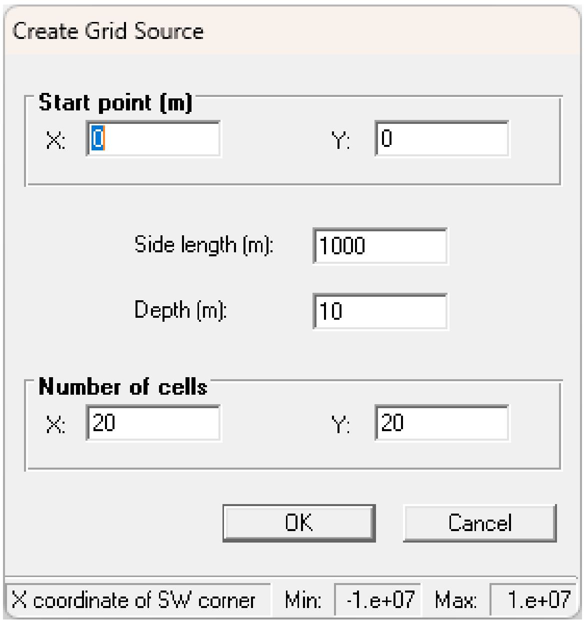


Figure 3.17 – The **Create Grid Source** screen.

Step 2 The default values shown in **Figure 3.17** create a 20 by 20 matrix of grid source cells, each of which may be considered as a volume source of side-length 1 000 m, depth 10 m resting on the ground, with the lower-left coordinate of the grid at (0,0). You can click on each of the input boxes to change the default values.

How should you determine the depth of a standard grid source?

The depth of a standard grid source should be determined by two factors:

- the height of the releases that comprise the residual emissions e.g. traffic on minor roads, low industrial stacks
- the depth through which those emissions are initially mixed, which depends on factors such as the height of buildings.

A typical value for large urban areas where most of the emissions are from traffic, such as London, is 10 m.

Step 3 Clicking **OK** creates the grid source (this procedure may take a few minutes if a large number of cells are defined). The screen shown in **Figure 3.18** appears. Each cell in the grid source is given a special name starting with the '#' character followed by two six-figure numbers corresponding to the lower left coordinate of the cell, separated by the '_' character. Note that the user cannot change the source name.

The columns of the grid source cells table are resizable.

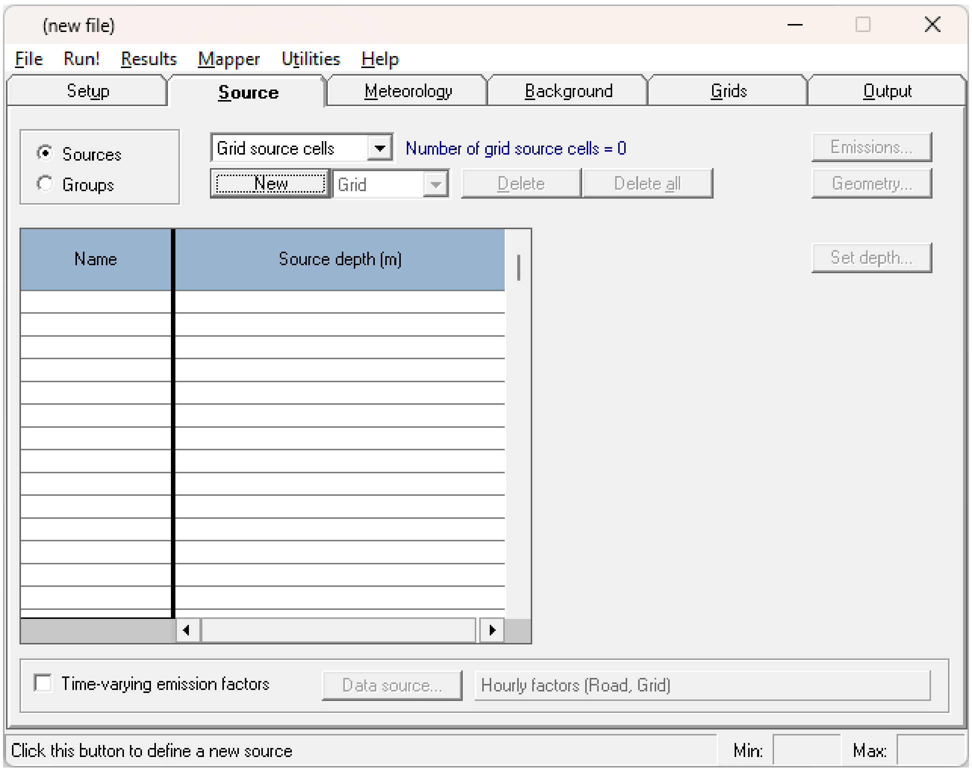


Figure 3.18 – The **Grid Source** screen.

Step 4 To specify the emissions of a grid source cell, click on the cell name to highlight it and click on the **Emissions...** button (see the G*rid source emissions* section).

Step 5 To look at the coordinates of a specific grid source cell, highlight the cell in the source table and then click on the **Geometry...** button (**Figure 3.19**).

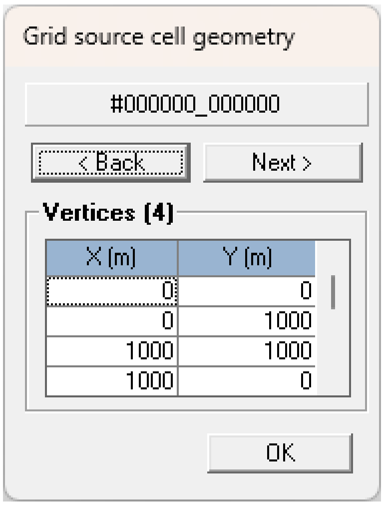


Figure 3.19 – The **Grid source cell geometry** screen.

Grid source emissions

As mentioned at the beginning of this section, the emissions of a particular grid source cell represent the total emissions of all sources within that cell, and therefore must contain all pollutants emitted by sources within that cell. For example, if one cell contains one explicitly modelled road source (emitting NO_X, VOC, PM_{10} and $PM_{2.5}$), one boiler stack (emitting NO_X, PM_{10} and SO_2) and residual emissions of NO_X, the grid source cell should have emissions of NO_X, VOC, PM_{10}, $PM_{2.5}$ and SO_2. This is important at all times, but is particularly important when modelling chemistry.

Grid source emissions are specified in g/m^2/s.

All NO_X emissions are assumed to be "NO_x as NO_2". Please refer to Appendix B for further information.

If NO_2 is being modelled, and NO_2 emissions are not explicitly defined for a grid source then the model assumes that the proportion of NO_x that is NO_2 is 10.5%. For details on how to change this value refer to Section 4.13.2.

3.2.4 Time-varying emissions

Unless otherwise specified, emission rates are assumed to be constant. However, it is possible in ADMS-Urban to model concentrations from emissions that vary with time. To select this option, check the **Time-varying emissions factors** box on the **Source** screen (as shown in **Figure 3.4**) and then click on the **Data Source...** button to enter the details of the time-varying emissions.

Full details of how to enter time-varying emissions data can be found in Section 4.1.

3.2.5 Groups

Sometimes it can be advantageous to group concentrations from a selection of sources together. ADMS-Urban has a facility that allows you to model up to 20 user-defined groups in any single model run, although more may be defined in the interface. Concentration output is then calculated separately for each group. Sources can be included in more than one group. In addition to the 20 user-defined groups it is also possible to select for output a group containing all of the sources; refer to Section 3.6.3 for details.

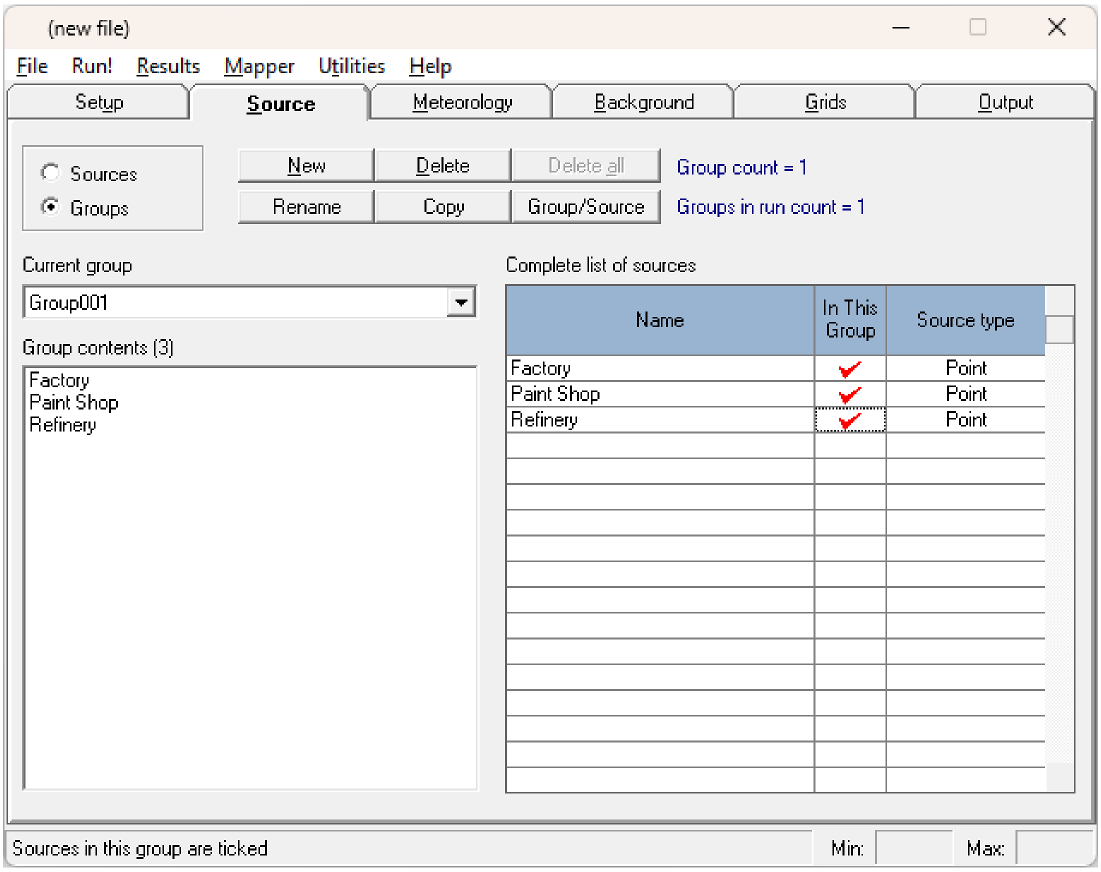


Figure 3.20 – The **Source** screen in **Create groups** mode.

Groups of sources are created by selecting **Groups** on the **Source** screen, shown in **Figure 3.20**. The **New** and **Delete** buttons are used to add and remove existing groups, respectively. The **Delete all** button removes all existing groups. The **Rename** button can be used to rename an existing group and the **Copy** button to duplicate an existing group. The **Group/Source** button will add a set of groups, one for each source of the chosen type(s) selected in the **Create group per source** screen (**Figure 3.21**).

The limit on the number of groups which can be defined is the same as the limit on the number of sources.

For runs with more than 20 sources, it may still be possible to obtain concentration output for each source separately in a single model run using the 'Output per source' option. See Section 3.6.5 for further details, including a list of restrictions that apply to this option.

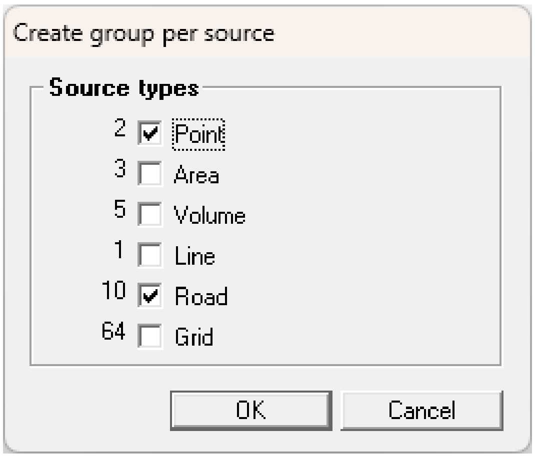


Figure 3.21 – The **Create group per source** screen.

Sources may be added to or removed from a group by selecting the group from the **Current group** list and then double-clicking in the corresponding cell of the **In This Group** column in the **Complete list of sources**. Alternatively, type **Y** or **N** to add and remove ticks respectively. Right-clicking in the **In This Group** column opens a menu providing further options to manipulate the membership of a group:

- **Select all**: add all sources to the group.
- **Clear selection**: remove all sources from the group.
- **Invert selection**: switch membership of the group so sources which were a member are no longer a member and vice-versa.
- **Select sources of type**: add all sources of the chosen type to the group, e.g. add all point sources.

The group membership of individual sources may also be defined in the **Sources** screen for industrial and road sources. Right-clicking on a source name allows the group membership of that source to be viewed and modified from the **Groups** section of the menu. Groups of which the source is a member are shown with a tick next to the name; clicking on the group name toggles the group membership for that source.

Group membership information can be imported and exported using the Import and Export facilities. Refer to Section 5 for more details.

3.3 Meteorology screen

The **Meteorology** screen shown in **Figure 3.22** allows details about the site and the meteorological conditions to be entered.

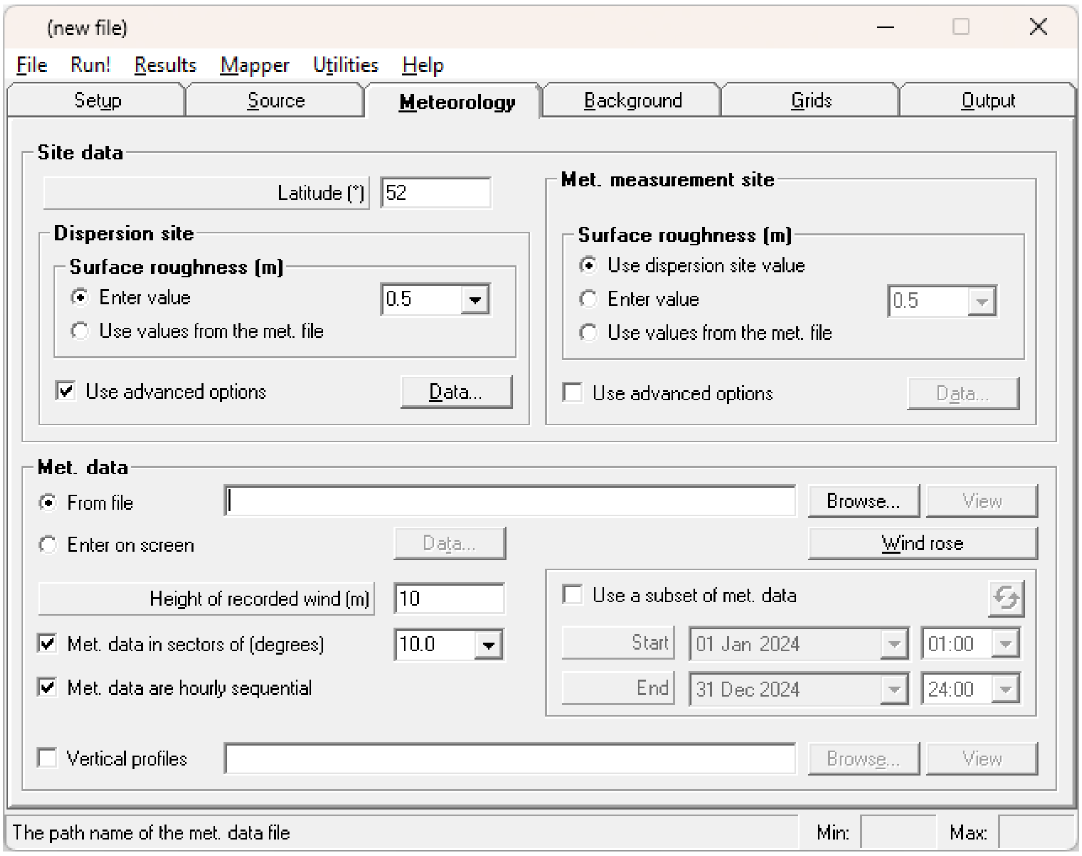


Figure 3.22 – The **Meteorology** screen in the ADMS-Urban interface.

There are two main boxes on the screen: the **Site data** section for entering details about the dispersion and meteorological measurement sites, and the **Met. data** section for entering details of the meteorological data.

The meteorological data can either be entered on screen or through a prepared meteorological data file. This file may be an example file supplied in the *<install_path>\Data* directory, it may be a file prepared by the user from their own data (for example from an on-site meteorological station) or it may be a pre-formatted file obtained from a supplier.

There are three different types of meteorological data that can be entered:

- a series of unrelated meteorological conditions,
- a series of hourly sequential data covering a certain period, for instance a year,
- statistical data covering a series of years (this can only be entered using a meteorological file).

Examples of each of these can be found in the *<install_path>\Data* directory. Some of the files in this directory such as *oneday.met* and *nsc.met* are for use in the worked examples and are not intended to be used in actual calculations.

There is a minimum amount of data that must be entered. These are the wind speed (`U`) and wind direction (`PHI`) and either

- the Julian day number, time of data and cloud cover (`TDAY`, `THOUR` and `CL`),

or

- sensible surface heat flux (`FTHETA0`),

or

- reciprocal of Monin-Obukhov length (`RECIPLMO`).

Additional meteorological parameters can be added to the meteorological data file, and should be included if known. Some model options require further parameters to be entered in this input file. Full details of the parameters that can be included can be found in Section 9.1.

3.3.1 Site data

The **Site data** box allows for the details about both the dispersion and meteorological sites to be entered. The **Latitude** of the site must be entered by the user and must lie between -90° and 90°. The default is 52°, which represents a site on a line roughly between Fishguard and Harwich in the UK, and would therefore be appropriate for southern to mid-England and for South Wales.

In the **Dispersion site** box, enter the **Surface roughness** at the dispersion site (in metres) based on land use. Either enter a value directly in the **Enter value** box, subject to the model limits (minimum = 10^{-7} m, maximum = 10 m), or alternatively use the list to select a value based on the land use − **Table 3.12** shows the land use types and corresponding values of surface roughness. The default value of surface roughness is 0.5 m, corresponding to a region of open suburbia. To enter hourly varying surface roughness values via the meteorological data file, select the **Use values from the met. file** option and add the surface roughness values to the file using the variable name ‘`Z0 (D)`’.

Land use	Surface roughness (m)
Large urban areas	1.5
Cities, woodlands	1
Parkland, open suburbia	0.5
Agricultural areas (max)	0.3
Agricultural areas (min)	0.2
Root crops	0.1
Open grassland	0.02
Short grass	0.005
Sea	0.0001

Table 3.12 − The surface roughness for the different land uses available on the **Meteorology** screen.

In general, the surface roughness length selected for the dispersion site is assumed to apply throughout the domain. However, ADMS-Urban also has advanced features that make it possible to define *spatially varying* surface roughness values over the domain, as described in Section 4.16, or for the roughness to be calculated from neighbourhood-scale building parameters via the *urban canopy flow* model option, as described in Section 4.18.

When using the urban canopy flow model option, the dispersion site roughness entered in the interface should be broadly representative of the roughness in the urban area, as it is used in locations where the urban canopy module does not apply a displacement height.

The **Met. measurement site** box allows surface roughness and other characteristics of the meteorological measurement site to be entered, if they are different from those at the dispersion site. To use the **Use values from the met. file** option, add surface roughness values to the meteorological data file using the variable name '`Z0 (M)`'.

For both the **Dispersion site** and **Met. measurement site**, **Use advanced options** can be selected to allow the user to alter the values of surface albedo, Priestley-Taylor parameter, minimum Monin-Obukhov length and for precipitation differences between the dispersion site and meteorological measurement site. Details of how to use these options are given in Section 3.3.2.

3.3.2 Advanced meteorological parameters

Advanced meteorological data can be entered for the dispersion site and/or the meteorological measurement site. To enter advanced meteorological data go to the **Meteorology** screen, select the **Use advanced options** box in either the **Dispersion site** or **Met. measurement site** boxes, then click on **Data...**. By default, **Use advanced options** is selected for the dispersion site with a value of 30 m entered for the minimum Monin-Obukhov length.

The **Advanced dispersion site data** screen is shown in **Figure 3.23**; the **Advanced met. site data** screen is shown in **Figure 3.24**.

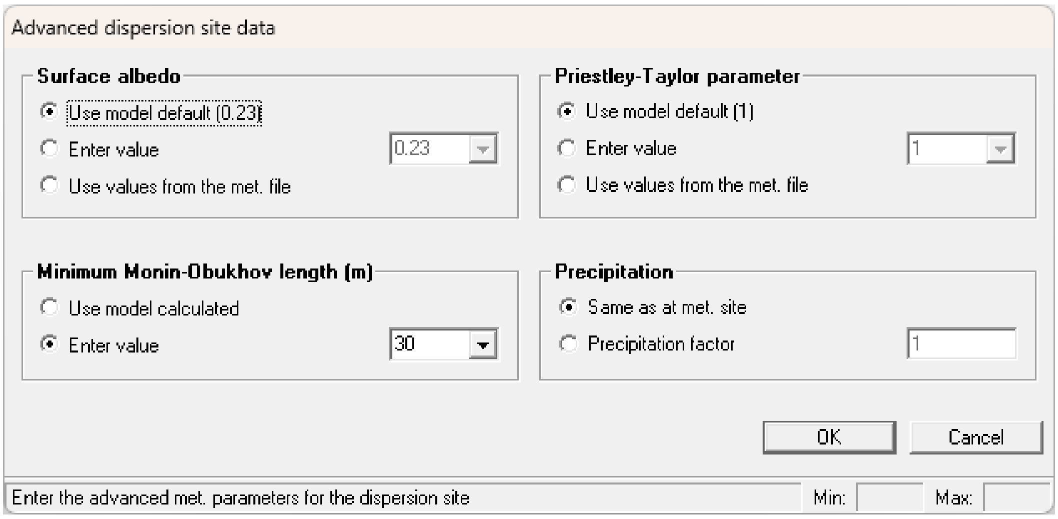


Figure 3.23 – The **Advanced dispersion site data** screen

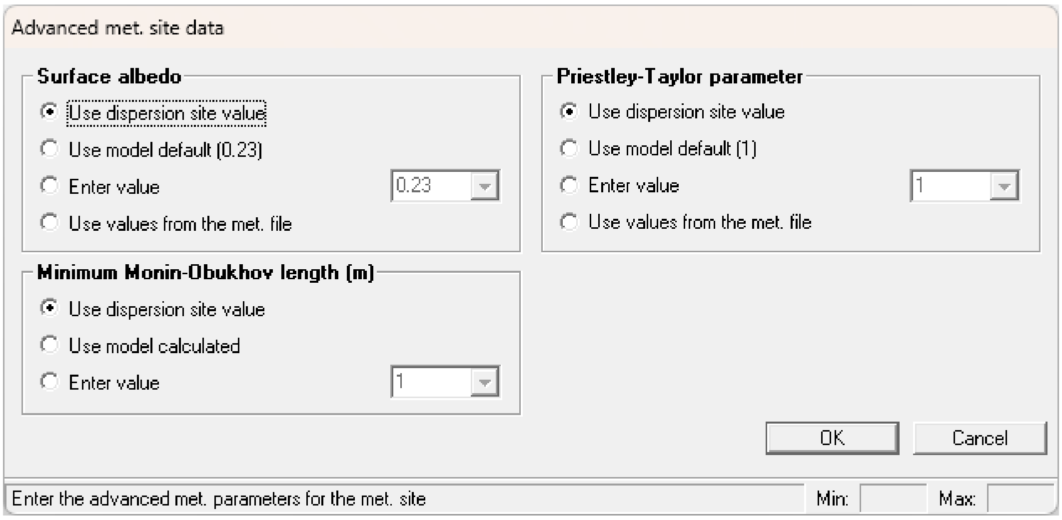


Figure 3.24 – The **Advanced met. site data** screen

The parameters used by ADMS-Urban to process the input meteorological data include the minimum value of the Monin-Obukhov length, the surface albedo and the Priestley-Taylor parameter. The default values of these parameters are defined for a typical urban UK site. ADMS-Urban includes an option to specify values of these parameters more suitable for the site being modelled.

Surface albedo

The surface albedo is the ratio of reflected to incident shortwave solar radiation at the surface of the earth. It therefore lies in the range 0 to 1. In particular, it takes a high value (high proportion of incident radiation reflected) when the ground is snow-covered. The default value for surface radiation is 0.23, i.e. not snow covered (Oke, 1987). The following options are available for the surface albedo.

Use model default (0.23): use the default value; this is the default option for the dispersion site.

Enter value: Enter a user-defined constant value. Either enter a value directly into the box or select a value from the list:

- **Snow-covered ground** = 0.6
- **Not snow-covered** = 0.23

Use values from the met. file: Use hourly varying values of surface albedo from the meteorological file. See Section 9.1 for details of the variable names to use in the meteorological file.

At the meteorological measurement site there is an additional option of **Use dispersion site value**; select this to ensure that the surface albedo at the dispersion site and the meteorological measurement site are the same. This is the default option for the meteorological measurement site.

Priestley-Taylor parameter

The Priestley-Taylor parameter is a parameter representing the surface moisture available for evaporation. The Priestley-Taylor parameter must be between 0 and 3 and the default value is 1 corresponding to moist grassland (Holtslag and van Ulden, 1983). The following options are available for the Priestley-Taylor parameter.

Use model default (1): use the default value; this is the default option for the dispersion site.

Enter value: Enter a constant value. Either enter a value directly into the box or select a value from the list:

- **Dry bare earth** = 0
- **Dry grassland** = 0.45
- **Moist grassland** = 1

Use values from the met. file: Use hourly varying values of Priestley-Taylor parameter from the meteorological file. See Section 9.1 for details of the variable names to be used in the meteorological file.

At the meteorological measurement site there is an additional option of **Use dispersion site value**; select this option to ensure that the Priestley-Taylor parameter at the dispersion site and the meteorological measurement site are the same. This is the default option for the meteorological measurement site.

Minimum Monin-Obukhov length

This is an option to specify the *minimum* value of the Monin-Obukhov length. This allows for the effect of heat production in cities, which is not represented by the meteorological data. The minimum Monin-Obukhov length provides a measure of the stability of the atmosphere (see Section 9.2). In very stable conditions in a rural area its value would typically be 2 to 20 m. In urban areas, there is a significant amount of heat generated from buildings and traffic, which warms the air above the town/city. For large urban areas this is known as the urban heat island. It has the effect of preventing the atmosphere from ever becoming very stable. In general, the larger the area the more heat is generated and the stronger this effect becomes. This means that in stable conditions the Monin-Obukhov length will never fall below some minimum value; the larger the city the larger the minimum value. The minimum Monin-Obukhov length should be between 1 and 200 m; the default value is 30 m corresponding to an urban area.

It is useful to note that the minimum value of the Monin-Obukhov length is used by the model to determine a minimum value for the turbulence parameters, $\sigma_{\min}$ (m/s), in the following way:

(3.1)
$$\sigma_{\min} = \begin{cases} 0.01 & L_{MOmin} \le 10 \\ 0.01 + 0.0095(L_{MOmin} - 10) & 10 < L_{MOmin} < 30 \\ 0.2 & L_{MOmin} \ge 30 \end{cases}$$

In addition, if complex terrain is modelled a further value between 0 and 0.1 m/s is applied, based on the variation of height within the terrain file. See Section 4.19 for further details regarding the minimum value for the turbulence parameters including a way to select the minimum value of the Monin-Obukhov length, the terrain file value, a user-defined value or any combination of the above.

The following options are available for the minimum Monin-Obukhov length:

Use model calculated: the model will calculate a value for the minimum Monin-Obukhov length based on the surface roughness.

Enter value: Enter a value, either directly into the box or select from the list:

- **Large conurbations > 1 million** = 100 m
- **Cities and large towns** = 30 m
- **Mixed urban/industria**l = 30 m
- **Small towns < 50,000** = 10 m

At the meteorological measurement site there is an additional option of **Use dispersion site value**; select this option to ensure the minimum Monin-Obukhov length at the dispersion site and the meteorological measurement site are the same. This is the default option for the meteorological measurement site.

Precipitation

If the precipitation rate at the source is known to be different from that at the site where the meteorological data were collected, then a constant factor can be applied to adjust the values. This option is available from the advanced meteorological screen for the dispersion site.

The precipitation factor is the ratio of the precipitation at the dispersion site to that at the meteorological site, e.g. if the precipitation factor is 0.5 then when the precipitation rate at the meteorological site is 2 mm/hr it will be 1 mm/hr at the dispersion site. As only one value is used, it would typically represent the ratio of annual rainfall at the source site to that at the dispersion site. The following options appear:

- **Same as at met. site**: the precipitation rate at the meteorological site is representative of that at the dispersion site. This is the default option.
- **Precipitation factor**: Enter a value for the precipitation factor. The precipitation factor should be between 0.2 and 5; the default value is 1.

Changing the precipitation rate is only necessary if wet deposition is modelled.

3.3.3 Met. data

The **Met. data** box on the **Meteorology** screen allows the data entry method for the meteorological data to be selected and other details about the meteorological data to be entered. The options available are as follows.

- **From file**: meteorological data are to be read in from a file (the name of the file should be entered by the user). Clicking **Browse...** displays the directory structure and lets the user select prepared meteorological data files, which, by convention, have the extension *.met*. Files can also be selected by dragging and dropping into the interface from Explorer. The **View** button allows the current *.met* file to be viewed; left click to open the file in the default application (as selected via the **File**, **Preferences**, **Viewing options** menu item) or right-click to choose between all available applications. Further information on entering the meteorological data using a *.met* file can be found in Section 3.3.4.

- **Enter on screen**: meteorological data are to be entered on screen. Click on the **Data…** button to enter the data; full details of this option can be found in Section 3.3.5.
- **Wind rose**: shows the wind rose for the meteorological data selected.
- **Height of recorded wind (m)**: height of the wind measurements (default = 10 m). A value of 0 m indicates the wind is a friction velocity measurement and a value of 1000 m indicates it is the geostrophic wind.
- **Met. data in sectors of (degrees)**: if the wind measurements are in sectors, check the box and enter the sector size, either by selecting one of the options from the list or entering a value directly into the box (refer to Section 9.1 for further details).
- **Met. data are hourly sequential**: check the box if the meteorological data entered are hourly sequential or if the meteorological data entered are to be interpolated to become hourly sequential (as described in Section 3.3.6). If a series of unrelated meteorological data are entered, for instance to examine what happens during specific meteorological conditions, this box must not be checked.
- **Use a subset of met. data**: check this box to only use a subset of the meteorological data; the dates and times of the first and last met. lines should be entered. The start and end dates can be automatically set to the limits of the meteorological data file by clicking the reset button . Additionally, right-click options exist that allow you to quickly set the **Start** date and time to the start of met. file or match the subset End date and time, and to set the **End** date and time to the end of met. file or match the subset Start date and time.
- **Vertical profiles**: enter user-defined vertical profile data for wind speed, turbulence parameters, temperature and specific humidity using a *.prf* file (for details of using this option refer to Section 4.21).

3.3.4 Entering meteorological data from a file

If the meteorological data are to be entered from a file, then on the **Meteorology** screen first select **From file**, then click on **Browse...** and search to find the file, or drag and drop the file from Explorer. This selected file can then be viewed using the **View** button; left click to open the file in the default application (as selected via the **File**, **Preferences, Viewing options** menu item) or right-click to choose between all available applications. Meteorological data files are comma-separated files with a *.met* extension, by convention. The format of the meteorological data file should then be as follows.

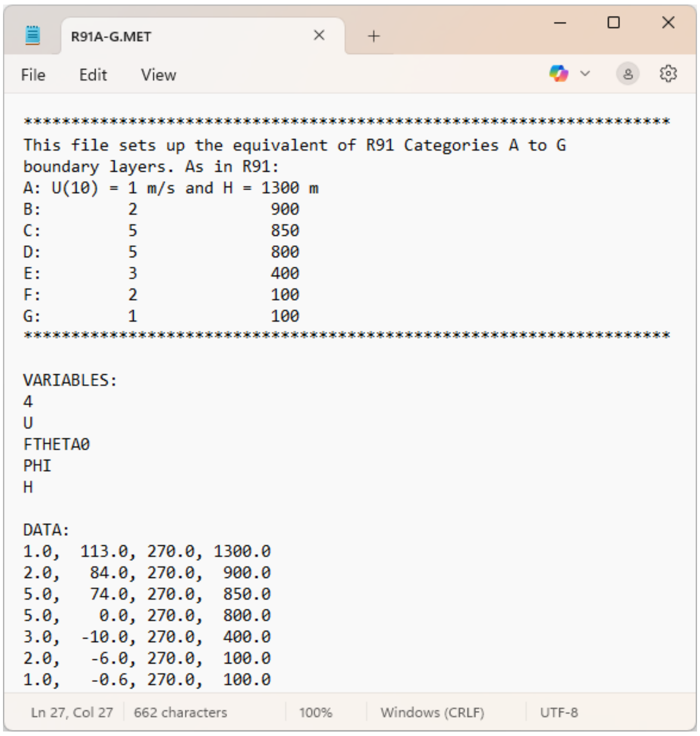
The top of the file may contain comments, after which must be a line containing the keyword “`VARIABLES:`”. The next line should contain the number of parameters to be entered in the *.met* file. Then follows a list of the parameters by name (one per line) for which data are entered in the *.met* file (details of the names to be used for each parameter can be found in Section 9.1). After this, further comments can be entered into the file. Then the data section starts with the keyword "`DATA:`", followed by the actual data. For each line of meteorological data, the parameters should be entered in a row in the order that they are listed in the "`VARIABLES:`" section. No blank lines should appear in the file except before the "`VARIABLES:`" section or between the "`VARIABLES:`" and "`DATA:`" sections. Further details about the format of the *.met* file can also be found by looking at *METDEMO.met* which can be found in the *<install_path>\Data* directory.

Any missing data should be identified using the value '-999'.

There are various different ways of using a *.met* file; details of each of these are given below.

Using one of the prepared example files

There are a number of example *.met* files provided with ADMS-Urban. They are installed in the *<install_path>\Data* directory of your computer. A particularly useful file is *R91A-G.met*, which contains seven lines of meteorological data that correspond approximately to the Pasquill-Gifford stability classes A-G. This file can be viewed in an editor such as Notepad or WordPad and is shown in **Figure 3.25**.




```
**********************************************************************
This file sets up the equivalent of R91 Categories A to G
boundary layers. As in R91:
A: U(10) = 1 m/s and H = 1300 m
B:         2              900
C:         5              850
D:         5              800
E:         3              400
F:         2              100
G:         1              100
**********************************************************************

VARIABLES:
4
U
FTHETA0
PHI
H

DATA:
1.0,  113.0, 270.0, 1300.0
2.0,   84.0, 270.0,  900.0
5.0,   74.0, 270.0,  850.0
5.0,    0.0, 270.0,  800.0
3.0,  -10.0, 270.0,  400.0
2.0,   -6.0, 270.0,  100.0
1.0,   -0.6, 270.0,  100.0
```

Figure 3.25 – The *.met* file *R91A-G.met*.

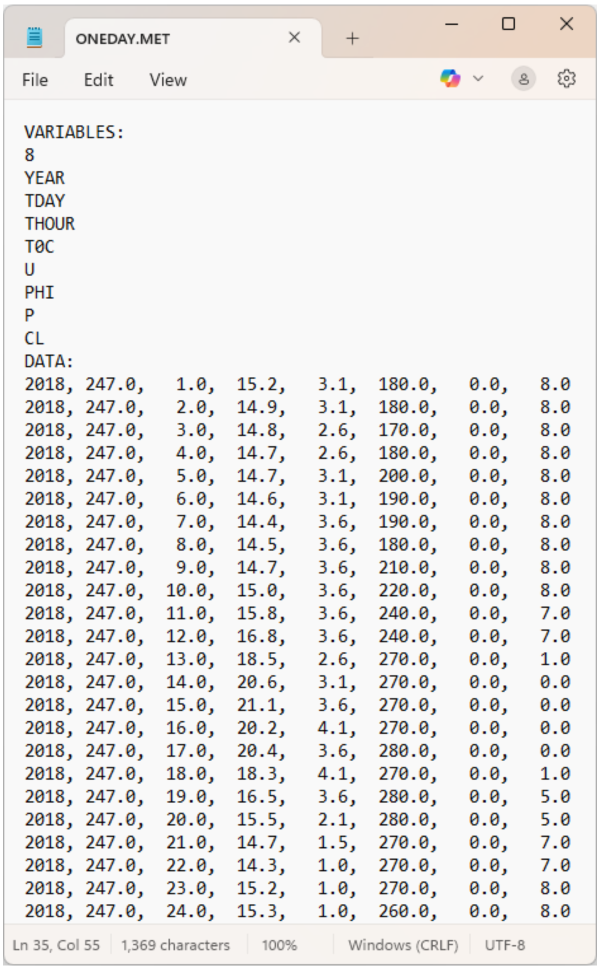
R91A-G.met can be used as it is, or edited, for instance by changing the wind direction (`PHI`) to investigate the effect of different wind directions on the concentration distribution around a building or in complex terrain. For a list of the variable names and their meanings, refer to Section 9.1 of this user guide. When using a file like this that represents a set of unrelated meteorological conditions, the **Met. data are hourly sequential** box must not be checked.

It is good practice to use this file in a preliminary model run to obtain a quick indication of what the maximum concentration from a source or group of sources may be, the location of the maximum concentration, and under which type of meteorological condition it occurs. This enables the user to obtain an early indication of likely results and, in particular, aids in setting up an appropriate calculation grid before performing a long-term average calculation.

Hourly sequential meteorological data

Hourly sequential data can either be entered using files prepared by the user, say from an on-site meteorological station, or using files obtained from a supplier. Hourly sequential files are normally available for periods of one year. Each line of data represents one hour of measurements, and therefore a data file for one year contains 8760 lines of data. This type of data is used for modelling historical pollution episodes where the actual hour-by-hour meteorological conditions during the episode are modelled. It can also be used to calculate peak hourly average concentrations and long-term average concentrations.

Alternatively, meteorological data which are not hourly sequential, but are in chronological order, can be input into ADMS-Urban. These data will then be interpolated to become hourly sequential, providing certain criteria regarding the variables used in the meteorological files are met. Please refer to Section 3.3.6 for more details.




```
VARIABLES:
8
YEAR
TDAY
THOUR
T0C
U
PHI
P
CL
DATA:
2018, 247.0,   1.0,  15.2,   3.1,  180.0,   0.0,   8.0
2018, 247.0,   2.0,  14.9,   3.1,  180.0,   0.0,   8.0
2018, 247.0,   3.0,  14.8,   2.6,  170.0,   0.0,   8.0
2018, 247.0,   4.0,  14.7,   2.6,  180.0,   0.0,   8.0
2018, 247.0,   5.0,  14.7,   3.1,  200.0,   0.0,   8.0
2018, 247.0,   6.0,  14.6,   3.1,  190.0,   0.0,   8.0
2018, 247.0,   7.0,  14.4,   3.6,  190.0,   0.0,   8.0
2018, 247.0,   8.0,  14.5,   3.6,  180.0,   0.0,   8.0
2018, 247.0,   9.0,  14.7,   3.6,  210.0,   0.0,   8.0
2018, 247.0,  10.0,  15.0,   3.6,  220.0,   0.0,   8.0
2018, 247.0,  11.0,  15.8,   3.6,  240.0,   0.0,   7.0
2018, 247.0,  12.0,  16.8,   3.6,  240.0,   0.0,   7.0
2018, 247.0,  13.0,  18.5,   2.6,  270.0,   0.0,   1.0
2018, 247.0,  14.0,  20.6,   3.1,  270.0,   0.0,   0.0
2018, 247.0,  15.0,  21.1,   3.6,  270.0,   0.0,   0.0
2018, 247.0,  16.0,  20.2,   4.1,  270.0,   0.0,   0.0
2018, 247.0,  17.0,  20.4,   3.6,  280.0,   0.0,   0.0
2018, 247.0,  18.0,  18.3,   4.1,  270.0,   0.0,   1.0
2018, 247.0,  19.0,  16.5,   3.6,  280.0,   0.0,   5.0
2018, 247.0,  20.0,  15.5,   2.1,  280.0,   0.0,   5.0
2018, 247.0,  21.0,  14.7,   1.5,  270.0,   0.0,   7.0
2018, 247.0,  22.0,  14.3,   1.0,  270.0,   0.0,   7.0
2018, 247.0,  23.0,  15.2,   1.0,  270.0,   0.0,   8.0
2018, 247.0,  24.0,  15.3,   1.0,  260.0,   0.0,   8.0
```

Figure 3.26 – Example of hourly sequential meteorological data file.

An example of the beginning of an hourly sequential file is shown in **Figure 3.26**. If this type of file is being used then the **Met. data are hourly sequential** box should be checked on the **Meteorology** screen. For hourly sequential files, wind direction measurements (`PHI`) are often reported to the nearest 10 degrees, i.e. the wind data are in 10 degree sectors; in these cases the **Met. data in sectors of (degrees)** option should also be used.

Hourly sequential files can be used for long-term average concentration and percentile calculations for comparison with regulatory standards. Since sequential data files consist of continuous hour-by-hour measurements, they are a better representation of actual meteorological conditions than statistically-averaged data files.

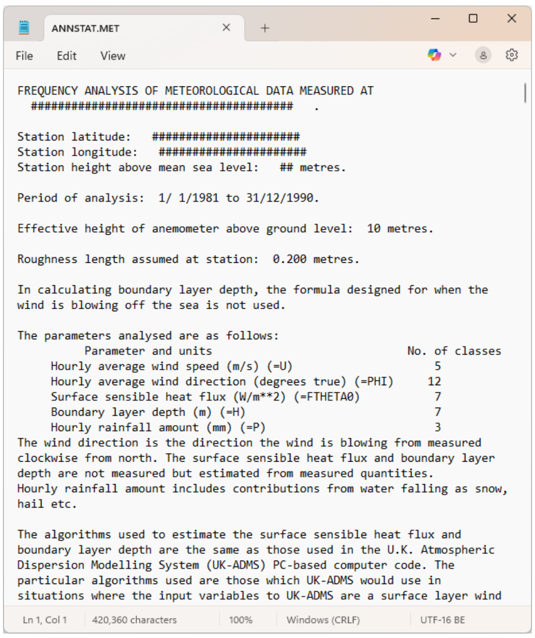
If other files are being used that contain the hour (e.g. background files or time-varying emissions files) then you should ensure that the definition of the hour is consistent between the files, i.e. that all files contain the hour number representing the hour ending time in local solar time.

However, it is important to note that year-to-year variations occur in meteorological conditions. Therefore, any conditions resulting in very high ground level concentrations one year would not necessarily have occurred in the previous year nor would they necessarily occur the following year. High percentiles of concentration will therefore vary according to the year of meteorological data used. Annual averages are less sensitive to the choice of year.

Up to ten years' worth of data may be included in a sequential data file.

Statistical meteorological data

Statistical data files normally include meteorological data for a period of ten years. They can be obtained from a supplier, e.g. the UK Met Office. They can be viewed in a text editor, and an example showing the beginning of a statistical file from the UK Met Office is given in **Figure 3.27**.




```
FREQUENCY ANALYSIS OF METEOROLOGICAL DATA MEASURED AT
  #######################################  .

Station latitude:   ######################
Station longitude:   ######################
Station height above mean sea level:   ## metres.

Period of analysis:  1/ 1/1981 to 31/12/1990.

Effective height of anemometer above ground level:  10 metres.

Roughness length assumed at station:  0.200 metres.

In calculating boundary layer depth, the formula designed for when the
wind is blowing off the sea is not used.

The parameters analysed are as follows:
          Parameter and units                          No. of classes
     Hourly average wind speed (m/s) (=U)                   5
     Hourly average wind direction (degrees true) (=PHI)   12
     Surface sensible heat flux (W/m**2) (=FTHETA0)         7
     Boundary layer depth (m) (=H)                          7
     Hourly rainfall amount (mm) (=P)                       3
The wind direction is the direction the wind is blowing from measured
clockwise from north. The surface sensible heat flux and boundary layer
depth are not measured but estimated from measured quantities.
Hourly rainfall amount includes contributions from water falling as snow,
hail etc.

The algorithms used to estimate the surface sensible heat flux and
boundary layer depth are the same as those used in the U.K. Atmospheric
Dispersion Modelling System (UK-ADMS) PC-based computer code. The
particular algorithms used are those which UK-ADMS would use in
situations where the input variables to UK-ADMS are a surface layer wind
```

Figure 3.27 − Example of statistical meteorological data file.

The purpose of using statistically analysed data is to take account of the variation in meteorological conditions from year to year. Each hourly measurement for that period is recorded and put into a bin or class representing a range of different values for different meteorological parameters. For example, in the file represented in **Figure 3.27**, the data were categorized into five classes of wind speed (`U`), twelve classes of wind direction (`PHI`) (i.e. a wind sector size of 30 degrees), seven classes of surface sensible heat flux (`FTHETA0`), seven classes of boundary layer depth (`H`), and three classes of precipitation (`P`).

There are typically over 2000 lines of data in a statistical meteorological data file, each line corresponding to a particular combination of meteorological conditions that occurred, for example the first line corresponds to the first class or bin of each of the five parameters. Each line has an associated frequency representing the number of hours that occurred during the period of the data (usually ten years) with meteorological conditions within the ranges described by that line of data. Detailed frequency data may also be included, giving the number of hours in ranges of months of the year and hours of the day during which the meteorological conditions were within the ranges described by that line of data. This is illustrated in the example file *ANNSTAT.MET.*

Users should ensure they read the information at the beginning of a statistical meteorological data file; particularly the 'Effective height of anemometer above ground'. For the majority of files this height is 10 m but occasionally it is some other height and the correct value should be entered into the ***Meteorology*** *screen of ADMS-Urban.*

Statistical data has a number of advantages over sequential data.

- There are about 2000 lines of data to process in ADMS-Urban with a statistical meteorological data file rather than 8760 in a one year sequential file; run times are therefore significantly shorter for statistical compared to sequential data.
- Statistical data are representative of the meteorological conditions which occurred over several years (usually ten); year-to-year variations in conditions are therefore accounted for. This is not true for sequential data.

A disadvantage of using statistical meteorological data is that because the data are categorised into bins with pre-defined upper and lower bounds, it is the mean for each bin rather than the actual hourly value of a parameter that is processed by ADMS-Urban. Thus extreme conditions are smoothed out. For example, there may be a few hours in a particular year when the surface heat flux is extremely high thus generating a lot of convective mixing and bringing an elevated plume down to the ground very quickly. This would normally result in high maximum ground-level concentrations near the source. Using sequential data, the data file would contain the actual high surface heat flux value for that particular hour, thus representing actual conditions. Using statistical data, the value of surface heat flux used would be the mean value of the appropriate class. In this case, this mean value would be less than the true hourly value and would thus underestimate the amount of convective turbulence, hence underestimating ground level concentrations for that hour.

In general, high percentiles obtained with sequential data are larger than those obtained with statistical data. Sequential data must be used when modelling plume chemistry and using background data from files. Sequential data are required for generating results for comparison with regulatory standards defined in terms of rolling or running means as statistical data are inappropriate for such calculations.

3.3.5 Entering meteorological data on screen

If the meteorological data are to be entered on screen, first select the **Enter on screen** option on the **Meteorology** screen and then click on the **Data...** button. This will bring up the **Meteorological Data** screen shown in **Figure 3.28**.

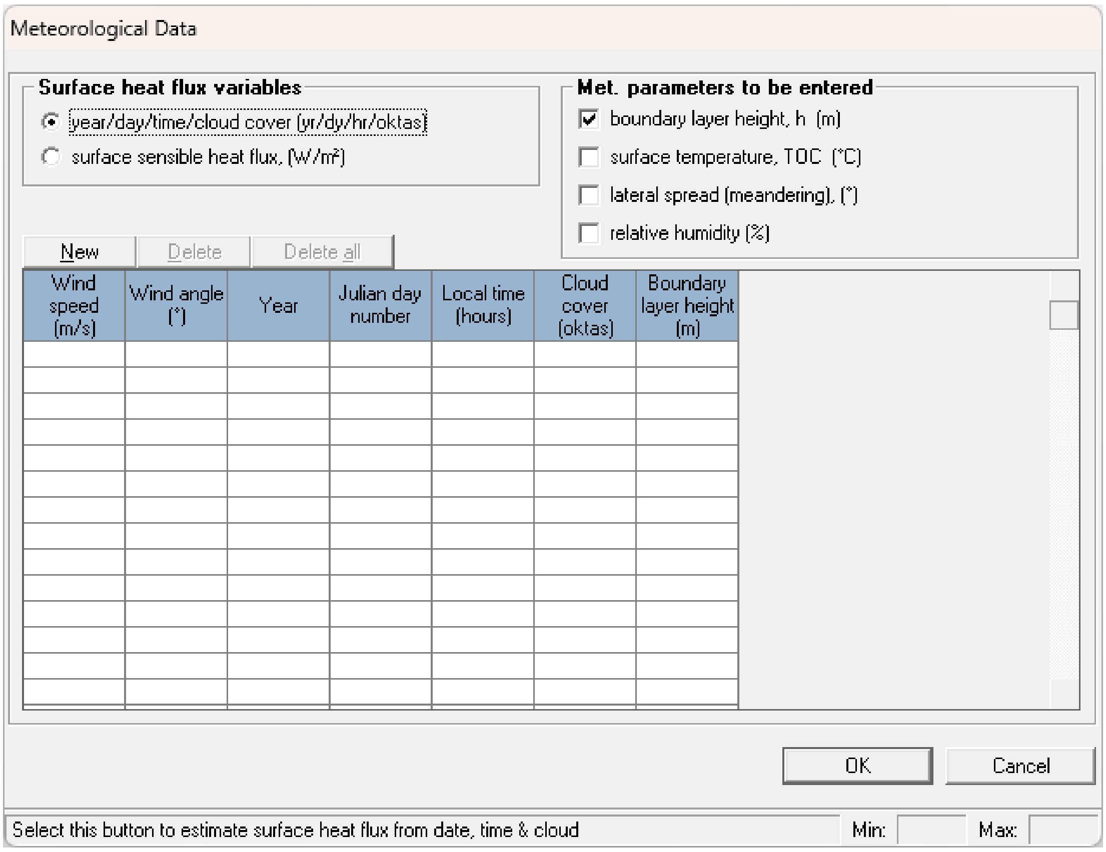


Figure 3.28 – The **Meteorological Data** screen used for entering meteorological data on screen.

This screen contains the table of data to be specified. The table always contains columns for wind speed and wind angle. It also includes the surface heat flux variables. The user *must* select one of the two options in the **Surface heat flux variables** box: either **year/day/time/cloud cover (yr/dy/hr/oktas)** or **surface sensible heat flux, (W/m²)**.

The choice of option determines the number of columns in the table.

In addition, data *may* be entered for one or more of the four options under **Met parameters to be entered** area if the user has accurate information. These are the **boundary layer height, h (m)**, the **surface temperature, TOC (°C)**, the **lateral spread (meandering), (°)** and the **relative humidity (%)**. Clicking in the check box beside each variable name causes the corresponding column to appear or disappear, as appropriate.

Click on the **New** button to add default lines of data, which may then be edited, click the **Delete** button to remove the selected line of data from the table, or click **Delete all** to remove all lines of data.

It is not possible to enter more than 99 lines of meteorological data by hand.

The columns of data are as follows:

- **Wind speed (m/s)**: wind speed at the measurement height.

  Minimum = 0 m/s
  Maximum = 100 m/s
  Default = 5 m/s

- **Wind angle (°)**: direction of the wind measured clockwise from north (e.g. 270° for a wind coming from the west)

  Minimum = 0°
  Maximum = 360°
  Default = 270°

- **Year**: year

  Minimum = 1900
  Maximum = 2100
  Default = Current year

- **Julian day number**: Julian day, e.g. Jan 1st = 1, Dec 31st = 365 or 366

  Minimum = 1
  Maximum = 366
  Default = 1

- **Local time (hours)**: time e.g. for hour ending at 4pm (16:00) hours enter 16

  Minimum = 0
  Maximum = 24
  Default = 12 (then increments by 1 hr with further clicks of the **New** button)

- **Cloud cover (oktas)**: cloud cover

  Minimum = 0
  Maximum = 8
  Default = 0

- **Surface heat flux (W/m²)**: surface sensible heat flux

  Minimum = -200 W/m^2
  Maximum = 1 000 W/m^2
  Default = 0 W/m^2

- **Boundary layer height (m)**: height of the boundary layer

  Minimum = 40 m
  Maximum = 3 000 m
  Default = 800 m

- **Surface temp. (°C)**: surface temperature

  Minimum = -100°C
  Maximum = 60°C
  Default = 15°C

- **Lateral spread (°)**: standard deviation of the mean wind direction

  Minimum = 0°
  Maximum = 90°
  Default = 7.5°

- **Relative humidity (%)**: relative humidity (this parameter is not used within the calculations in ADMS-Urban at present)

  Minimum = 0%
  Maximum = 100%
  Default = 50%

The user should refer to Section 9.1 for more detailed information on the parameters appearing on the **Meteorology** screen.

3.3.6 Interpolation of meteorological data

In order to model averaging times greater than one hour, hourly sequential meteorological data are required. If hourly sequential data are not available, but a chronological sequence of meteorological data are available then, providing certain constraints are met, the data will be interpolated to become hourly sequential. To enable this option, enter your meteorological data as described in Section 3.3.4 or 3.3.5 and check the **Met. Data are hourly sequential** box.

In order for the meteorological data to be interpolated, they must meet the following criteria:

- Contain year, day and hour;
- Be in chronological order, with gaps no larger than 9 hours;
- Contain wind speed, direction and cloud cover; and
- Not contain any variables that cannot be interpolated – please refer to Table 9.1 for details.

If the meteorological data entered are hourly sequential and the ***Met. Data are hourly sequential*** *box is checked, no interpolation will be carried out and the restrictions regarding which variables can be entered do not apply.*

If the chronological sequence of meteorological data entered can be interpolated by the model to a series of hourly sequential data covering the same period, the interpolated meteorological data will then be modelled. If other files containing year, day and hour are included in the run (e.g. background or time-varying files) then these must include data for each of the hours that will be modelled and not just the hours which are in the input meteorological data i.e. the model only interpolates meteorological data files.

3.4 Background screen

The **Background** screen shown in **Figure 3.29** allows the user to enter background data for any of the pollutants entered in the **Palette of Pollutants**.

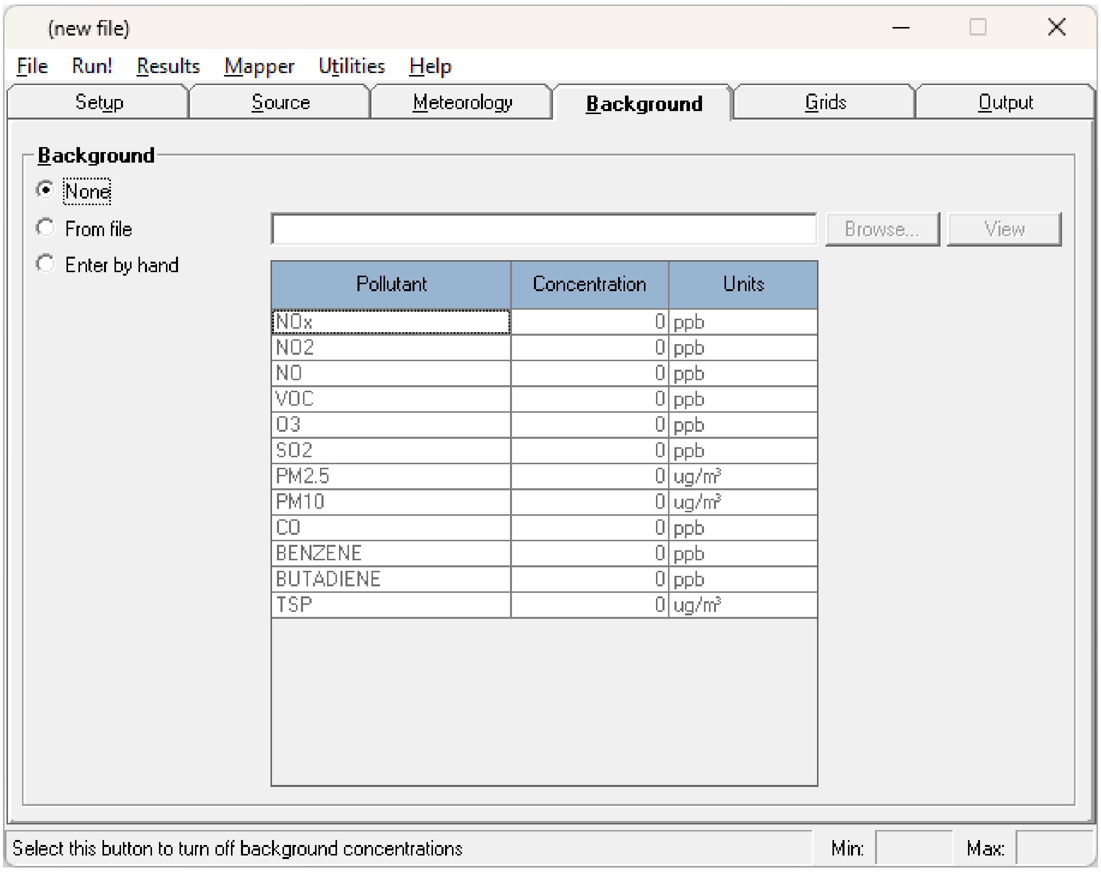


Figure 3.29 – The **Background** screen.

When predicting pollutant contributions it may be important to take into account the underlying, or background, levels of pollutants in the atmosphere. In particular, if chemistry is being modelled then background information must be included in the run.

Care should be taken that the appropriate background data are selected for each modelling scenario. If all of the local major sources of pollution are included in the model run, for instance an isolated industrial complex, then background from a rural site outside (and ideally upwind of) the modelling domain should be chosen, to avoid double-counting the effect of modelled sources. On the other hand, if not all of the local sources of pollution are to be modelled, for instance an industrial complex in an urban area, then background concentration values taking into account those sources which are not being modelled should be used, although again care should be taken not to double-count any sources that are being modelled.

The **Background** screen presents three options:

- **None**: no background concentrations.
- **From file**: hourly varying background data from a prepared background file (see Section 3.4.1 for more details).

- **Enter by hand**: constant background values entered directly on screen (see Section 3.4.2 for more details).

3.4.1 Entering background data from a file

To specify the file containing the background data, select the **From file** option. Either click on **Browse...** to bring up a screen that allows you to browse through the directory structure to find the appropriate background file (*.bgd* extension by convention), or drag and drop a background file from Explorer. The **View** button allows the currently selected background file to be viewed; left click to open the file in the default application (as selected via the **File**, **Preferences**, **Viewing options** menu item) or right-click to choose between all available applications. The format of the background file is as described below.

Background data files are in a comma-separated format. If you create your data file in a spreadsheet package such as Microsoft Excel and then save it as a comma-separated (.csv) file, you should subsequently open it in an editor such as Notepad to remove any trailing commas from the header lines.

The first line gives the version number of the file. Currently the version number is 2 and so the first line should read "`BackgroundVersion2`". This line must always be included at the start of the file.

The next line states the number of pollutants for which background data are contained in the file. The names of the pollutants are then listed (one per line) and should match the pollutant definitions in the **Palette of Pollutants**. Any pollutants that are not listed in the background file are assumed to have zero background concentrations, with the exception of NO if modelling chemistry, which the model will calculate internally as the difference between the NO_X and NO_2 background concentrations.

The keyword "`UNITS:`" must appear after the pollutant names. This is then followed by a list of the concentration units for each of the pollutants, in the same order as the pollutant names were listed.

If odours are modelled then the background values must be in units of ou_e/m^3.

The pollutant data are listed after the keyword "`DATA:`" in the following format:

```
Year, Day number, Hour, Pollutant data
```

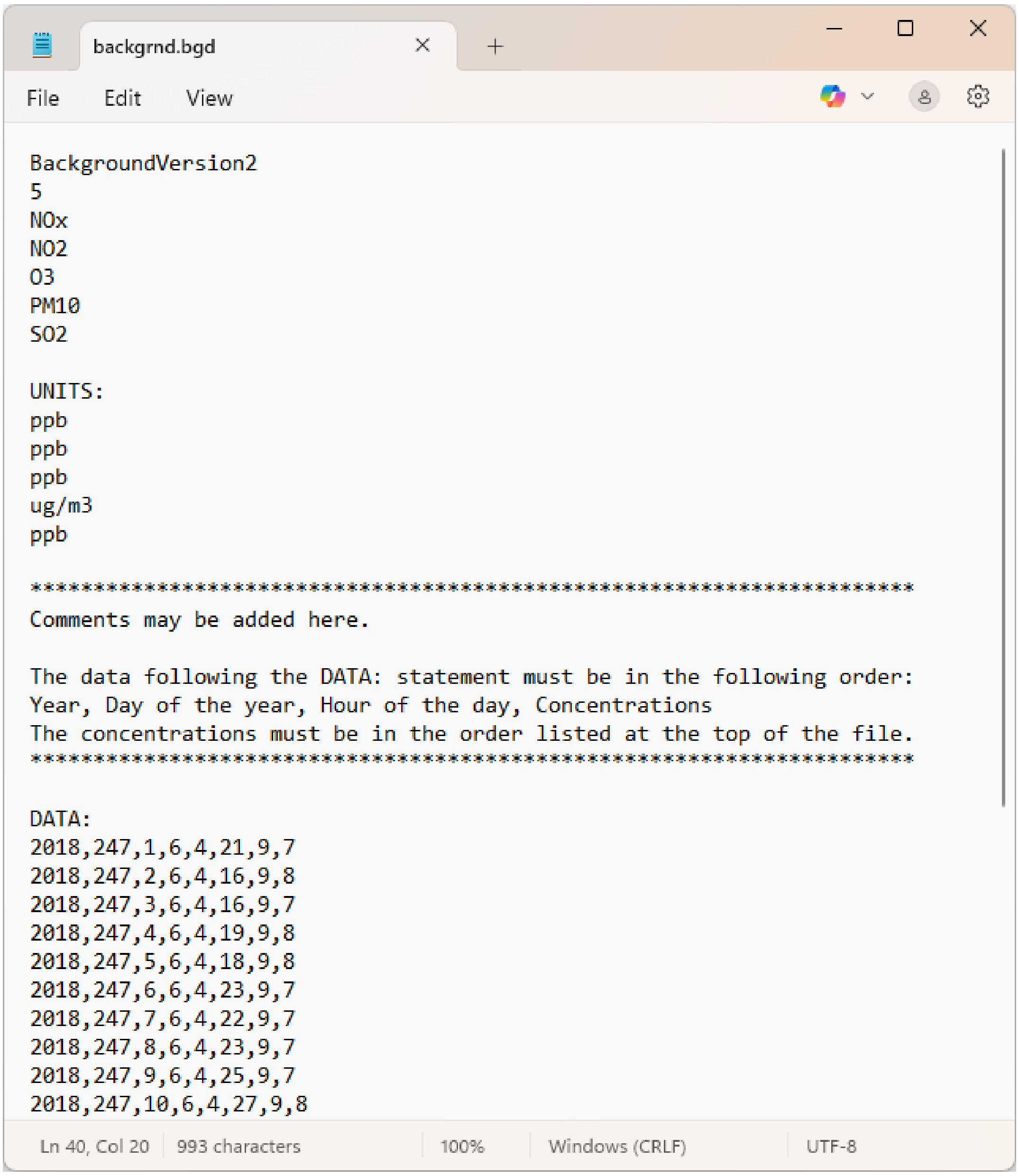
Background pollutant data must be given as hourly sequential values and must be supplied for each hour to be modelled. An example of a background file is shown in **Figure 3.30**.

If NO_X data are included in the background file and chemistry is being modelled, then NO_2 data must also be included.

Care should be taken to ensure that the quality of background data is good – large amounts of missing data may cause inaccuracies in model results. If any data are missing for a particular pollutant and hour, then the model will use the previous value for the pollutant. It may be preferable for the user to fill in any gaps in the data before running the model, particularly if there are several consecutive missing values. Missing data for a particular hour could be filled in by, for example, interpolating the non-missing values or using the annual average value for that hour of the day.

If there are more than 24 consecutive lines of missing data for any pollutant, then the model will stop with an error message.

Any missing data should be identified using the value '-999'.




```
BackgroundVersion2
5
NOx
NO2
O3
PM10
SO2

UNITS:
ppb
ppb
ppb
ug/m3
ppb

***********************************************************************
Comments may be added here.

The data following the DATA: statement must be in the following order:
Year, Day of the year, Hour of the day, Concentrations
The concentrations must be in the order listed at the top of the file.
***********************************************************************

DATA:
2018,247,1,6,4,21,9,7
2018,247,2,6,4,16,9,8
2018,247,3,6,4,16,9,7
2018,247,4,6,4,19,9,8
2018,247,5,6,4,18,9,8
2018,247,6,6,4,23,9,7
2018,247,7,6,4,22,9,7
2018,247,8,6,4,23,9,7
2018,247,9,6,4,25,9,7
2018,247,10,6,4,27,9,8
```

Figure 3.30 – An example showing the format of the background file.

3.4.2 Entering background data by hand

To enter constant values of background data directly on screen, first select the **Enter by hand** option. The table then contains all of the pollutants in the **Palette of Pollutants** and the background values, along with the units of these background values; they can then be edited in the table.

Background concentrations of particulate pollutants can only be specified in mass units.

Background concentrations of NO are non-editable; the value is automatically calculated as the difference between the NO_X and NO_2 background concentrations.

3.4.3 Special consideration for PM10 and PM2.5

Particulate matter consists of the following components:

- Coarse particulates

  These are emissions from non-combustion sources such as suspended soils and dusts, sea salt, biological particles and particles from construction work.

- Secondary particulates

  These are formed in the atmosphere, following the release of emissions in the gaseous phase; examples are sulphates and nitrates. Secondary particulates include emissions from combustion sources outside the study area.

- Primary particulates

  These are from local traffic and industrial sources within the study area. Local traffic emissions may include both exhaust contributions and non-exhaust contributions such as resuspension or brake, tyre and road wear.

ADMS-Urban models all primary particulates, and also some secondary particulates that are created within the modelling domain (for further details of the sulphate chemistry module, please refer to Sections 4.13 and 9.17). All other particulates should be included as a background value entered into the model.

Particulate measurements

In order for measured particulate data to be used as input to the model in the form of background concentrations, the data must be in units consistent with the model i.e. mass units. Devices that measure particulate concentrations ‘by mass’ are known as gravimetric samplers. A number of other devices are available that also record particulate concentrations, for example TEOM monitors, but measurements are not always consistent with those from gravimetric devices.

Users should ensure that they know what units their measured particulate data are in, and if the data have not been measured by a gravimetric sampler, then conversion factors should be applied as necessary.

All EU particulate air quality objectives are in gravimetric units.

3.5 Grids screen

The **Grids** screen shown in **Figure 3.31** allows the user to define output points at which concentration values will be output by the model. Output points may be defined as a grid (**Gridded output points**), individually within the interface (**Specified points**) and/or individually via a file (**Specified points file**). Grids of output points may be defined with regular or variable spacing.

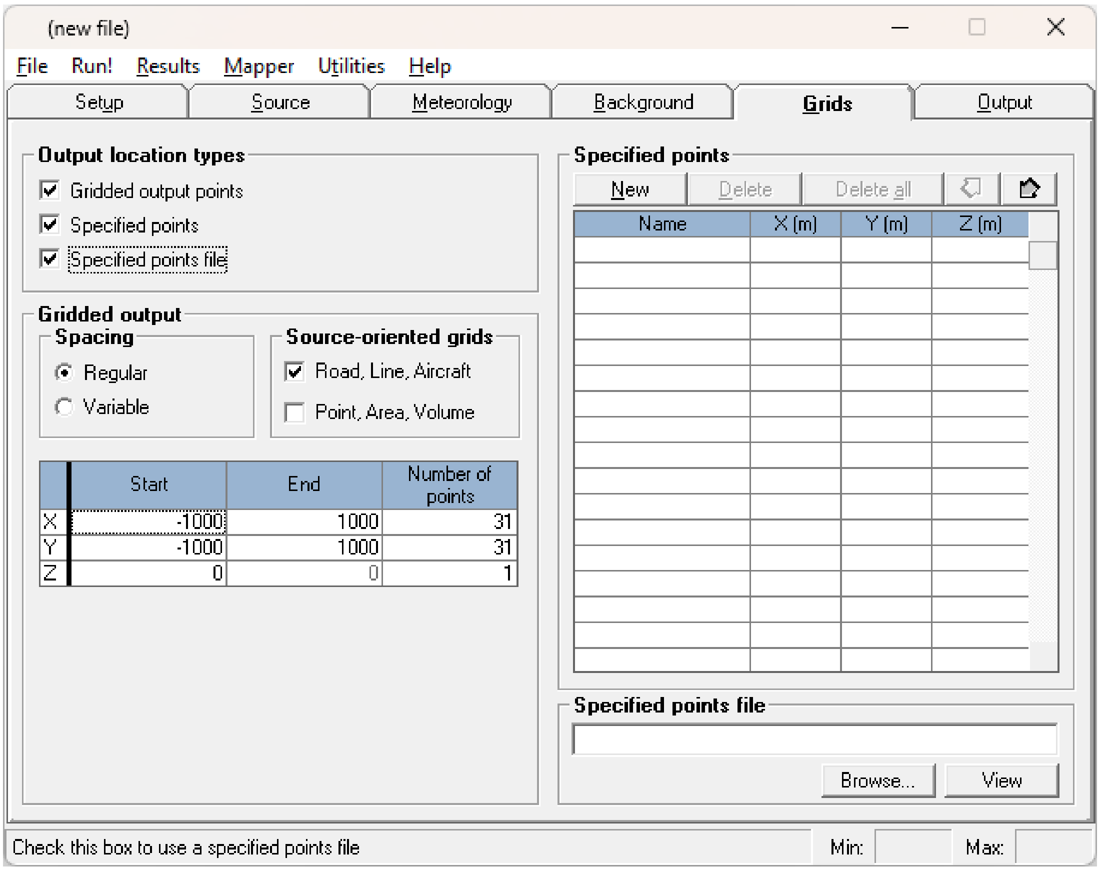


Figure 3.31 – The **Grids** screen for selecting an output grid and/or specified output points.

The units of output points are metres and they may be defined anywhere in 3D space, but must be on or above the ground and are subject to the following restrictions:

- Horizontal coordinates (**X** and **Y**):

  Minimum: 10^{-7} m
  Maximum: 10^{7} m

- Vertical coordinate (**Z**):

  Minimum: 0 m
  Maximum: 15 000 m

3.5.1 Gridded output

Within the **Gridded output points** option, there is some flexibility about the spacing between the individual receptor points. Grids may have **Regular** spacing, or **Variable** spacing.

Only Cartesian, orthogonal output grids may be defined in ADMS-Urban.

Regular spacing

To define the extent of the regular grid, enter coordinates for the maximum and minimum values of **X**, **Y** and **Z**.

By default, ADMS-Urban creates a regular Cartesian grid of 31 by 31 by 1 receptor points within the rectangular region specified.

To increase or reduce this resolution, edit the values in the **Number of points** column. The maximum resolution in each horizontal direction is 2 001 points, with up to 501 vertical levels. Care should be taken with high-resolution output grids since the model run time and memory requirements partly depend on the number of receptor points.

Variable spacing

To specify a variably spaced Cartesian grid choose the **Variable** option in the **Gridded output spacing** box. The **Grids** screen will now appear similar to **Figure 3.32**.

To enter **X**, **Y** and **Z** values, click on the small text box in that column, type the value then press the **SPACE** bar. To remove points, click on them in the larger, upper text box, and press the **DELETE** key. Results will be calculated on a grid of points at the specified **X**, **Y** and **Z** values. For example, if three **X** values, four **Y** values and one **Z** value are specified, results will be calculated on a grid of 12 points. The maximum size of the variable grid in any dimension is 101 points.

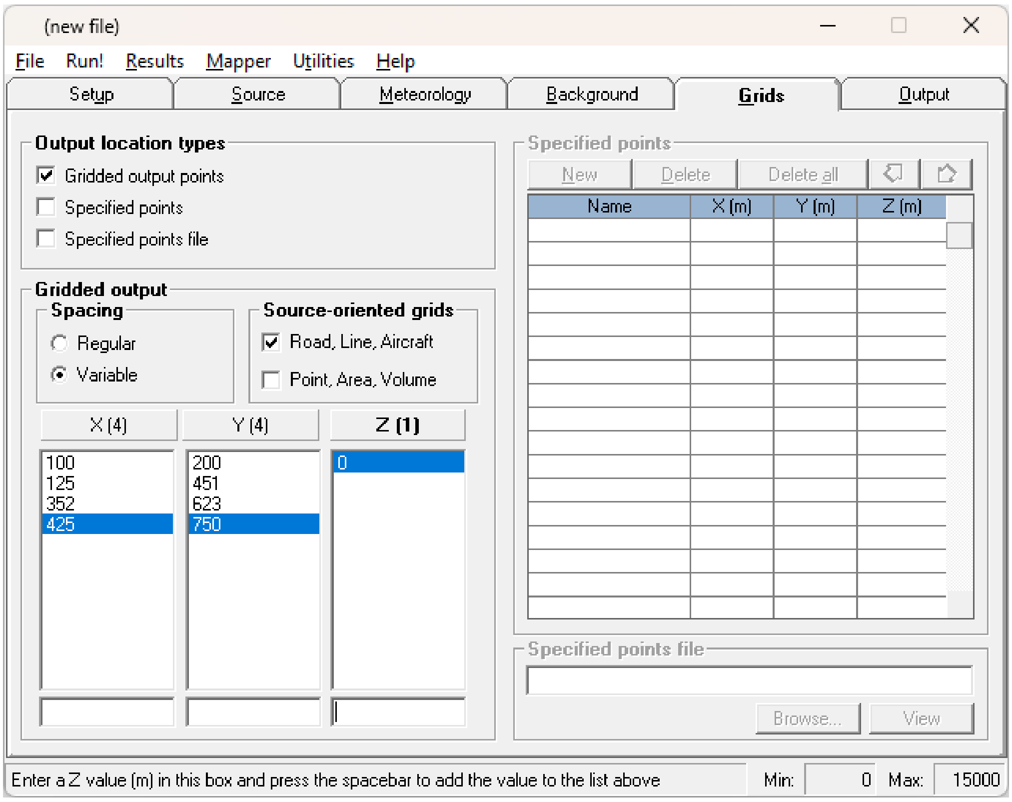


Figure 3.32 – The **Grids** screen when a variable Cartesian grid is specified. The numbers in brackets in the column headers indicate the number of points in each column box.

Source-oriented grids

If gridded output is being modelled, then as well as the user-defined grid, further output points can be added automatically based on the sources contained in the run. The way these points are added is dependent on the source type. Details on how these points are added for road and line sources are given in Section 3.5.2, and for point, area and volume sources in Section 3.5.3.

It may be desirable to export the locations of the source-oriented grid points so that they can be used in a separate model run with a different source configuration. ADMS-Urban has an additional input file option that makes this possible; see Section 4.28 for further details.

3.5.2 Source-oriented grids – Road, Line, Aircraft sources

If you have a large number of road or line sources within your project, a regular grid may not capture all the features of the pollutant concentration distribution within the study area. ADMS-Urban can add additional receptor points close to these sources, where the pollutant concentration gradients are often greatest. To use this option, tick the **Road, Line, Aircraft** checkbox in the **Source-oriented grids** section. This is selected by default when modelling road or line sources.

ADMS-Urban will add the additional receptor points to the user-defined grid in two stages:

1. Firstly, a number of extra receptor points will be added in and around road and line sources, and the model will calculate concentrations at these points in the same way as at any other standard grid point or specified point. These form rows of points aligned with the source segments themselves. For details on the perpendicular distances of these rows of points from the source centrelines, refer to the paragraph *Across-source locations* below. For details on the number and spacing between points on the same row, refer to the paragraph *Along-source locations* further below. For details on which road and line sources receive these extra points, refer to the paragraph *Included sources* further below.
2. Secondly, after the model has performed the concentration calculations, it will add three additional interpolated points between each 'pair' of points from the first stage, i.e. between neighbouring points on the same row of a given source segment, unless (i) only one point was added to that segment row during the first stage, or (ii) the distance between a pair of points is large compared with the resolution of the standard grid. Concentrations at these additional points are calculated by linearly interpolating between the values at the points from the first stage. However, if the model deems that a linear interpolation between a given pair of points would be inappropriate, it will replace the three interpolated points with three actual points (i.e. points at which the full concentration calculations are performed). By default, this second stage of adding interpolated points will always be carried out (unless buildings are modelled), but it can be disabled via the additional input file; refer to the paragraph *Along-source locations* further below.

Across-source locations

By default, four rows of extra points are added along standard road/line sources, and six along (two-sided) advanced street canyon road sources. For a standard road source, the points are placed at perpendicular distances of 0.45 and 2.0 times the road width from the road centreline on either side of the road, i.e. two lie inside the road and two lie outside it. For an advanced street canyon road source, they are placed at a perpendicular distance of 0.45 times the road width (i.e. inside the road) and 0.9 and 1.1 times the canyon half-width from the road centreline on either side of the road (i.e. just either side of each canyon wall). This ensures that the large concentration gradients that often exist across the canyon walls are well captured. If the two rows of points at 0.45 times the road width and 0.9 times the canyon half-width from the road centreline are sufficiently close (within 1 m), the points at 0.9 times the canyon half-width will be discarded. For a road source with a canyon on only one side, there will be two rows of points on the non-canyon side and three on the side with the canyon.

It is possible to modify the default across-source locations of the source-oriented grid points via the additional input file. To do this, create a new additional input file or edit the existing additional input file (refer to Section 3.1.8) and enable the **Across-source locations** option (keyword IGPACROSSLOCATIONS) under the **Source-oriented grids** folder, as shown in **Figure 3.33**.

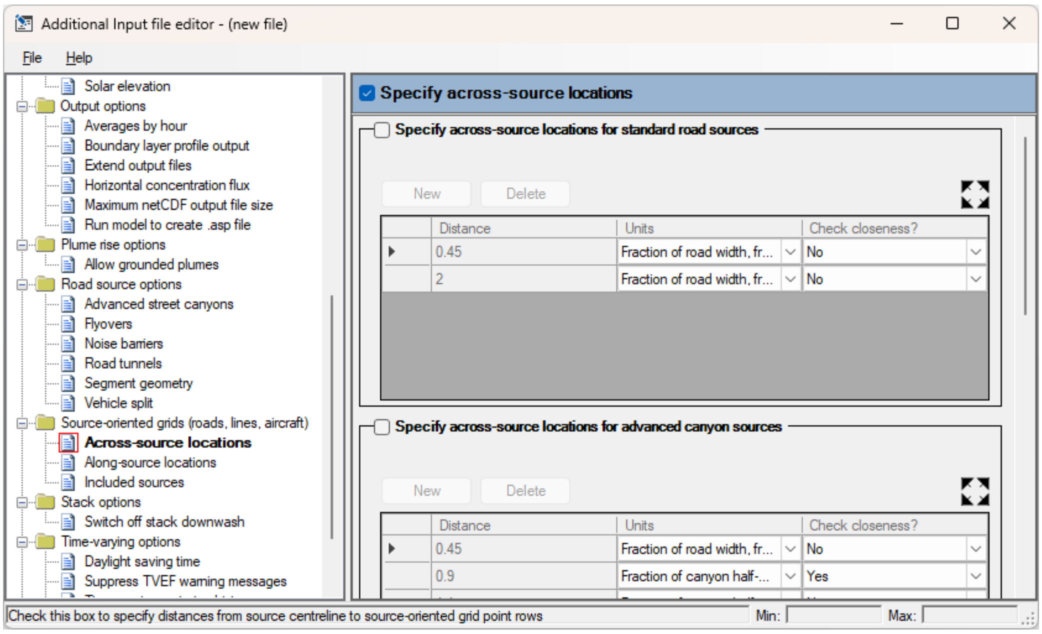


Figure 3.33 – The **Across-source locations** screen of the **Additional Input file editor**.

The right-hand side of this screen then allows you to alter the following parameters for standard road sources, advanced street canyon road sources, line sources and (licence permitting) aircraft sources separately:

- **Specify across-source locations for [standard road/advanced canyon/line/aircraft] sources**: Tick the check box to enable the relevant table to be modified. You can then use the **New** and **Delete** buttons to add new entries to/delete the highlighted entry from the table, respectively. A maximum of five entries can be added to each table. Each entry in the table corresponds to two rows of source-oriented grid points, one either side of the source at the given distance, unless the row is along the source centreline itself.

- **Distance**: This column specifies the perpendicular distance for this row(s) of source-oriented grid points (units and reference point depend on the choice of **Units**). This value must be greater than or equal to zero.

- **Units**: The following options are available:

  - **Absolute, from source edge (m)** – the value in the **Distance** column is taken as the absolute distance (in metres) from the source edge, i.e. the distance from each kerb in the case of a standard/advanced canyon road source.

  - **Absolute, from source centreline (m)** – the value in the **Distance** column is taken as the absolute distance from the source centreline; this option must be used to specify any points that are within the source if using absolute distances.

  - **Fraction of source width, from centreline** – the value in the **Distance** column is taken as a fractional distance of the source width from the source centreline, e.g. a value of 2 with **Fraction of road width, from centreline** selected would mean add points either side of the road centreline at a distance of two times the road width.

- **Fraction of canyon half-width, from centreline** (only available for advanced canyon sources) – the value in the **Distance** column is taken as a fractional distance of the canyon half-width from the source centreline. This can take a different value on each side of the source centreline if the values of *Width_L* and *Width_R*, as specified in the advanced street canyon input data file (see Section 4.2), are different.

- **Check closeness?**: If this is set to **No**, this row(s) of points will always be added. If this is set to **Yes**, each row will only be added if it is further than 1 m away from the row on the same side of the road associated with the previous entry.

Along-source locations

By default, the maximum number of source-oriented grid points (not including interpolated points) that are added to road sources is around 5 000. For line sources, the number is around 1 000. The default average spacing between pairs of points (again, not including interpolated points) is the maximum of:

(a) The spacing that would be achieved if the maximum number of source-oriented grid points were added to all source segments of that type that fall within the output grid

(b) 0.5% of the output grid length (defined as the geometric mean of the output grid length in X and Y)

(c) The source width

As already stated, the model run-time depends partly on the number of receptor points, and 5 000 source-oriented grid points is over five times as many receptor points as a standard 31 x 31 output grid. In some cases, the number of source-oriented grid points could be reduced significantly without visibly reducing the resolution of contour plots. Conversely, for a large modelling region or a large number of road sources, it may be desirable to increase the number of source-oriented grid points in order to improve the resolution of contour plots. It is therefore possible to modify the defaults described above via the additional input file. To do this, create a new additional input file or edit the existing additional input file (refer to Section 3.1.8) and enable the **Along-source locations** option (keyword IGPALONGLOCATIONS) under the **Source-oriented grids** folder, as shown in **Figure 3.34**.

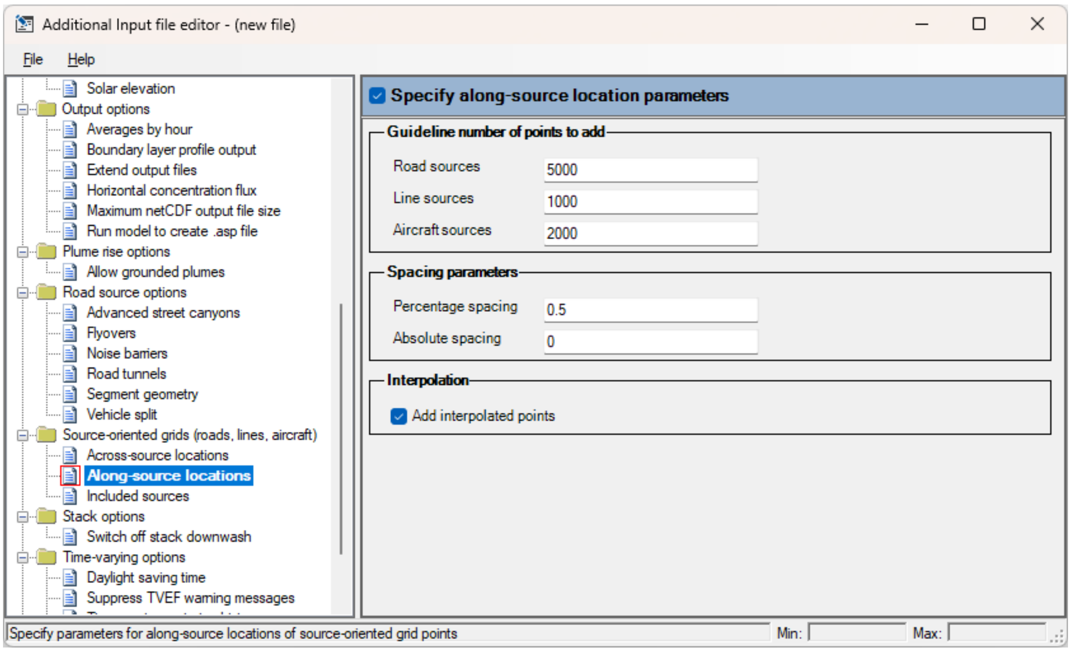


Figure 3.34 – The **Along-source locations** screen of the **Additional Input file editor**.

The right-hand side of this screen then allows you to alter the following parameters:

- **Guideline number of points to add**: The guideline maximum number of extra output points that can be added along road sources, line sources and (licence permitting) aircraft sources during the first stage of source-oriented gridding. These values must be positive.
- **Percentage spacing**: The minimum along-source spacing between the extra output points added during the first stage of source-oriented gridding, as a percentage of the output grid length, defined above. This value must be no less than 0.001 and no greater than 5.0.
- **Absolute spacing**: The actual along-road distance (in metres) between the extra output points added during the first stage of source-oriented gridding. This value will only be used if it is greater than (a) and (b) above (where (b) uses the specified **Percentage spacing** and not necessarily the default value of 0.5%).
- **Add interpolated points**: If the check box is ticked, the second stage of source-oriented gridding (adding additional points by interpolation) will be carried out. If unticked, these additional points will be not added. To obtain the best results for contouring output, it is recommended that the check box is ticked.

Included sources

By default, the model will add source-oriented grid points to all the road and line sources that lie within the output grid. To modify this behaviour, create a new additional input file or edit the existing additional input file (refer to Section 3.1.8) and enable the **Included sources** option (keyword IGPINCLUDESOURCES) under the **Source-oriented grids** folder, as shown in **Figure 3.35**.

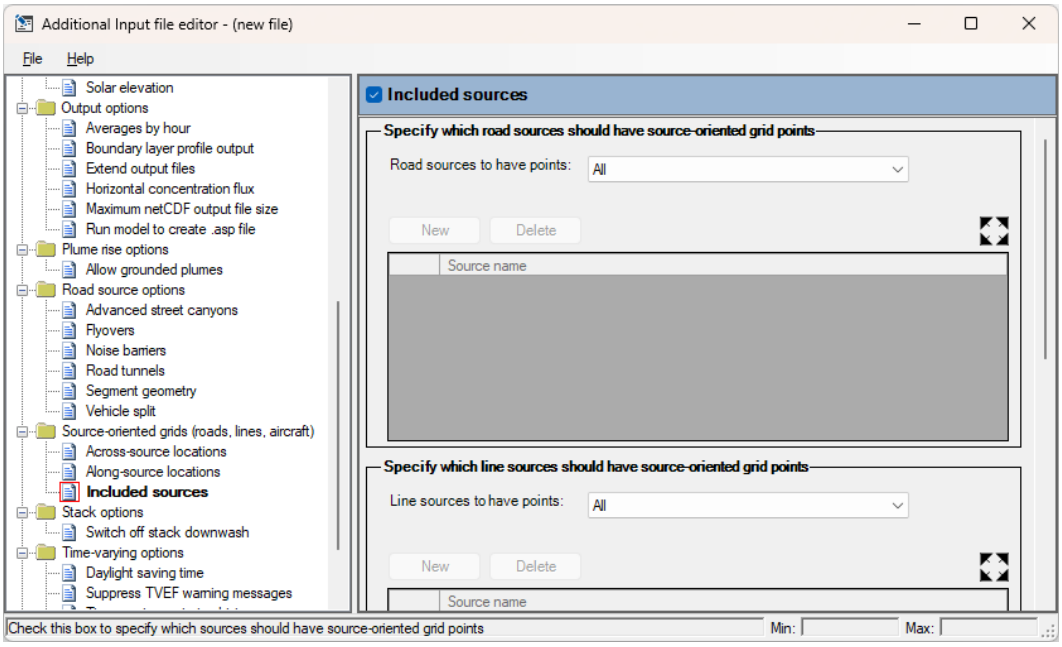


Figure 3.35 – The **Included sources** screen of the **Additional Input file editor**.

The right-hand side of this screen then allows you to alter the following parameters for road sources, line sources and (licence permitting) aircraft sources separately:

- **[road/line/aircraft] sources to be used**: Select **All** to add extra grid points to all applicable sources of that type or **None** to add no extra grid points to any sources of that type. Selecting **All but** or **Only** allows you to add source names to the relevant **Source name** table. If **All but** is selected, extra grid points will be added to all but the sources listed in this table; if **Only** is selected, extra grid points will be added to only those sources listed in this table.
- **Source name** table: The names typed into this table should exactly match the source names in the **Source** screen of the main ADMS-Urban interface. Use the **New** and **Delete** buttons to add new sources to/delete the highlighted source from the table, respectively.

3.5.3 Source-oriented grids – Point, Area, Volume sources

In addition to the user-defined grid, it may be desirable to have a high resolution grid added around point, area and volume sources. To use this option select **Point, Area, Volume** from the **Source-oriented grids** section. With this option, grid(s) extending 1 000 m from each source (or cluster of close-by sources) are added with a resolution determined by the number of grids to be added and the height of the sources. Up to 10 201 additional output points may be added using this option. The motivation for using this option is to create a high resolution output grid; therefore a warning is issued if the spacing of the nested grid points is greater than 100 m and the model will stop if the spacing is greater than 200 m.

ADMS-Urban will add the additional receptor points to the user-defined grid in the following way:

- The point, area and volume sources are grouped into clusters, where a source is part of a cluster if it is within 1 km of any other source in that cluster.
- For each cluster, the size of the area to be covered is calculated, as the size of the region covered by the sources extended in all directions by 1 km. The number of points available for that cluster is then calculated based on the maximum number of points available and the proportion of the area that the cluster occupies.
- The spacing in the X and Y directions are then calculated to allow for the maximum number of points possible in that cluster, with a minimum spacing applied based on the height of the lowest source in that cluster.
- The additional receptor points are then added to each of the clusters.

3.5.4 Specified points output

Specified points may be added to or deleted from the table on the right-hand side of the **Grids** screen by clicking on the **New** and **Delete** buttons, respectively, if the **Specified points** option is selected in the **Output location types** box. The **Delete all** button can also be used to remove all specified points from the table. There is also a right-click option to **Delete all but this** point from the table. The maximum number of points is 100. Points may be given a name by the user (e.g. “point 1”, “School”) and have to be specified with **X, Y** and **Z** coordinates. The names given to the specified points must be no longer than 30 characters and must not contain commas. It is also strongly recommended to make these names all unique. Right-clicking on an existing specified point allows you to **Copy** that point; the name and coordinates of the new point should then be modified accordingly.

If you want to specify a receptor point in a street canyon, it must be located between the road centreline and the edge of the buildings.

As well as entering up to 100 specified points in the interface, you can enter specified points using an additional specified points file by selecting the **Specified points file** option in the **Output location types** box. The additional specified points file is a comma-separated text file with the extension *.asp* that contains a line of data for every extra specified point required. Each line contains the point name, the X coordinate, the Y coordinate, and Z coordinate, in that order. The location of the *.asp* file is entered in the interface on the bottom right hand corner of the **Grids** screen (as shown in **Figure 3.36**). The **View** button allows the current *.asp* file to be viewed; left click to open the file in the default application (as selected via the **File**, **Preferences**, **Viewing options** menu item) or right-click to choose between all available applications.

An example *.asp* file (*Example.asp*) is supplied in the *<install_path>\Data* directory. There is no limit on the number of points which may be included in the .asp file, however if a very large number of points is included the model run time and memory requirements may be substantial. The specified points entered into the interface can be exported to or imported from an *.asp* file using the and buttons, respectively.

A utility is provided to help create specified points in a rectangular grid aligned with the coordinate system, a circular grid around an origin or a rectangular grid around a specified line. The created output points can be saved to a new .asp file or appended to an existing one; refer to Section 7.7 for more details.

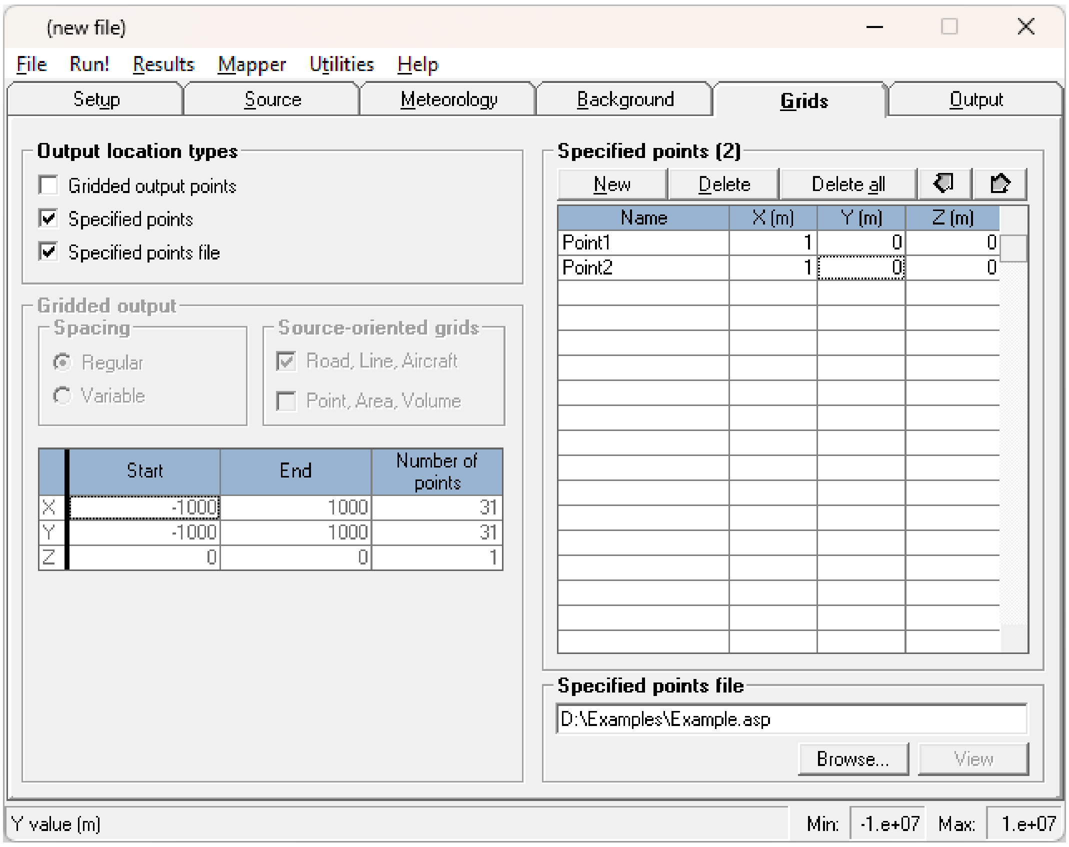


Figure 3.36 – The **Grids** screen showing specified points entered via the interface and an *.asp* file.

3.6 Output screen

The pollutants and sources for which output is to be calculated, together with the type of output required, are selected in the **Output** screen, shown in **Figure 3.37**.

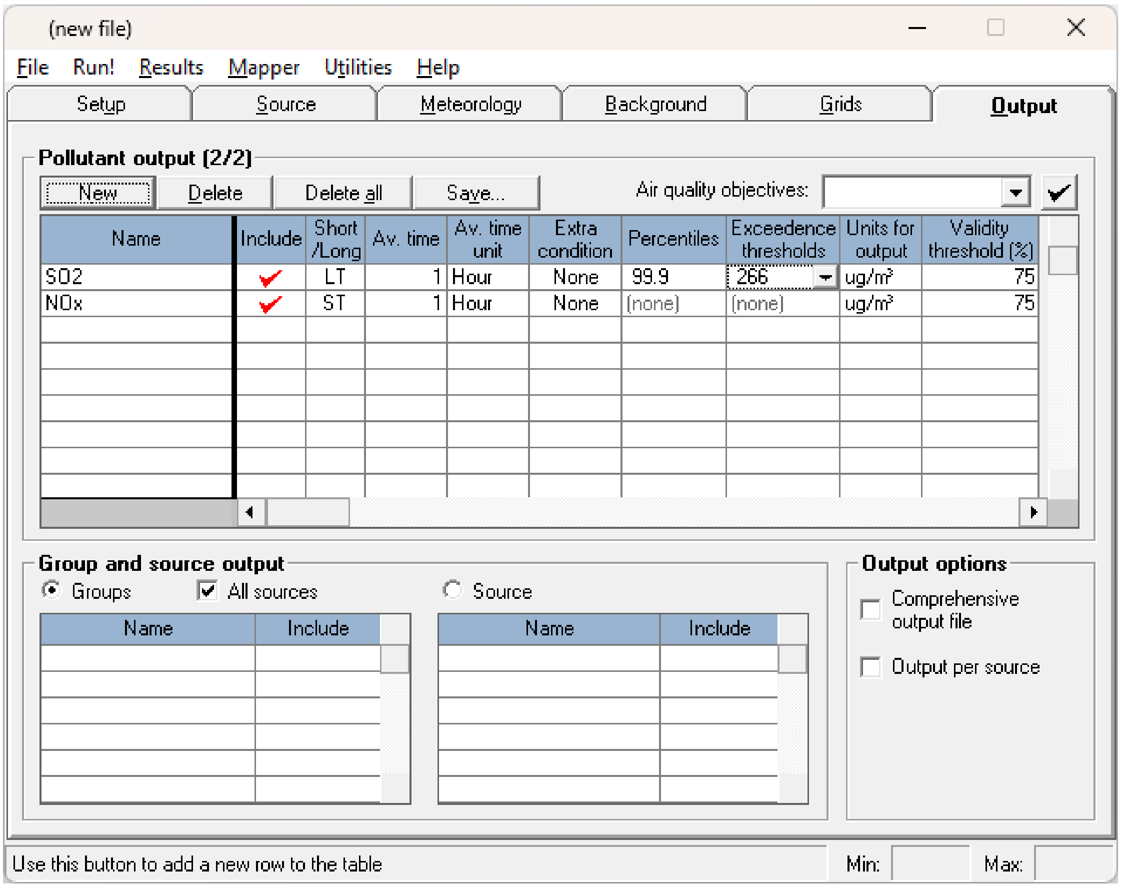


Figure 3.37 – The **Output** screen shown for the case of a long-term and short-term output of SO_2 and NO_X, respectively

3.6.1 Pollutant output

Pollutants may be added to or deleted from the **Pollutant output** table by clicking on the **New** and **Delete** buttons. Note that not all these parameters are visible at the same time in the table. They can all be displayed by scrolling to the left and right.

*The columns in the **Pollutant output** table are resizable.*

For each pollutant, parameters to be specified are described below. The user **must** select the pollutant **Name** (1), indicate whether the pollutant has to be included in the output (2), whether short or long-term calculations are required (3), select the averaging time (4), the output units (8) and the met. data validity threshold (9). The other parameters (5, 6, 7) are optional.

1. **Name**

   From the drop-down list, the name of the pollutants for which concentrations are to be calculated can be selected. The list contains all pollutants from the **Palette of Pollutants**.

   Default = **NOx**

   The same pollutant name may be entered more than once in the table in order to calculate, for example, short-term statistics for one entry and long-term statistics for another entry.

2. **Include**

   A tick must be present in this column for the model to calculate output for the corresponding pollutant. Click in the cell to toggle between tick or no tick, or type **Y** in the box to add a tick, **N** to remove it.

   Default = **Y**

   *Right-clicking within the **Include** column will open a menu allowing for the manipulation of the inclusion status for all outputs.*

3. **Short/Long**

   For each output pollutant, the type of average (short-term or long-term) to be calculated must be specified. Click in this cell to toggle between **ST** (short-term) and **LT** (long-term). Following these first three columns, various other options are available in the rest of the table depending on whether **ST** or **LT** is selected, as illustrated in **Figure 3.37**.

   If short-term averages are selected, then any gridded concentration output files (e.g. *filename.gst*) will contain output for the first 24 lines of meteorological data only. If specified point output is selected, the file *filename.pst* will contain a set of concentrations for every line of meteorological data.

   If long-term averages are selected, then the concentration output files (e.g. *filename.glt* for gridded output and *filename.plt* for specified points) will contain *a single set of concentration data*, averaged over all the lines of meteorological data. A file containing data at all output points for each meteorological line can be created for the long-term output by selecting the **Comprehensive Output file** option; see Section 3.6.4.

   Default = **ST**

4. **Av. time (value, unit)**

   Select the appropriate averaging time over which concentrations should be averaged for each pollutant. Note that ADMS-Urban allows the user to select different averaging times for different pollutants or the same pollutant *in the same run* (e.g. 15 minutes for SO_2 and 1 hour for NO_X). A drop-down list presents commonly-used averaging times, and may be accessed by clicking on the **Av. time** cell and then pressing the **ALT** and ↓ keys.

---

*If **Chemistry** is selected on the **Setup** screen, then the averaging times chosen for any pollutants involved in the NO_X chemistry calculations (NO_X, NO_2, NO, O_3, VOC) must be either all the same, or all greater than or equal to 1 hour, and similarly for the pollutants involved in the sulphate chemistry calculations (SO_2, PM_{10}, $PM_{2.5}$)*

Minimum = 0 second
Maximum = 1 week for hourly sequential meteorological data, 1 hour for statistical data
Default = 1 hour

---

*It is **compulsory** to have **Met. Data are hourly sequential** checked on the **Meteorology** screen in order to be able to select an averaging time greater than 1 hour.*

For averaging times greater than 1 hour, rolling averages of concentration may be calculated. For example, for an averaging time of 8 hours over a 24 hour period, concentrations *without* rolling averages would be calculated for hours 1 to 8, 9 to 16 and 17 to 24 (i.e. 3 sets of calculations). However, if rolling averages are required, concentrations are calculated for hours 1 to 8, 2 to 9, 3 to 10, and so on. To enter a rolling average select **Hour rolling** in the **Unit** cell.

5. **Extra condition**

The options for an **Extra condition** are **None** and **Maximum daily**. If **None** is selected, no extra condition is applied. If **Maximum daily** is selected, then for short-term output, for each day, the maximum concentration (or deposition) value for the valid averaging periods which end on that day is output. For long-term output, the maximum concentration per day (based on valid averaging periods which end on that day) is used in the percentiles and exceedence calculations.

Long-term average concentration and deposition values are always calculated using all valid averaging periods and not just the maximum for each day.

Default=**None**

6. **Percentiles** (long-term average only)

If long-term averages are being calculated, then up to 20 percentiles of concentration may be calculated for each pollutant. The N^{th} percentile is the concentration below which *N*% of the values fall. As an example, the 50^{th} percentile of hourly average concentrations over a 24-hour period would be the 12^{th} highest (i.e. 50% of 24) of the 24 one-hour averages. In this case, the **Av. time** would be one hour and the meteorological data for a period of one day would be used.

Different percentiles may be selected for different pollutants *in the same run* (e.g. 100^{th} and 99.9^{th} percentiles for SO_2 and 100^{th} and 98^{th} percentiles for NO_x). Worst case concentrations may be found by calculating the 100^{th} percentile (i.e. the maximum concentration).

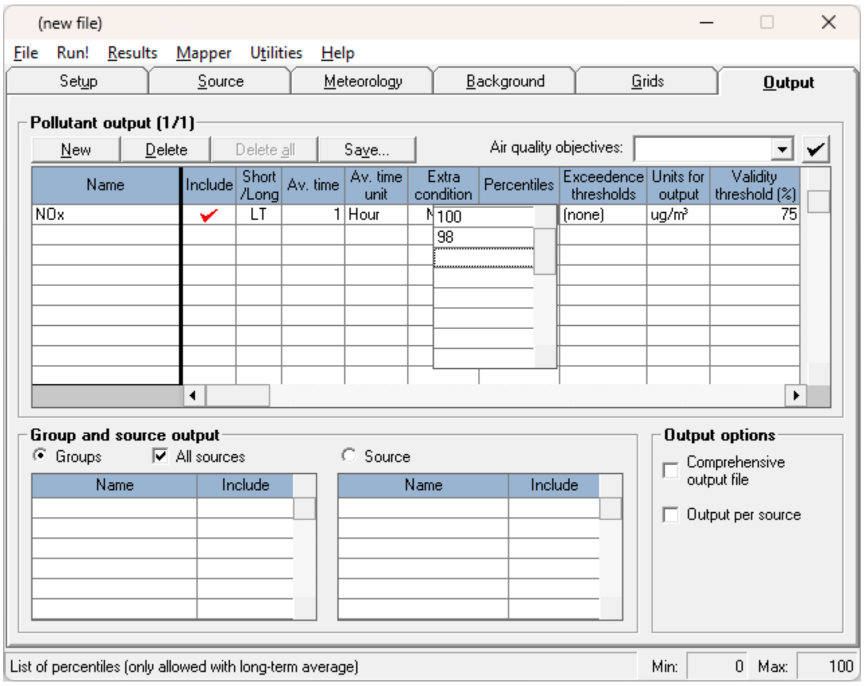


Figure 3.38 – Entering data into the **Percentiles** table.

Percentile values are entered via a table (**Figure 3.38**) which is made available when the drop-down arrow in the **Percentiles** value cell is clicked. Click on the first blank cell and type the desired percentile value. Pressing the down arrow moves to the next blank cell to allow further values to be entered, or pressing **Enter** exits the table, retaining the values already entered. Pressing the **Delete** key on a particular table entry deletes that particular percentile, whereas pressing the **Delete** key on the **Percentiles** cell when the table is minimised allows you to delete all percentiles from the table. The resulting list of percentiles will then be used for the given pollutant. Values are automatically sorted into descending order for convenience.

7. **Exceedence thresholds** (long-term average only)

   If long-term averages are being calculated, then up to 20 exceedence values may be entered. For each pollutant, the exceedence values are entered in the same units as displayed in the **Units for output** cell. The equivalent number of exceedences per annum will then be calculated by the model at each output point for each exceedence value entered. This is a useful way of looking at the results from the model because many air quality standards are stated in terms of number of exceedences per year (refer to Section 3.6.2 and Appendix D for more details).

   As for percentiles, different exceedence values may be entered for different pollutants. Exceedence values are entered and/or deleted via a table in the same way as percentiles.

8. **Units for output**

   For each output pollutant, the units may be selected from the list available by clicking on this cell:

ng/m^3	nanograms per cubic metre (1 ng = 10^{-9} g)
ug/m^3	micrograms per cubic metre (1 μg = 10^{-6} g)
mg/m^3	milligrams per cubic metre (1 mg = 10^{-3} g)
g/m^3	grams per cubic metre
ppb	parts per billion by volume
ppm	parts per million by volume

Default = ug/m^3

---

ppb and ppm are not available as output units for particulate pollutants as they are only applicable to gases.

If the **Odours** option has been checked on the **Setup** screen, the units of output are ou_e/m^3.

9. **Validity threshold (%)**

The value in this column states the percentage of met. lines in a given averaging period that need to be valid in order for that averaging period to be considered valid. For example, for an **Av. time** of 8 hours, if the validity threshold was set at 75%, you would need at least 6 met. lines to be valid in any given averaging period for that averaging period to be considered valid. Concentration/deposition values for any invalid averaging periods will be output as -999 in the (short-term) output files and not used in any subsequent calculations of maximum daily values (short-term or long-term), long-term averages, percentiles or exceedences (long-term). A maximum daily value is itself only considered valid if there is at least the specified percentage of valid averaging periods within that day. If invalid, it will also be output as -999 in the (short-term) output files and not used in any subsequent calculations of percentiles or exceedences (long-term). Conversely, long-term averages, percentiles and exceedence values will always be given in the long-term output files as long as there is at least one valid averaging period from which to calculate these statistics; however a warning will be issued if fewer than the specified percentage of averaging periods were valid.

Minimum = 0 %
Maximum = 100 %
Default = 75 % (in line with various monitoring guidelines)

3.6.2 Air quality objectives

Many users are interested in comparing model results with air quality objectives. An option is available to facilitate this for some standard sets of objectives. Click on the **Air quality objectives** list on the **Output** screen, select a set of objectives and click the tick (see **Figure 3.39**). This action will populate the output table with the relevant air quality objective values (see **Figure 3.40**). When using this tool, the user is advised to check that the objectives are up-to-date.

---

*Using the **Air quality objectives** tool will delete all previous entries in the output table.*

A full list of the EU, UK, French, Lithuanian, US and WHO air quality objectives is given in Appendix D.

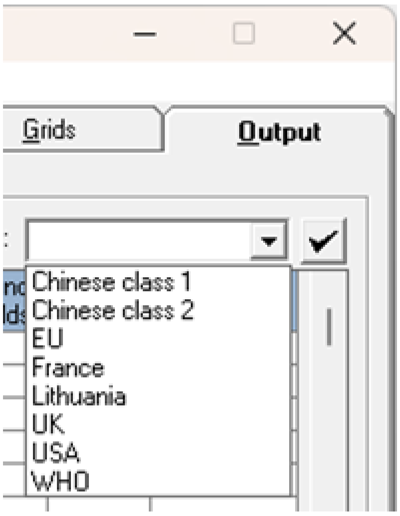
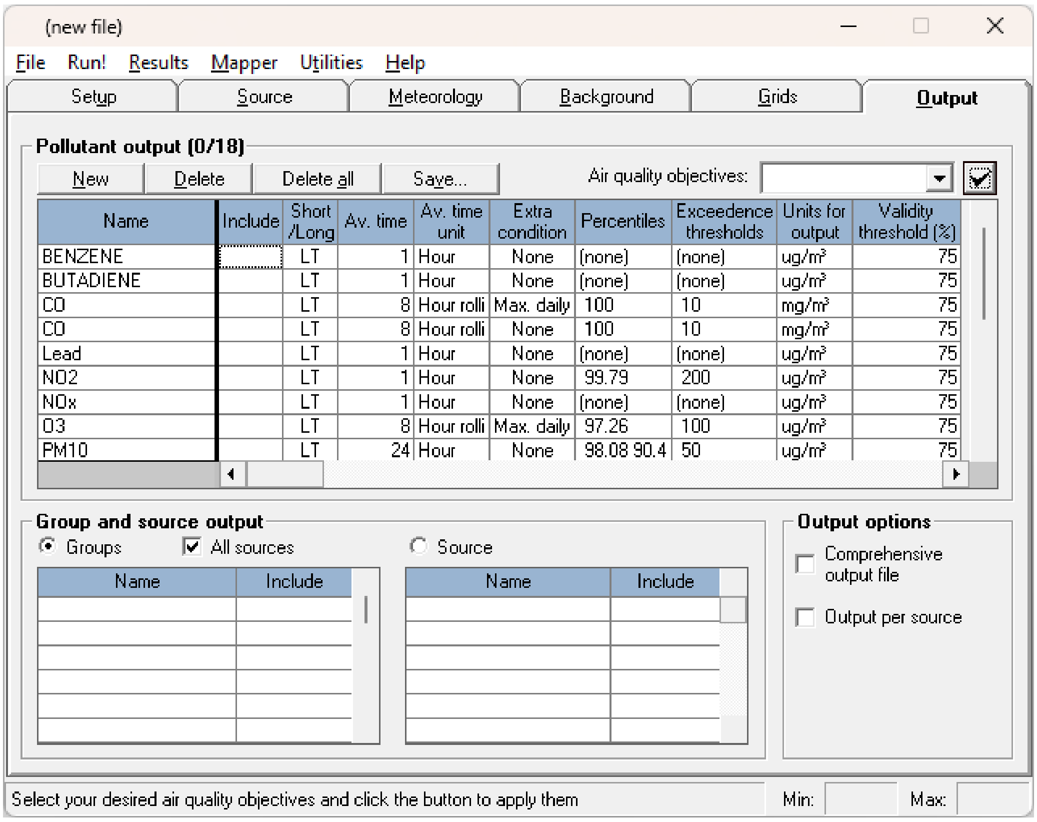


Figure 3.39 – The **Air quality objectives** list.

Pollutant output *table entries added using the* ***Air quality objectives*** *tool are, by default, not included in the run. The user should put a tick in the* ***Include*** *column for those outputs required.*



Name	Include	Short /Long	Av. time	Av. time unit	Extra condition	Percentiles	Exceedence thresholds	Units for output	Validity threshold (%)
BENZENE		LT	1	Hour	None	(none)	(none)	ug/m³	75
BUTADIENE		LT	1	Hour	None	(none)	(none)	ug/m³	75
CO		LT	8	Hour rolli	Max. daily	100	10	mg/m³	75
CO		LT	8	Hour rolli	None	100	10	mg/m³	75
Lead		LT	1	Hour	None	(none)	(none)	ug/m³	75
NO2		LT	1	Hour	None	99.79	200	ug/m³	75
NOx		LT	1	Hour	None	(none)	(none)	ug/m³	75
O3		LT	8	Hour rolli	Max. daily	97.26	100	ug/m³	75
PM10		LT	24	Hour	None	98.08 90.4	50	ug/m³	75

Figure 3.40 – The **Output** screen showing the effect of selecting **UK** from the **Air quality objectives** list and then clicking the tick.

Limitations

It is not possible to precisely define all the air quality objectives given in Appendix D using the **Pollutant output** table. Many of these can however be calculated using a Comprehensive Output File and the Comprehensive Output File Processor (see Section 3.6.4). Explanations for why some of the air quality objectives have not been defined exactly are given below:

- For air quality objectives defined in terms of a running annual mean, the calculated statistic in the **Pollutant output** table is defined as a long-term average. Therefore, the calculated statistic will agree exactly with the objective if one year of meteorological data is used; for longer time periods, the calculated statistic differs from the objective. A running annual mean can be calculated for longer periods of meteorological data using a Comprehensive Output File and the Comprehensive Output File Processor.
- Air quality objectives defined as an average of a particular statistic over a number of years are specified in the **Pollutant output** table so that the result for one year will be calculated, if one year of meteorological data are used. In order to model a concentration that can be compared directly with the objective, several years of meteorological data should be run independently and the resulting values averaged. Alternatively, multiple years of meteorological data can be run to produce a Comprehensive Output File and the Comprehensive Output File Processor used to generate averages for each year.
- Air quality objectives based on 3-month, quarterly or seasonal averages are omitted from the **Pollutant output** table. Comparison to these objectives can be made by running each of the required meteorological periods separately, for instance, by using the meteorological subset option, and requesting a long-term average in each case. Alternatively, the full period of meteorological data can be run to generate a Comprehensive Output File and then the Comprehensive Output File Processor used to generate averages for the required averaging times.
- Air quality objectives based on AOT40 cannot be calculated directly from the model. If comparison to these objectives is required, it is recommended that concentrations for each hour are output by the model (by selecting short-term output) and that these concentration values are processed manually.

Saving air quality objective files

It is also possible to save your own set of **Air quality objectives**. Firstly, the required outputs should be entered into the **Pollutant output** table. Selecting **Save...**. then brings up the **Save pollutant outputs to file** screen (**Figure 3.41**). In this screen enter:

- the **Caption**, to appear in the **Air quality objectives** list, and
- the **Description**, to appear in the helpline,

then click **OK**. A **Save As** dialogue will appear. The file must be saved in the ADMS-Urban installation directory for the set to appear in the **Air quality objectives** list.

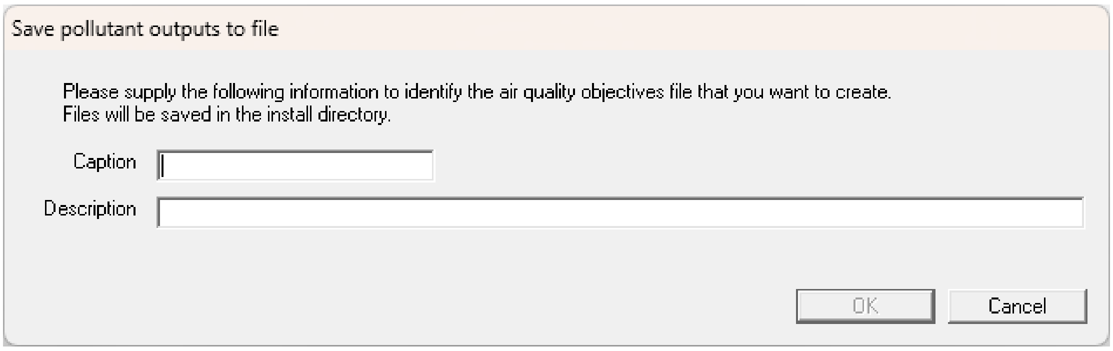


Figure 3.41 – The **Save pollutant outputs to file** screen.

3.6.3 Group and source output

For each model run, the user may choose to calculate concentrations either for one specific source or for one or more groups of sources.

To obtain output for a specific source, select the **Source** option above the right-hand lower table on the **Output** screen (see **Figure 3.37**). The table will show in the **Name** column a list of all possible individual sources available for the current model run. *Only one* of these sources may be selected (click in the **Include** cell next to the required source or enter **Y** in that cell). Choosing the single source option generates files containing model parameters on the plume centreline, e.g. σ_y, σ_z, that may be useful to view using the **Line Plot** utility (see Section 6.3 for further details).

If more than one source is to be modelled, then select the **Groups** option above the left-hand lower table. This table shows a list of all the user-defined groups that have been set up for the current model run in the **Name** column. Up to 20 of these groups may be selected for each run (click in the **Include** cell next to the required group or enter **Y** in that cell). In addition to these user-defined groups a group containing all of the defined sources can be modelled. To include this in the run check the **All sources** check box. Except when chemistry is modelled, the concentration output for a group consists of a linear superposition of the concentrations from its constituent sources.

A group may be defined to contain only one source, so that output may be generated for several individual sources if required. For runs with more than 20 sources, it may still be possible to obtain concentration output for each source separately in a single model run using the 'Output per source' option. See Section 3.6.5 for further details, including a list of restrictions that apply to this option.

3.6.4 Comprehensive output file

Selecting long-term output means that the output files contain a single set of output data, for example, the average or maximum over all of the lines of meteorological data. By selecting the **Comprehensive output file** option, an additional file containing output data for each met line, for all the pollutants selected for long-term output, is also created. This file is created in netCDF format and a full description of the file is given in Section 6.1.12.

The netCDF file holds datasets containing the concentration and dry or wet deposition data (if selected) for each unique pollutant and averaging time combination selected for long-term output, at all standard grid points, specified points and source-oriented (but not interpolation) grid points. For averaging times greater than one hour, the averaging time used in creating this file is one hour. Percentile or exceedence data are not output to the netCDF file. As well as containing the concentration (and deposition) data, the netCDF file also contains basic meteorology relating to each of the lines in the met. file.

A Comprehensive Output File Processor is supplied with ADMS-Urban and can be launched from the **Results**, **Process comprehensive output** *menu. A full User Guide for the Comprehensive Output File Processor can also be launched via the* **Help** *menu.*

The Comprehensive Output File Processor is able to calculate longer averaging periods, percentiles and exceedences and output the results in standard ADMS format output files.

The Comprehensive Output File Processor cannot currently process files containing odour units.

The netCDF files produced by this option can be very large. By default the run will stop if a netCDF file of greater than 20 Gb is required, based on the output options selected. The file size limit can be altered using the additional input file. This is done by creating an additional input file or editing the existing additional input file (refer to Section 3.1.8), enabling the **Maximum NetCDF output file size** option (keyword NETCDFOUTPUT) and entering the **Maximum file size** (in Mb), as shown in **Figure 3.42**.

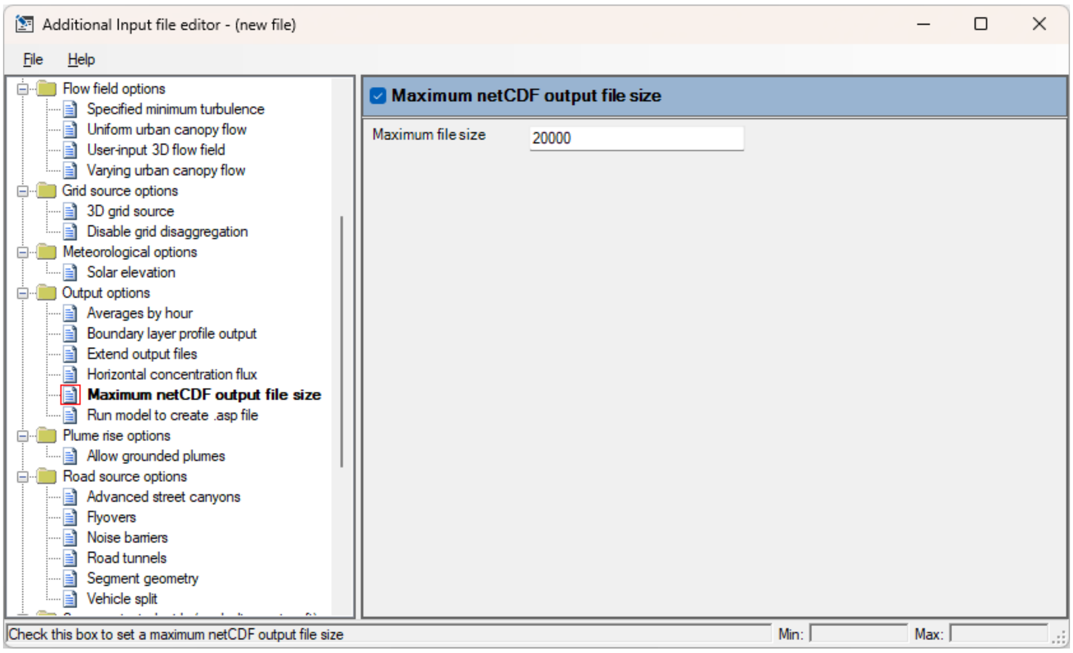


Figure 3.42 – The **Maximum NetCDF output file size** screen in the **Additional Input file editor**.

3.6.5 Output per source

The contribution to pollutant concentration or deposition values at particular output locations from emissions of individual sources or source types is of interest for source apportionment and for developing targeted mitigation strategies. This type of output can be obtained using groups containing one or more sources but limits on the number of user-defined groups which can be included in a model run make this approach impractical for large urban areas with significant numbers of sources.

The **Output per source** option is designed to allow the assessment of the concentration and/or deposition contributions of large numbers of sources to a small number of output locations. Note that the memory requirements and the output file size may become large for this option, especially for short-term output.

To enable this option, tick the **Output per source** checkbox on the **Output** screen. Note that this checkbox will appear greyed out if any incompatible options are currently selected in the interface, namely:

- **Specified points** output must be selected in the **Grids** screen.
- The **Groups** output must be selected and the **All sources** checkbox ticked in the **Output** screen.
- The **Chemistry** option cannot be used in combination with this option.

The **Output per source** output data is written to an *.sst* (source short-term) and/or *.slt* (source long-term) file. A full description of the format of both these output file types is provided in Section 6.1.4. Note that no background concentrations are included in the concentration values given in the *.sst* and *.slt* files.

The **Results\Display footprint** menu item can also be used to graphically display the data within *.sst*/*.slt* files in the Mapper. This colour-codes the sources themselves based on their individual contribution to a specified output point. Further details are provided in Section 6.4.

3.6.6 Long-term averages for each hour of the day

In addition to the output options available in the **Output** screen of the ADMS-Urban interface it is possible to output the long-term average for each hour of the day, i.e. the average of all of the hour 1 concentrations, the average of all of the hour 2 concentrations etc. This option is enabled using an additional input file and applies to all long-term outputs included in the **Output** screen of the ADMS-Urban interface. To enable this option create a new additional input file or edit the existing one (see Section 3.1.8) and enable the **Long-term averages for each hour** section, as shown in **Figure 3.43** (keyword CALCAVGSBYHR).

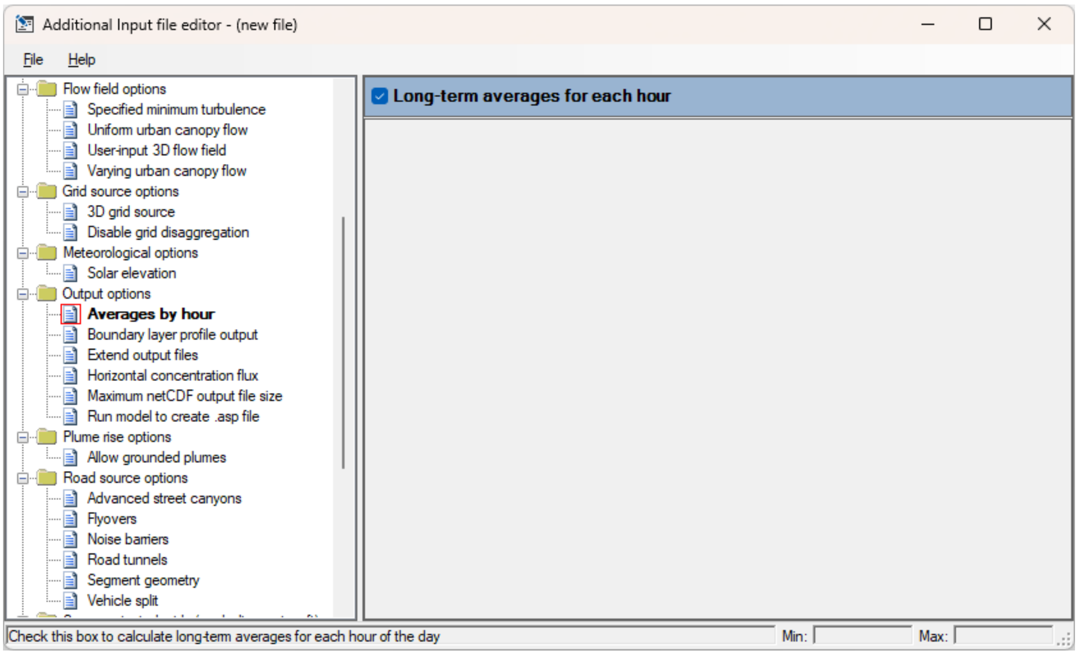


Figure 3.43 – The **Long-term averages for each hour** option enabled in the **Additional input file editor**.

SECTION 4 Additional Model Options

This section describes the additional model options available in ADMS-Urban.

Options such as deposition, chemistry, odours, buildings, complex terrain, coastline, spatial splitting and source exclusion are available in the various sections of the **Setup** screen, as illustrated in **Figure 4.1**. Options related to the meteorological data are available in the **Meteorology** screen. Time-varying data options are available in the **Source** screen. Other options are available by means of the additional input file; refer to Section 3.1.8 for detailed instructions for creating an additional input file.

Some of these model options require only basic model input, some require further information to be entered in the ADMS-Urban interface and some require information to be entered both in the ADMS-Urban interface and in an additional input file.

The permitted combinations of all options are given in Appendix A.

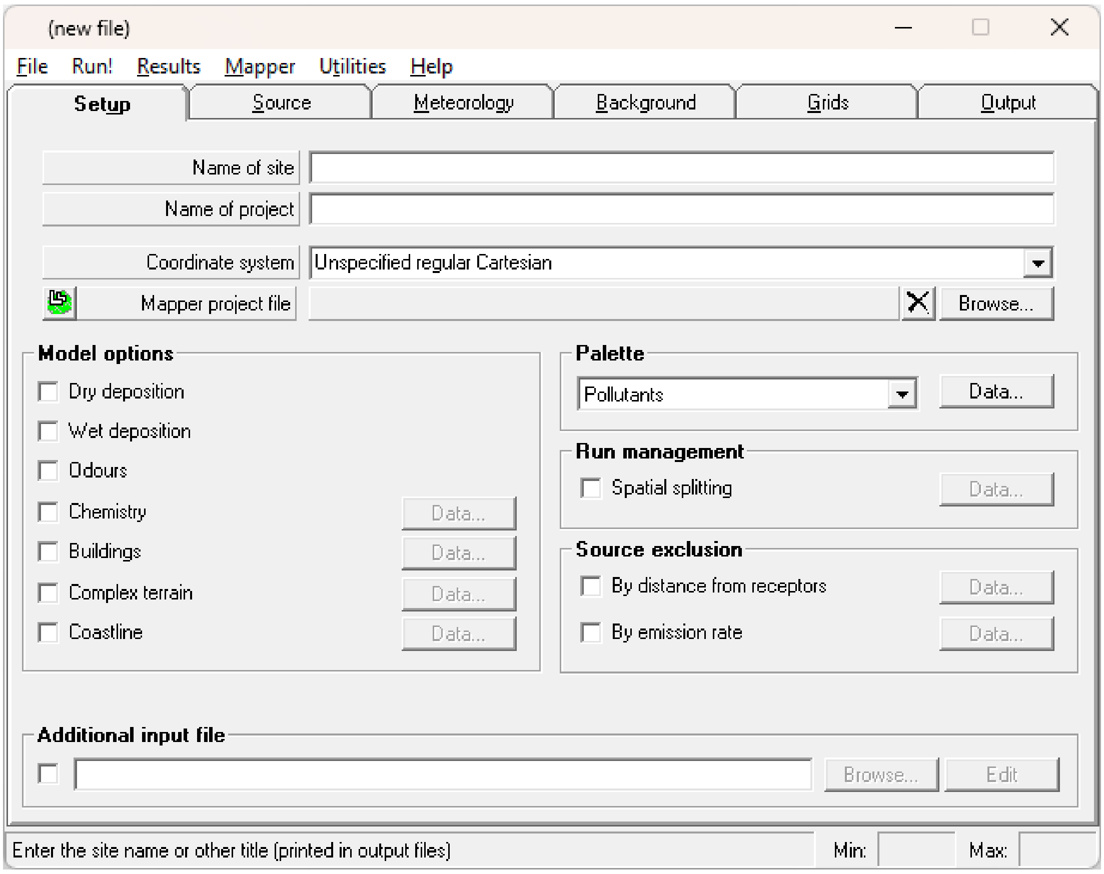


Figure 4.1 – The ADMS-Urban **Setup** screen.

4.1 Time-varying emissions

ADMS-Urban includes the option to model concentrations from emissions that vary with time. To do this the user should check the **Time-varying emission factors** box in the **Source** screen and then click on the **Data Source...** button to enter the choice of time-varying options on the **Time-varying emission factors** screen, **Figure 4.2**. The current selection is also shown in the information panel next to the **Data Source...** button.

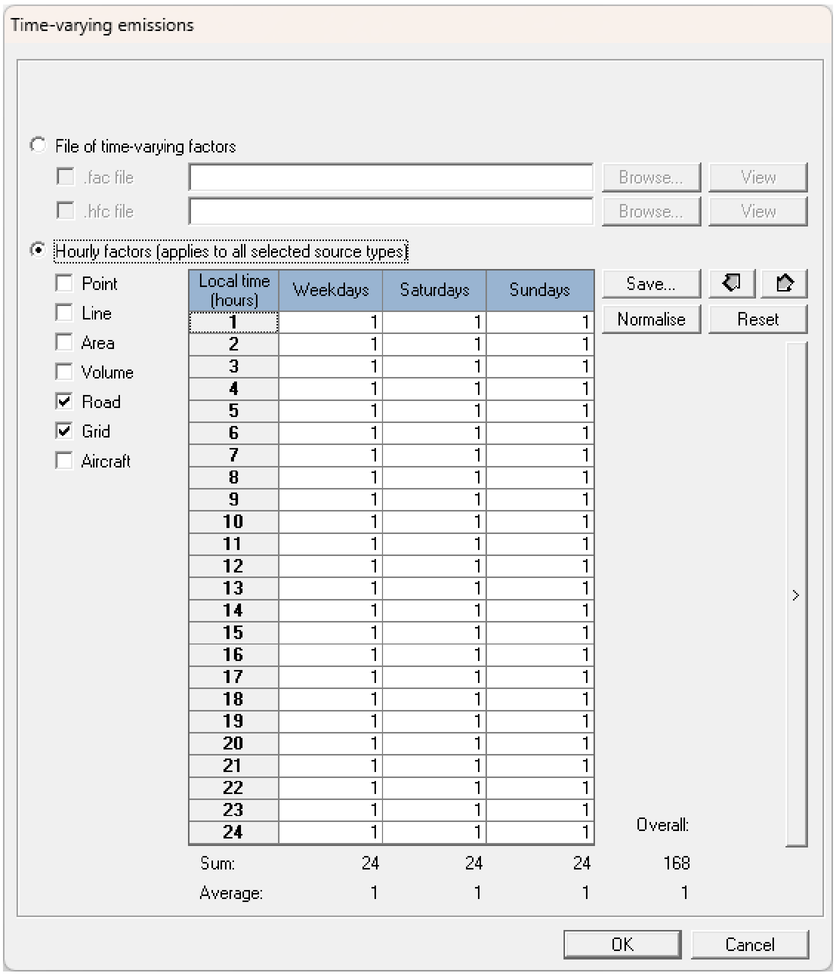


Figure 4.2 – The **Time-varying emission factors** screen

There are two options given on the **Time-varying emission factors** screen:

- A single set of hourly factors can be entered in the interface by hand for weekdays, Saturdays and Sundays. If selected, these factors will be applied to all sources of the selected types, refer to Section 4.1.3 for details.
- Alternatively, more detailed data may be entered using a file of time-varying factors. Two types of file may be used:
    - *.fac*: In which profiles of hourly factors for each day of the week (7 x 24 hours) or weekdays (24 hours), Saturdays (24 hours) and Sundays (24 hours) (i.e. diurnal profiles) and monthly profiles can be defined. These profiles can then be assigned to specific sources or source-pollutant combinations. Additionally, each source or source-pollutant combination can be defined to only be in operation for a specific

range of wind directions. For more details on using a *.fac* file, see Section 4.1.1.

- *.hfc:* In which profiles of hour-by-hour varying emission factors (8760 factors for one year of met data) can be defined and then assigned to specific sources or source-pollutant combinations. More details on the *.hfc* file can be found in Section 4.1.2.

Each source can be listed in either the *.hfc* file or the *.fac* file but not both.

Table 4.1 shows which source types can be used with each type of time-varying emission factors.

	.fac file	*.hfc* file	Interface
Road	✓	✓	✓
Industrial	✓	✓	✓
Grid*	✓	✗	✓

Table 4.1 – Time-varying emissions options for different source types. *Refers to standard grid source (3D grid sources do not require time-varying emission factors)

Meteorological data must contain the year, day and hour if the ***Time-varying emission factors*** *option is selected.*

4.1.1 Emission factors from a .fac file

ADMS-Urban can take account of diurnal and seasonal changes in emissions for each source or source-pollutant combination, and can model sources or source-pollutant combinations that are only in operation for a specific range of wind directions. This is done by setting up an extra input file, *.fac*, which contains the emission factor profiles and data linking each source or source-pollutant combination to the appropriate emission factor profile(s). The format of the *.fac* file is described below.

The full functionality of this option is as follows:

- Hourly factors can be defined for weekdays, Saturdays and Sundays (‘3-day diurnal profiles’) or for each day of the week (‘7-day diurnal profiles’). A different diurnal profile can be applied to each source or source-pollutant combination.
- Monthly factors can be defined for each month (‘monthly profiles’). A different monthly profile can be applied to each source or source-pollutant combination.
- Each source or source-pollutant combination can be defined to be in operation only for a specific range of wind directions.
- A single set of profiles (per pollutant, or across all pollutants) can be applied to all road sources not explicitly listed in the *.fac* or *.hfc* file, by including a section(s) headed ‘default road’.
- A single set of profiles (per pollutant, or across all pollutants) can be applied to all of the grid cells by including a section(s) headed ‘grid’.

To use a *.fac* file in a modelling run, select **File of time-varying factors** on the **Time-varying emission factors** screen in the ADMS-Urban interface and tick the appropriate box. In the text box, type the full path name of the required *.fac* file, or use the **Browse** button to find available *.fac* files, or drag and drop a *.fac* file from Explorer. The **View** button allows the current *.fac* file to be viewed; left click to open the file in the default application (as selected via the **File**, **Preferences**, **Viewing options** menu item) or right-click to choose between all available applications.

.fac file format

An example time-varying emission factors *.fac* file is shown in **Figure 4.3**. This file is also supplied in the *<install_path>\Data* directory. The file contains two 3-day diurnal profiles, one 7-day diurnal profile and one monthly profile, followed by data for two point sources, *point1* and *point2*, a road source *road1*, a default road setting *default road* and a grid source *grid*. Note that the data in the file are factors, i.e. the emission rates entered on the **Emissions** screen of the interface will be multiplied by these values. The data in the file are as follows.

1. **Version**

   '`FacVersion4`' is the current file version identifier.

   Note that ADMS-Urban 5.1 will also accept .fac *files with the FacVersion2 or FacVersion3 identifier, but future versions may not.*

   ---

2. **3-day diurnal profiles**

   Line 1: Number of 3-day diurnal profiles included in the file
   Line 2: Name of the 3-day diurnal profile, up to 30 characters
   Line 3: The 24 hourly factors for weekdays, in order, from hour 1 to hour 24
   Line 4: The 24 hourly factors for Saturdays, in order, from hour 1 to hour 24
   Line 5: The 24 hourly factors for Sundays, in order, from hour 1 to hour 24

   Line 1 must always be included, even if no 3-day diurnal profiles are included in the file (enter a zero in this case). Lines 2-5 are repeated for each 3-day diurnal profile.

   After all the 3-day diurnal profile data, there is one blank line before the 7-day diurnal profile data.

3. **7-day diurnal profiles**

   Line 1: Number of 7-day diurnal profiles included in the file
   Line 2: Name of the 7-day diurnal profile, up to 30 characters
   Line 3: The 24 hourly factors for Mondays, in order, from hour 1 to hour 24
   Line 4: The 24 hourly factors for Tuesdays, in order, from hour 1 to hour 24
   Line 5: The 24 hourly factors for Wednesdays, in order, from hour 1 to hour 24
   Line 6: The 24 hourly factors for Thursdays, in order, from hour 1 to hour 24
   Line 7: The 24 hourly factors for Fridays, in order, from hour 1 to hour 24
   Line 8: The 24 hourly factors for Saturdays, in order, from hour 1 to hour 24
   Line 9: The 24 hourly factors for Sundays, in order, from hour 1 to hour 24

Line 1 must always be included, even if no 7-day diurnal profiles are included in the file (enter a zero in this case). Lines 2-9 are repeated for each 7-day diurnal profile.

After all the 7-day diurnal profile data, there is one blank line before the monthly profile data.

4. **Monthly profiles**

Line 1: Number of monthly profiles included in the file
Line 2: Name of the monthly profile, up to 30 characters
Line 3: The 12 monthly factors for the 12 months, in order, from January to December

Line 1 must always be included, even if no monthly profiles are included (enter a zero in this case). Lines 2-3 are repeated for each monthly profile.

After all the monthly profile data, there is one blank line before the source-specific data.

5. **Source-specific data**

Line 1: `SrcName[|PolName], HourlyFlag, MonthlyFlag, WindFlag`
Line 2: Name of the diurnal profile to be applied to this source/source-pollutant combination (only included if `HourlyFlag` = 1)
Line 3: Name of the monthly profile to be applied to this source/source-pollutant combination (only included if `MonthlyFlag` = 1)
Line 4: `PhiStart`, `PhiEnd` (only included if `WindFlag` = 1)

where

`SrcName` is the source name (this should be 'grid' for the grid source).

`PolName` (optional) is the pollutant name. If included, it should be separated from the source name by the vertical bar ('|') character.

`HourlyFlag` = 1 if a diurnal profile (3-day or 7-day) should be applied to this source/source-pollutant combination, 0 otherwise.

`MonthlyFlag` = 1 if monthly factors should be applied to this source/source-pollutant combination, 0 otherwise.

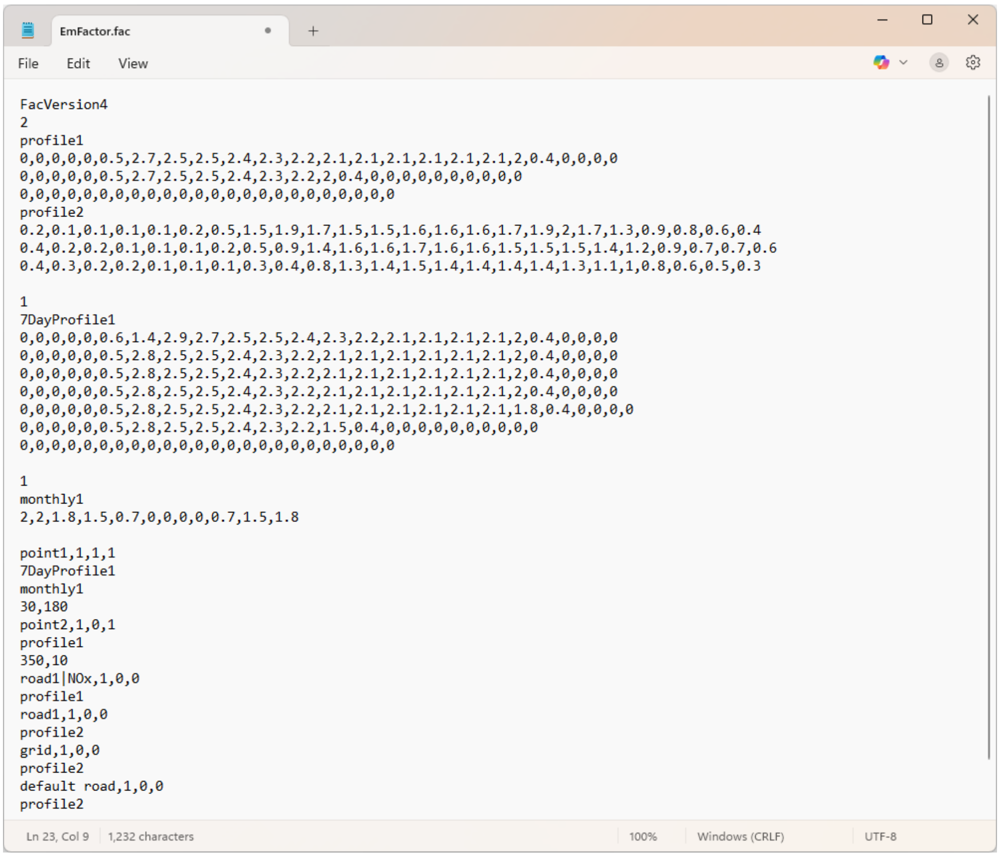
`WindFlag` = 1 if the source/source-pollutant combination is only operational for a specific range of wind directions, 0 if it is operational for all wind directions.

`PhiStart` is the start of the range of surface wind directions for which the source/source-pollutant combination is operational. Values are in degrees moving clockwise from North.

`PhiEnd` is the end of the range (inclusive). Values are in degrees moving clockwise from North.

Lines 1, and 2-4 if appropriate, are repeated for each desired source/source-pollutant combination.

In the example shown in **Figure 4.3**, all pollutants emitted by *point1* have variation for each day of the week, using *7DayProfile1*; for each month of the year, using *monthly1*; and are restricted to operating when the wind direction is between 30 and 180 degrees. All pollutants emitted by *point2* vary diurnally, using *profile1*, and the stack is only operational if the wind direction is greater than or equal to 350° or less than or equal to 10°. The NO_X emissions from *road1* vary diurnally using *profile1*, whereas all other pollutants emitted by *road1* vary diurnally using *profile2*. All other road sources (*default road*) vary diurnally using *profile2*. The grid source varies diurnally using *profile2*.




```
FacVersion4
2
profile1
0,0,0,0,0,0.5,2.7,2.5,2.5,2.4,2.3,2.2,2.1,2.1,2.1,2.1,2.1,2.1,2,0.4,0,0,0,0
0,0,0,0,0,0.5,2.7,2.5,2.5,2.4,2.3,2.2,2,0.4,0,0,0,0,0,0,0,0,0,0
0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0
profile2
0.2,0.1,0.1,0.1,0.1,0.2,0.5,1.5,1.9,1.7,1.5,1.5,1.6,1.6,1.6,1.7,1.9,2,1.7,1.3,0.9,0.8,0.6,0.4
0.4,0.2,0.2,0.1,0.1,0.1,0.2,0.5,0.9,1.4,1.6,1.6,1.7,1.6,1.6,1.5,1.5,1.5,1.4,1.2,0.9,0.7,0.7,0.6
0.4,0.3,0.2,0.2,0.1,0.1,0.1,0.3,0.4,0.8,1.3,1.4,1.5,1.4,1.4,1.4,1.4,1.3,1.1,1,0.8,0.6,0.5,0.3

1
7DayProfile1
0,0,0,0,0,0.6,1.4,2.9,2.7,2.5,2.5,2.4,2.3,2.2,2.1,2.1,2.1,2.1,2,0.4,0,0,0,0
0,0,0,0,0,0.5,2.8,2.5,2.5,2.4,2.3,2.2,2.1,2.1,2.1,2.1,2.1,2.1,2,0.4,0,0,0,0
0,0,0,0,0,0.5,2.8,2.5,2.5,2.4,2.3,2.2,2.1,2.1,2.1,2.1,2.1,2.1,2,0.4,0,0,0,0
0,0,0,0,0,0.5,2.8,2.5,2.5,2.4,2.3,2.2,2.1,2.1,2.1,2.1,2.1,2.1,2,0.4,0,0,0,0
0,0,0,0,0,0.5,2.8,2.5,2.5,2.4,2.3,2.2,2.1,2.1,2.1,2.1,2.1,2.1,1.8,0.4,0,0,0,0
0,0,0,0,0,0.5,2.8,2.5,2.5,2.4,2.3,2.2,1.5,0.4,0,0,0,0,0,0,0,0,0,0
0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0

1
monthly1
2,2,1.8,1.5,0.7,0,0,0,0,0.7,1.5,1.8

point1,1,1,1
7DayProfile1
monthly1
30,180
point2,1,0,1
profile1
350,10
road1|NOx,1,0,0
profile1
road1,1,0,0
profile2
grid,1,0,0
profile2
default road,1,0,0
profile2
```

Figure 4.3 − Example time-varying emission factors file (*.fac*).

Important points to note:

- The name used to identify the grid source must always be ‘grid’
- None of the text in the file is case-sensitive.
- Source, pollutant and profile names can contain spaces, but not commas.
- The file must be in comma-separated format.
- There should be no blank lines, except for the lines between the 3-day diurnal profile data and the 7-day diurnal profile data, between the 7-day diurnal profile data and the monthly profile data, and between the monthly profile data and the source-specific data.

- Any source that is completely omitted from the *.fac* file is assumed to have constant emissions across all pollutants emitted by that source, unless an *.hfc* file is also used, see Section 4.1.2.
- If only the source name is specified in a given entry, the listed profile(s) and/or operational wind direction data are assumed to apply to all pollutants emitted by that source, apart from any pollutants for which profiles are listed in a separate source-pollutant entry for that source.
- If profiles are provided for some but not all pollutants emitted by a given source (and there is no non-pollutant-specific entry for that source), the model will assume that the missing pollutants have constant emission, and a warning message will be issued (unless the **Suppress time-varying emission factor warning messages** option is selected; see Section 4.1.6). Note that this applies to road sources even when the 'default road' source is specified.
- When using the chemistry model option, if a given source has a profile specified for NO_x but not NO_2, the NO_x profile will be used for NO_2, and vice-versa.
- Each source/source-pollutant combination can use either a 3-day diurnal profile, with separate hourly factors for weekdays, Saturdays and Sundays, or a 7-day diurnal profile, with separate hourly factors for each day of the week, but not both. Both 3-day and 7-day diurnal profiles can be included in the same *.fac* file, for use by different sources/source-pollutant combinations.
- Any wind direction sector limits are applied to the wind direction following the pre-processing of the meteorological data, i.e. the wind direction given in the output *.mop* file rather than the input *.met* file.
- When modelling multiple sources including a 2D-grid source, the model applies the time-varying emission factors by disaggregating once at the start of the run, then applying the time-varying factor to the residual grid emissions and to each industrial or road source for each met line. However, care must be taken when defining the emissions in the interface. The method works on the basis that the emission rates entered in the **Source** screen for each industrial or road source are the emission rates used to calculate the gridded emissions entered for the grid source. This ensures the disaggregation at the start of the run is correct. Note that the time-varying factors for the grid source will be applied to the residual emissions only.
- When modelling multiple sources including a 3D-grid source, the model disaggregates on a per met line basis taking into account the time varying emissions for each source. Therefore, care must be taken that the 3D-grid source emissions for each cell and met line represent the total emissions for that met line for that grid cell.

4.1.2 Emission factors from .hfc file

As well as modelling diurnal and monthly changes in emissions, ADMS-Urban can also model emissions that change for every hour of the year. This is done by setting up an extra input file, *.hfc*, with format as described below. To use an *.hfc* file in a modelling run, select **File of time-varying factors** on the **Time-varying emission factors** screen in the ADMS-Urban interface and tick the appropriate box. In the text box, type the full path name of the required *.hfc* file, or use the **Browse** button to find available *.hfc* files, or drag and drop an *.hfc* file from Explorer. The **View** button allows the current *.hfc* file to be viewed; left click to open the file in the default application (as selected via the **File**, **Preferences**, **Viewing options** menu item) or right-click to choose between all available applications.

It is possible to use both a .fac file and an .hfc file in the same run, but there must not be any sources that are included in both files.

.hfc file format

An example annual hourly time-varying profile file is shown in **Figure 4.4**. This file is also supplied in the *<install_path>\Data* directory. The *.hfc* file is a comma separated variable file and is split into two parts. The top part of the file defines a profile and its emission factors. This will usually be 8761 lines long if modelling one year (8760 hours in one year plus a header line). The bottom part lists the sources or source-pollutant combinations and the profile to use for each.

1. **Version**

   '`HfcVersion2`' is the current file version identifier.

   Note that ADMS-Urban 5.1 will also accept .hfc *files in the previous format (which had no version identifier line), but future versions may not.*

2. **Profiles data**

   A header line in the form 'year,day,hour' followed by a list of the profile names separated by commas.

   The profile data in the form:

```
Year, day, hour, EmissionFactor1, EmissionFactor2,…
```

   where *EmissionFactor1* is the emission factor for *profile1* for that met line, etc. This is then repeated for each met line in the meteorological file. After all of the profile data there should be a blank line.

3. **Source-specific data**

   A line for each source or source-pollutant combination in the form:

```
SrcName[|PolName], -999, ProfileName
```

   where

   `SrcName` is the source name.

   `PolName` (optional) is the pollutant name. If included, it should be separated from the source name by the vertical bar ('|') character.

   `-999` indicates that this column is not used by ADMS-Urban.

`ProfileName` is the name of the profile from the first section of the *.hfc* file to apply to this source/source-pollutant combination. This should be repeated for each desired source/source-pollutant combination to be included in the *.hfc* file.

In the example shown in **Figure 4.4**, all pollutants emitted by *Source1* varying hourly using *Profile1*, all pollutants emitted by *Source2* varying hourly using *Profile2*, the NO_X emissions from *Source3* vary hourly using *Profile1*, and all other pollutants emitted by *Source3* vary hourly using *Profile3*.

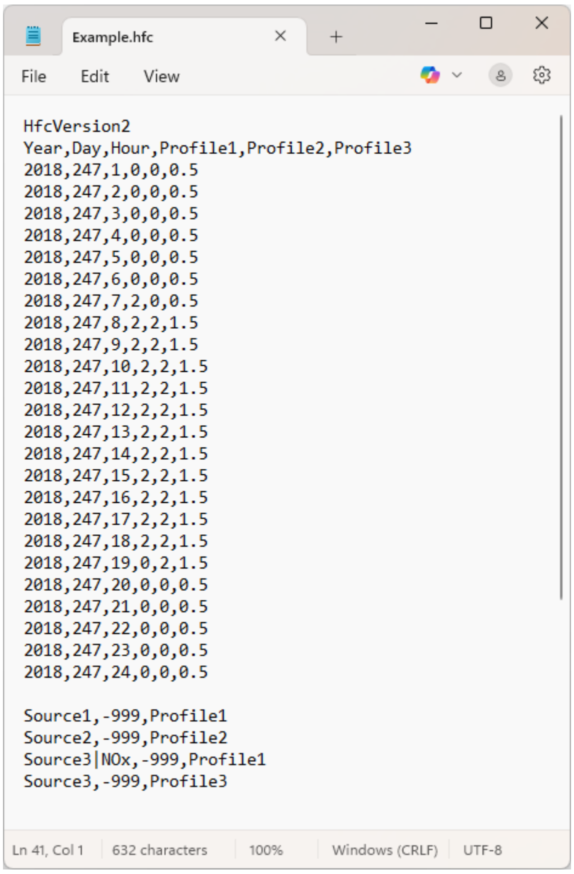


Figure 4.4 − Example annual hourly time-varying emission factors file (*.hfc*).

Important points to note:

- None of the text in the file is case-sensitive.
- Source, pollutant and profile names can contain spaces, but not commas.
- A source must not be listed in both the *.fac* and *.hfc* files.
- The file must be in comma-separated format.
- There should be no blank lines, except for the lines between the profiles data and the sources-specific data.
- Any source that is completely omitted from the *.hfc* file is assumed to have constant emissions across all pollutants emitted by that source, unless a *.fac* file is also used, see Section 4.1.1.
- If only the source name is specified in a given entry, the listed profile is assumed to apply to all pollutants emitted by that source, apart from any

pollutants for which profiles are listed in a separate source-pollutant entry for that source.

- If profiles are provided for some but not all pollutants emitted by a given source (and there is no non-pollutant-specific entry for that source), the model will assume that the missing pollutants have constant emission, and a warning message will be issued (unless the **Suppress time-varying emission factor warning messages** option is selected; see Section 4.1.6).
- When using the chemistry model option, if a given source has a profile specified for NO_x but not NO_2, the NO_x profile will be used for NO_2, and vice-versa.
- If a default road profile is used in the *.fac* file then this default profile is applied to any road sources that are not explicitly listed in either the *.fac* or *.hfc* file. A default road profile cannot be used in the *.hfc* file.
- The grid source cannot use a profile in the *.hfc* file.

4.1.3 Emission factors from screen

To directly enter a set of emission factors, first check the **Time-varying emission factors** option and then click on the **Data source...** button in the lower section of the **Source** screen to display the screen shown in **Figure 4.5**. From this screen, select the **Hourly factors (applies to all selected source types)** option.

On the left hand side of the **Time-varying emission factors** screen, place a check next to the source type(s) for which the emission factors should apply. Any source types which are not checked will have constant emissions.

Factors may be entered into the table for each hour in a 24-hour period, with different factors according to whether it is a weekday, Saturday or Sunday. The release rate given in the **Emissions** screen for each source of the selected types will be multiplied by the appropriate factor for each hour. The sum and average of each set of factors (Weekdays, Saturdays, Sundays) are displayed below the table. It is also possible to copy a set of factors by right-clicking on the header of the column you want to copy and selecting the column you want to copy to.

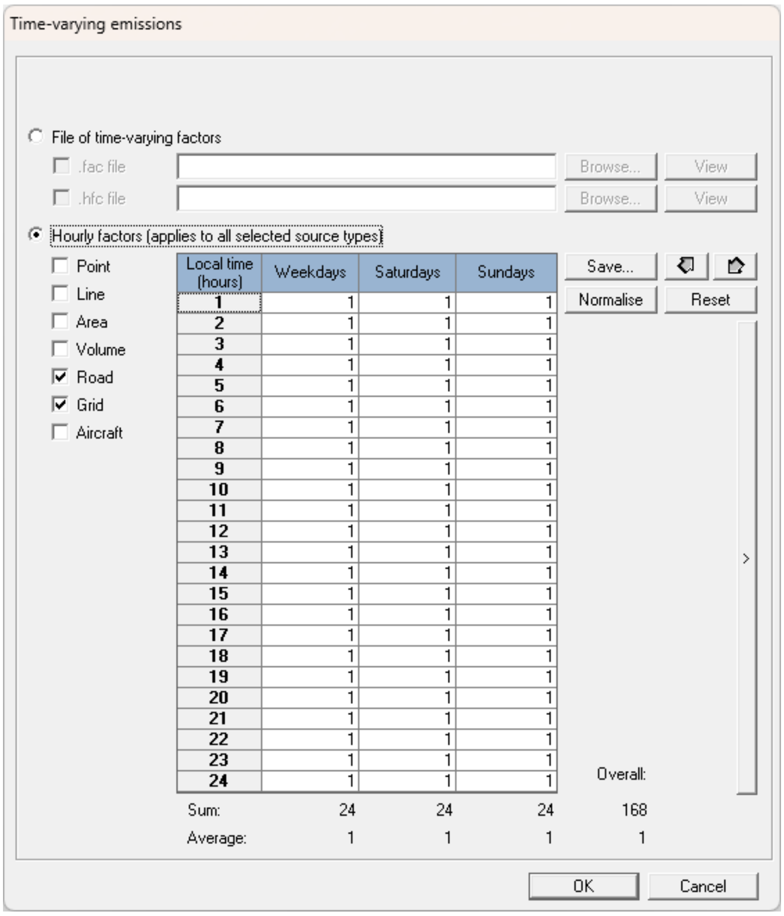


Figure 4.5 – **Time-varying emission factors** screen applied to **Road** and **Grid** sources.

The buttons to the right of the table provide the following additional functionality:

- **Save**: Allows the user to save the current set of factors to a *.fac* file (see Section 4.1.1) as a single 3-day diurnal profile. For the industrial source types to which the emission factors apply, any currently-defined industrial sources in the **Source** screen will be set to use this profile in the *.fac* file. If the factors are to be applied to road sources, the 'default road' profile will also be set to use this profile in the *.fac* file. Similarly, if the factors are to be applied to grid sources, the 'grid' profile will also be set to use this profile in the *.fac* file.

-  **and** : Allow the user to export the current set of factors to, and import a set of factors from, a comma-separated values (*.csv*) file, respectively. An example *.csv* file is shown in **Figure 4.6**. The format of this file consists of an optional comment line (or set of comment lines) that must start with a '!' character, a header line in the form 'Hour, Weekdays, Saturdays, Sundays', and 24 lines of data (one for each hour of the day) giving the hour and the weekday, Saturday and Sunday emission factors for that hour.

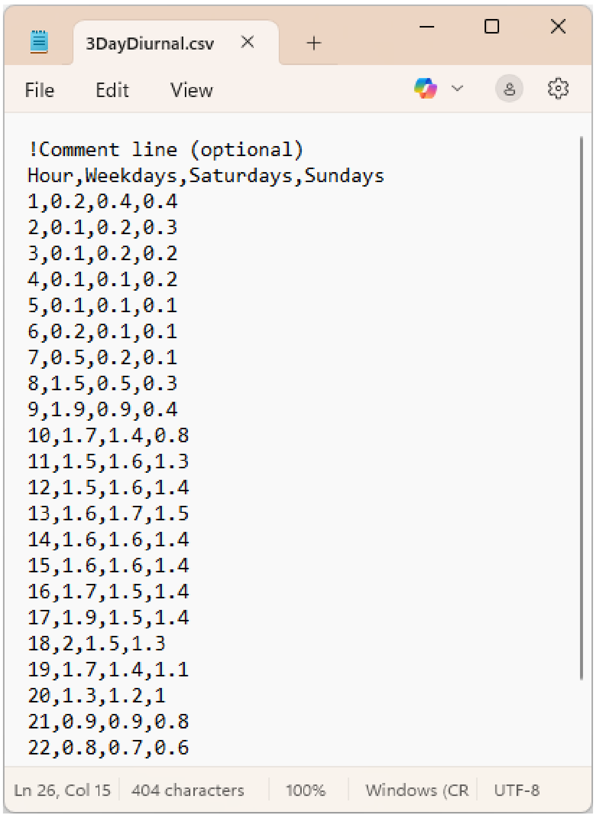


Figure 4.6 – Example 3-day diurnal profile *.csv* file.

- **Normalise**: The emission rates entered in the **Source** screen typically correspond to time-averaged values, in which case it is important that any hourly factors entered in the table are normalised, i.e. they average out to 1 over the entire week. The **Normalise** button can be used to ensure this; it divides each factor by the current overall average.

- **Reset**: Resets all values to 1 in the table.

Clicking the large arrow on the far right hand side of the **Time-varying emission factors** screen reveals a time-series plot of the current emission factors, as shown in **Figure 4.7**. This is a useful tool for checking that the entered emission factors are sensible. Each profile (weekdays, Saturdays, Sundays) is plotted using a different line and/or symbol colour. The appearance of the lines/symbols can be altered by right-clicking on the plot to bring up the **Graph Control** dialogue box.

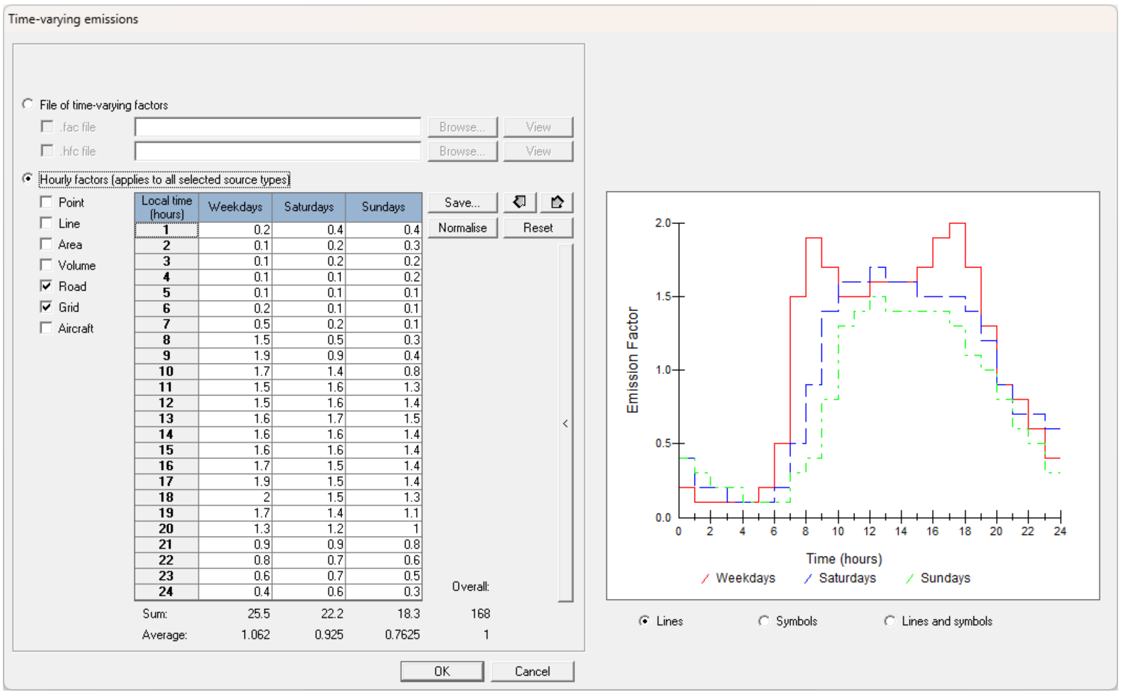


Figure 4.7 – **Time-varying emission factors** screen showing the emission factor time-series plot.

4.1.4 Daylight saving time

Times entered into ADMS-Urban should be entered as hour ending values in local solar time and should be consistent between all input files including the time-varying emission factors. In many countries daylight saving time is used during the summer to shift the times forward by one hour. This changes the relationship between local solar time and clock time during the daylight saving time periods.

For input files where the year, day and hour are input, including the *.hfc* file, the input times should all remain in local solar time. For emission factors entered on screen or using a *.fac* file it may be required to include effects of the one hour shift during daylight saving time.

Daylight saving time is included in the model by creating a new additional input file, or editing the existing additional input file (refer to Section 3.1.8) and enabling the **Daylight saving time** option (keyword ADJUSTFACTORS_DST), as shown in **Figure 4.8**.

If the desired daylight saving option corresponds to British Summer Time (BST), select the **BST** option from the dropdown menu as shown in **Figure 4.8**. Otherwise, select the **Custom** option from the dropdown menu and enter the **START** and **END** times of the daylight saving periods into the table, using the **New** button using the format `YYYY, MM, DD, HH`. (e.g. for 2 am on the 30th March 2025 enter `2025, 03, 30, 02)`. The start and end periods must be entered in sequential order. During daylight saving time the emission factor for one hour ahead of local solar time will be used for the source.

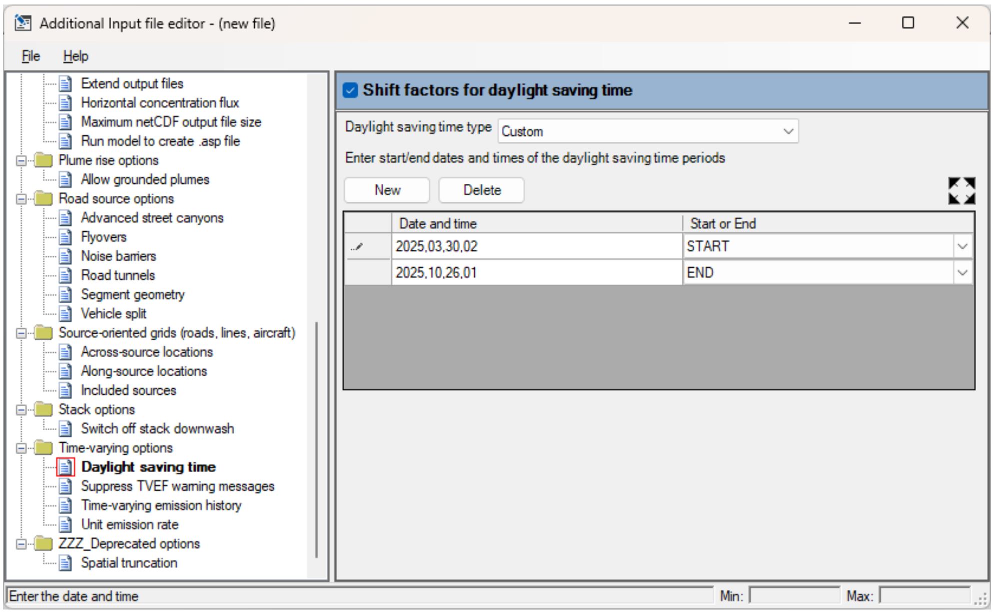


Figure 4.8 – **Daylight saving time** option in the **Additional Input file editor** showing the **Custom** type.

4.1.5 Time-varying emission history

By default, ADMS-Urban models each source assuming that the conditions do not change for that meteorological data line. If the effects of sources are being examined over a large distance then this assumption becomes less valid. For certain source types, for instance road sources with rush hour traffic peaks, the emissions are likely to be the quantity which varies the fastest with time. ADMS-Urban includes a time-varying emissions history option which allows for the effect of the temporal variation of the emission to be taken into account for each meteorological data line.

When the time-varying emission history option is used, for each source, the emission factor used will vary depending on the travel time, and therefore the distance, from the source to each output point. The input emission factors are taken to remain constant over the hour for which they are specified and the emission factor used is calculated based on the one hour average of the input emission factors taking into account the travel time to the output point. All other parameters, including the meteorology, remain constant for that meteorological data line.

For instance, if the current hour is *t* then for an output point:

- at the source: the emission factor is F(t)
- a quarter of an hour from the source: the emission factor is (3F(t)+F(t-1))/4
- half an hour from the source : the emission factor is (F(t)+F(t-1))/2
- one hour from the source: the emission factor is F(t-1)
- two and a half hours from the source: the emission factor is (F(t-2)+F(t-3))/2

If an .hfc *file is being used to specify the emission factors then any emission factors required for times before the first met line are considered to match those from the first met line.*

To enable the time-varying emission history option, create a new additional input file, or edit the existing additional input file (refer to Section 3.1.8) and enable the **Time-varying emission history** option (keyword TIMEVARYQ), **Figure 4.9**. There are no further parameters required for this option.

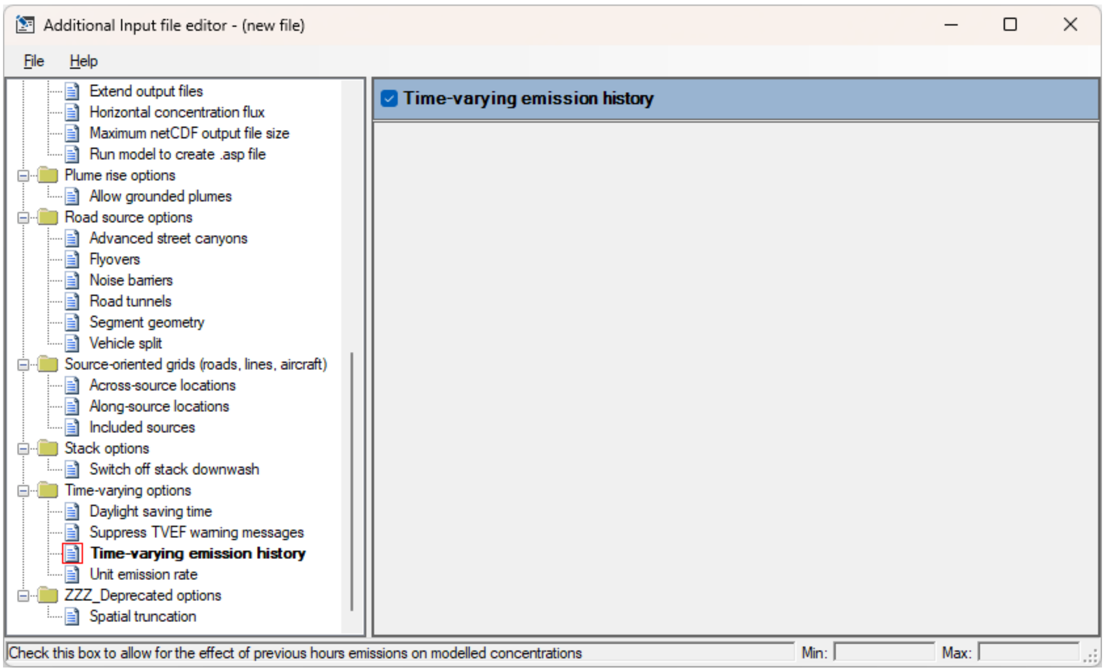


Figure 4.9 – **Time-varying emission history** option in the **Additional input file editor**.

Output averaging times which are not a multiple of one hour cannot be used with the time-varying emissions history option.

4.1.6 Suppress time-varying emission factor warning messages

It is possible to prevent time-varying emission factor warning messages from being written to the log file via an additional input file option. To do this, create a new additional input file, or edit the existing additional input file (refer to Section 3.1.8) and enable the **Suppress TVEF warning messages** option (keyword SUPPRESSTVEFWARNINGS), as shown in **Figure 4.10**. There are no further parameters required for this option.

When using a time-varying emission factors data file for the first time, users are advised to examine the warning messages generated from the file processing in order to check for any possible inconsistencies in the data. For subsequent runs with a corrected file, the option to suppress warnings will result in a more compact log file.

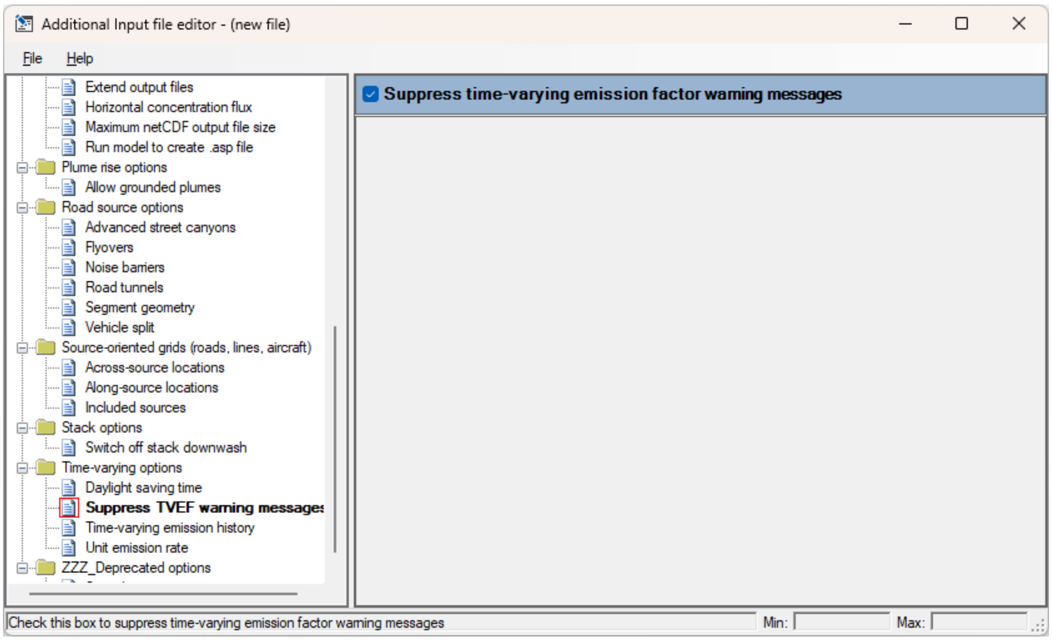


Figure 4.10 – **Suppress TVEF warning messages** option in the **Additional input file editor**.

4.2 Advanced street canyon

The advanced street canyon modelling option in ADMS-Urban modifies the dispersion of pollutants from a road source according to the presence and properties of canyon walls on one or both sides of the road. It differs from the 'basic canyon' modelling (described in Sections 3.2.1 and 9.8.1) in the following ways:

- The model has been formulated to consider a wider range of canyon geometries, including the effects of tall canyons and of canyon asymmetry;
- The concentrations predicted by the model vary with height within the canyon;
- Emissions may be restricted to a subset of the canyon width so that they may be specified only on road lanes and not on pedestrian areas; and
- Concentrations both inside and outside a particular street canyon are affected when running this model option.

A technical summary of the modelling method for advanced street canyons is given in Section 9.8.2.

To use the advanced street canyon module in a run, first create an additional input file or edit the existing additional input file (as described in Section 3.1.8) so that the **Advanced street canyons** option (keyword ADVANCEDSTREETCANYONS) is enabled, as shown in **Figure 4.11**.

Note that when using the advanced street canyon modelling option, the road width entered on the Source screen of the interface should represent the road carriageway width and the road centreline should be located at the centre of the carriageway.

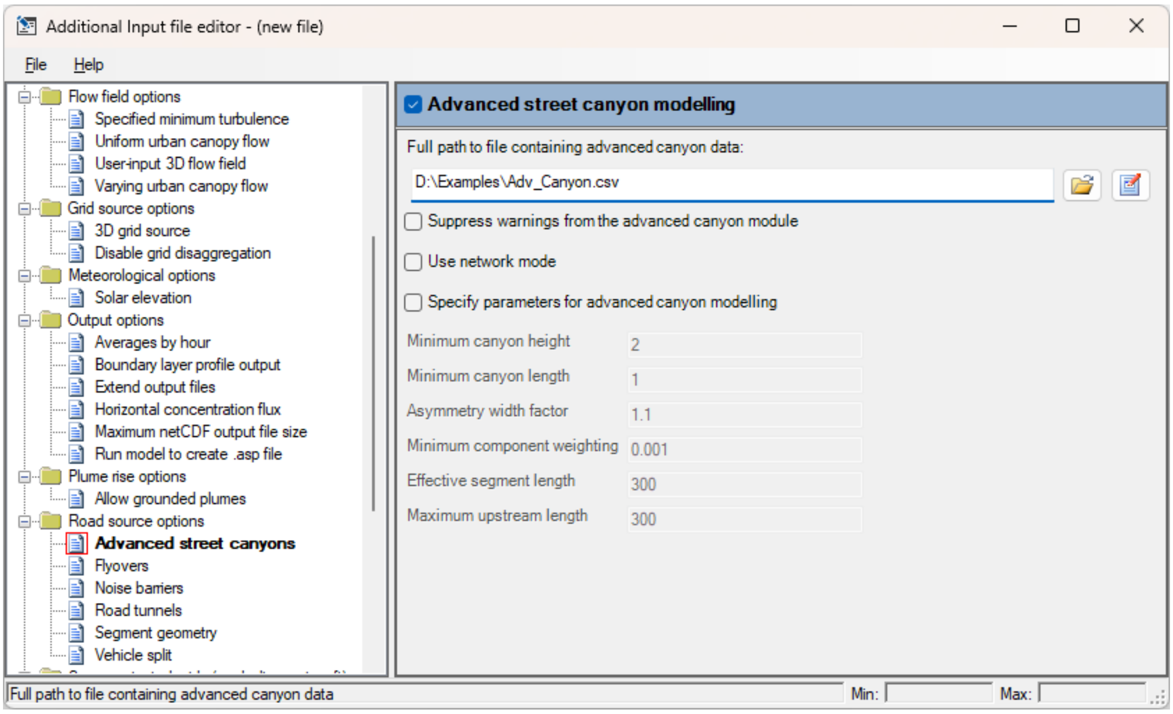


Figure 4.11 The **Advanced street canyons** section enabled in the **Additional Input file editor**

The user must specify:

- **Full path to file containing advanced canyon data**: Use the button to browse to an input file containing advanced canyon data for road sources, please refer to Section 4.2.1 for details of the input file format and contents, and Section 4.2.2 for how to generate a template input file automatically. Use the button to view the file in the default application, as selected by the **File\Preferences...\Viewing** menu option in the **Additional input file editor**;
- **Suppress warnings from the advanced canyon module**: Check this box to prevent source-specific warning messages from being written to the log file;

When using an advanced canyon data file for the first time, users are advised to examine the warning messages generated from the advanced canyon processing in order to check which sources are associated with invalid data. For subsequent runs with a corrected file, the option to suppress warnings will result in a more compact log file.

- **Use network mode**: Check this box to enable network mode; see Section 4.2.3 for more details on the choice of mode;
- **Specify parameters for advanced canyon modelling**: Check this box to enter values for user-defined parameters as listed below;
- **Minimum canyon height**: Enter the minimum canyon height (in metres) for which the advanced canyon module should be used (default value 2 m);
- **Minimum canyon length**: Enter the minimum length of road (in metres) for which the advanced canyon module should be used (default value 1 m);
- **Asymmetry width factor**: Enter the factor which should be used to define the width of a canyon on a side where there is no canyon wall, relative to the width of the other side of the canyon (default value 1.1);
- **Minimum component weighting**: Enter the minimum weighting value where the modelling for the associated component source should be carried out (default value 0.001);

Lower values of the minimum component weighting are associated with greater accuracy but also longer run times.

- **Effective segment length**: Enter the value (in metres) of segment length to use if the standard mode is in use (default value 300 m);
- **Maximum upstream length**: Enter the value (in metres) of maximum upstream length to use if network mode is in use (default value 300 m).

4.2.1 Advanced street canyon input data file format

The structure of the advanced street canyon input data file consists of a version string and a header line followed by one line of data for each road source that will be modelled as an advanced street canyon. In some cases, comment lines beginning with the `//` symbol may appear between the header and data lines. Road sources that are not included in the model file may be included in the advanced street canyon input file, but they will not be modelled. Road sources that are in the model file but not in the advanced street canyon input file will not be modelled with advanced street canyon dispersion. An example data file is shown in **Figure 4.12**, which is also supplied in the *<install_path>\Data* directory.

The Mapper contains a tool to create advanced street canyon data files from a buildings data file and a road centreline data. Please refer to Section 6.4 of the Mapper User Guide for more details.

The variables that must be input by the user to characterise the properties of each street canyon are as follows:

- `Name`: Source name used for matching between advanced canyon and model file sources;
- `X1, Y1, X2, Y2`: x and y coordinates of the first and second (or last and penultimate) vertices of the road centreline (m), in order to indicate orientation of left and right canyon sides and for matching with the model file source data;

Then, for each side of the canyon, where N can be L (left) or R (right):

- `Width_N`: the distance from the road centreline to canyon wall (m), if present;
- `AvgHeight_N`: the average height of buildings in the canyon wall (m). If this is below the minimum canyon height, no canyon will be modelled for this side;
- `MinHeight_N`: the minimum height of buildings in the canyon wall (m);
- `MaxHeight_N`: the maximum height of buildings in the canyon wall (m);
- `BuildLength_N`: the total length of road with adjacent buildings on this side (m).
- `Porosity_N`: the fractional length of road without adjacent building on this side.

For each side of the canyon, exactly one of `BuildLength_N` and `Porosity_N` must be specified. It is not required to choose the same parameter for both sides.

Then:

- `FracCovered`: the fractional plan-view area of the street canyon that is covered by overhanging features, e.g., balconies, walkways, curved noise barriers, etc. This should take a value between zero and one (though it is advisable to model the street canyon as a road tunnel instead for any values approaching one).

If there are no buildings adjacent to a road on one side, values of zero can be entered for all parameters for that side.

AC_v3Comments.csv

File Edit View

```
AdvancedCanyonVersion3
//Comment
//Comment
Name,X1,Y1,X2,Y2,width_L,avgHeight_L,minHeight_L,maxHeight_L,porosity_L,width_R,avgHeight_R,minHeight_R,maxHeight_R,buildLength_R,fracCover
ed
Road001,0,-50,0,50,6,10,10,10,0,6,10,10,10,100,0
```

Ln 9, Col 1 | 237 characters | 100% | Windows (CRLF) | UTF-8

Figure 4.12 Example Advanced street canyon data input file

The full file format is as follows:

- 'AdvancedCanyonVersion3' Version string, must be the first line in the file

---

Note that ADMS-Urban version 5.1 will also accept files with the AdvancedCanyonVersion1 and AdvancedCanyonVersion2 version string, but future versions may not. If using an older version file, the model will assume that FracCovered is zero for all entries in the file.

- Header line, must be the second line in the file, excluding any comment lines. The file's columns must contain the following labels, with comma separation and optional capitalisation and order:
  - `Name`
  - `X1`
  - `Y1`
  - `X2`
  - `Y2`
  - `Width_L`
  - `AvgHeight_L`
  - `MinHeight_L`
  - `MaxHeight_L`
  - `BuildLength_L or Porosity_L`
  - `Width_R`
  - `AvgHeight_R`
  - `MinHeight_R`
  - `MaxHeight_R`
  - `BuildLength_R or Porosity_R`
  - `FracCovered`
- One line of comma-separated data per advanced canyon road source containing the variables as labelled in the header line. Additional information may also be included in optional extra columns, but will be ignored.

Blank lines may be used after the header line to separate groups of road sources if desired.

4.2.2 Create advanced canyon template

To generate a template advanced street canyon input data file based on the road sources currently defined in the **Source** screen, select the **Utilities/Create advanced canyon template** menu option and browse to save the file in the desired location, giving it a sensible name.

This option will automatically populate the data file, with one entry per road source, where:

- The `Name` will be set to match the source name in the **Source** screen.
- `X1, Y1, X2, Y2` will be set to match the first and second vertices of the road source, as listed in the **Road geometry** screen. Note that this sets the orientation of the left and right canyon sides; if the user requires the orientation to be the other way around, they should manually modify these values to be the last and penultimate x and y coordinates of the road instead.
- The left and right canyon half widths (`Width_L`, `Width_R`) will both be set to half the width of the road, as specified in the **Source** screen. If either canyon half widths are greater than this (typically they will be, due to pavements between the edge of the road carriageway and the building facades), the user should increase these values manually.
- The rest of the columns (`AvgHeight_N`, `MinHeight_N`, `MaxHeight_N` `BuildLength_N`, `FracCovered`) will be set to zero and the user should enter the appropriate values manually.

4.2.3 Choice of modes

The advanced street canyon model option can be run with two different modes, standard and network mode. Standard mode assumes that each canyon is part of a continuous road network of canyons with similar properties, whereas network mode analyses the road network to determine the amount of transport of pollutants between adjoining street canyons. Standard mode is appropriate for most scenarios whilst network mode should be used for detailed local analysis. Section 9.8.2 provides more details on the differences between standard and network mode.

When network mode is used it is important that the road geometry is accurate at road junctions. Adjoining road sources must have the same coordinates for their adjoining ends.

Analysis of the transport between street canyons only takes place at the ends of road sources (i.e. not at intermediate vertices), so for junctions to be considered correctly the junction should be an end vertex for each road source in the junction.

4.2.4 Restrictions

The advanced street canyon option cannot be run in combination with the time-varying emissions history option.

The canyon width on each side of the canyon with a canyon wall should be greater than half the road carriageway width.

The advanced street canyon modelling does not affect output deposition values.

4.3 Noise barriers

Ground-level concentrations downwind of noise barriers are lower than those in the absence of a barrier, because the barrier effectively raises the source height. The effect of noise barriers can be included in a model run, with barriers defined on one or both sides of a road, on up to 100 roads. Note that the noise barrier module only affects concentrations at output points ‘outside’ the road, where ‘outside’ means the non-road side of the barrier. The noise barrier module is used by supplying a noise barrier (*.nbr*) file.

To use the noise barrier model option create a new additional input file or edit an existing additional input file (see Section 3.1.8) and enable the **Noise barriers** option, as shown in **Figure 4.13** (keyword NBRFILE). The path to the noise barrier file should then be specified.

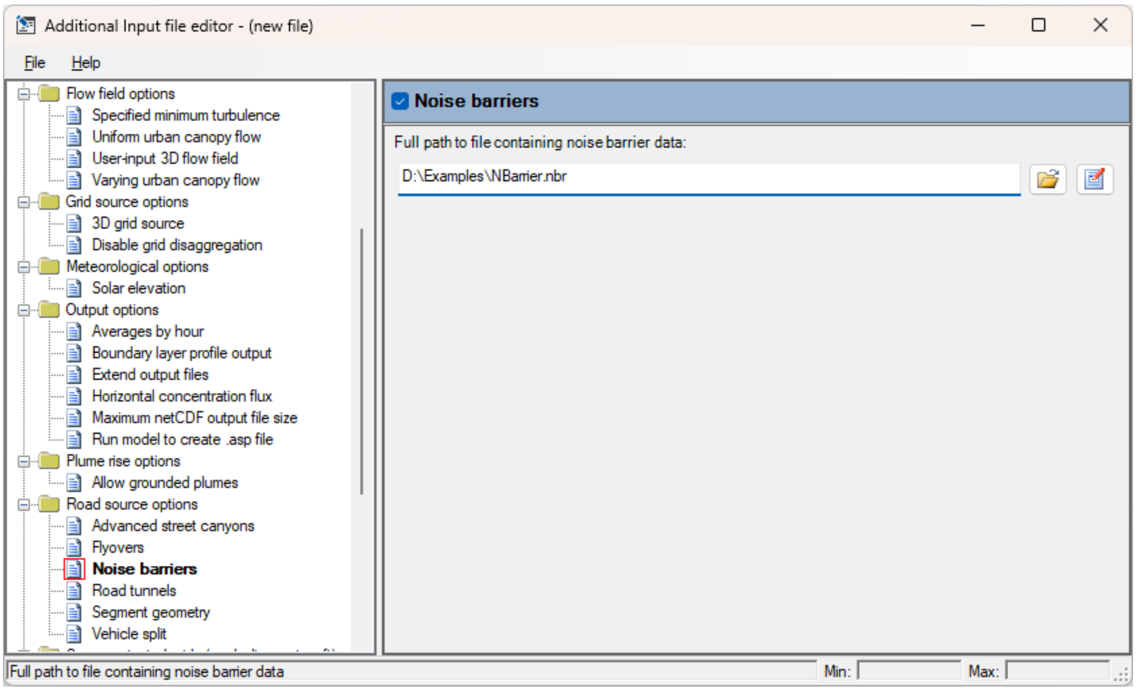


Figure 4.13 – The **Noise barriers** screen of the **Additional input file editor**.

The noise barrier file is a comma-separated text file and contains a line of data for every road that has a noise barrier associated with it. Each line of data contains the following information:

```
sName, xref, yref, HB1, HB2, d1, d2
```

where

- **sName** is the name of the road source. This must be identical to the name used in the interface.
- (**xref**, **yref**) is the location of a reference point outside the road, close to one of the barriers.
- **HB1** is the height of the barrier on the side of the road closest to the reference point.
- **HB2** is the height of the barrier on the other side of the road.

- **d1** is the distance of the barrier with height HB1 from the centre of the road.
- **d2** is the distance of the barrier with height HB2 from the centre of the road.

Figure 4.14 shows an example cross-section of a road, with noise barriers of heights **HB1** and **HB2** on either side of the road, at distances **d1** and **d2** respectively from the road centreline.

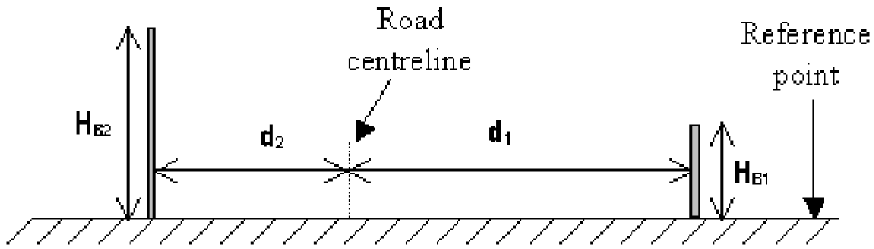


Figure 4.14 – Noise barriers on either side of a road

Figure 4.15 shows an example *.nbr* file, which is also supplied in the *<install_path>\Data* directory.

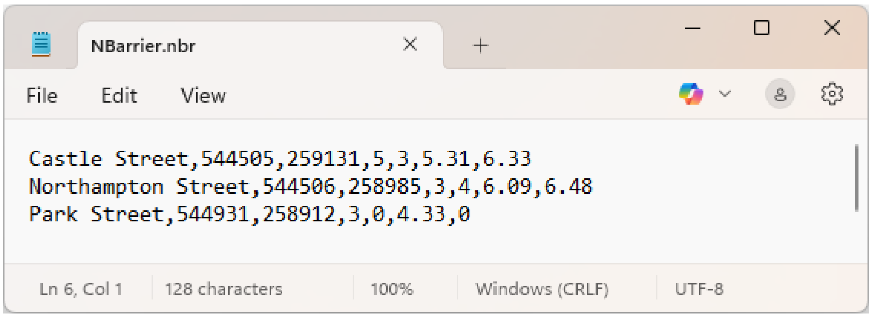


Figure 4.15 – Example *.nbr* file

If barriers are present on both sides of the road, it may be appropriate to model the road as a basic street canyon as well as including the noise barriers, as the noise barrier module only affects concentrations outside the road, and the basic street canyon module only affects concentrations within the road. Alternatively, the advanced street canyon module could be used in place of using noise barriers (see Section 4.2).

When elevated roads are modelled, the noise barrier height corresponds to the height above the road surface and not height above ground level.

4.4 Road tunnels

The road tunnel modelling option in ADMS-Urban modifies the dispersion of pollutants from a road source to take into account dispersion from the tunnel portal(s) and any vents. A technical summary of the modelling method for road tunnels is given in Section 9.10.

Note that this option models the effects of emissions from traffic within road tunnels on concentrations outside the tunnel; it does not model the concentrations which occur within the tunnel.

By default, the emissions from a road tunnel are considered to occur at the end(s) where traffic leaves the tunnel. The initial dispersion may follow the traffic flow along an outflow road due to pollution entrainment in vehicle wakes, with this effect being particularly significant at low wind speeds.

There is an option to also model emissions from a road tunnel via point or area vent sources. Such additional ventilation systems may be implemented for long tunnels in order to improve the air quality within the tunnel, or where there is particular sensitivity to high concentrations near a tunnel portal. A vent source may extract emissions from more than one tunnel, for example from both bores of a twin-bore tunnel, and/or a tunnel may have more than one associated vent source. Vent source emissions are set to a specified proportion of the road tunnel emissions, but all other source properties set in the interface for a vent source will be retained.

Note that any emissions defined for vent sources in the model file will not be used by the model. If there are emissions from a vent source which do not originate from a road tunnel, a duplicate source should be defined with these emissions.

To use the road tunnels model option in a run, first create an additional input file or edit an existing additional input file (as described in Section 3.1.8) so that the **Road tunnels** option (keyword `ROADTUNNELS`) is enabled, as shown in **Figure 4.16**.

Note that when using the road tunnel modelling option, the road width entered on the ***Source*** *screen of the interface should represent the tunnel bore width and the road centreline should be located at the centre of the tunnel bore.*

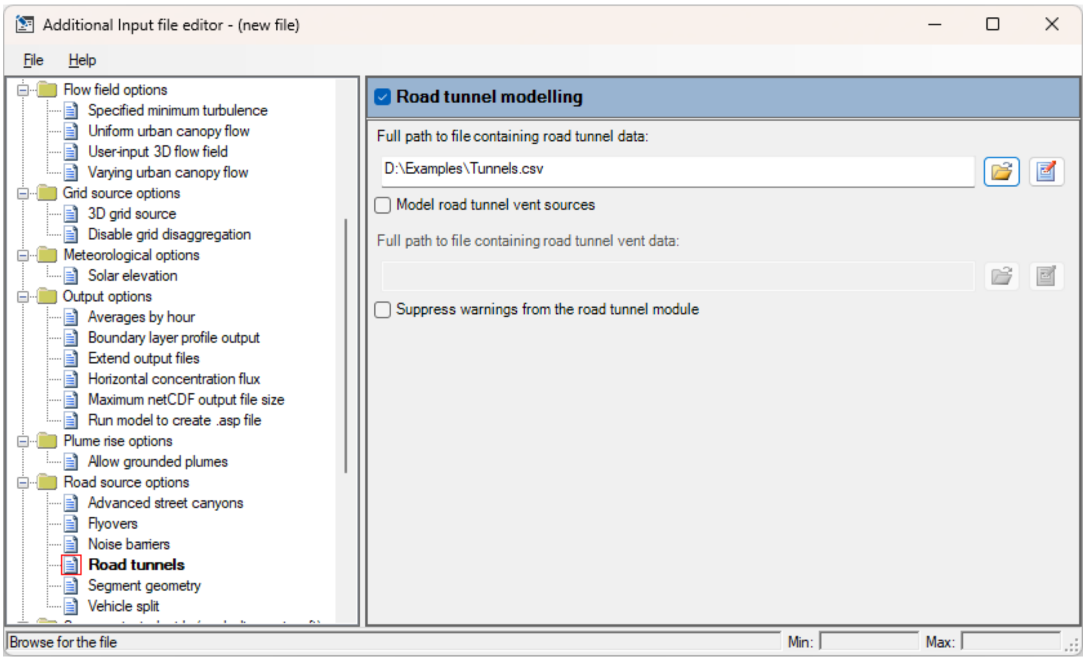


Figure 4.16 - The **Road tunnels** section enabled in the **Additional Input file editor**.

The user must specify:

- **Full path to file containing road tunnel data**: Use the button to browse to an input file containing road tunnel data for road sources. Use the button to view the file in the default application, as selected by the **File\Preferences...\Viewing** menu option in the **Additional input file editor.** Refer to Section 4.4.1 for details of the input file format and contents;
- **Model road tunnel vent sources**: Whether to include modelling of road tunnel vent sources;
- **Full path to file containing road tunnel vent data**: If road tunnel vent modelling is required, use the button to browse to an input file containing road tunnel vent data. Use the button to view the file in the default application, as selected by the **File\Preferences...\Viewing** menu option in the **Additional input file editor.** Refer to Section 4.4.2 for details of the input file format and contents;
- **Suppress warnings from the road tunnel module**: Check this box to prevent source-specific warning messages from being written to the log file;

When using a road tunnel data file for the first time, users are advised to examine the warning messages from the road tunnel modelling in order to check which sources are associated with invalid data. For subsequent runs with a corrected file, the option to suppress warnings will result in a more compact log file.

4.4.1 Road tunnel input data file format

The structure of the road tunnel input data file consists of a version string and a header line followed by one line of data for each road source that will be modelled as a tunnel. Road sources which are not included in the model file may be included in the road tunnel input file, but they will not be modelled. Road sources which are in the model file but not in the road tunnel input file will not be modelled as a tunnel. An example data file is shown in **Figure 4.17**, which is also supplied in the *<install_path>\Data* directory.

The variables that must be input by the user to characterise the properties of each road tunnel are as follows:

- `Name`: Source name used for matching between road tunnel and model file sources;
- `X1`, `Y1`, `X2`, `Y2`: x and y coordinates of the first and last (or last and first) vertices of the road centreline (m), in order to indicate orientation of traffic flow and for matching with the model file source data;
- `NumTrafficDir`: number of traffic flow directions within the tunnel bore (1 or 2), if 1 the traffic flow is taken to be from X1, Y1 to X2, Y2 and emissions will only be modelled from the portal at X2, Y2;

Further, for each tunnel portal, where the portal number N can be 1 (corresponding to X1, Y1) or 2 (corresponding to X2, Y2):

- `BoreDepthN`: the vertical extent of the tunnel bore (m);
- `PortalBaseElevN`: the elevation of the road surface at the tunnel portal, relative to local average ground level (m);
- `OutflowRoadN`: the name of the road source used by traffic leaving the tunnel portal, as given in the model file;

  The outflow road will be used to define the direction of initial dispersion from the tunnel portal only if it has an end vertex within 1 m of the relevant tunnel portal vertex.

- `OutflowWidthN`: the ground-level width of the outflow road (may be larger than the carriageway width if the portal base elevation is non-zero) (m); and
- `OutflowWallN`: whether an anti-recirculation wall is present at this portal (yes or no).

The bore depth, portal base elevation and outflow width are illustrated in **Figure 4.18** for three example configurations. If the tunnel traffic flow is in one-direction, values entered for parameters relating to the first portal will not be used by the model.

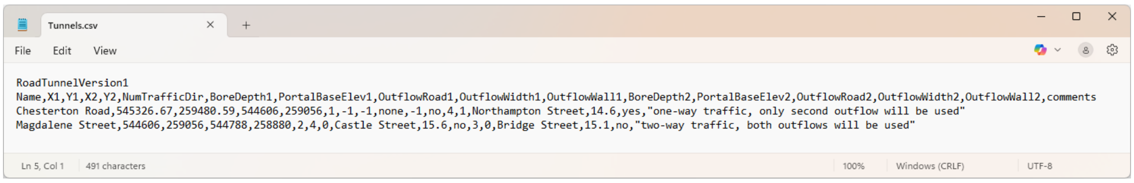


Figure 4.17 - Example road tunnel data input file

The full file format is as follows:

- `RoadTunnelVersion1` Version string, must be the first line in the file
- Header line, must be the second line in the file. The first 16 columns must

contain the following labels, with comma separation and optional capitalisation:

* `Name`
* `X1`
* `Y1`
* `X2`
* `Y2`
* `NumTrafficDir`
* `BoreDepth1`
* `PortalBaseElev1`
* `OutflowRoad1`
* `OutflowWidth1`
* `OutflowWall1`
* `BoreDepth2`
* `PortalBaseElev2`
* `OutflowRoad2`
* `OutflowWidth2`
* `OutflowWall2`

Additional labels may be added at the end of the line.

- One line of comma-separated data per road source containing the variables as labelled in the header line. Additional information may also be included in optional extra columns at the end of the file.

Blank lines may be used after the header line to separate groups of road sources if desired.

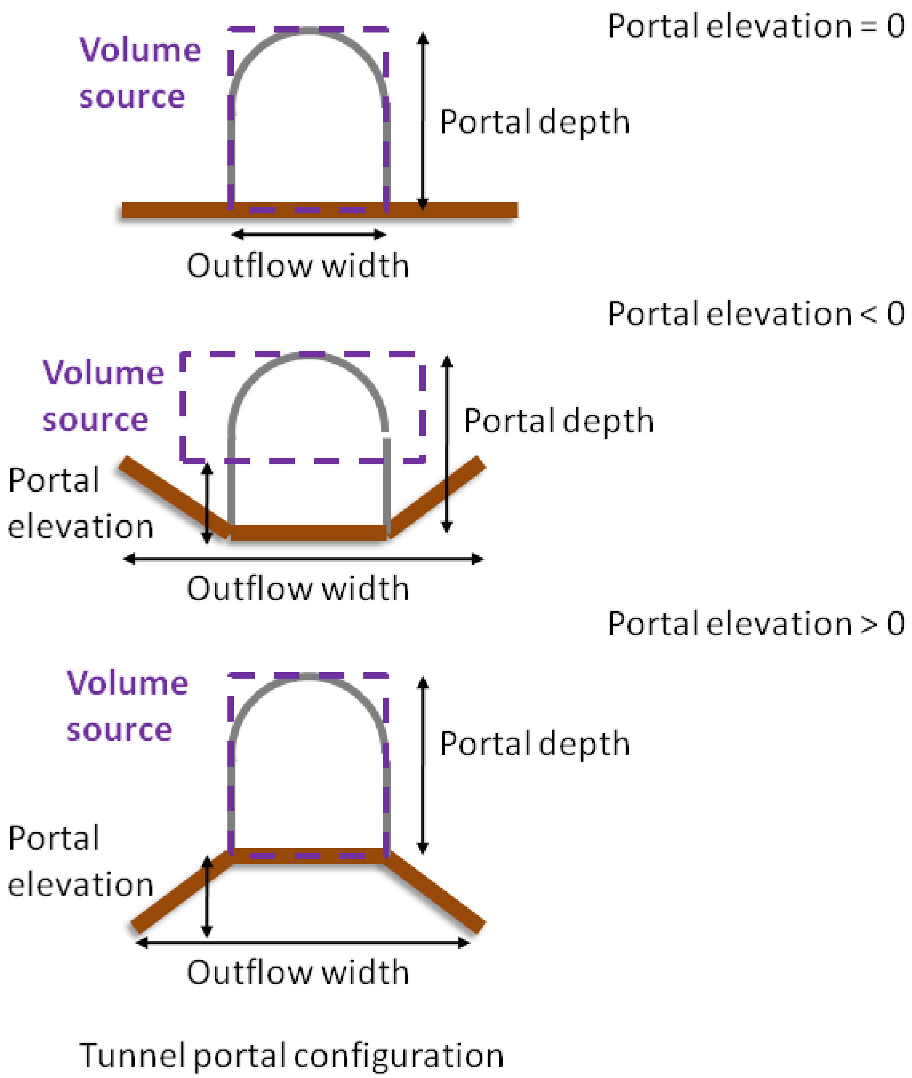


Figure 4.18 - Illustration of tunnel portal and outflow geometry parameters for three example configurations.

4.4.2 Road tunnel vent input data file format

The structure of the road tunnel vent input data file consists of a version string and a header line followed by one line of data for each combination of road tunnel and vent source. Tunnel and vent sources which are not included in the model file may be included in the road tunnel vent input file, but they will not be modelled. An example data file is shown in **Figure 4.19**, which is also supplied in the *<install_path>\Data* directory.

The variables that must be input by the user to characterise the properties of each road tunnel are as follows:

- `TunnelName` Source name for the road tunnel source as used in the model file;
- `VentName` Source name for the road tunnel vent source as used in the model file; and

Note that only point or area sources can be used as tunnel vent sources

- `VentFraction` base fraction of road tunnel emissions which are extracted by the vent, must be between 0 and 1.

The vent fraction for each vent source can be altered by time-varying emission factors applied to the vent source (using any of the standard methods for point/area sources), for example if active ventilation systems are switched on only at peak times.

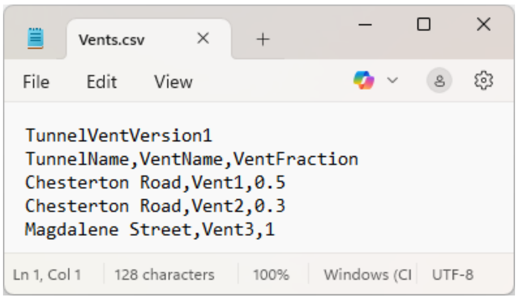


Figure 4.19 - Example road tunnel vent data input file

The full file format is as follows:

- `TunnelVentVersion1` Version string, must be the first line in the file
- Header line, must be the second line in the file. The first 3 columns must contain the following labels, with comma separation and optional capitalisation:

  * `TunnelName`
  * `VentName`
  * `VentFraction`

  Additional labels may be added at the end of the line.

- One line of comma-separated data per tunnel/vent source combination containing the variables as labelled in the header line. Additional information may also be included in optional extra columns at the end of the file.

Blank lines may be used after the header line to separate groups of vent sources if desired.

4.4.3 Restrictions

- The road tunnel option cannot be run in combination with the time-varying emissions history option.
- The outflow width must be greater than or equal to the road carriageway width.
- Plume depletion due to deposition or chemical transformations is not modelled within the tunnel. If either of these effects are thought to be significant, the user is advised to modify the road tunnel source emissions to reflect the conditions at the tunnel portal.
- A tunnel vent source must not be selected for single source output.

- Tunnel vent sources will only contribute concentrations to groups where the tunnel source is also included in the group.
- The total fraction of emissions extracted by vent sources from any tunnel cannot be greater than 1. If time-varying factors lead to a total fraction greater than 1, the fraction of emissions extracted by each vent will be scaled down proportionally to give a total fraction of 1.
- It is not possible to define pollutant-specific time-varying emission factors for vent sources in the *.fac* or *.hfc* file since the fraction of emissions extracted from the tunnel should be pollutant-independent.
- No explicit consideration is given to recirculation of pollutants between bores with a twin-bore tunnel. The presence of an anti-recirculation wall affects the geometry of the volume sources used to represent dispersion from a tunnel portal. The user may wish to alter the road tunnel source emissions to reflect recirculation effects if required.
- Tunnel portal emissions are assumed to occur at ambient conditions, without consideration of any temperature difference between the tunnel and atmospheric conditions.
- If using the spatial splitting option (Section 4.24) or the source exclusion by distance from receptors option (Section 4.25.1), vent sources and their associated tunnels will always be modelled explicitly (i.e. they will never be truncated/excluded) in case a tunnel source falls inside the truncation/inclusion region but its vent sources fall outside it, or vice-versa. This is also the case if using the source exclusion by emission rate option (Section 4.25.2).

4.5 Flyovers

The standard approach to modelling an elevated line/road source in Gaussian type dispersion models such as ADMS-Urban is to allow material to disperse freely through the source itself (**Figure 4.20 (a)**). In reality, the downward dispersion of vehicular emissions on an elevated road will only occur once the material has been advected past the downwind edge of the road. The flyovers model option accounts for this by limiting the downward spread of the plume during its traversal over the road surface (**Figure 4.20 (b)**). The vertical concentration profile is calculated using two joined half-Gaussian profiles with different spreads. A more detailed technical summary is provided in Section 9.11.

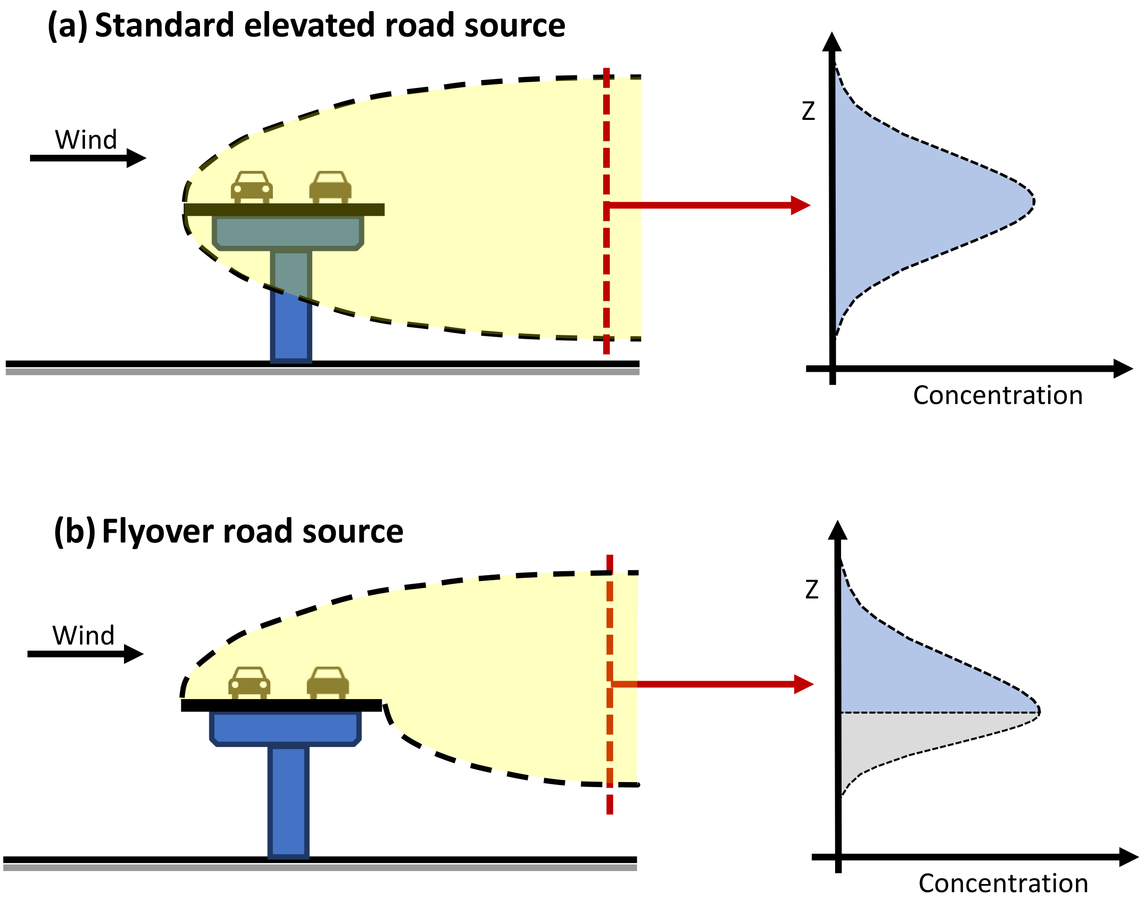


Figure 4.20 – Schematic of modelling approach for (a) a standard elevated road source and (b) a flyover road source.

To use the flyovers option, first create a *.uai* file or edit the existing *.uai* file (as described in Section 3.1.8) so that the **Flyovers** option (keyword FLYOVERS) is enabled, as shown in **Figure 4.21**.

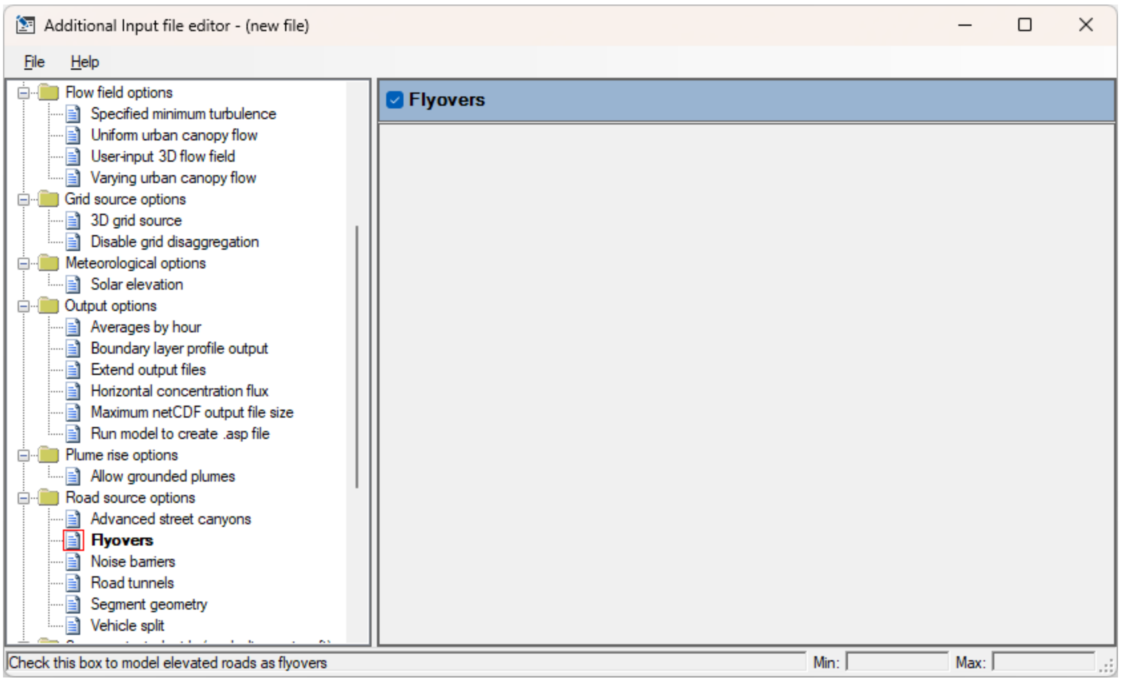


Figure 4.21 – The **Flyovers** section enabled in the **Additional Input file editor**

The model will then use the flyovers approach for any road source that satisfies all the following criteria:

- has a positive **Elevation of road (m)** value in the **Source** screen of the interface
- has a **canyon height (m)** value of zero in the **Source** screen of the interface
- is not already being modelled as either an advanced canyon road source or a road tunnel source

Note that a flyover road source can still be used in combination with the **Noise barriers** option. In this case, dispersion from the raised source at height H_s that affects concentrations on the non-road side of the barrier (refer to Section 9.9) uses the standard elevated road approach.

4.6 Road geometry output

Road and line sources consisting of multiple segments are modelled using trapezoidal, rather than rectangular, geometry. This avoids overlap (and gap) regions at segment joins, as illustrated in **Figure 4.22**. The shared trapezium sides at a segment join lie along the bisector of the angle between the two source segment centrelines. A similar process is also applied between adjoining pairs of road or line sources. If more than two road or line sources meet at a point, the standard rectangular ends are used for each of these sources. For advanced street canyon road sources, the trapezia extend out to the canyon walls either side of the road.

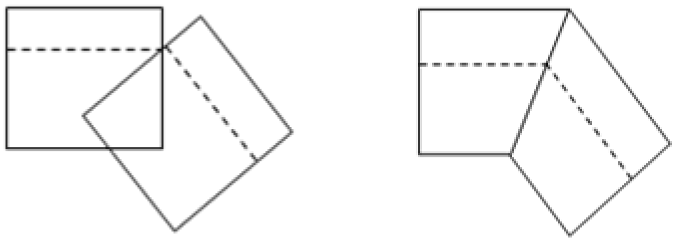


Figure 4.22 – Segment geometry using rectangles (left) and trapezia (right; as used in ADMS-Urban). The dashed line shows the source centreline.

The coordinates of these trapezoidal segments can be output for road and advanced street canyon sources via an additional input file option. Create an additional input file, or edit the existing additional input file (refer to Section 3.1.8), so that the **Segment geometry** option (keyword SEGMENTGEOMETRY) is enabled and the **Output road geometry (.rds) file** checkbox is ticked, as shown in **Figure 4.23**.

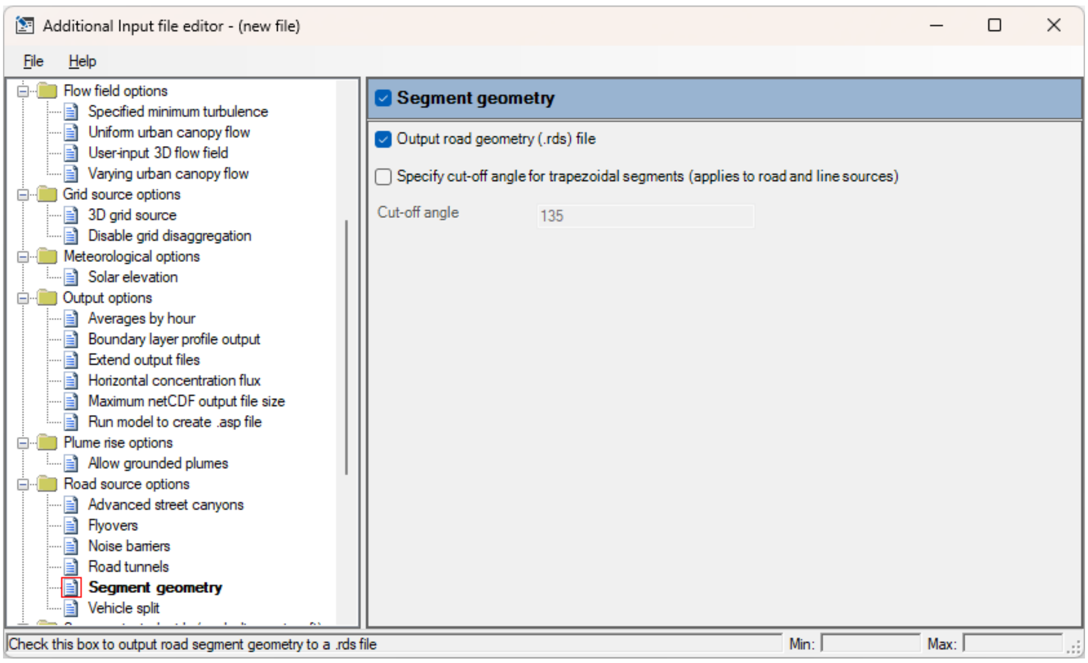


Figure 4.23 – The **Segment geometry** section enabled in the **Additional Input file editor**

Section 6.1.23 describes the format of the *.rds* file. The contents of this file can be easily viewed in the Mapper using the **Add layer** facility; please refer to the Mapper User Guide for full details, which can be accessed from the **Help** menu of the Mapper.

The **Segment geometry** additional input file option also allows the user to specify the maximum external angle between two line or road segments before the model will revert from trapezoidal to rectangular geometry. The default is 135°; to change this, check the **Specify cut-off angle for trapezoidal segments** box and modify the value in the **Cut-off angle** field as necessary.

4.7 3D grid source

The standard ADMS-Urban grid source is a useful way of modelling aggregated pollutant emissions from relatively uniform sources, where all of the emissions have similar time-varying profiles and vertical distribution. If emissions from more heterogeneous sources need to be aggregated, it may be better to use a 3D grid source, where hourly emissions can be specified per grid cell per pollutant on a 3D grid. One or more netCDF files following 'Climate and Forcast' (CF) metadata conventions are used to define the 3D grid source geometry and hourly emissions for ADMS-Urban. This approach is similar to that used in regional models such as CMAQ and WRF-Chem. The full definition of the file format is given in Section 4.7.1. A conversion utility has been created which will convert CMAQ or WRF-Chem emissions files into ADMS-Urban 3D grid emission files, as described in the ADMS-Urban Regional Model Link User Guide.

The method used to aggregate the emissions onto the 3D grid affects how the explicit source emissions should be disaggregated by the ADMS-Urban model. Emissions may be included in the 3D grid at the physical source height or there may be additional considerations:

- Road emissions may all be included in the surface layer, even if they are elevated above the height of the surface layer; and
- large point source emissions may be spread vertically using an estimate of initial plume rise.

There are ADMS-Urban model options to allow disaggregation corresponding to both of these approaches.

To use a 3D grid source emissions file in a run, first create an additional input file or edit the existing additional input file (as described in Section 3.1.8) so that the **3D grid source** option (keyword 3DGRIDSOURCE) is enabled, as shown in **Figure 4.24**.

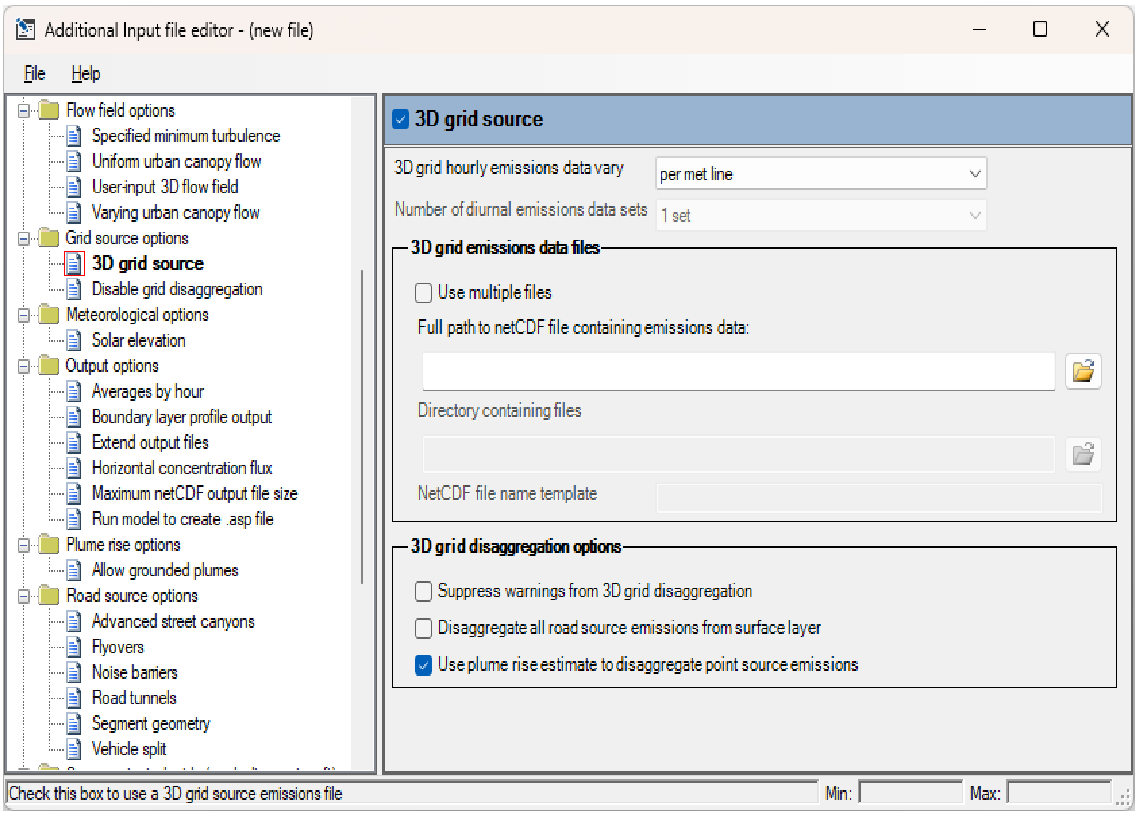


Figure 4.24 – The **3D grid source** option enabled in the **Additional Input file editor**

The user must specify:

- **3D grid hourly emissions data vary**: Use the drop-down list to select whether the 3D grid emissions data vary **per met line** or **diurnally**. If **per met line** is chosen, the data file(s) should contain emissions data for every line of meteorological data being run. If **diurnally** is chosen, the user should also select the **number of diurnal emissions data sets** from the second drop-down list; each data file should then contain exactly 24 times this number of hours of emissions data. When using diurnal data, the exact dates in the netCDF files do not matter. For example, if using 3 sets of diurnal data, a netCDF file would be valid as long as it contained a full 24 hours of data from any weekday, Saturday and Sunday.

If **Use multiple files** is unticked:

- **Full path to netCDF file containing emissions data**: Use the button to browse to the single netCDF file containing the 3D grid emissions data.

If **Use multiple files** is ticked:

- **Directory containing files**: Use the button to browse to the date-independent parent directory containing the 3D grid emissions data files.
- **NetCDF file name template**: Enter the date-dependent part of the file path, using 'tags' to replace the actual dates/times in the file path. Tags consist of a percent sign (%) and a single letter to indicate particular date or time components, for example %Y is used to represent a four-digit year value. **Table 4.2** provides the full list of available tags.

Tag	Description	Example values	
		1 am 1st January 2014	2 am 6th January 2010
%Y	Four-digit year	2014	2010
%M	Two-digit month	01	03
%D	Two-digit day of month	01	06
%J	Three-digit Julian day	001	065
%h	Two-digit hour	01	14
%m	Two-digit minute	00	00
%s	Two-digit second	00	00

Table 4.2 – Tags used to indicate date and time information in the file name template

Note that the tag character is case-sensitive to distinguish between %M for month and %m for minute. The values for minute and second tags are always set to zero.

These tags are replaced by their actual values at run time. For example, if the **Directory containing files** is set to `D:\MyData\` and the **NetCDF file name template** set to `3DGrid_%Y%M%D_%h0000.nc`, then when the met line corresponding to 4pm on 22nd February 2019 is run, the model will look for a file with the path `D:\MyData\3DGrid_20190222_160000.nc`.

If the hour tag is used, each file should contain no more than 24 hours of data, and the hour specified in the file name should correspond to the first hour in the file.

If the emissions data vary diurnally, only the month tag can be used in the file name template.

- **Suppress warnings from 3D grid disaggregation** Select this option to prevent disaggregation warning messages being issued for every 3D grid cell, pollutant and time-step, which may generate a large log file if there are substantial inconsistencies between the explicit and gridded emissions.
- **Disaggregate all road source emissions from surface layer** Select this option to disaggregate all explicit road source emissions from the surface layer, if this approach was used to aggregate the emissions.
- **Use plume rise estimate to disaggregate point source emissions** Select this option to disaggregate explicit point source emissions from the vertical layers of the 3D grid using an estimate of initial plume rise, if this corresponds to the aggregation approach. Section 9.7 provides more technical details about this option.

4.7.1 3D grid emissions input data file format

The structure of an input 3D grid emissions file follows CF metadata conventions. It contains a variable for each emitted pollutant species, with emission rates for each cell of the 3D grid for each hour of the model period. In addition, there are coordinate variables which define the horizontal centre of each cell and coordinate system definition attributes which allow the data to be viewed as geo-referenced maps in standard netCDF viewing utilities such as Panoply (https://www.giss.nasa.gov/tools/panoply/).

Note that CF-compliance requires that all variable, dimension and attribute names should start with a letter and include only letters, numbers and underscores. Note that the variable and attribute names are case-sensitive.

The dimensions included in an ADMS-Urban 3D grid file are described in **Table 4.3**, the variables in **Table 4.4** and the global and variable attributes in **Table 4.5**. An example 3D grid emissions file called *3DGrid.nc* is supplied with ADMS-Urban in the *<install_path>\Data* directory.

Name	Description
x	Number of grid cells in x horizontal coordinate direction
y	Number of grid cells in y horizontal coordinate direction
z	Number of vertical layers of grid cells
time	Number of time-steps in file (should be defined as the 'unlimited' dimension)

Table 4.3 – Dimensions in a 3D gridded emissions *.nc* file.

Name	Dimensions	Description
x	x	Cell centre x coordinates in projected coordinates
y	y	Cell centre y coordinates in projected coordinates
z	z	Cell top z coordinate in m above ground level
time	time	Hour-ending time in days since midnight on 1st January 1900
ADMS_grid_mapping	(none)	Dummy variable containing coordinate projection information attributes
<emitted pollutant species>	x, y, z, time	One variable for each emitted pollutant species, emissions in $g/m^2/s$ for each grid cell and each hour

Table 4.4 – Variables in a *.nc* file. Dimensions are listed with the fastest varying first.

Associated with	Name	Example value	Comment
Global	Conventions	CF-1.6	May be used by viewing utilities
Global	Title	ADMS-Urban 3D Emissions File	
Global	File_version	ADMS_3DEF_v1.0	Checked by model
Global	History		May be useful for QA
ADMS_grid_mapping	Grid_mapping_ name	lambert_conformal_conic	Projection type[1]
ADMS_grid_mapping	Earth_radius	6.370e+006	
ADMS_grid_mapping	(projection parameters)		(depend on projection used)
x, y	standard_name	Projection_x_coordinate	Allows viewing utilities to identify coordinates
x, y	axis	X	
x, y	units	m	
x, y	grid_mapping	ADMS_grid_mapping	
x, y	resolution_m	1000.0	Allows simple extraction of grid geometry
z	long_name	height of top of cell above local ground level	
z	units	m	
z	positive	up	
time	long_name	local time at end of emissions period	
time	units	days since 1900-01-01 00:00:00	
<emitted pollutant species>, eg. PM2_5	long_name	Gridded emissions of PM2.5	
<emitted pollutant species>	ADMS_species	PM2.5	Species name as used in the ADMS-Urban pollutant palette[2]
<emitted pollutant species>	units	g m^-2 s^-1	Emission units of $g/m^2/s$
<emitted pollutant species>	coordinates	x y z	
<emitted pollutant species>	grid_mapping	ADMS_grid_mapping	
<emitted pollutant species>	_FillValue	-999.0	

Table 4.5 – Global and variable attributes in an *.nc* file. [1]Projection types and required parameters for CF convention compliance are given in Appendix F of the CF Metadata Conventions document (Eaton et al., 2011). [2]Pollutant emissions will only be used in ADMS-Urban if the ADMS_species attribute is present and its value matches the name of a species defined in the pollutant palette.

4.7.2 Restrictions

The following restrictions apply to a 3D grid source:

- Only one grid source can be included in a run, so if a 3D grid source file is in use, no 2D grid source can be defined in the model file. If only gridded sources are required, this will lead to a model file containing no source data.
- Aircraft sources cannot be included in a run with a 3D grid source file.
- The use of hourly emission rates from the 3D grid source file means that no time-varying factors are required for this source type. The time-varying emissions history option cannot be used with 3D grid sources.
- The trajectory model chemistry option cannot be used with 3D grid sources.
- When modelling chemistry, NO_X and NO_2 emissions must be included in the 3D grid source file.

4.8 Disable grid disaggregation

By default, in ADMS-Urban the explicitly defined source emissions are disaggregated from the grid source emission rates. In some cases, it may be desirable to disable disaggregation for some, or all, explicit sources.

To enable this option, create a new additional input file or edit an existing additional input file (refer to Section 3.1.8) so that the **Disable grid disaggregation** option (keyword DISABLEDISAGGREGATION) is enabled as shown in **Figure 4.25**.

From the dropdown menu select to disable grid disaggregation for either **All explicit sources** or **Specified sources.** If **Specified sources** is selected enter the names of the sources for which grid disaggregation should be disabled into the table.

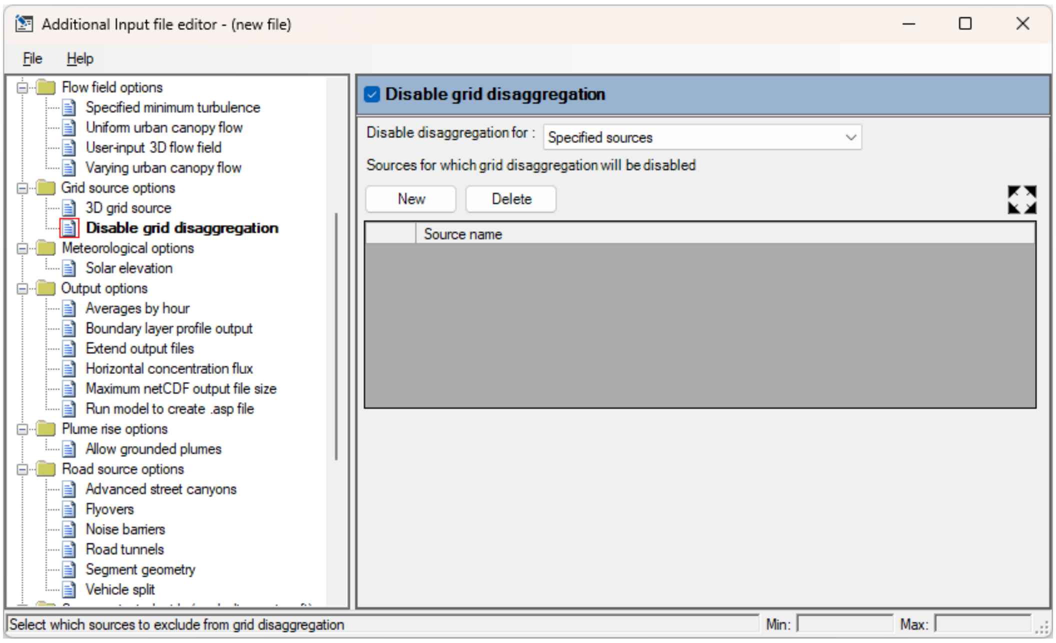


Figure 4.25 – The **Disable grid disaggregation** option in the **Additional Input file editor**.

4.9 Switch off stack downwash

For point sources, ADMS-Urban incorporates by default the reduction in the mean height of the plume just downwind of the source caused by the flow and pressure fields around the stack; refer to Section 9.4.1 for more details. In some cases this stack-induced downwash may not be appropriate; it is therefore possible to turn off the stack-induced downwash for selected sources.

To turn off the stack-induced downwash for selected sources, create a new additional input file or edit an existing additional input file (refer to Section 3.1.8) so that the **Stack downwash** option (keyword NOSTACKDOWNWASH) is enabled, as shown in **Figure 4.26**. Either check the box to switch off stack downwash for **All point sources** or enter the names of the sources for which stack downwash should not be modelled into the table.

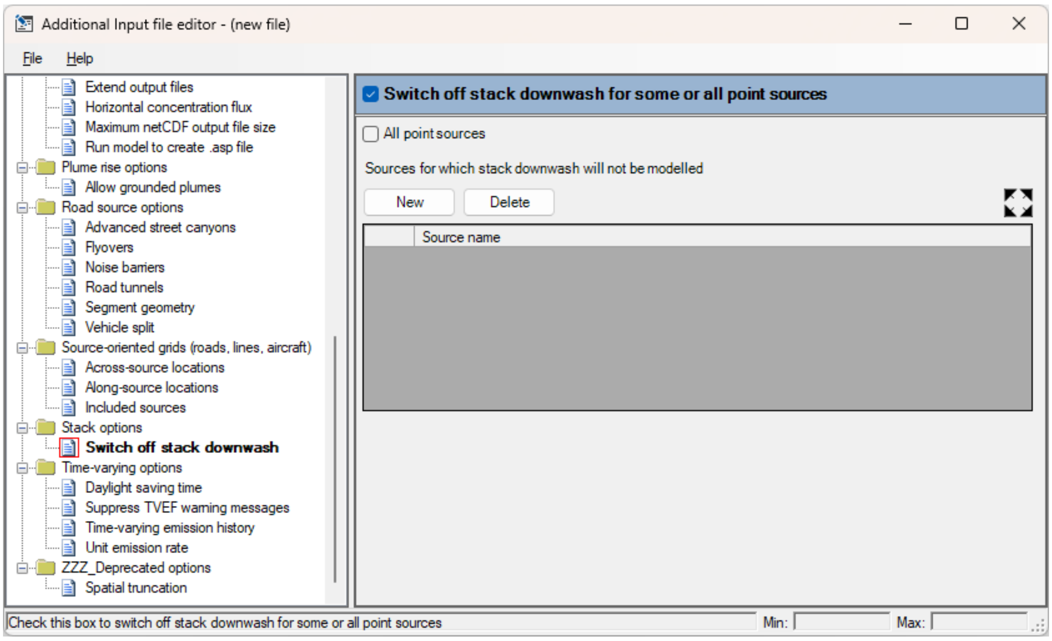


Figure 4.26 – The **Stack downwash** option, which is used to disable stack downwash for certain sources, in the **Additional Input file editor**.

4.10 Allow grounded plumes

By default, in ADMS-Urban if a plume impacts the ground and is too dense, then the model will either stop processing that meteorological data line (in short term) or stop with an error (in long term). In some cases, it may be desirable to continue modelling in such circumstances. This can be accomplished by using **Allow grounded plumes** option.

To enable this an *.uai* file should be created or the existing *.uai* file edited, see Section 3.1.8, so the **Allow grounded plumes** section (keyword ALLOWGROUNDEDPLUMES) is enabled, **Figure 4.27**. No further parameters are required for this section.

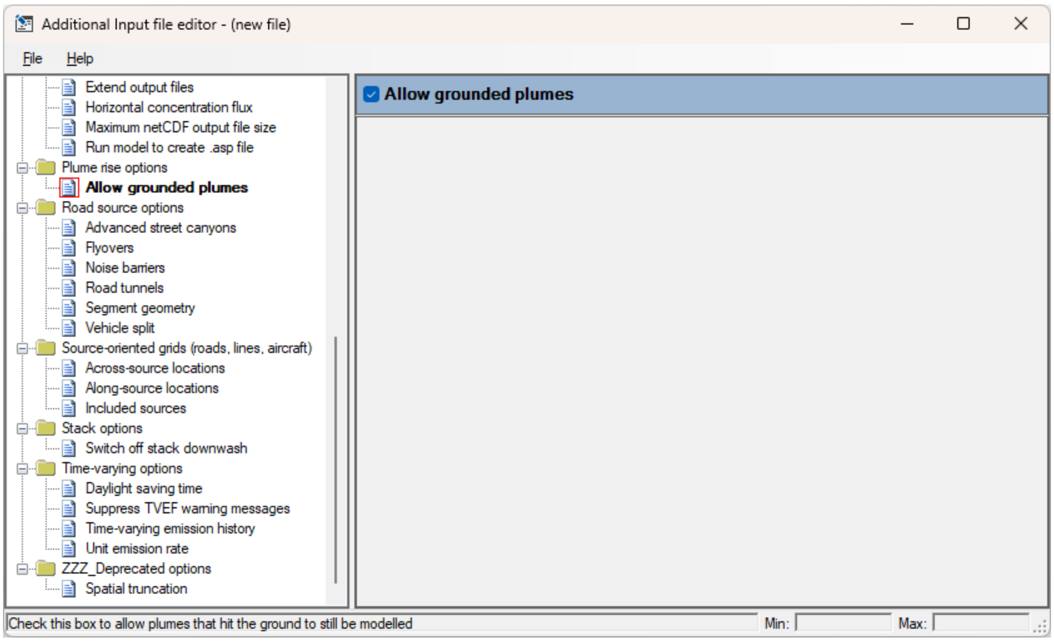


Figure 4.27 – **Allow grounded plumes** section in the **Additional Input file editor**.

When this option is being used, the plume rise calculations will be switched off and the plume modelled as if it is passive from the point it hits the ground. No account will be made of gravitational slumping or other dense gas effects.

An alternative model like GASTAR may be more appropriate for modelling a dense release, contact CERC to discuss the specifics of the situation you are trying to model.

4.11 Dry deposition

Dry deposition occurs when material is lost from the plume at the surface of the ground with a certain deposition velocity. This changes the airborne concentration of a pollutant compared with the case of no deposition by depleting the plume concentration with increasing distance downstream from the source and by adjusting the vertical profile of concentration through removal of material at the surface.

The rate of dry deposition is assumed to be proportional to the near-surface concentration, i.e.:

(4.1) $$F = v_d \, C(x, y, 0)$$

where F is the rate of deposition per unit area per unit time, v_d is the deposition velocity, and C is the predicted airborne concentration at the position $(x,y,0)$.

The deposition velocity contains a diffusive part, which is commonly referred to as the deposition velocity itself, and the terminal velocity of a particle, which is due to gravitational settling in the plume. The reciprocal of the diffusive part is called the resistance. This resistance is calculated as a sum of aerodynamic, sub-layer and surface resistances.

Both the deposition velocity (diffusive part) and the terminal velocity may be input directly in the interface. If the values are unknown, they can be estimated by the model, based on gas type for gaseous substances or particle size and particle density for particulate substances. For gaseous substances, if the deposition velocity (diffusive part) is not known, the value of the surface resistance used in the calculation of the deposition velocity (diffusive part) is determined by the model from the gas type.

Further details of the dry deposition calculation are given in Section 9.15.

4.11.1 Modelling dry deposition

To model dry deposition effects with ADMS-Urban, pollutant deposition parameters must be specified in the **Palette of Pollutants**.

The palette can be accessed by clicking on the **Data...** button in the **Palette** section of the **Setup** screen or by clicking **Pollutants...** on the **Emissions** screen of the **Source** screen.

If spatially and/or hourly varying deposition parameters are required, then the additional input file should be used (refer to Sections 4.11.2 and 4.11.3 for more details about how to do this).

It is only necessary to enter dry deposition parameters if deposition modelling has been selected, otherwise the model will not use the deposition parameters in the table so the user may use the default values. As an exception, ozone deposition parameters will always be used in the Trajectory Model chemistry scheme, whether deposition modelling has been selected or not. Users should therefore ensure that sensible values are entered for these parameters if using this scheme.

The **Palette of Pollutants** is shown in **Figure 4.28** for the case of a particulate pollutant for which both deposition and terminal velocities are unknown.

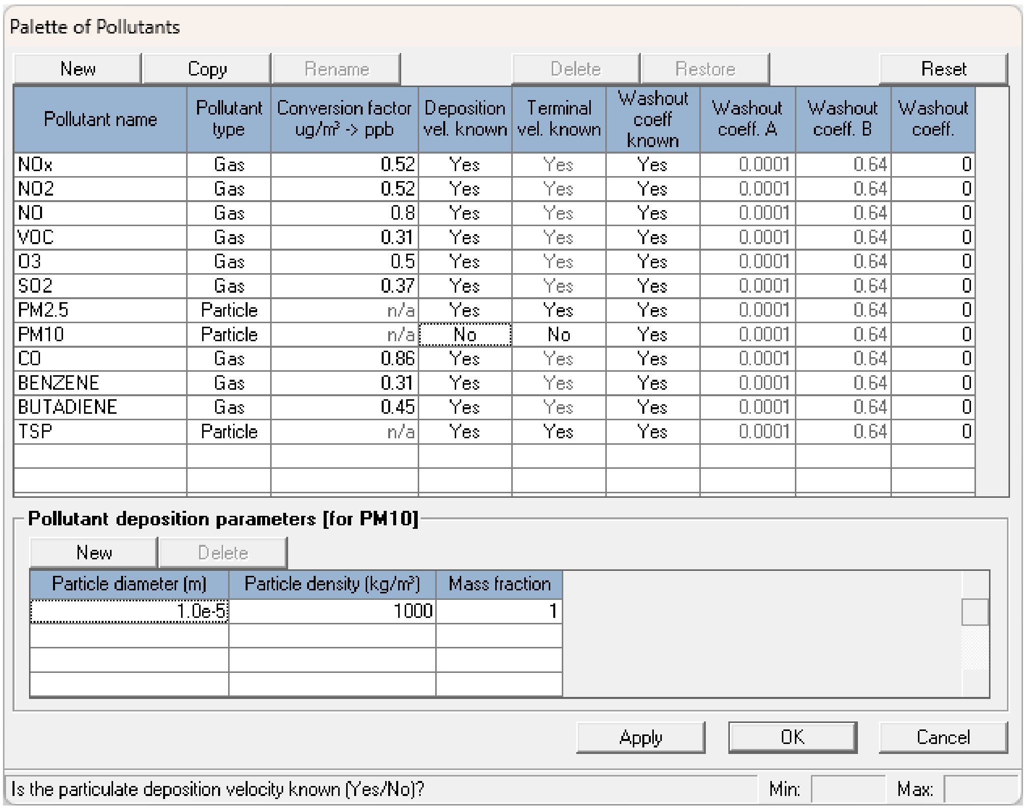


Figure 4.28 − The **Palette of Pollutants** screen showing additional deposition parameters for a particle pollutant.

The deposition parameters that must be defined depend on whether the pollutant type has been specified as **Gas** or **Particle**.

Each parameter is defined below and **Table 4.6** summarises the input options available for each pollutant type.

1. **Deposition vel. known** (required for both gases and particles)

   Select **Yes** or **No** depending on whether the deposition velocity is known.

   Default = **Yes** (for all pollutants)

2. **Terminal vel. known** (required for particles only)

   Select **Yes** or **No** depending on whether the terminal velocity is known.

   Default = **Yes** (for all particles)

3. **Deposition velocity (m/s)**: rate at which material is deposited to the ground due to diffusion

   For both **Gas** and **Particle** emissions if a constant value is to be specified, this should be done in the lower table of **Figure 4.28** (the column **Deposition velocity (m/s)** will appear in the lower table if **Yes** was chosen in the upper table for **Deposition vel. known**).

Minimum = 0 m/s
Maximum = 10 m/s
Default = 0 m/s

4. **Nature of gas** (gases only)

   If the deposition velocity of a gas is unknown (**Deposition vel. known** = **No**), then the gas nature should be specified with one of the following:

   Reactive: gas expected to undergo significant chemical reaction with the surface (for example HCl)

   Non-Reactive: gas not undergoing significant chemical reaction with the surface (for example CO_2, CH_3I, O_3, SO_2, NO_2)

   Inert: noble gas (for example He, Ne, Ar, Kr, Xe, Rn)

   ADMS-Urban uses the above information to estimate the surface resistance.

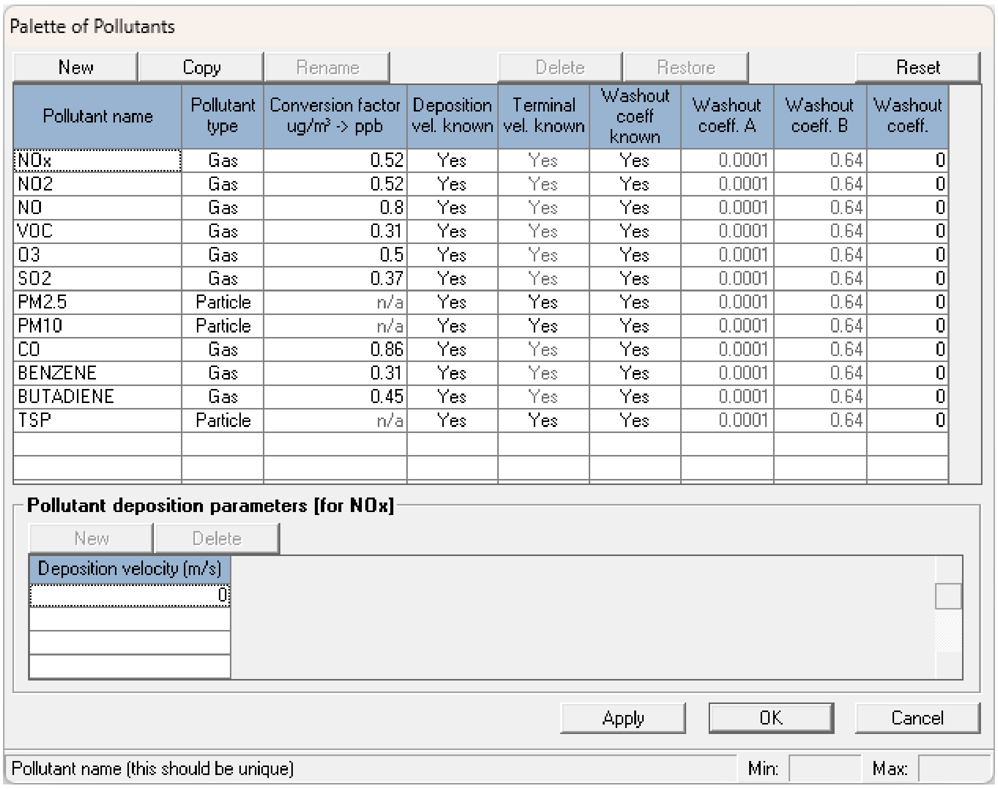


Figure 4.29 – The **Palette of Pollutants** screen showing additional deposition parameters for a gaseous pollutant.

5. **Terminal velocity (m/s)** (particles only)

   This represents the velocity at which material is deposited to the ground due to gravitational settling of particles.

   Minimum = 0 m/s
   Maximum = 10 m/s
   Default = 0 m/s (for both pre-defined and user-defined particles)

6. **Particle diameter (m)** (particles only)

   More than one particle diameter may be defined for each particulate pollutant.

   Minimum = 10^{-9} m (= 0.001 μm)
   Maximum = 0.01 m
   Default = 10^{-5} m (= 10 μm) for PM_{10}, 2.5 ×10^{-6} m (= 2.5 μm) for $PM_{2.5}$, 10^{-6} m (= 1 μm) for user-defined particles

   This is used in the estimation of deposition velocity and/or terminal velocity, for particulate emissions.

7. **Particle density (kg/m³)** (particles only)

   Density of material of which particles are composed. More than one particle density may be defined for each particulate pollutant.

   Minimum = 1 kg/m^3
   Maximum = 1000000 kg/m^3
   Default = 1000 kg/m^3

   This is used in the estimation of terminal velocity for particulate emissions.

8. **Mass fraction** (particles only)

   Mass fraction of the particles with specified velocities/diameters (must sum to 1 ± 0.05 over all particle sub-sets).

   Minimum = 10^{-10}
   Maximum = 1
   Default = 1

   This is always used for particle emissions and allows the user to model particulate mixtures. As long as the total mass fraction sums to 1, up to 10 different particle size/density combinations can be specified by the user.

The greater the number of particle size/density combinations, the longer the run time required for the calculation.

Pollutant type	Dep. vel. known	Term. vel. known	Dep. velocity	Nature of gas	Term. velocity	Particle diameter	Particle density	Mass fraction
Gas	Yes	-	✓	-	-	-	-	-
Gas	No	-	-	✓	-	-	-	-
Particle	Yes	Yes	✓	-	✓	-	-	✓
Particle	No	Yes	-	-	✓	✓	-	✓
Particle	Yes	No	✓	-	-	✓	✓	✓
Particle	No	No	-	-	-	✓	✓	✓

Table 4.6 − Dry deposition parameters that must be specified for each pollutant type.

4.11.2 Using spatially varying dry deposition parameters

It is often the case that the dry deposition parameters vary spatially due to land use changes over the model domain. For both gaseous and particulate pollutants, spatially varying values of the deposition velocity can be used by the model. If these are to be entered for a particular pollutant, then the deposition velocity should be set to known in the **Palette of Pollutants**. For gaseous pollutants, if the deposition velocity is unknown, but the spatial variation of the surface resistance parameter is known, then this information can be used by the model to calculate the deposition velocity. In this case, the deposition velocity should be set to unknown in the **Palette of Pollutants**.

One set of deposition velocity and surface resistance values that vary over the model domain can be entered into a *.din* file and the path of this entered into the additional input file. The values specified will be used for all pollutants listed in the relevant section in the additional input file. Details of how to set up the additional input file and the format of a *.din* file are given below.

Additional input file

Spatially varying values of the deposition parameters are entered using an additional input file. To use these options, create a new additional input file or edit the existing additional input file (refer to Section 3.1.8) and enable the **Spatial and temporal variation of deposition parameters** section (keyword VARDEPOSITION), as shown in **Figure 4.30**.

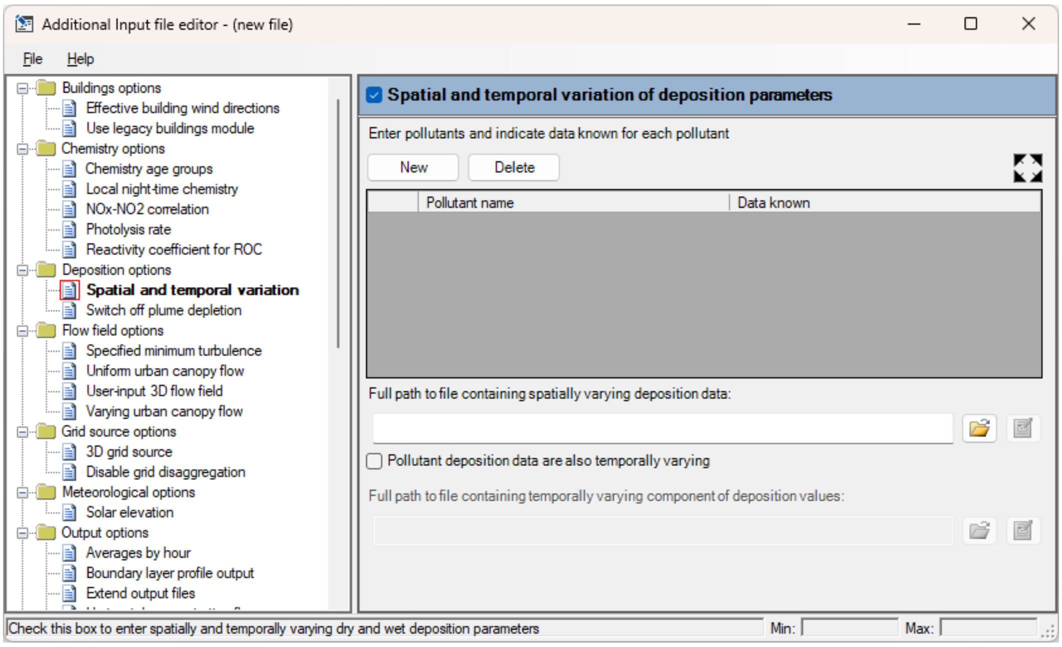


Figure 4.30 – The **Spatial and temporal variation of deposition parameters** section enabled in the **Additional Input file editor**

Each of the pollutants for which spatially varying deposition parameters are to be entered should be entered into the table with the choice of dry deposition data type as follows:

- **Deposition velocity** if **Deposition vel. known** is set to **Yes** in the **Palette of Pollutants**
- **Surface resistance** if **Deposition vel. known** is set to **No** in the **Palette of Pollutants**

The path to the *.din* file must also be entered; the format of the *.din* file is described below. The button can be used to view the file in the viewer selected from the **File\Preferences** menu. For details on specifying that the deposition data are also temporally varying see Section 4.11.3.

Spatially-varying file format

Spatially-varying values of the deposition velocity or surface resistance are entered using a *.din* file. Note that this file also contains the associated wet deposition parameter values. Further details of how to model the spatial variation of wet deposition parameters is given in Section 4.12.2.

The first line contains the file version number; this is currently `VarDepVERSION1`. The data are then entered in the following format:

```
N, X, Y, Vd, Rs, Lambda, A, B
```

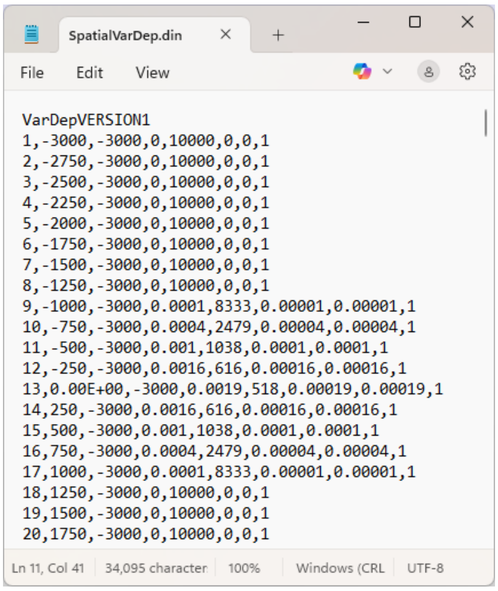
where:

- `N` is an incrementing counter for each line,
- `X` is the X coordinate of the data point,
- `Y` is the Y coordinate of the data point,
- `Vd` is the deposition velocity at this data point (m/s),
- `Rs` is the surface resistance at this data point (s/m),
- `Lambda` is the washout coefficient at this data point (1/s) (refer to Section 4.12 for details),
- `A` is the washout coefficient A at this data point (refer to Section 4.12 for details), and
- `B` is the washout coefficient B at this data point (refer to Section 4.12 for details).

Note that data for all of the parameters must be entered even if some of the parameters are not to be used. The data values for the unused columns can be set to 0.

Data point coordinates should be in metres, in the same coordinate system as all other input coordinate data (e.g. source locations, output grid etc).

An example *.din* file is shown in **Figure 4.31**, which is also supplied in the *<install_path>\Data* directory.



```
VarDepVERSION1
1,-3000,-3000,0,10000,0,0,1
2,-2750,-3000,0,10000,0,0,1
3,-2500,-3000,0,10000,0,0,1
4,-2250,-3000,0,10000,0,0,1
5,-2000,-3000,0,10000,0,0,1
6,-1750,-3000,0,10000,0,0,1
7,-1500,-3000,0,10000,0,0,1
8,-1250,-3000,0,10000,0,0,1
9,-1000,-3000,0.0001,8333,0.00001,0.00001,1
10,-750,-3000,0.0004,2479,0.00004,0.00004,1
11,-500,-3000,0.001,1038,0.0001,0.0001,1
12,-250,-3000,0.0016,616,0.00016,0.00016,1
13,0.00E+00,-3000,0.0019,518,0.00019,0.00019,1
14,250,-3000,0.0016,616,0.00016,0.00016,1
15,500,-3000,0.001,1038,0.0001,0.0001,1
16,750,-3000,0.0004,2479,0.00004,0.00004,1
17,1000,-3000,0.0001,8333,0.00001,0.00001,1
18,1250,-3000,0,10000,0,0,1
19,1500,-3000,0,10000,0,0,1
20,1750,-3000,0,10000,0,0,1
```


Figure 4.31 − An example of a *.din* file.

Restrictions

The **Spatial and temporal variation of deposition parameters** option cannot be used in combination with either the **Chemistry** or **Urban canopy flow** option. Additionally, the spatially and/or hourly varying deposition data are not applied to grid sources; the fixed deposition parameters as specified in the interface will be used instead.

4.11.3 Combining hourly and spatially varying dry deposition velocities

Sometimes it is also useful to be able to model both the hourly and spatial variation of the deposition velocity at the same time, for example when modelling deposition of ammonia. The spatially varying deposition model option can also be combined with hourly variation. However, this option is restricted to modelling the variation of deposition velocity only and to one pollutant per model run.

In order to model a combination of hourly and spatially varying dry deposition velocities, a *.din* file should be set up representing the required spatial variation of deposition velocities, as described in Section 4.11.2 above. The hourly variation of the deposition velocity is introduced by a scaling factor entered into a *.dnf* file and the path of this entered into the additional input file. The additional input file should be set up as outlined in Section 4.11.2 but with the **Pollutant deposition data are also temporally varying** option checked and the path of the *.dnf* file entered into the appropriate section. Details of the format of a *.dnf* file are given below.

If this option of modelling the hourly variation of deposition velocity is used then the meteorological data must contain values of the year, day and hour.

Hourly scaling factor file format

Hourly-varying scaling factors to be applied to the spatially varying values of the deposition velocity in the *.din* file are entered using a *.dnf* file. The file format is outlined below.

The first line contains the file version number; this is currently `VarDepFacVersion1`. Then any comments may be added. The data are then entered: firstly there is a line containing the key word "`DATA:`", then for each line of meteorological data, the data are entered in the following format:

```
Year, Day, Hour, Factor
```

where `Factor` is the scaling factor by which the deposition velocity specified in the *.din* file should be multiplied. An example *.dnf* file is shown in **Figure 4.32**, which is also supplied in the *<install_path>\Data* directory.

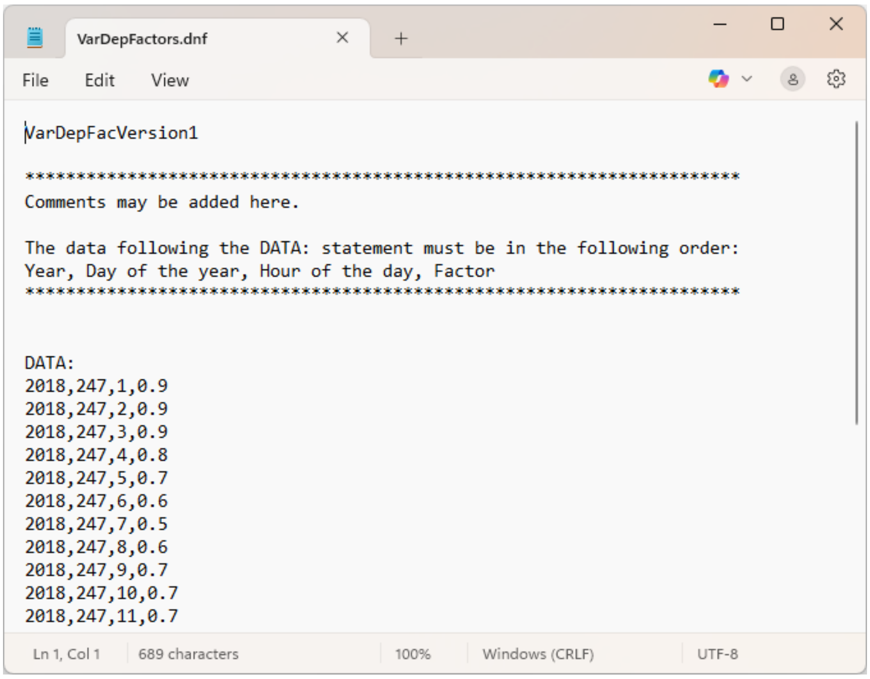


Figure 4.32 – An example of a *.dnf* file.

The restrictions that apply to this option are given in Section 4.11.2.

4.11.4 Modelling without plume depletion

It is possible to switch off the modelling of plume depletion due to dry deposition by the use of an additional input file. When this option is enabled, the modelled concentrations are identical to those that would be predicted if the dry deposition option were not selected. The exception to this is if a particulate pollutant is modelled, in which case when using this option the effects of gravitational settling are still taken into account. The values of dry deposition calculated by the model are equal to the product of the dry deposition velocity and the ground level concentration.

To switch off the plume depletion for dry deposition, create an additional input file or edit the existing additional input file (refer to Section 3.1.8) enable the **Plume depletion** section (keyword SWITCHOFFDEPLETION) and check the **Switch off plume depletion for DRY deposition** option, as shown in **Figure 4.33**.

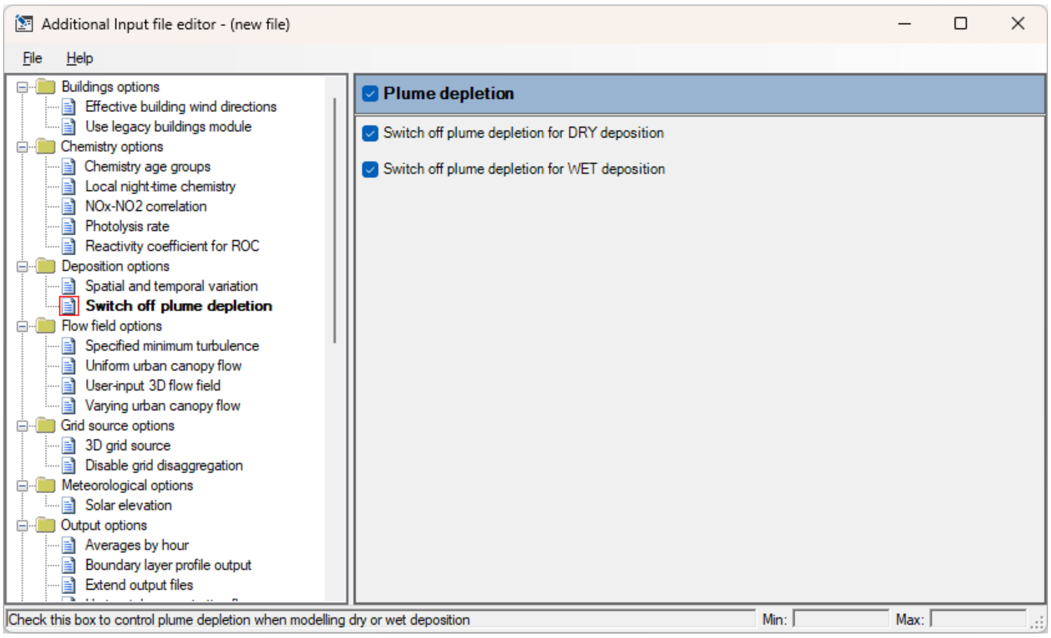


Figure 4.33 – **Additional Input file editor** screen for switching off plume depletion for dry deposition.

4.11.5 Output

Dry deposition results are output to *.gst/.pst* and *.glt/.plt* files for short-term and long-term runs, respectively. The output units are mass units/m^2/s, e.g. µg/m^2/s. These are units of deposition flux or rate.

Dry deposition results do not include deposition of background concentrations.

If wet and dry deposition have both been calculated then total deposition results are also output. Total deposition is the sum of the dry and wet deposition rates.

The values of dry deposition velocity and terminal velocity entered by the user and/or calculated by the model for each pollutant are output to the *.dep* file, for the first 24 lines of meteorological data (or for all met. lines if using the 'Extend output files' option, see Section 4.26).

4.11.6 Guidance on dry deposition parameters

Dry deposition velocities depend on the nature of the surface. **Table 4.7** contains suggested deposition velocities for short vegetation (such as grass) for NO_2, NO, SO_2, and NH_3.

Parameter	NO_2	NO	SO_2	NH_3
deposition velocity (m/s)	0.0015	0.00015	0.012	0.02

Table 4.7 − Suggested deposition velocities for short vegetation.

A conservative estimate for both concentrations and deposition rates of NOx can be obtained by setting the gas property to Non-reactive and switching off plume depletion as described in Section 4.11.4.

4.12 Wet deposition

Wet deposition occurs when there is precipitation (rain, sleet, snow, etc) and material is washed out of the plume. Wet deposition is modelled through a washout coefficient Λ such that the mass of material captured by the precipitation and therefore removed from the plume is ΛC per unit volume per unit time, where C is the local airborne concentration. Λ is dependent on the nature of the pollutant, the rainfall rate and droplet size.

The washout coefficient Λ can be taken to have a constant value, in which case the wet deposition is independent of the rainfall rate. In fact, even if there is zero rainfall, or if rainfall rate is not a parameter in the meteorological file, a constant washout coefficient can be specified and the wet deposition can be calculated. This is equivalent to assuming a constant drizzle.

Alternatively, the washout coefficient Λ can depend on the rainfall rate P, and the model will assume a relationship between Λ and P of the form

(4.2) $$\Lambda = AP^{B}$$

where A and B are constants entered in the interface (see **Figure 4.34**). If Λ is specified in this way then the rainfall rate in mm/hr must be included as a variable in the meteorological data.

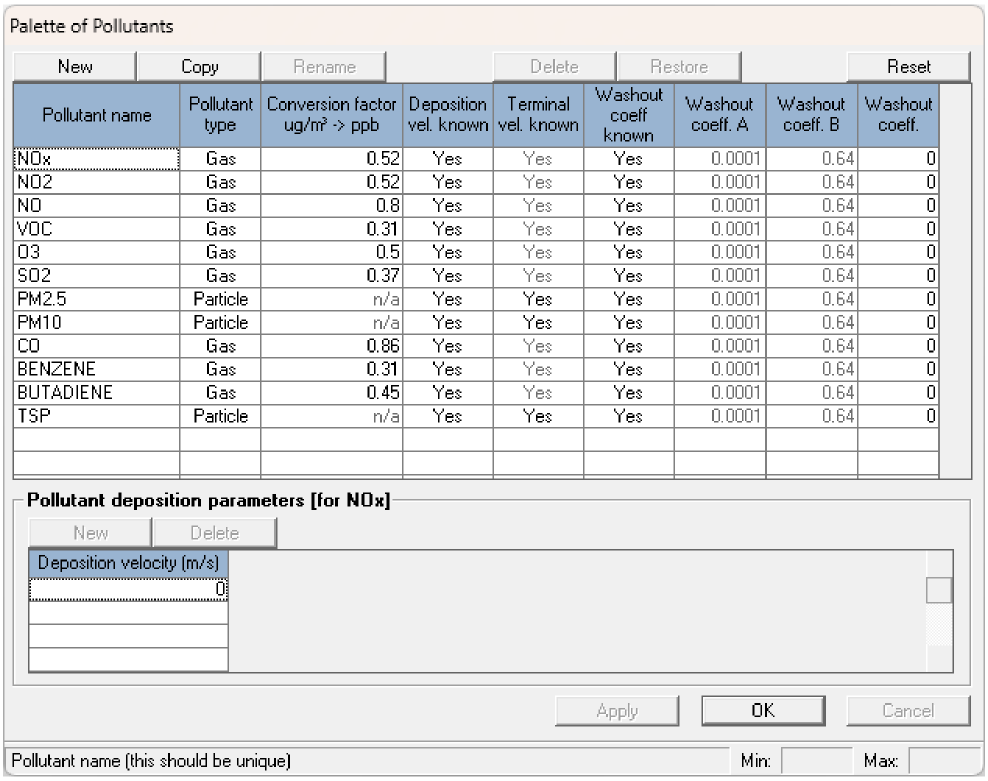


Figure 4.34 − The **Palette of Pollutants** screen showing the parameters required to model wet deposition.

4.12.1 Modelling wet deposition

To model the effect of wet deposition, the appropriate pollutant deposition parameters must be specified in the **Palette of Pollutants**, shown in **Figure 4.34**. The palette can be accessed via the **Emissions** screen (launched from the **Source** screen) or by clicking on the **Data...** button in the **Palette** section of the **Setup** screen of the ADMS-Urban interface.

The columns in the upper and lower tables in the ***Palette of Pollutants*** *are resizable.*

It is only necessary to enter wet deposition parameters if deposition modelling has been selected, otherwise the model will not use the parameters and the user may use the default values.

Default values apply for all pollutants including user-defined pollutants.

1. **Washout coeff known**

   The user should select **Yes** or **No** depending on whether the *washout coefficient* Λ (refer to item 4 below) is known. If **Yes**, wet deposition will be calculated using a constant washout coefficient which is independent of the rainfall rate. If **No**, it will be calculated using parameters A and B in equation (4.2).

   Default = **Yes**

2. **Washout coeff. A**

   This parameter is required if the washout coefficient is not known (refer to 1 above) and represents A in equation (4.2). If this parameter is used, wet deposition is taken to be a function of the rainfall rate.

   Minimum = 0
   Maximum = 100
   Default = 0.0001

3. **Washout coeff. B**

   This parameter is required if the washout coefficient is not known (refer to 1 above) and represents B in equation (4.2). If this parameter is used, wet deposition is taken to be a function of the rainfall rate.

   Minimum = 10^{-10}
   Maximum = 100
   Default = 0.64

   The default values of A *and* B *may be used unless the user has specific data e.g. from experiments for the particular pollutant.*

4. **Washout coeff.**

   If the washout coefficient is known, wet deposition is independent of the rainfall rate.

   Minimum = 0 s^{-1}
   Maximum = 1 s^{-1}
   Default = 0 s^{-1}

The washout coefficient values are not used for NO as the wet deposition of NO is always assumed to be 0.

4.12.2 Using spatially varying wet deposition parameters

Wet deposition parameters may vary spatially due to changes in precipitation rates across the model domain. ADMS-Urban is able to model this variation, with details given below.

For both gaseous and particulate pollutants, spatially varying values of the washout coefficients Λ, or A and B can be used by the model. If these are to be entered for a particular pollutant, then the **Washout coeff known** option should be set up as appropriate in the **Palette of Pollutants.**

One set of washout coefficients Λ, A and B that vary over the model domain can be entered into a *.din* file, and the path of this entered into the additional input file. The values specified will be used for all pollutants listed in the relevant section in the additional input file. Details on setting up the additional input file, the format of a *.din* file and the restrictions that apply to this option are given in Section 4.11.2.

4.12.3 Modelling without plume depletion

It is possible to switch off the modelling of plume depletion due to wet deposition by use of an additional input file. When this option is enabled, the modelled concentrations are identical to those which would be predicted if the wet deposition option were not selected.

To switch off the plume depletion for wet deposition, create an additional input file or edit an existing additional input file (refer to Section 3.1.8) enable the **Plume depletion** section (keyword SWITCHOFFDEPLETION) and check the **Switch off plume depletion for WET deposition** option, as shown in **Figure 4.35**.

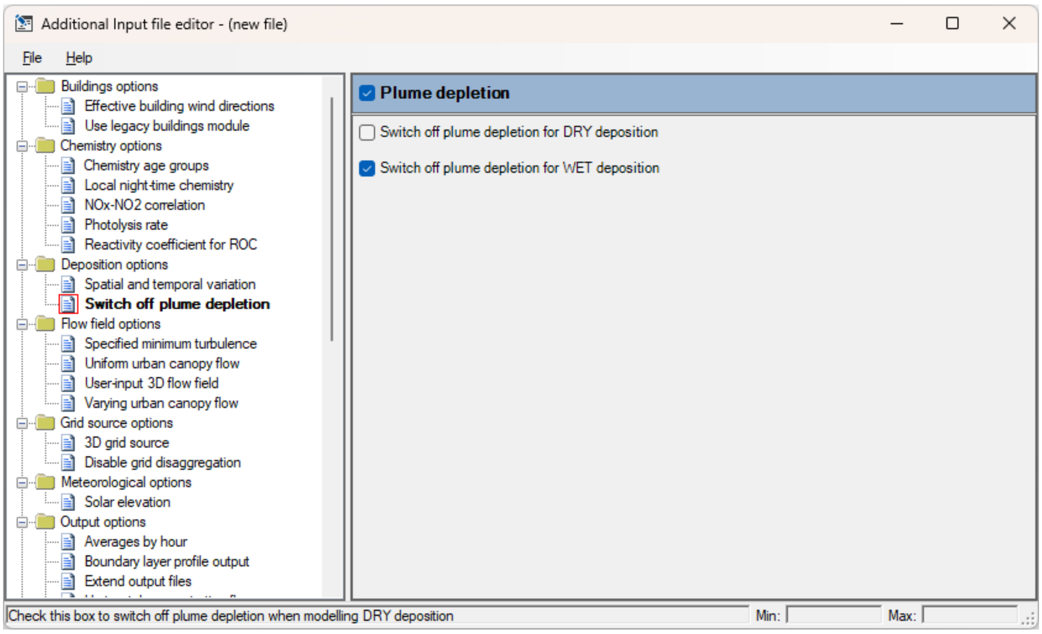


Figure 4.35 – **Additional Input file editor** screen for switching off plume depletion for wet deposition.

4.12.4 Output

Wet deposition rates are output to *.gst/.pst* and *.glt/.plt* files for short-term and long-term runs respectively. The output units are mass units/m^2/s as this is a deposition flux or rate. Note that dry deposition (if selected) and total deposition results are also available in the same way. Total deposition is the sum of dry and wet deposition rates.

The wet deposition washout coefficients entered by the user and/or calculated by the model for each pollutant are output to the *.dep* file for the first 24 lines of meteorological data (or for all met. lines if using the 'Extend output files' option, see Section 4.26).

4.12.5 Guidance on wet deposition parameters

Table 4.8 gives some suggested values for wet deposition parameters for NO_2, NO, NH_3 and HCl. Note that NO is very insoluble and the wet deposition rate will be 0 even if different values for the wet deposition parameters are entered. The deposition rate of NO_2 is also small.

Parameter	NO_2	NO	NH_3	HCl
A	10^{-4}	0	0.005	0.0003
B	0.64	n/a	0.64	0.66

Table 4.8 − Deposition parameters for NO_2, NO, NH_3 and HCl.

For further information on wet deposition parameters, users are advised to consult the reports of the Review Group on Acid Rain (RGAR). The following advice on wet deposition parameters for SO_2 and sulphate (SO_4^{2-}) is based on RGAR reports (CLAG, 1994).

For SO_2, if P < 103 mm/year, $\Lambda = 1.0 \times 10^{-6} (P / 500)$
if P > 103 mm/year, $\Lambda = 1.0 \times 10^{-6} \left[2 + (0.336 / 0.84)(P - 1000) / 500\right]$

For SO_4^{2-}, if P < 103mm/year, $\Lambda = 1.3 \times 10^{-5} (P / 500)$
if P > 103mm/year, $\Lambda = 1.3 \times 10^{-5} \left[2 + (0.336 / 0.84)(P - 1000) / 500\right]$

P is the rainfall rate in mm/year, and Λ is the constant value of the washout coefficient, independent of hourly rainfall rate.

4.13 Chemistry

ADMS-Urban includes the facility to calculate the chemical reactions in the atmosphere between nitric oxide (NO), nitrogen dioxide (NO_2), ozone (O_3) and volatile organic compounds (VOC). The conversion of sulphur dioxide (SO_2) to sulphate particles is also modelled. For full details of the chemistry schemes refer to Section 9.17.

To model these chemical reactions, check the box on the **Setup** screen marked **Chemistry**. Click **Data...** to bring up the **Chemistry** screen, shown in **Figure 4.36**.

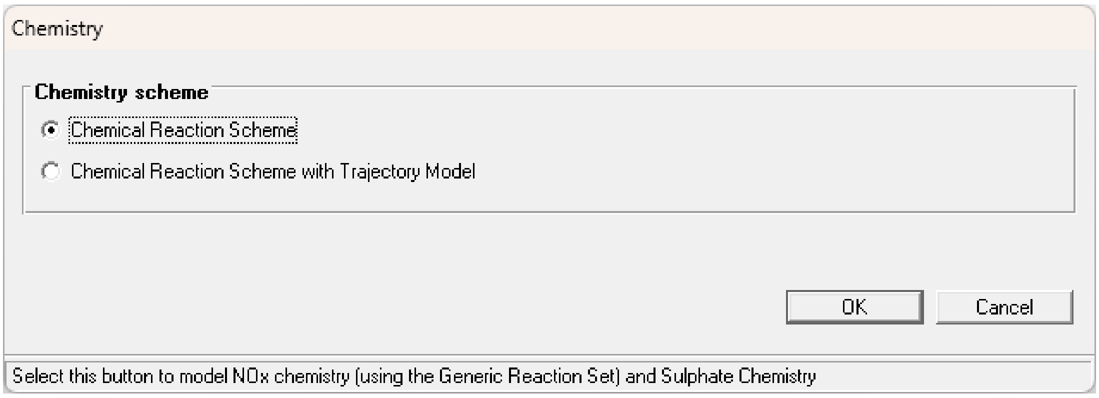


Figure 4.36 – Chemistry screen

4.13.1 The Chemistry Schemes

Chemical Reaction Scheme

The Chemical Reaction scheme incorporates both the set of eight reactions known as the ‘Generic Reaction Set’ for the photochemical reactions between NO, NO_2, VOC and O_3, and also the reactions governing the oxidation of SO_2 that lead to the formation of ammonium sulphate particles, i.e. PM_{10} and $PM_{2.5}$. In order to use this scheme, background concentration data are required for the critical pollutants NO_X, NO_2, O_3 and SO_2. (If no background concentration data are available for SO_2 then values of 0 may be input.) Please refer to Section 3.4 for more advice on selecting background data.

Chemical Reaction Scheme with Trajectory Model

This is an extension of the Chemical Reaction Scheme that is used to represent the physical and chemical processes which can occur in the atmospheric boundary layer over a very large urban area. When modelling large conurbations, the surface ozone concentrations may vary considerably from the background value. In these cases, a more representative and spatially variable ozone field is required as input to the Chemical Reaction Scheme in ADMS-Urban. The Trajectory Model calculates this spatially varying ozone field (as well as the NO_X, VOC and PM_{10} fields). This allows the ADMS-Urban output calculation grid to be nested within a much larger domain, as in the case of a modelling study area in the centre of London or Birmingham, for example.

The Trajectory Model assumes that the initial background data are representative of the upwind edge of the area being modelled. In order to run the Trajectory Model, a grid source must be defined covering the whole study area. The Trajectory Model aggregates the emissions in each grid cell into 5 km x 5 km grid squares. The emissions, meteorological conditions and deposition fluxes are assumed constant over each 5 km Trajectory Model grid square.

Note that the Trajectory Model always uses the ozone deposition parameters defined in the ***Palette of Pollutants*** *when calculating the deposition fluxes, whether deposition modelling has been selected or not. Users should therefore ensure that sensible values are entered for these parameters.*

Background concentration data are required for the critical pollutants NO_X, NO_2, O_3 and SO_2. As the Trajectory Model can only be used if a grid source is modelled, and the grid source should contain emissions for all the sources in the study area, background concentrations from a rural site outside the study area should be used.

4.13.2 Primary NO2

When using the chemistry model option, emissions of NO_2 as well as NO_x are required by the model.

The EFT v13.1, 12.1, 12.0, 11.0, 10.1, 9.0 and 8.0 datasets include emission factors for NO_2. To use these factors to calculate explicit NO_2 emission rates, ensure that NO_2 is included in the list of emission pollutants on the **Emissions** screen for each road source.

By default, if NO_2 emissions are not included explicitly, then the model will assume that a certain percentage of the NO_X emissions are NO_2 when modelling chemistry. The **Primary NO2 (%)** is specified on the **Source** screen of the interface (**Figure 4.37**) when **Chemistry** is enabled, with different values for each source type. The default values for each of the source types are shown in **Table 4.9**.

The ***Primary NO2 (%)*** *field does not apply to 3D grid sources (see Section 4.7). Therefore,* NO_X *and* NO_2 *emissions must be included in the 3D grid file when* ***Chemistry*** *is enabled.*

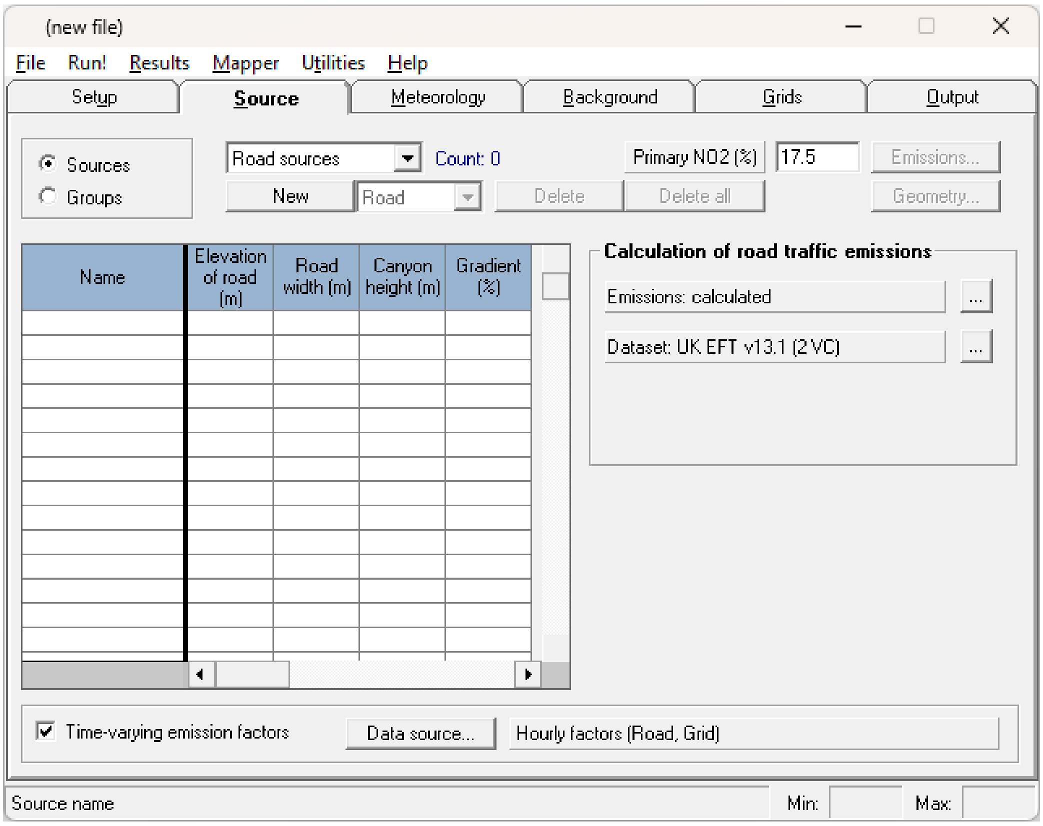


Figure 4.37 – **Source** screen for **Road** sources when **Chemistry** is enabled.

Source type	**Primary NO_2**
Industrial	5%[1]
Roads	17.5%[2]
Grids	10.5%[3]

Table 4.9 – Default values of primary NO_2 per source type.

When modelling chemistry, it is important that the primary NO_2 values are specified correctly for each of the sources. In particular when modelling with a grid source, the primary NO_2 values used when aggregating the emissions onto the grid must be consistent with those that will be used when modelling the sources to ensure that the disaggregation of the NO_2 emissions is performed correctly.

4.13.3 Additional meteorological data requirements

In order to use the Chemical Reaction Scheme, or Chemical Reaction Scheme with Trajectory Model, some additional meteorological data must be input. The additional data required for each chemistry scheme are summarised in **Table 4.10**. These data

[1] The suggested value for typical installations given in the UK Environment Agency note RM-QG05 on Monitoring oxides of Nitrogen (2012)
[2] The value for ‘All UK traffic’ for 2024 from the NAEI Primary NO_2 factors spreadsheet https://naei.energysecurity.gov.uk/sites/default/files/2024-10/PrimaryNO2%20factors_NAEIBase%202024_v1.xlsx
[3] Calculated assuming the grid consists solely of residual emissions split between minor roads, domestic gas and industrial sources.

are used by ADMS-Urban to calculate the chemical reaction rates. Note that these requirements are additional to those required for all model runs, which are described in Section 9.1 of the User Guide.

Chemistry scheme	Additional meteorological data required
Chemical Reaction Scheme (NOx pollutants)	Temperature Solar radiation, or all of cloud cover, year, day and hour
Chemical Reaction Scheme with Trajectory Model (NOx pollutants)	Temperature Solar radiation or cloud cover Year, day and hour Met data must be hourly sequential
Chemical Reaction Scheme (sulphate pollutants)	Year, day and hour
Chemical Reaction Scheme with Trajectory Model (sulphate pollutants)	Year, day and hour Met data must be hourly sequential

Table 4.10 - Additional met. data requirements for chemistry calculations.

4.13.4 Photolysis rate

By default, ADMS-Urban calculates the photolysis rate based on the assumption that background concentration values are in equilibrium. In some cases, it may be desirable to calculate the photolysis rate based on solar radiation values instead. This can be accomplished by using the **Photolysis rate** option.

To enable this option, create a new additional input file or edit the existing additional input file (refer to Section 3.1.8) and enable the **Photolysis rate** section (keyword R3REACTIONRATE), as shown in **Figure 4.38**. The user can pick either option from the dropdown menu.

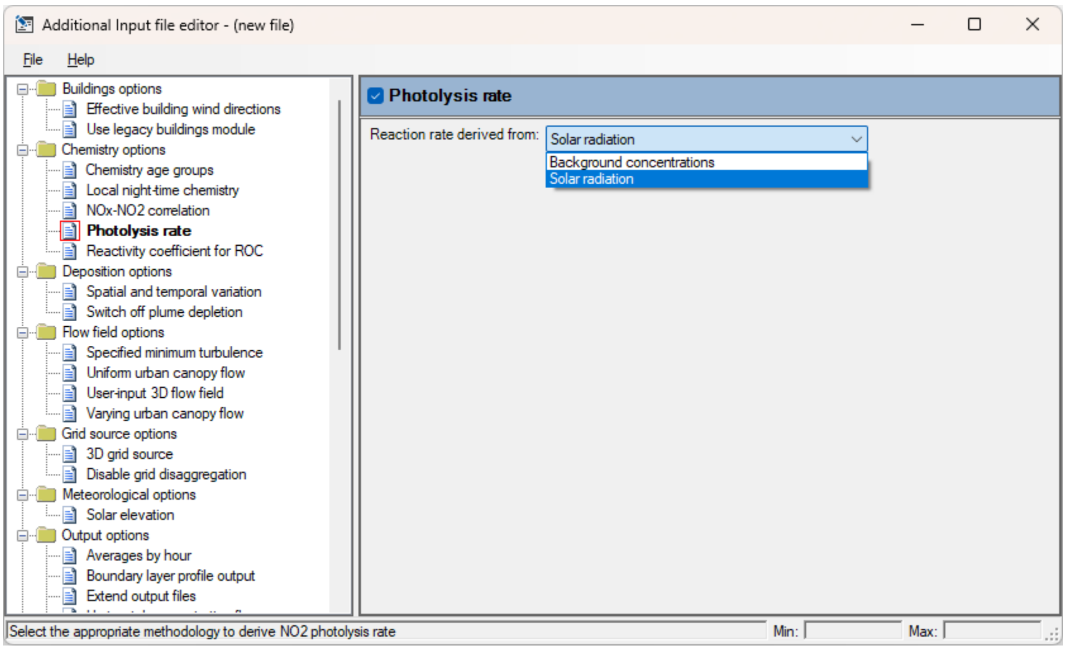


Figure 4.38 – **Photolysis rate** scheme in the **Additional Input file editor**.

4.13.5 Local night-time chemistry

During the day the model ensures the background values are in equilibrium, either by calculating the photochemical reaction rate based on the background values, or by adjusting the background values directly if the photochemical reaction rate is calculated from solar radiation. This adjustment cannot be performed during the night as there is no photochemical equilibrium. A local night-time chemistry scheme is available, which allows the effects of chemistry on the emitted NO_X and NO_2 to be taken into account without spuriously affecting the background values. Technical details of this scheme are given in Section 9.17. As this is a local chemistry scheme, it cannot be used with the Trajectory model.

To use the local night-time chemistry scheme, the **Chemistry** option should be enabled in the **Setup** screen of the interface and the **Chemical reaction scheme** selected on the **Chemistry** screen. An additional input file should then be created or the existing additional input file edited (refer to Section 3.1.8) so the **Local night-time chemistry** section (keyword NIGHTCHEM) is enabled, as shown in **Figure 4.39**. No further parameters are required for this section.

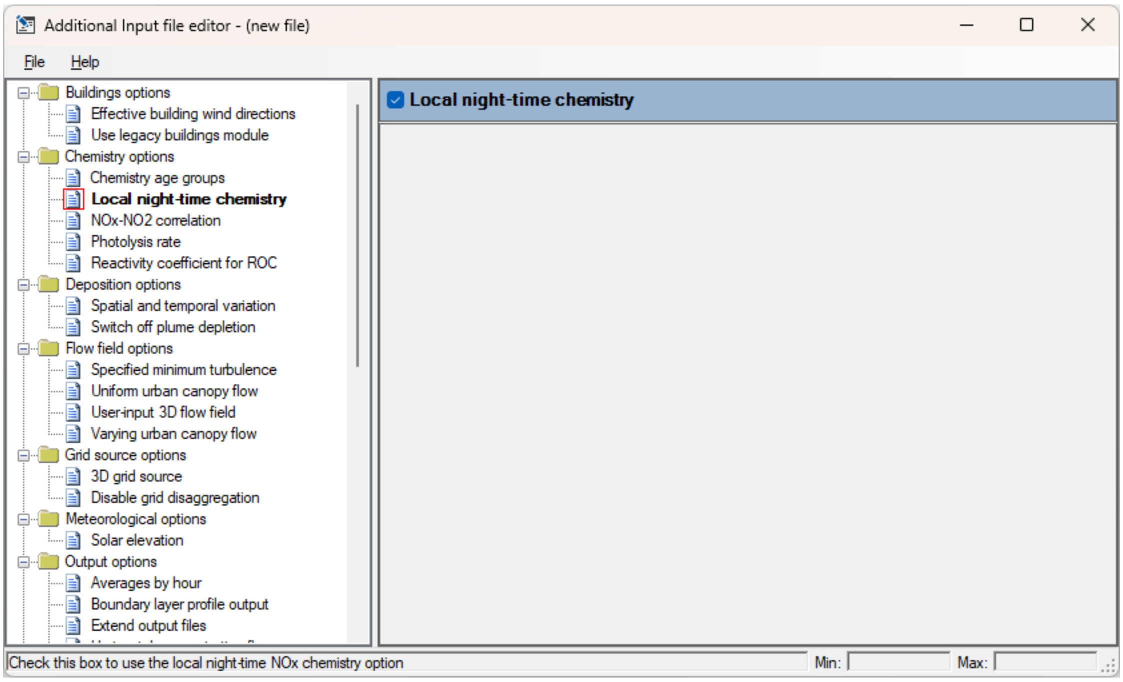


Figure 4.39 – **Local night-time chemistry** scheme in the **Additional Input file editor**.

4.13.6 Reactivity coefficient for ROC

Reaction 1 in the GRS scheme (see Section 9.17) involves the production of radicals via the photolysis of reactive organic compounds (ROCs), which are a subset of VOCs. The proportion of VOCs that contribute to this reaction is controlled by a weighted reactivity coefficient. By default, this coefficient is set to 0.05 as a result of validation work. However, it is possible for the user to modify this default value via an additional input file option. To use this option, create a new additional input file or edit an existing additional input file (refer to Section 3.1.8) so that the **Reactivity coefficient for ROC** option (keyword AROCUSERDEF) is enabled, as shown in **Figure 4.40**. Enter a reactivity coefficient value for ROC between 0 and 1.

The reactivity coefficient needs to be adjusted to account for the presence of methane in the VOC emissions and concentrations.

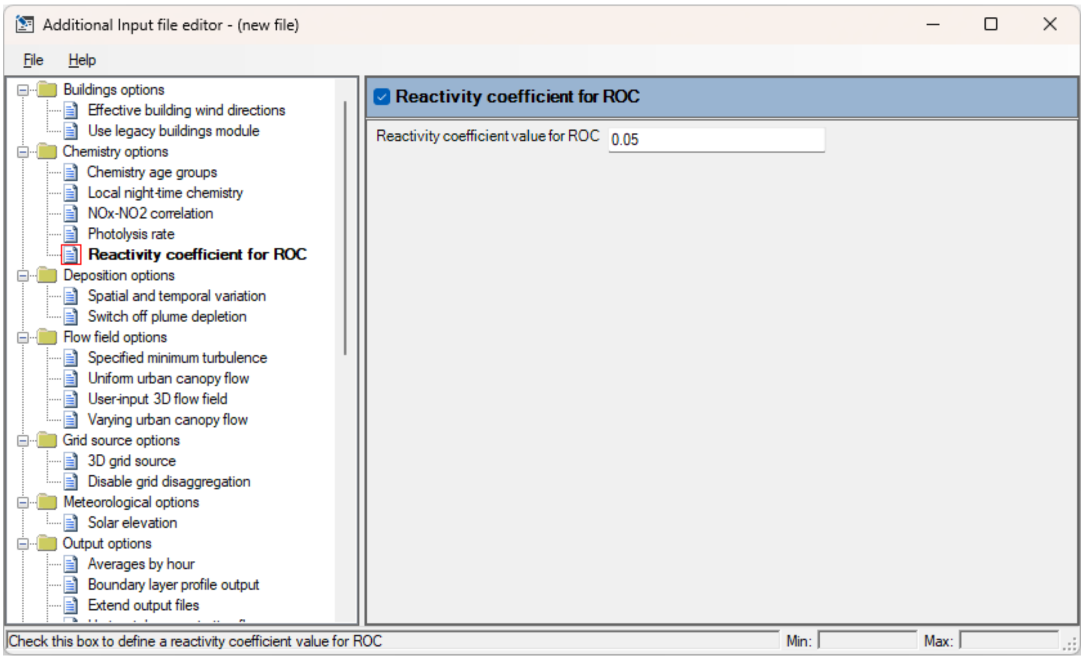


Figure 4.40 – The **Reactivity coefficient for ROC** option enabled in the **Additional Input file editor**.

4.13.7 Chemistry age groups

When the **Chemistry** option is selected on the **Setup** screen of the ADMS-Urban Interface, it will automatically split sources into near and far field based on the travel time (refer to Section 9.17). The **Chemistry age groups** option allows changing when that boundary occurs and splitting into more chemistry age groups where the user can define the age range of each group. To enable this option, create a new additional input file or edit the existing additional input file (refer to Section 3.1.8) and enable the **Chemistry age groups** section (keyword CHEMISTRYSTEPS), as shown in **Figure 4.41**. The user may add or remove upper time bounds in the table using the **New** and **Delete** buttons, respectively.

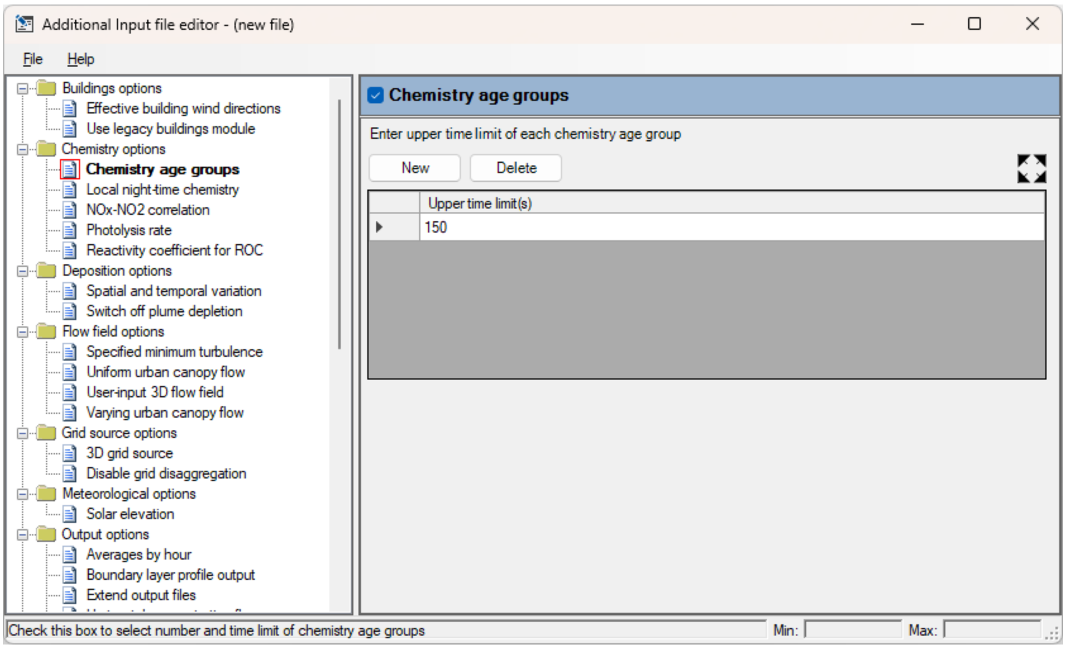


Figure 4.41 – **Chemistry age groups** scheme in the **Additional Input file editor**.

4.13.8 NOx-NO2 correlation

Instead of the chemical reaction scheme discussed above it is possible to use a relatively simple function, the Derwent-Middleton Correlation, to estimate the concentration of NO_2 from a given concentration of NO_X. No additional data are required for this scheme. Note that only NO_X background data are required – NO_2 background data will be ignored if input. This scheme does not include the formation of sulphate particles.

This option is not a valid method for UK regulatory modelling. It is also not suitable for rural areas and has only been validated with data from cities within the United Kingdom. Refer to Section 9.17.3 for more details.

The NO_X-NO_2 correlation is enabled through the use of an additional input file. First ensure that **Chemistry** is selected on the **Setup** screen of the ADMS-Urban Interface, with any of the choices on the **Chemistry** screen selected. Then create a new additional input file or edit the existing additional input file (refer to Section 3.1.8) and enable the **NOx-NO2 correlation** section (keyword NOX-NO2CORR), as shown in **Figure 4.42**. No further parameters are required for this section.

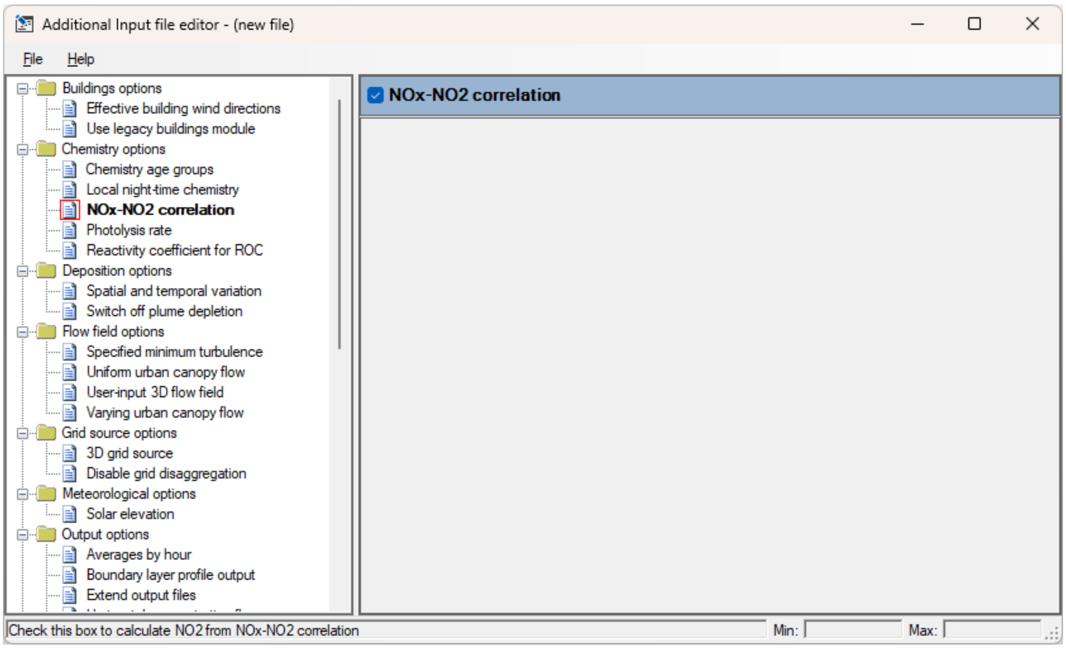


Figure 4.42 – **NOx-NO2 correlation** scheme in the **Additional Input file editor**.

4.14 Odours

Odours are typically measured in ‘odour units’. The **Odours** option in ADMS-Urban uses the odour unit (ou_E) defined in the CEN standard (EN 13725:2003). One ou_E is the mass of pollutant that, when evaporated into 1 m^3 of odourless gas at standard conditions, is at the detection limit.

4.14.1 Input data

When the **Odours** option is selected on the **Setup** screen, the emission rate and output units are changed to be based on ou_E; refer to Sections 3.2 and 3.6.1 for more details.

4.14.2 Restrictions

- The odours option may not be used in combination with the **Chemistry** option.
- When using the odours option, the **Calculate emissions using traffic flows** option for road sources cannot be used.
- Import, Export and the emissions inventory may not be used with the odours option.

4.15 Buildings

4.15.1 Building effects

Chimney stacks often protrude from the roof of a building or are attached to the side of a building on industrial sites.

Buildings can have a significant effect on the dispersion of pollutants from sources and can increase the maximum predicted ground level concentrations. The main effect is to entrain pollutants into the cavity region in the immediate leeward side of the building, bringing them rapidly down to ground level. As a consequence, concentrations near the buildings are increased but further away they are decreased.

Up to 25 buildings may be modelled in an ADMS-Urban model run. However, this does not mean that this many buildings *should* be included in a given run and consideration must be given as to which buildings to include.

For each source, the program combines the individual buildings into a single effective building for each wind direction in the meteorological file, the height of which is the height of the building specified as the main building for that source. (This is described in more detail in the Section 9.18.)

The effect of buildings on dispersion can only be modelled for point sources. Other types of sources may be included in the modelling run, but the effect of the buildings on dispersion from these sources will not be modelled. In order to model building effects for a line, area, volume or road source, the source should be modelled as a number of point sources.

The effects of buildings/structures alongside road sources should be modelled using the street canyon, or advanced street canyon model options.

4.15.2 Defining buildings

Select the **Buildings** button on the **Setup** screen and click **Data...** to go to the **Buildings** screen shown in **Figure 4.43**.

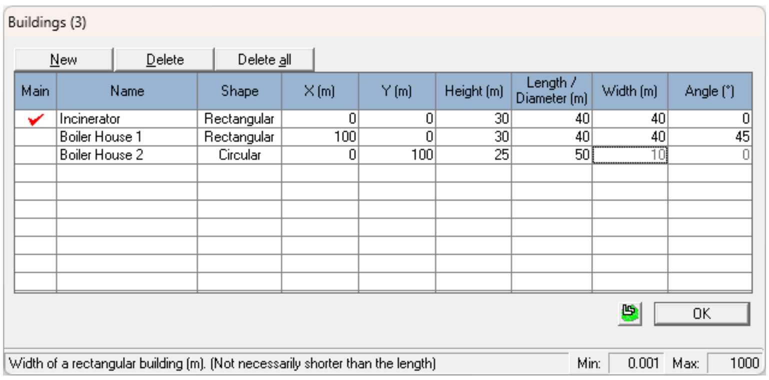


Figure 4.43 − The **Buildings** screen.

The user may add or remove buildings in the table using the **New**, **Delete** and **Delete all** buttons, respectively. There is also a right-click option to **Delete all but this** building from the table. At least one building and up to 25 may be specified, each defined by the following parameters.

1. **Main**: The user must define one 'main building' by placing a tick in this column. This is likely to be the building that has the most significant effect on dispersion. To change the main building, double click in the **Main** column or type **Y** or **N** to display or remove the tick.

   Default = a tick appears next to the first building created.

   Note that the main building defined in this screen is the default main building. Source-specific main buildings are defined on the **Source** screen (refer to Section 4.15.3 for further details).

2. **Name**: The model will use a default name when a new building is added (e.g. Building001). The user is advised to change this to something more meaningful (e.g. boiler house). Building names must be no more than 30 characters and must not contain commas.

3. **Shape**: Shape of the building, either **Rectangular** or **Circular**. To change the shape of the building click on the cell.

   Default = **Rectangular**.

4. **X (m)**: X coordinate of the centre of the building
   Y (m): Y coordinate of the centre of the building

   Minimum = -9 999 999 m
   Maximum = 9 999 999 m
   Default = 0 m

   The large maximum value allows the user to input UK National Grid coordinates or worldwide UTM (Universal Transverse Mercator) coordinates. Note that the building position should be specified in the same coordinate system as the source position and output grid.

5. **Height (m)**: Height of the building.

   Minimum = 0.001 m
   Maximum = 500 m
   Default = 10 m

6. **Length (m)**: Length of a rectangular building or diameter of a circular building. For a rectangular building this is simply one horizontal dimension.

   Minimum = 0.001 m
   Maximum = 1 000 m
   Default = 10 m

7. **Width (m)**: Width of a rectangular building, not necessarily smaller than the length. This parameter is not used for a circular building.

   Minimum = 0.001 m

Maximum = 1 000 m
Default = 10 m

8. **Angle (°)**: The angle between north and the previously defined **Length**, measured clockwise from north. This is not used for a circular building. See Section 9.18 for a diagram and more information.

   Minimum = 0°
   Maximum = 360°
   Default = 0°

---

*To duplicate an existing building right-click on the name of the building and select **Copy**.*

*To delete all buildings except one, right-click on the name of that building and select **Delete all but this**.*

4.15.3 Defining the 'main building' for each source

The user may specify a 'main building' for each source, i.e. indicate which building is likely to have the most effect on dispersion from that source. **Figure 4.44** shows the **Source** screen when **Buildings** are selected in the **Setup** screen. In the **Main building** column for each source the user may select a building from the list to define as the **Main building** for that source. The list includes all the named buildings defined in the **Buildings** screen, as well as **"(Auto)"**, **"(Main)"** and **"(None)"**.

If **"(Auto)"** is chosen then the main building will be determined automatically from the available buildings. The main building may vary between different sources and different meteorological data lines.

If **"(Main)"** is chosen then the building defined as the **Main building** in the **Buildings** screen will be the main building for this source. This is the default selection.

If **"(None)"** is selected then building effects will not be modelled for this source. This might be useful, for example, where the user knows that a source lies outside the region of influence of all buildings and wishes to keep run-times to a minimum.

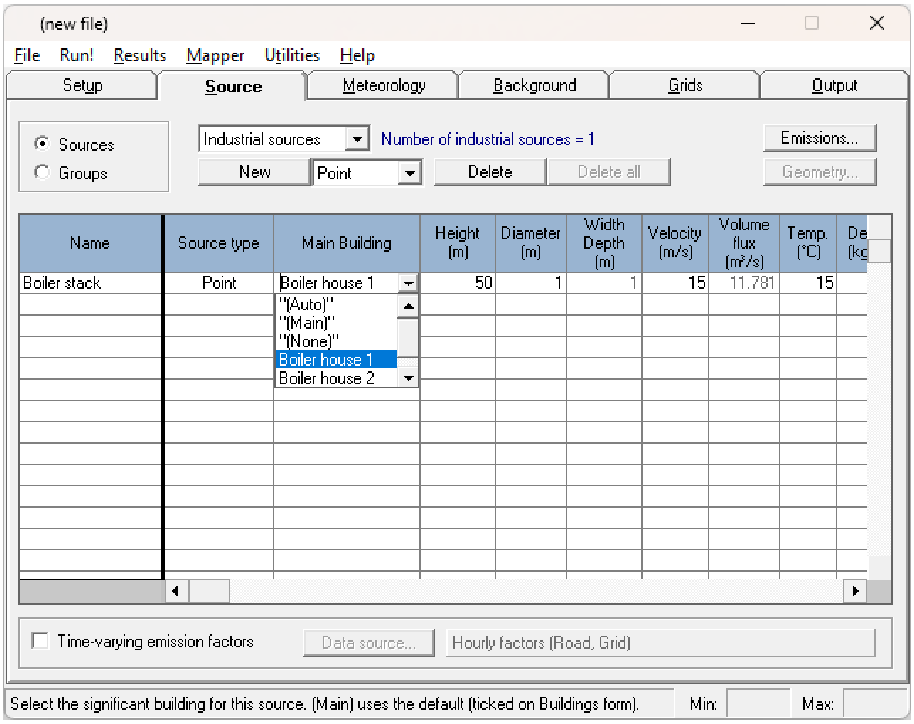


Figure 4.44 – The **Source** screen when a building is present.

4.15.4 Effective building wind directions

When modelling buildings ADMS-Urban combines a subset of the user entered buildings into a single effective building for each source and meteorological data line, see Section 9.18. During a model run ADMS-Urban outputs a description of each of these effective buildings (for the first 24 lines of meteorological data) to the *.bef* file, see Section 6.1. During a verification run a representative *.bef* file is produced based on neutral meteorological conditions and a range of wind directions. By default, these wind directions are 10° to 360° in 10° increments.

To alter this list of wind directions an *.uai* file should be created or the existing *.uai* file edited, see Section 3.1.8, so the **Effective building wind directions** section (keyword BEFWINDDIRECTIONS) is enabled, **Figure 4.46**. Enter the wind directions for which an effective building should be calculated into the table.

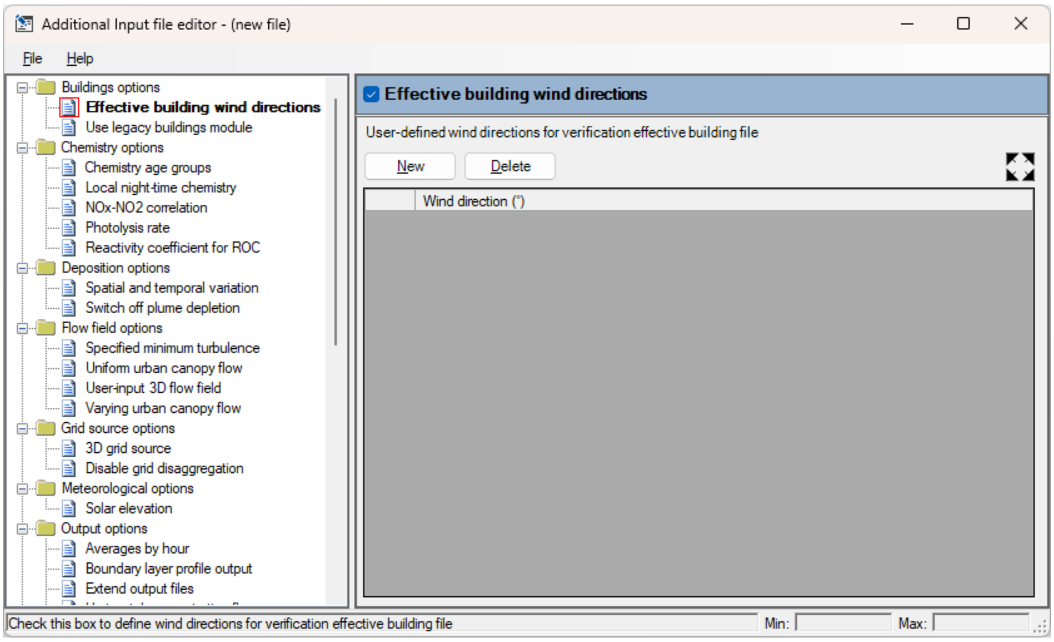


Figure 4.45 – Effective building wind directions section in the **Additional Input file editor**.

4.15.5 Use legacy buildings module

ADMS-Urban 5.1 introduced a range of changes to the way the dispersion around buildings is modelled, see the *What's New in ADMS-Urban 5.1* document for details. While it is recommended to use this new dispersion behaviour, for continuity with older results an option to revert to the ADMS-Urban 5 dispersion methodology is available.

To enable this option an *.uai* file should be created or the existing *.uai* file edited, see Section 3.1.8, so the **Use legacy buildings module** section (keyword LEGACYBLDBEHAVIOUR) is enabled, **Figure 4.46**. No further parameters are required for this section.

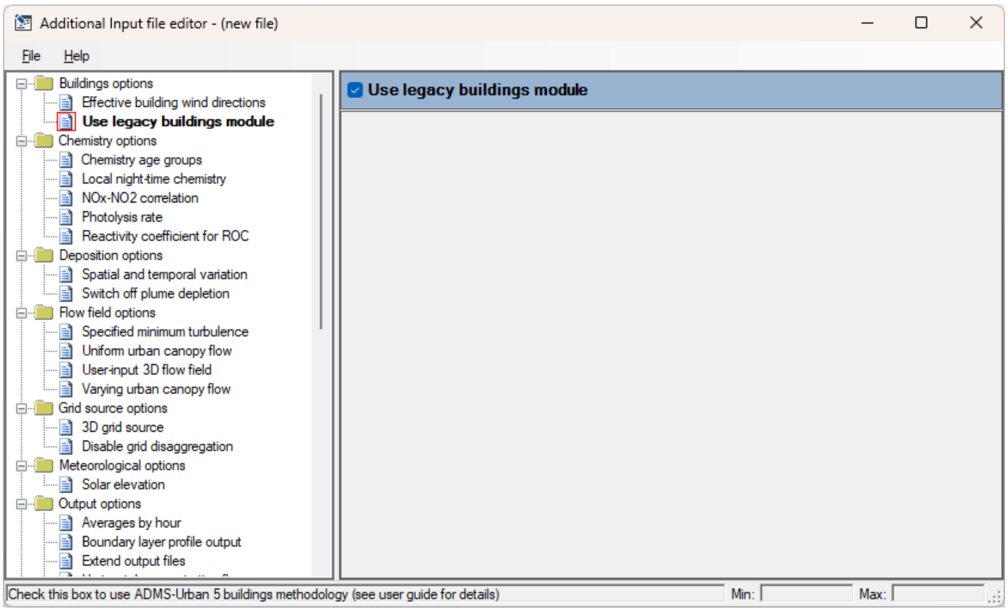


Figure 4.46 – Use legacy buildings module section in the **Additional Input file editor**.

4.15.6 Guidance

Wherever possible, a complicated site with a large number of buildings should be simplified. The user should consider the building or buildings to which the sources are attached or to which they are nearest.

A few hints and tips for modelling buildings are listed below.

- Buildings such that the building height, H, is less than a fraction ($1/\alpha$) of the source height are ignored when setting up the effective building, so need not be entered. Where $\alpha = 1 + 2*min(1,W/H)$, where W is its crosswind width.
- Where there is a cluster of similar buildings on a site, such as a row of warehouses, the user may decide to enter them as a single building.
- If there are a large number of buildings on a large site, the user should consider whether to include those that are nearest to/attached to the sources and/or those that will have the greatest effect on dispersion (tallest/largest), or consider a higher surface roughness, which can be entered in the **Meteorology** screen, as a means of representing the buildings in a complex site.
- The effect of buildings on dispersion can only be modelled for point sources. Other types of sources may be included in the modelling run, but the effect of the buildings on dispersion from these sources will not be modelled. In order to model building effects for a line, area or volume source, the source should be modelled as a number of point sources.
- Sources inside buildings cannot be modelled, though sources on the sides or roof of a building can be modelled.

4.15.7 Restrictions

The buildings option cannot be used with the **Coastline** or **Urban canopy flow** options.

4.16 Complex terrain

For many urban sites located in areas of complex terrain, the effect of the terrain height and/or the spatial variation of surface roughness on plume dispersion should be considered.

The effect of complex terrain is modelled by changing the plume trajectory and dispersion to account for disturbances in the air flow due to the terrain. This may increase or decrease the concentrations calculated.

It is not always necessary to include the effects of surrounding terrain in a modelling calculation: usually it is advisable to include terrain height effects if the gradients exceed 1:10 in the model domain. The model can typically be used for gradients up to about 1:2 but may not be reliable close to isolated slopes with higher gradients or more generally if large parts of the modelling domain have slopes greater than 1:2. The influence of the terrain will vary with the source height and position and the local meteorology.

When modelling the effect of complex terrain, the input flow parameters (wind speed and wind direction) in the meteorological input data should be representative of the oncoming (ideally unperturbed) flow field upwind of the modelling region.

When modelling spatially varying terrain height an additional minimum turbulence can be applied based on the variation in terrain height, see Section 4.19 for a method of supplying a user defined minimum turbulence value.

4.16.1 Terrain height and surface roughness files

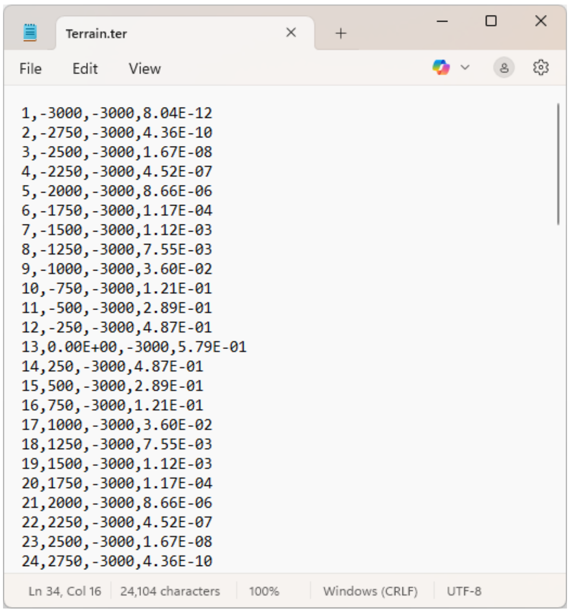
To run the **Complex terrain** option, additional data file(s) are required which contain the spatially varying terrain height and/or surface roughness data; these are called the *.ter* and *.ruf* files respectively. The file(s) should consist of a series of lines with

```
N, X, Y, Z
```

entered on each line, where `N` is an incrementing counter for each line, `X` is the X coordinate of the data point, `Y` is the Y coordinate of the data point, and `Z` is:

- the height of the terrain (in metres) at that data point for the *.ter* file (see example in **Figure 4.47**), and
- the surface roughness value (in meters) at that data point in the *.ruf* file.

The .ter/.ruf files should be comma-separated and the maximum number of data points that may be included in each file is 770,000.




```
1,-3000,-3000,8.04E-12
2,-2750,-3000,4.36E-10
3,-2500,-3000,1.67E-08
4,-2250,-3000,4.52E-07
5,-2000,-3000,8.66E-06
6,-1750,-3000,1.17E-04
7,-1500,-3000,1.12E-03
8,-1250,-3000,7.55E-03
9,-1000,-3000,3.60E-02
10,-750,-3000,1.21E-01
11,-500,-3000,2.89E-01
12,-250,-3000,4.87E-01
13,0.00E+00,-3000,5.79E-01
14,250,-3000,4.87E-01
15,500,-3000,2.89E-01
16,750,-3000,1.21E-01
17,1000,-3000,3.60E-02
18,1250,-3000,7.55E-03
19,1500,-3000,1.12E-03
20,1750,-3000,1.17E-04
21,2000,-3000,8.66E-06
22,2250,-3000,4.52E-07
23,2500,-3000,1.67E-08
24,2750,-3000,4.36E-10
```

Figure 4.47 − The start of the terrain file *terrain.ter*.

The data points do not need to be regularly spaced, but the area described should be rectangular. Note that the coordinate system used should be the same as that in which the sources, output points, etc., are defined. An example *terrain.ter* file describing a bell-shaped hill is supplied in the *<install_path>\Data* directory; the first few lines of this file are shown in **Figure 4.47**. This file may be edited to create new *.ter* files using a desktop editor. Section 7.6 provides further details of how to create new terrain files using the **ADMS Terrain Converter** utility and Section 6.1 of the Mapper User Guide (which can be accessed from the **Help** menu of the Mapper) provides details on using the Mapper to create terrain files. An example *ROUGHNES.RUF* file is also supplied in the *<install_path>\Data* directory.

The surface roughness file should cover the same area as the terrain file. An easy way to create a roughness file is to edit the Z values of the terrain file being used so that the same domain, with the same X and Y coordinates, is used for both files. This can be carried out using a spreadsheet package such as Microsoft Excel.

The values in the terrain file (.ter*) should not all be equal, or the program will fail.*

4.16.2 Modelling the effects of Complex terrain

To model the effects of hills, go to the **Setup** screen. Select **Complex terrain** from the model options and click on **Data...** to go to the **Complex terrain** screen shown in **Figure 4.48**.

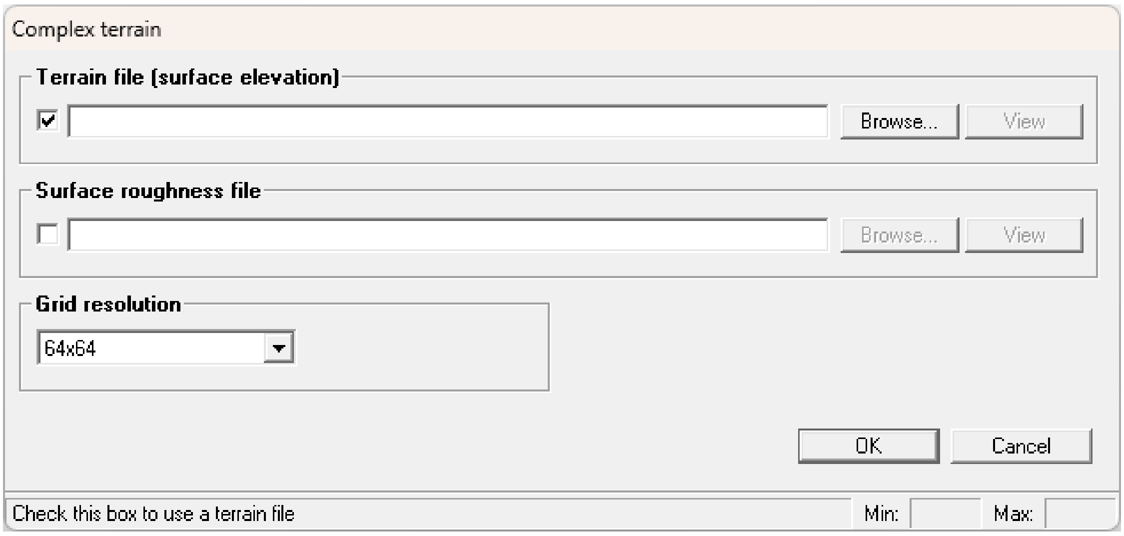


Figure 4.48 − The **Complex terrain** screen.

1. **Terrain file (surface elevation)**: Select the check box if this option is required.

   In the text box, type the full path name of the required terrain file, or use the **Browse** button to find available *.ter* files, or drag and drop a *.ter* file from Explorer. The **View** button allows the current *.ter* file to be viewed; left click to open the file in the default application (as selected via the **File**, **Preferences, Viewing options** menu item) or right-click to choose between all available applications

2. **Surface roughness file**: Select the check box if this option is required.

   In the text box, type the full path name of the required surface roughness file, or use the **Browse** button to find available *.ruf* files, or drag and drop a *.ruf* file from Explorer. The **View** button allows the current *.ruf* file to be viewed; left click to open the file in the default application (as selected via the **File, Preferences, Viewing options** menu item) or right-click to choose between all available applications

3. **Grid Resolution**: The following choices of internal calculation grid sizes are offered:

   * **512 x 512** – this option should only be used if the complex terrain input data are highly resolved, and cover large or complicated areas of terrain; run times may be prohibitive.
   * **256 x 256** – again, only use this option if the complex terrain input data are highly resolved.
   * **128 x 128**
   * **64 x 64** – this is the default
   * **32 x 32**
   * **16 x 16** - for testing only

For most calculations the 32 x 32 or 64 x 64 grids are sufficient.

Note that the grid resolution here refers to the internal grid used to calculate the flow field and is different to that in the **Grids** screen, which refers to the grid on which concentrations are output. The terrain grid must always cover a larger area than the output grid.

4.16.3 Restrictions

The complex terrain option may not be used with the **Coastline** or **Urban canopy flow** options.

The geographical extent covered by the terrain data must be larger than the modelling region, where the modelling region comprises the region covering all of the industrial and road sources, buildings, specified output points and the output grid, extended by the maximum dimension of any source. It is not necessary for the terrain data to be larger than the grid source. Specifically:

- For runs with a wind sector size of less than or equal to 15° the terrain region must be at least 15% larger than the modelling region.
- For runs with a wind sector size equal to or greater than 15° the terrain region must be at least 50% larger than the modelling region.
- Additionally the terrain region must be at least 500 m larger than the modelling region in each direction.

If the terrain requirements are not met then the model will stop with a message giving the minimum region that must be covered by the terrain to run with the current road and industrial sources, buildings, output points and model options.

4.17 User-input 3D flow field data

In ADMS-Urban, users can input their own 3D flow fields to be used in place of the standard flat terrain or hills flow fields. For example, users can enter flow fields output from a mesoscale meteorological model or a CFD program.

Details of the format of the flow field data that can be input into the ADMS-Urban model are given in Section 4.17.1 below; Section 4.17.2 outlines how these data can be included in a model run using an *.aai* file. Section 4.17.3 gives some guidance on the use of user-input 3D flow field data with ADMS-Urban, and Section 4.17.4 summarises some restrictions of this option.

4.17.1 Format of the user-input 3D flow field file

When entering user-input 3D flow fields it is necessary to enter values for three velocity components and additionally either the turbulent kinetic energy or three turbulence components.

The tab- or comma-separated flow field data should be in a text file with the following format (there is no header row):

```
NX NY NZ
X Y Z U V W TKE
…
```

or

```
NX NY NZ
X Y Z U V W SigU SigV SigW
…
```

On the first row, `NX` is the number of grid points in the X horizontal direction, `NY` is the number of grid points in the Y horizontal direction, and `NZ` is the number of grid points in the vertical direction.

On the second and subsequent rows, `X`, `Y`, and `Z` are the coordinates of the grid point in metres, using the same coordinate system as the rest of the modelling input. `Z` should be measured in metres from the ground. `U`, `V`, and `W` are the X, Y and Z components of the wind velocity (m/s). `TKE` is the turbulent kinetic energy (m^2/s^2). `SigU`, `SigV`, and `SigW` are the X, Y and Z components of the turbulence velocity (m/s).

The data must be ordered as follows: data for each (x,y) location must be grouped together, with corresponding vertical heights being distinct, and increasing in magnitude.

The first few lines of an example file using `TKE` are shown in **Figure 4.49**.

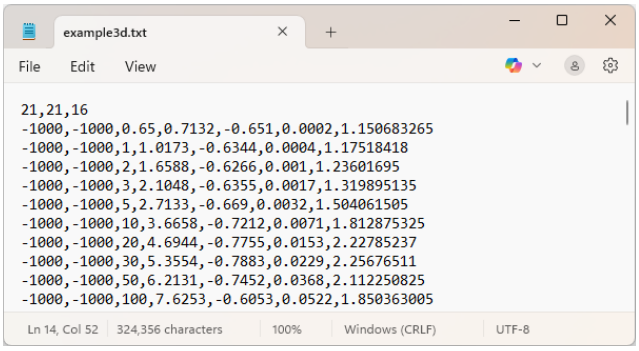


Figure 4.49 – Example user-input 3D flow field file using `TKE`.

4.17.2 Additional input file

To use user-input 3D flow field data create a new additional input file or edit an existing additional input file (see Section 3.1.8), so the **User-input 3D flow field** option (keyword MESO) is enabled (**Figure 4.50**).

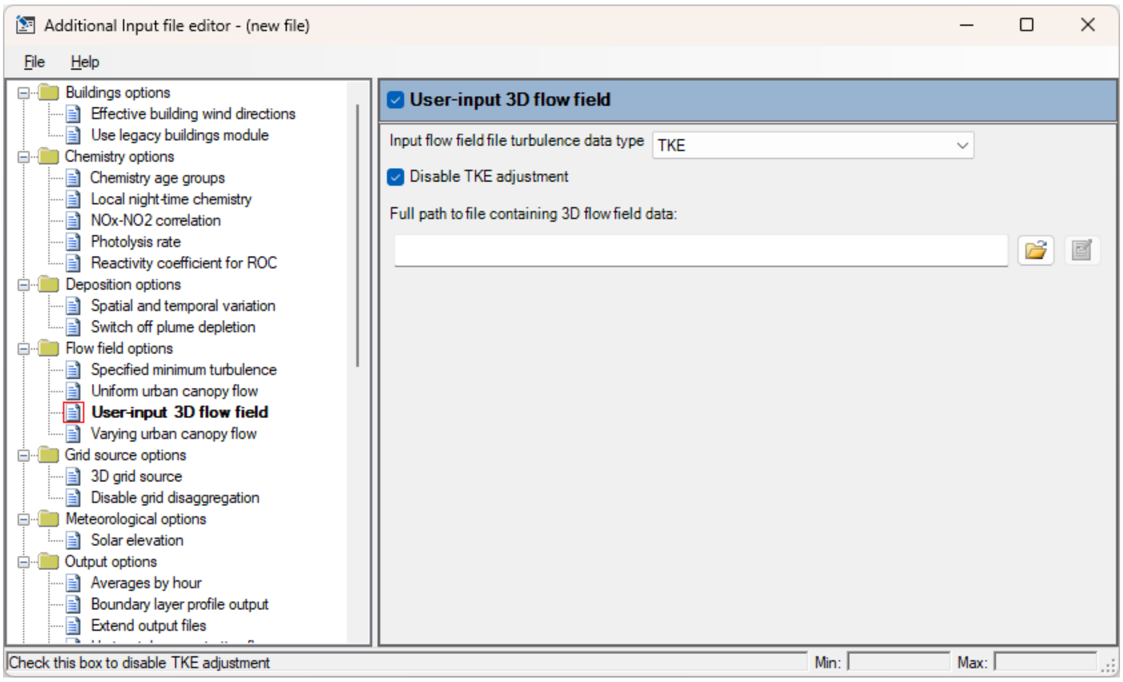


Figure 4.50 – **User-input 3D flow field** option enabled in the **Additional Input file editor**.

The type of turbulence data being entered in the file should be selected, either:

- **TKE** if turbulent kinetic energies are being entered or
- **Turbulent velocities** if turbulent velocities are being entered.

If the **TKE** option is selected, it is also possible to disable the adjustments by selecting the **Disable TKE adjustment** check box. The path to the file containing the 3D flow field data must also be entered. Use the  button to browse to the file and the  button to view the file in the default application, as selected by the **File\Preferences...\Viewing** menu option in the **Additional input file editor**.

4.17.3 Guidance

All input values are used without modification in the model apart from `TKE`, unless the **Disable TKE adjustment** check box is ticked, which is adjusted to account for the fact that mesoscale numerical models have a tendency to predict `TKE` values that are substantially lower than those typically used in ADMS-Urban. Further details of this adjustment are given in Section **9.20**.

Users should note that:

- In addition to the 3D flow field, the user must provide a standard ADMS-Urban *.met* file containing the meteorological conditions upstream of the domain covered by the 3D flow field.
- The user-input 3D flow field data should include data for at least one height below 50 m and at least three heights below 100 m. Higher vertical resolution in the region of the source height will improve modelling results.

4.17.4 Restrictions

There are a number of restrictions with the user-input 3D flow field option:

- User-input 3D flow field data can only be used for short-term runs with one line of meteorological data.
- Using the notation introduced above, the maximum number of distinct (x,y) points, `NX×NY`, is 1000. At each (x,y) point, up to 50 vertical points (`NZ`) are allowed.
- The model assumes that the flow field data are defined on a rectangular grid, although this grid does not have to be north-south / east-west aligned.
- The geographical extent covered by the flow field data must be larger than the modelling region, where the modelling region comprises of all of the sources, buildings, specified output points and the output grid. Specifically, the flow field data must cover a region at least 10% larger than the modelling region, extended by the maximum dimension of any source. Additionally the region covered by the flow field data must be at least 500 m larger than the modelling region in each direction.
- If the flow field data does not cover a sufficiently large area then the model will stop with a message giving the minimum region that must be covered by the flow field data to run with the current sources, buildings, output points and model options.
- The **Complex terrain** module cannot be used at the same time as user-input 3D flow field data.

4.18 Urban canopy flow

ADMS-Urban offers two urban canopy flow options for calculating changes in vertical profiles of velocity and turbulence by the presence of buildings in an urban area: a spatially varying option (see Section 4.18.1) and a homogeneous option (see Section 4.18.2). It is generally accepted that as wind approaches a built-up urban area, the profile is displaced vertically by a height related to the mean height of the buildings, while the flow within the building canopy is slowed by the buildings (Belcher *et al.* 2013).

4.18.1 Spatially varying urban canopy flow option

To use the spatially varying urban canopy flow field in a run, first create an additional input file or edit an existing additional input file (refer to Section 3.1.8) so that the **Spatially varying urban canopy flow** option (keyword URBANCANOPYFLOW) is enabled, as shown in **Figure 4.51**.

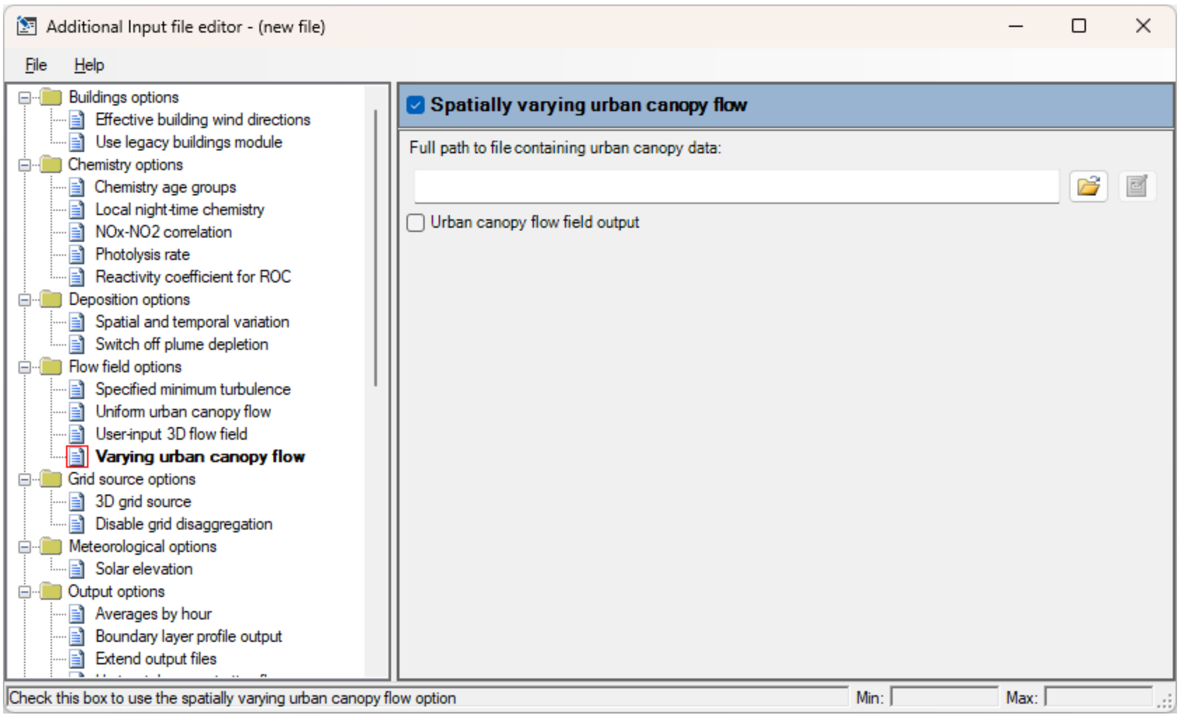


Figure 4.51 – The **Spatially varying urban canopy flow** section enabled in the **Additional Input file editor**.

The user must specify:

- **Full path to file containing urban canopy data**: Use the  button to browse to an input file containing gridded urban canopy data. Use the  button to view the file in the default application, as selected by the **File\Preferences...\Viewing** menu option in the **Additional input file editor.** Refer to Section 4.18.1.1 for details of the input file format and contents;
- **Urban canopy flow field output**: Whether the model should write values of velocity and turbulence at user-specified output locations to files; please refer to

Sections 6.1.17 to 6.1.22 for details of the output flow field files and variables.

4.18.1.1 Input data file format

The structure of the input data file for the urban canopy flow option is similar to the ADMS meteorological file format, with a list of variable names followed by a table of data. At the beginning of the data file, an additional section specifies the grid layout of the urban canopy data. An example data file is shown in **Figure 4.52**, which is also supplied in the *<install_path>\Data* directory.

The Mapper contains a tool to create urban canopy data files from a buildings data file, with a choice of grid layout and lambdaF wind directions. Please refer to Section 6.4 of the Mapper User Guide for more details.

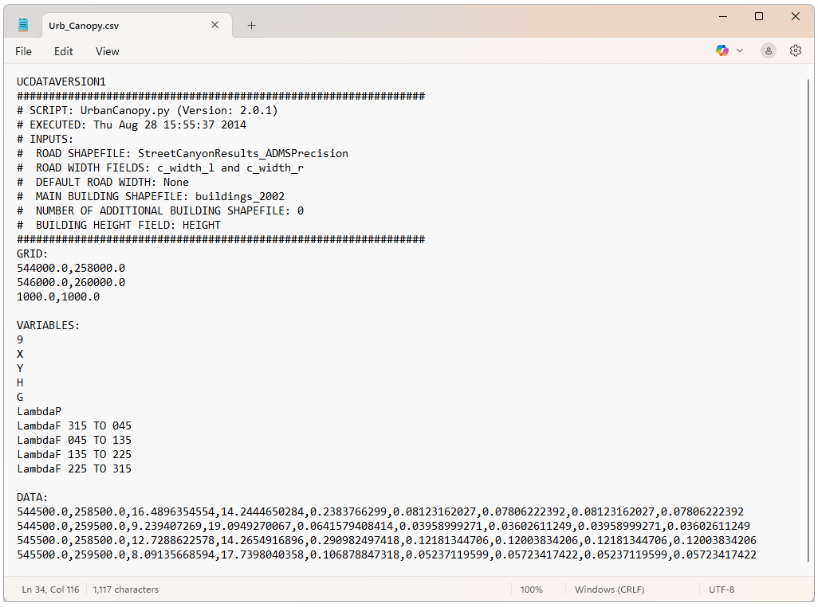
The input variables used to determine the urban canopy flow parameters in each cell of a regular x, y urban canopy grid, which may have the same layout as the ADMS-Urban grid source but is not required to do so, are as follows:

- `H` average building height within the cell
- `G` average street canyon width within the cell
- `LambdaP` ratio of plan area occupied by buildings to total plan area within the cell (must be between 0 and 1)
- `LambdaF phi1 to phi2` ratio of total frontal area of buildings perpendicular to a wind direction to total plan area within the cell (may be greater than 1), for a range of wind directions phi1 to phi2 in degrees

There must be one line of data for each urban canopy grid cell. If there are no buildings within a particular urban canopy grid cell, the average building height and value of lambdaP will be zero and the urban canopy flow field calculations will not be run, so upstream flow parameters will be used in this cell.

The LambdaF angle thresholds have the following restrictions:

- The upper (‘to’) threshold of one sector must be equal to the lower (‘from’) threshold of the next (sectors must be in order).
- The first ‘from’ threshold must be equal to the last ‘to’ threshold, to describe a complete range of wind directions. Wind directions will be considered to belong to the sector where they are greater than or equal to the ‘from’ threshold and less than the ‘to’ threshold.
- Each threshold value must be greater than or equal to 0 and less than 360 degrees.
- Exactly one sector may have a ‘from’ threshold greater than or equal to its ‘to’ threshold (crossing 0 degrees).
- The number of sectors must be between one and forty.



```
UCDATAVERSION1
#################################################################
# SCRIPT: UrbanCanopy.py (Version: 2.0.1)
# EXECUTED: Thu Aug 28 15:55:37 2014
# INPUTS:
#  ROAD SHAPEFILE: StreetCanyonResults_ADMSPrecision
#  ROAD WIDTH FIELDS: c_width_l and c_width_r
#  DEFAULT ROAD WIDTH: None
#  MAIN BUILDING SHAPEFILE: buildings_2002
#  NUMBER OF ADDITIONAL BUILDING SHAPEFILE: 0
#  BUILDING HEIGHT FIELD: HEIGHT
#################################################################
GRID:
544000.0,258000.0
546000.0,260000.0
1000.0,1000.0

VARIABLES:
9
X
Y
H
G
LambdaP
LambdaF 315 TO 045
LambdaF 045 TO 135
LambdaF 135 TO 225
LambdaF 225 TO 315

DATA:
544500.0,258500.0,16.4896354554,14.2444650284,0.2383766299,0.08123162027,0.07806222392,0.08123162027,0.07806222392
544500.0,259500.0,9.239407269,19.0949270067,0.0641579408414,0.03958999271,0.03602611249,0.03958999271,0.03602611249
545500.0,258500.0,12.7288622578,14.2654916896,0.290982497418,0.12181344706,0.12003834206,0.12181344706,0.12003834206
545500.0,259500.0,8.09135668594,17.7398040358,0.106878847318,0.05237119599,0.05723417422,0.05237119599,0.05723417422
```


Figure 4.52 – Example Urban canopy data input file

The file format is as follows, where all starred items are required:

- ‘`UCDATAVERSION1`’ Version string, must be the first line in the file
- (Comments may be included before the first keyword)
- Grid information section
  - * ‘`GRID:`’ Keyword
  - * `XMIN, YMIN` Minimum x,y coordinates of the urban canopy grid in units of metres, in a coordinate system consistent with all other spatial inputs to the ADMS-Urban run
  - * `XMAX YMAX` Maximum x,y coordinates of the urban canopy grid in units of metres, in a coordinate system consistent with all other spatial inputs to the ADMS-Urban run
  - * `XSPACING, YSPACING` The resolution of the urban canopy grid in metres
- (Comments may be added between sections)
- Variable information section
  - * ‘`VARIABLES:`’ Keyword
  - * `nVar` number of variables included in the file
  - * List of nVar variable names, each on a separate line: must include X, Y, G, H, LambdaP, and at least one sector of LambdaF, may include other variables eg. labels. Refer to additional information above about definitions of variables and restrictions on LambdaF sectors
- (Comments may be added between sections)
- Data section
  - * ‘`DATA:`’ Keyword

* One line per urban canopy grid cell of comma-separated data values for nVar variables, in the order listed in the Variable information section

Blank lines or comments may be included before the first section, between sections or after the last section. There should be no blank lines or comments within each section

4.18.2 Uniform urban canopy flow option

To use the uniform urban canopy flow field in a run, first create an additional input file or edit an existing additional input file (refer to Section 3.1.8) so that the **Uniform urban canopy flow** option (keyword URBANCANOPYFLOW_ONECELL) is enabled, as shown in **Figure 4.53**.

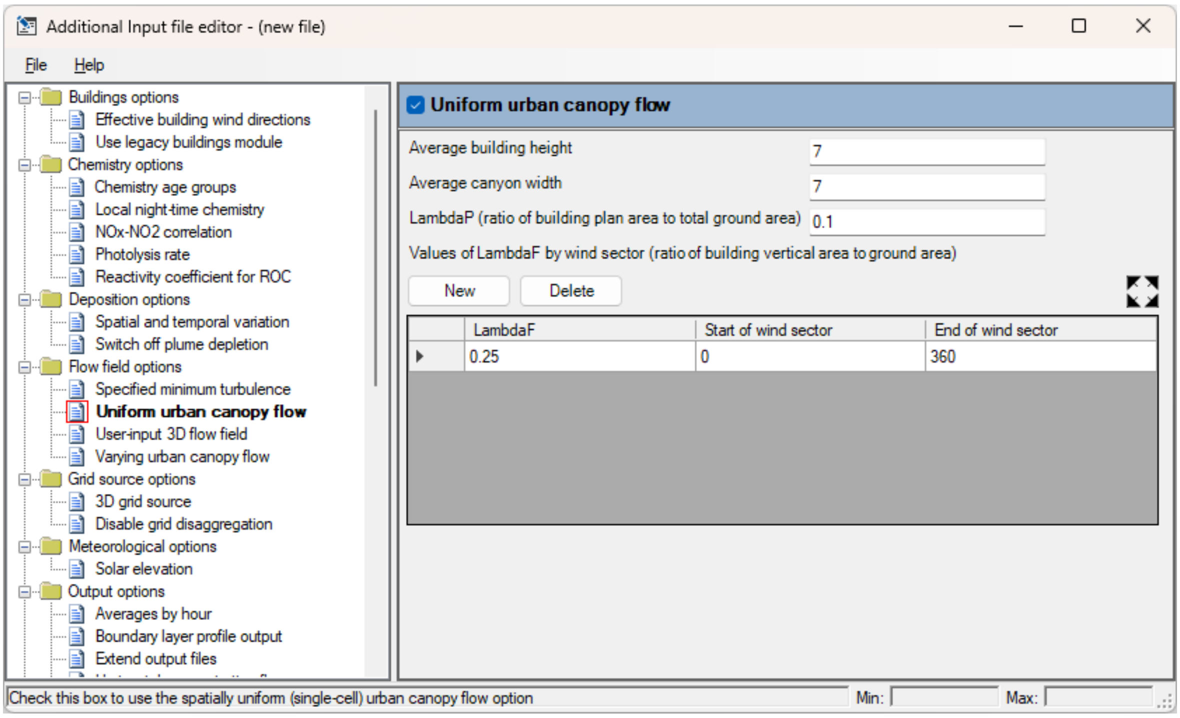


Figure 4.53 – The **Uniform urban canopy flow** section enabled in the **Additional Input file editor**.

The user must specify **Average building height**, **Average canyon width**, **LambdaP** and the values of **LambdaF** for specified wind direction sectors (refer to Section 4.18.1.1 for details on these parameters).

4.18.3 Restrictions

Only one of the **Spatially varying urban canopy flow** or **Uniform urban canopy flow** options can be selected at a time and none of these options can be used in conjunction with the **Complex Terrain**, **Buildings** or **Coastline** options.

4.19 User-defined minimum turbulence

As part of the dispersion calculations, a minimum turbulence value is applied. This minimum turbulence value varies between 0.01 m/s and 0.2 m/s depending on the minimum value of the Monin-Obukhov length specified in the **Advanced dispersion site data** screen; see **Figure 3.23**. Additionally, a further minimum value between 0 and 0.1 m/s is applied if variable terrain height is being modelled depending on the terrain height variation in the file.

An option exists to select how the minimum value of the turbulence parameters should be calculated. First create a new additional input file or edit an existing additional input file (refer to Section 3.1.8) so the **Specified minimum turbulence** option (keyword DEFINEMINTURBULENCE) is enabled, as shown in **Figure 4.54**. Then select which of these to use: **Minimum LMO**, **Terrain file** or **Minimum turbulence value** and enter a value.

If multiple options are selected, the maximum value of the selected one will be used.

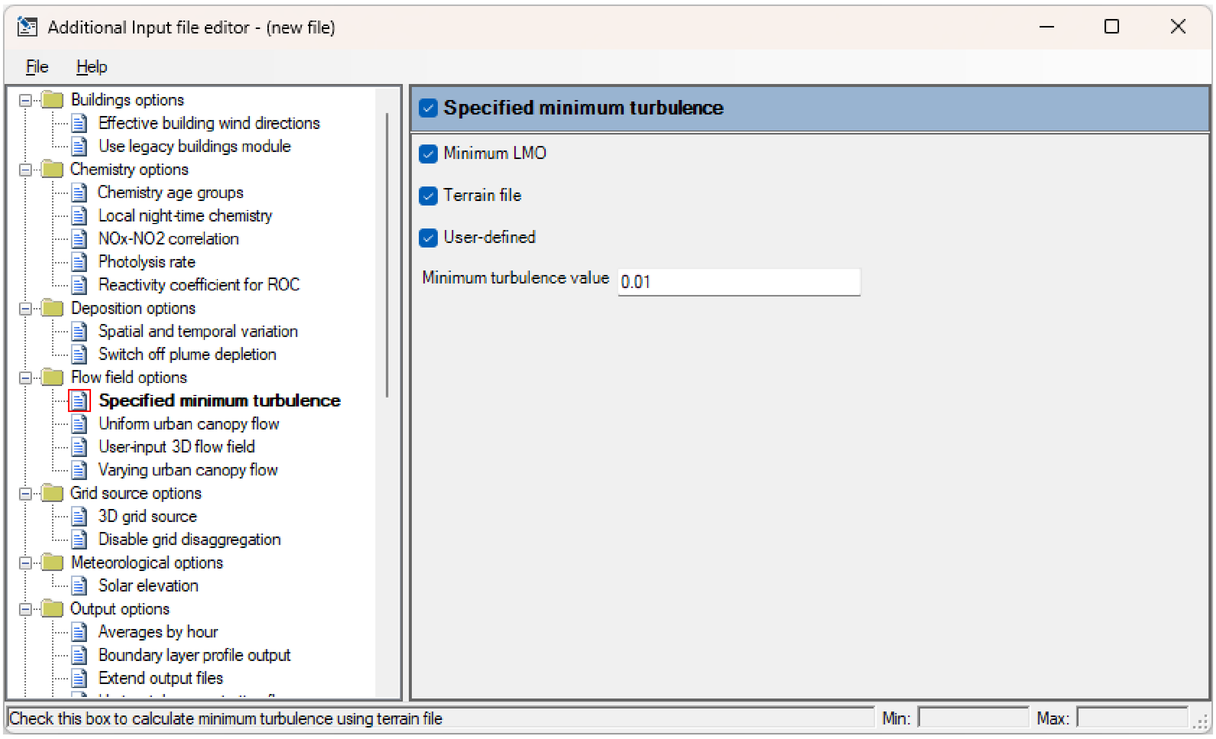


Figure 4.54 – The **Specified minimum turbulence** section in the **Additional Input file editor**.

4.20 Coastline

The effect of a coastline will generally only be significant for emissions from elevated point sources that are within a few kilometres (up to a maximum of 5 km) of the coast. The coastline module is only invoked when:

- the wind is blowing onshore (from the sea to the land),
- the land is warmer than the sea,
- the meteorological conditions over the land are unstable (convective).

When the **Coastline** option is selected, these conditions are checked automatically by the model for each line of meteorological data used in the calculation. If they are not satisfied then the coastline module is disabled for that meteorological data line. In addition to the usual meteorological data variables, the meteorological file used when modelling coastline effects must contain either:

`DELTAT`: the difference in temperature (in degrees) between the sea surface temperature and the near surface temperature over land (e.g. as in *coast.met*, supplied in the *<install_path>\Data* directory); **or**

both `TSEA` (the sea surface temperature) and `T0C` (the surface temperature over land).

It is not possible to enter meteorological data by hand when modelling coastal effects because the necessary sea temperature data above are not included in the table.

To model coastal effects, select the **Coastline** option on the **Setup** screen and click **Data...** to go to the **Coastline** screen shown in **Figure 4.55**. The coastline position and orientation (sense) is specified by means of the coordinates of two points lying on the coastline and one point on the land. The model will assume that the coastline is a straight line between the two points on the coastline. This is illustrated in **Figure 4.56**.

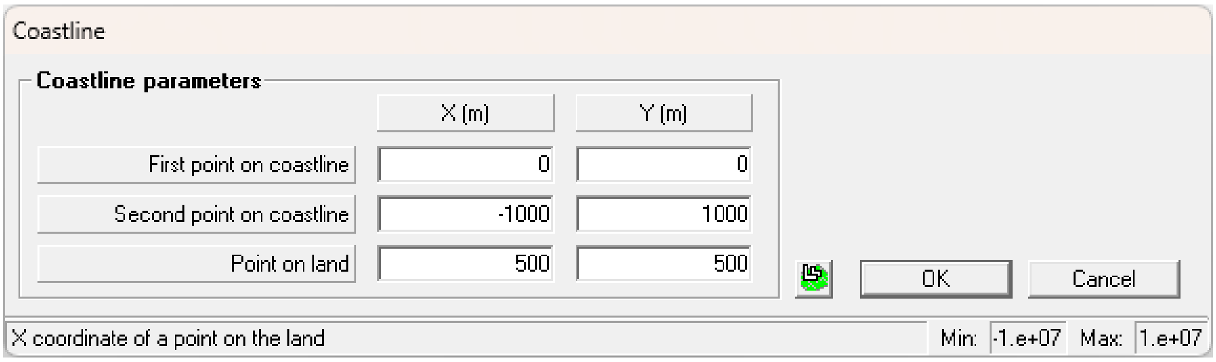


Figure 4.55 − The **Coastline** screen.

The following parameters should then be defined.

1. **First point on the coastline**: X and Y coordinates of the first point on the coastline

   Minimum = -9,999,999 m
   Maximum = 9,999,999 m

Default = (0,0)

2. **Second point on the coastline**: X and Y coordinates of the second point on the coastline

   Minimum = -9,999,999 m
   Maximum = 9,999,999 m
   Default = (-1000,1000)

3. **Point on land**: X and Y coordinates of a point on land (X_L, Y_L in **Figure 4.56**)

   Minimum = -9,999,999 m
   Maximum = 9,999,999 m
   Default = (500,500)

The coordinate system used to define the coastline should be the same as that used for the rest of the model input file.

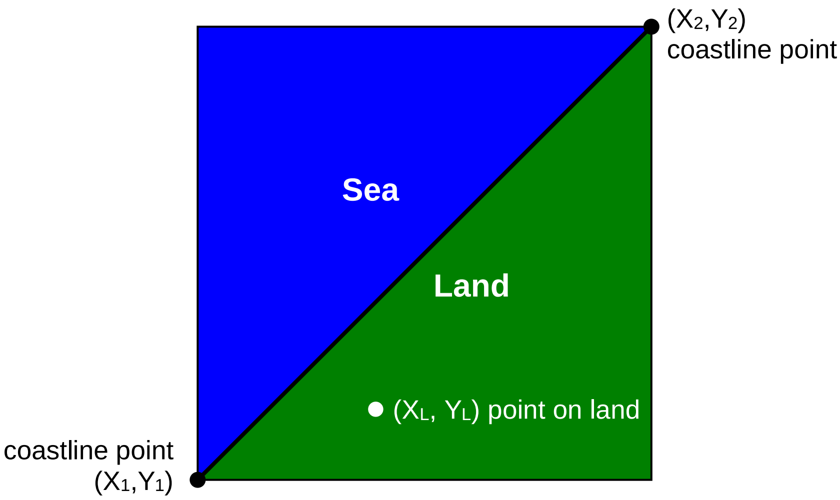


Figure 4.56 − An example of the coastline coordinates to be specified.

Section 9.21 and Technical Specification document P15/01 (CERC, 2025) provide more technical details about this option.

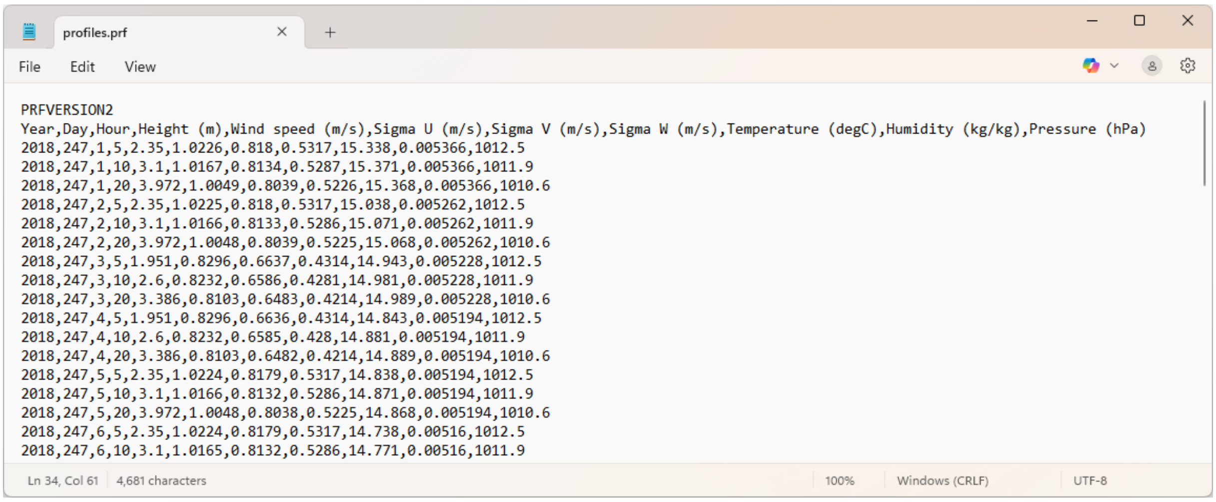
4.20.1 Restrictions

The coastline option cannot be used with the **Buildings**, **Complex Terrain** or **Urban canopy flow** options.

4.21 User-input vertical profiles of meteorological data

User-defined vertical profiles of wind speed, turbulence, temperature and specific humidity can be entered. These data are entered using a *.prf* file, and then interpolated for use by the model using the algorithm outlined in Section 9.1.

To use this option first select the **Vertical profiles** option on the **Meteorology** screen, then **Browse…** to select the *.prf* file required, or drag and drop the *.prf* file from Explorer.




```
PRFVERSION2
Year,Day,Hour,Height (m),Wind speed (m/s),Sigma U (m/s),Sigma V (m/s),Sigma W (m/s),Temperature (degC),Humidity (kg/kg),Pressure (hPa)
2018,247,1,5,2.35,1.0226,0.818,0.5317,15.338,0.005366,1012.5
2018,247,1,10,3.1,1.0167,0.8134,0.5287,15.371,0.005366,1011.9
2018,247,1,20,3.972,1.0049,0.8039,0.5226,15.368,0.005366,1010.6
2018,247,2,5,2.35,1.0225,0.818,0.5317,15.038,0.005262,1012.5
2018,247,2,10,3.1,1.0166,0.8133,0.5286,15.071,0.005262,1011.9
2018,247,2,20,3.972,1.0048,0.8039,0.5225,15.068,0.005262,1010.6
2018,247,3,5,1.951,0.8296,0.6637,0.4314,14.943,0.005228,1012.5
2018,247,3,10,2.6,0.8232,0.6586,0.4281,14.981,0.005228,1011.9
2018,247,3,20,3.386,0.8103,0.6483,0.4214,14.989,0.005228,1010.6
2018,247,4,5,1.951,0.8296,0.6636,0.4314,14.843,0.005194,1012.5
2018,247,4,10,2.6,0.8232,0.6585,0.428,14.881,0.005194,1011.9
2018,247,4,20,3.386,0.8103,0.6482,0.4214,14.889,0.005194,1010.6
2018,247,5,5,2.35,1.0224,0.8179,0.5317,14.838,0.005194,1012.5
2018,247,5,10,3.1,1.0166,0.8132,0.5286,14.871,0.005194,1011.9
2018,247,5,20,3.972,1.0048,0.8038,0.5225,14.868,0.005194,1010.6
2018,247,6,5,2.35,1.0224,0.8179,0.5317,14.738,0.00516,1012.5
2018,247,6,10,3.1,1.0165,0.8132,0.5286,14.771,0.00516,1011.9
```

Figure 4.57 − An example *.prf* file.

The *.prf* file should be comma-separated. Part of an example *.prf* file is shown in **Figure 4.57** to illustrate the format of the file. The full file is also supplied in the *<install_path>\Data* directory. The first line of the file is a version string in the form `PRFVERSION2`.

Note that ADMS-Urban 5.1 can read version 1 .prf files but later versions may not.

The next line is the header line which indicates which variables are to be read in. The first variables must be either '`year, day, hour`' or `MetLine` and the next must be `Height (m)`. Then the included meteorological parameters be specified; these can be any of `Wind speed (m/s)`, `Sigma U (m/s)`, `Sigma V (m/s)`, `Sigma W (m/s)`, `Temperature (degC)`, `Humidity (kg/kg)` or `Pressure (hPa)`. Some of these parameters will only effect the dispersion calculations if specific modelling options are chosen. Any parameters not included in the *.prf* file will take the standard ADMS-Urban boundary layer vertical profiles.

The following lines contain the data in the same order as the header. Data should be entered for at least two heights for every line of data in the meteorological file. The lines should be in order with the heights in ascending order for each meteorological data line. If values are unknown for any parameter then a value of -999 should be entered.

The variables which can be entered in the vertical profiles file are as follows:

1. **`Year`**: the year
2. **`Day`**: the day of the year

3. **`Hour`**: the hour of the day
4. **`MetLine`**: the meteorological line number corresponding to this line of data

*If the **Met. Data are hourly sequential** checkbox is ticked in the **Meteorology** screen, Year/Day/Hour must be used within the .prf file.*

5. **`Height`**: the height in metres (must be greater than 0 m)
6. **`Wind speed`**: the wind speed in m/s

   The wind speeds entered should be consistent with the wind speeds in the meteorological file.

   Minimum = 0 m/s
   Maximum = 100 m/s

7. **`Sigma U`**: The along-wind turbulence in m/s

   Minimum > 0 m/s
   Maximum = 1 000 m/s

8. **`Sigma V`**: The cross-wind turbulence in m/s

   Minimum > 0 m/s
   Maximum = 1 000 m/s

9. **`Sigma W`**: The vertical turbulence in m/s

   Minimum > 0 m/s
   Maximum = 1 000 m/s

10. **`Temp.`**: the temperature in °C

    Minimum = -100°C
    Maximum = 60°C

11. **`Humidity (kg/kg)`**: the specific humidity in kg/kg

    Minimum = 0 kg/kg
    Maximum = 0.1 kg/kg

12. **`Pressure (hPa)`**: the pressure in hPa

    Minimum = 0 hPa
    Maximum = 2000 hPa

4.22 Solar elevation calculation

As part of the meteorological calculations the meteorological processor calculates the solar elevation based on the time of day and day of year. As times entered in ADMS-Urban are hour ending and to best represent conditions across the whole hour the time used in the solar elevation calculation is half an hour before the time of the meteorological data line, i.e. the middle of the hour that data line represents. In some situations, ADMS-Urban may be used to model individual meteorological conditions where it would be more appropriate to use the time specified by the meteorological data line. This can be accomplished by using the **Solar elevation** option.

To enable this option an *.uai* file should be created or the existing *.uai* file edited, see Section 3.1.8, so the **Solar elevation** section (keyword SOLARELEVATION) is enabled, **Figure 4.58**. Then select **End of met. hour** to **Calculate solar elevation at**.

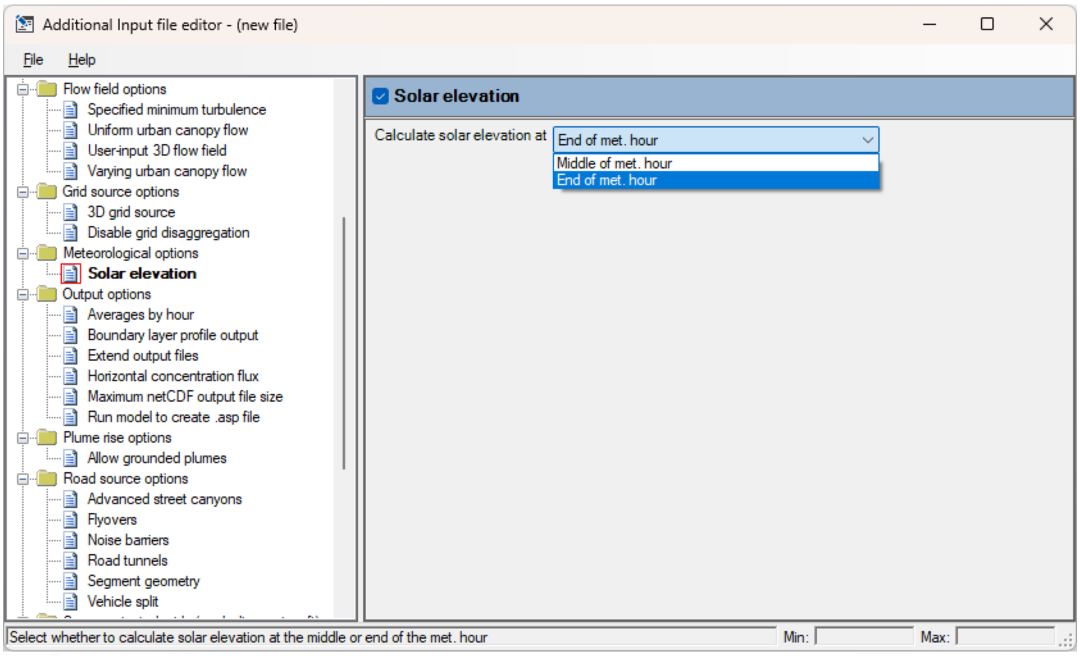


Figure 4.58 –**Solar elevation** section in the **Additional Input file editor**.

4.23 Boundary layer profile output

ADMS-Urban calculates internally a vertical profile of the boundary layer properties at the dispersion site, and an option exists to output these properties at a range of heights to a *.pro* file using an additional input file. Section 6.1.13 describes the format of the *.pro* file and Section 9.2 together with the Technical Specification document P09/01 (CERC, 2025) give more details regarding the calculation of these variables.

To output a *.pro* file, first create an additional input file or edit the existing additional input file (refer to Section 3.1.8) so that the **Boundary layer profile output** option (keyword BLPROFILE) is enabled, as shown in **Figure 4.59**.

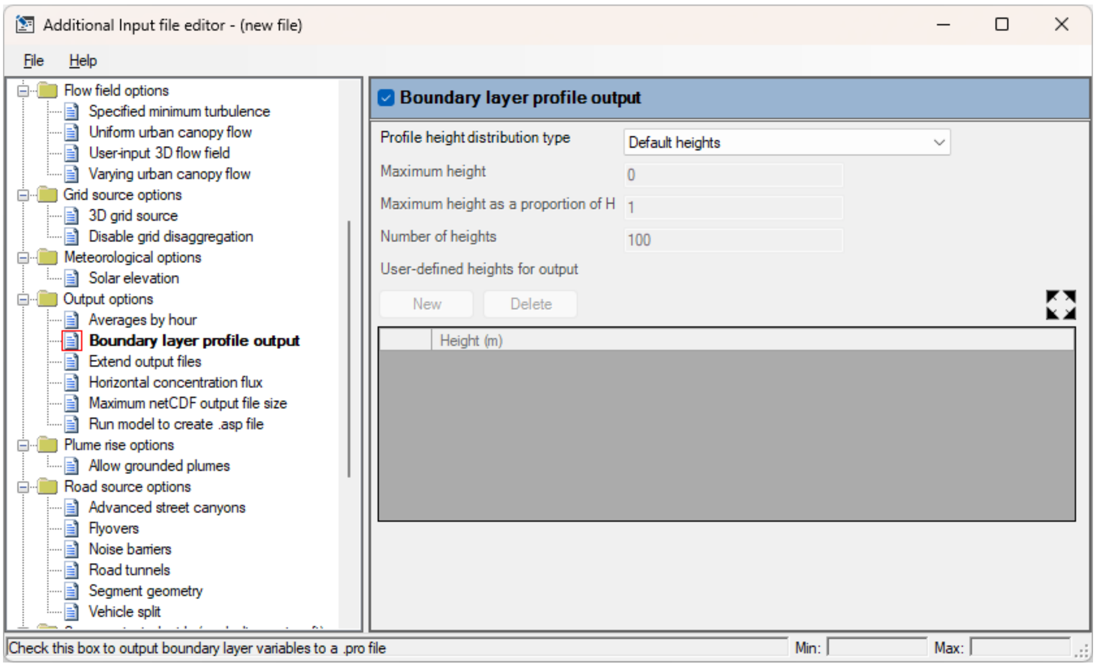


Figure 4.59 – The **Boundary layer profile output** section enabled in the **Additional Input file editor**.

A choice of the **Profile height distribution type** to use should then be made:

- **Default heights**: A geometric series of 100 heights up to the boundary layer height will be used.
- **Define absolute max height**: A geometric series of the specified number of heights up to the specified maximum height will be used.
- **Define max height relative to H**: A geometric series of the specified number of heights up to the specified proportion of the boundary layer will be used.
- **Define all heights**: The heights defined in the table will be used. Up to 200 heights may be defined.

If only one height is defined, the wind speed and direction in the resulting .pro file can be plotted using the ***Wind rose viewer*** *utility (Section 7.2). In particular, it is possible to view the wind rose at the recorded wind height (usually 10 m) at the dispersion site, which may differ from the wind rose at the met. site depending on the options chosen in the* ***Meteorology*** *screen.*

4.24 Spatial splitting

The spatial splitting option can be used to divide a large modelling domain into multiple contiguous smaller regions, each of which can be run separately and potentially simultaneously. This option is useful if long run times and/or source limits imposed by the model licence are found to be inhibitive.

The ADMS-Urban licence limits are applied to the number of sources in each separate run (i.e. each region), not to the number of sources in the model file.

Alternatively, this option could be used to specify just one spatial subset of the sources and/or output points defined in a model file to be run by the model. For example, all sources in a large urban area could be included in a model file, to allow a single template model file to be used for all studies in that urban area. Then, for a particular study, if concentrations from a small area are required, this can be easily defined. In ADMS-Urban, some ground-level explicit sources (road, line, area or volume) can be adequately modelled by inclusion in the grid source if they are far enough from the output area; spatial splitting will improve the efficiency of the run in this case.

When multiple regions are to be run, the model file can either be run using CERC's Run Manager software tool or a batch file (which can be generated automatically using the **Batch File Creator** utility; see Section 7.4). Concentration contours from the resulting set of output files can then be plotted simultaneously using the **Contour Plotter** utility (see Section 6.2).

The spatial splitting option supersedes (and extends) the 'Spatial truncation' additional input file option used in previous versions of the model. Any old additional input files that contain this option can still be used in ADMS-Urban 5.1, but this may not be the case in future versions.

To use spatial splitting in a run, tick the **Spatial splitting** check box under the **Run management** section of the **Setup** screen, and click the **Data...** button. This brings up the **Run Splitting** screen, as shown in **Figure 4.60**.

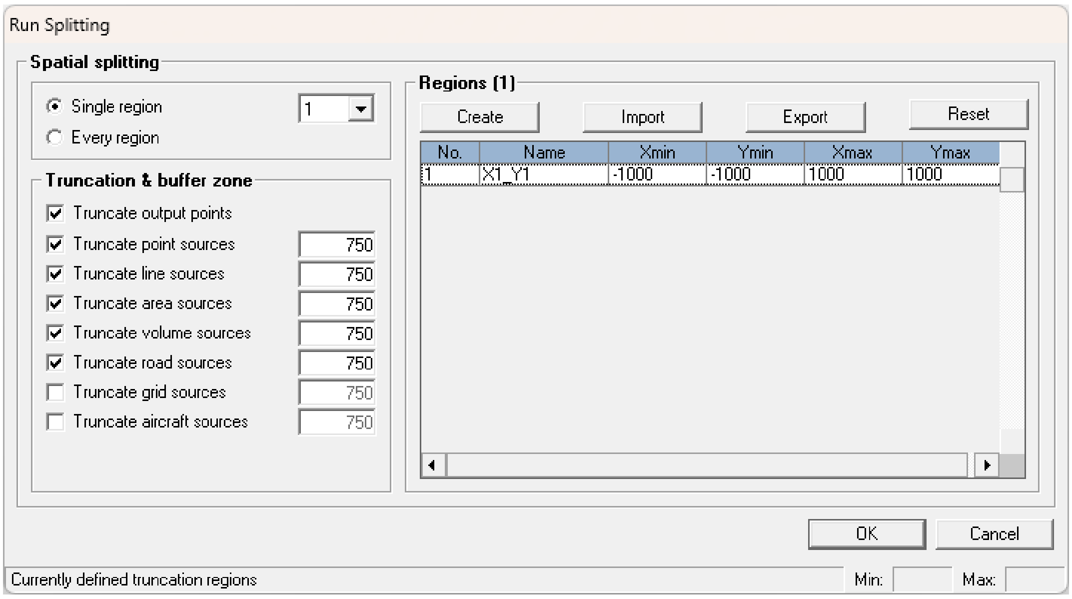


Figure 4.60 – The **Run splitting** screen.

‘Single region’ and ‘Every region’ modes

In the top left section of the **Run Splitting** screen, there are two options to choose from:

- **Single region**: This option should be selected if only one region of the modelling domain is to be run. The drop down list can be used to select the region number from those listed in the **Regions** section.

- **Every region**: This option should be selected if all the regions listed in the **Regions** section are to be run. If this option is selected, the user must either run the model file using CERC’s Run Manager software tool or the **Batch File Creator** utility (rather than using the **Run!** menu option). The **Batch File Creator** utility is accessed via the **Utilities, Create/run batch file** menu option (see Section 7.4 for more details). This utility can be used to generate a batch file that will automatically run ADMS-Urban for each region *consecutively*. Alternatively, if Run Manager is used to run the model file, all regions will be run automatically and, given enough licenced runs machines are available, *simultaneously*. Further information about Run Manager can be found at www.cerc.co.uk/RunManager.

  Output files from an **Every region** run use a naming convention that includes the region number between the file stem and the file extension, e.g. *example.00001.glt*. The **Plot all** feature of the **Contour Plotter** utility (see Section 6.2) can then be used to simultaneously plot all output files with the same file stem. Each region number is also written to the relevant *.log* file.

Output points on the North and East boundary of a cell will be excluded to avoid duplication. This means that output points on the North and East boundary of the output grid will be missing if the extent of the spatial splitting regions exactly matches the output grid extent. In such cases, it is advisable to extend the spatial splitting extent slightly to account for the described phenomenon.

Verification mode

When **Single region** is selected, the model verifies only the specified region. In contrast, selecting **Every region** prompts the model to verify all regions and generate a *.prc* file containing the region name, region number, the number of output points per region and the number of sources for each source type per region (refer to Section 6.1.16).

Truncation and buffer zone

The **Truncation & buffer zone** section of the **Run Splitting** screen is used to select whether output points that fall outside the region are to be truncated, which source types are to be truncated if they fall outside the region, and the size of the buffer zone used for each source type.

The buffer zone extends the region in all directions by the given length (m); any sources that fall within this extended region will not be truncated. If any part of an explicit source (point, line, area, volume, road) is within this extended region it will be included. It may be desirable to include some explicit sources near to, but not within, the region in order to allow smooth modelling at the region edges. The default buffer zone size is 750 m for each included source type. Setting a buffer zone size of zero will mean that no buffer zone is used at all for that source type.

The default source types selected for truncation are all the explicit source types (point, road, line area, volume). Typically, in ADMS-Urban, grid sources should not be truncated in order to maintain the same total emissions over the entire modelling domain (the emissions from any explicit sources that are truncated will be aggregated into the grid source). Care should be taken with large point sources when using 2D grid sources as the emissions will be brought closer to the ground.

Regions

The **Regions** section of the **Run Splitting** screen shows how the modelling domain is to be split up. By default, one region covering the entire output grid extent (as defined in the **Grids** screen) is defined. The **Reset** button can also be used to set the **Regions** table back to this default at any point.

*The columns in the **Regions** table are resizable.*

The **Create** button brings up the **Create regions** screen, as shown in **Figure 4.61**, which can be used to change the number and extent of the regions within the modelling domain. To change the overall extent of the regions from the default (the output grid extent), modify the values in the **Overall Min.** and **Overall Max.** fields accordingly for both X and Y. The **Region count** fields can be used to split the overall extent into contiguous regions of initially equal size in X and (separately) Y. Changing either of these values and pressing Enter/space-bar (or moving the cursor to another field) will automatically (re)populate the **Intermediate values** list, which give the X and Y values of the boundary lines between neighbouring regions. Further intermediate values can also be added to each table using the appropriate **Add** field, and existing intermediate values can be removed from the table by selecting them and pressing Delete. Once you are happy with the list of values, click the **Apply** and **OK** buttons to populate the **Regions** table and return to the **Run Splitting** screen, respectively. This will also give each region a unique name, which is non-editable.

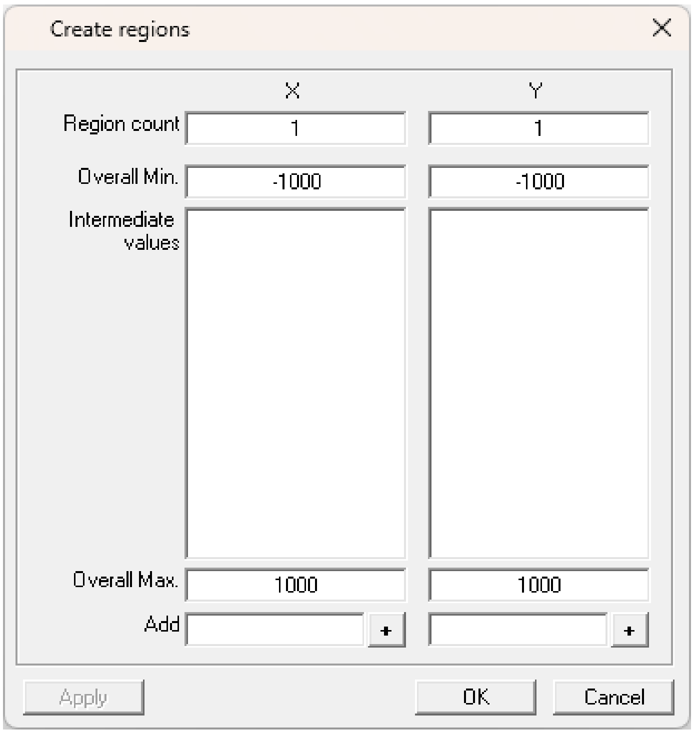
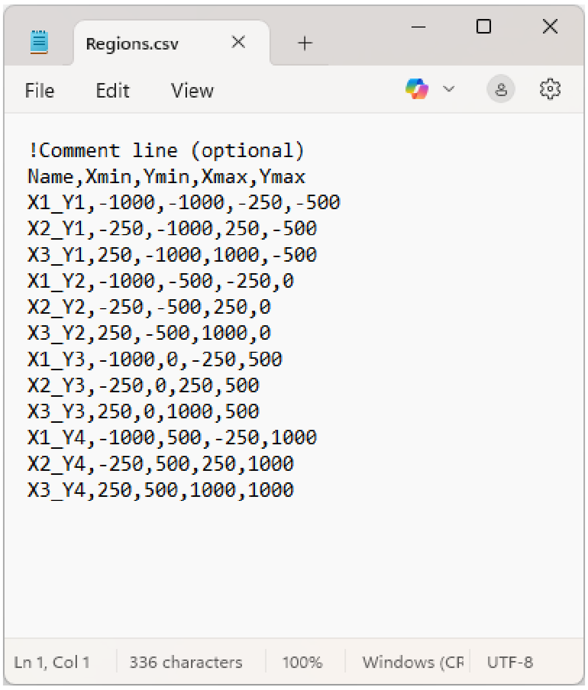


Figure 4.61 – The **Create regions** screen.

Alternatively, the **Import** button can be used to import a pre-defined list of regions from a comma-separated values (*.csv*) file. An example *.csv* file is shown in **Figure 4.62**. The format of this file consists of an optional comment line (or set of comment lines) that must start with a '!' character, a header line in the form 'Name, Xmin, Ymin, Xmax, Ymax', and one line of data per region giving the (unique) region name and the X and Y coordinates of the lower-left and upper-right corners of the region. Note that the regions defined in the *.csv* files do not have to be contiguous, and may even overlap, although most model set-ups will typically use contiguous regions. The **Export** button can also be used to export the current list of regions to a *.csv* file of the same format.

*The regions can be visualised in the Mapper, which will also highlight which region is selected for output when using the **Single region** option.*




```
!Comment line (optional)
Name,Xmin,Ymin,Xmax,Ymax
X1_Y1,-1000,-1000,-250,-500
X2_Y1,-250,-1000,250,-500
X3_Y1,250,-1000,1000,-500
X1_Y2,-1000,-500,-250,0
X2_Y2,-250,-500,250,0
X3_Y2,250,-500,1000,0
X1_Y3,-1000,0,-250,500
X2_Y3,-250,0,250,500
X3_Y3,250,0,1000,500
X1_Y4,-1000,500,-250,1000
X2_Y4,-250,500,250,1000
X3_Y4,250,500,1000,1000
```

Figure 4.62 – Example regions *.csv* file.

4.24.1 Restrictions

The **Spatial splitting** option cannot be run in combination with the **Disable grid disaggregation** option if truncating any explicit source types and not truncating grid sources. It also cannot be used with the **Source exclusion by distance from receptors** option.

When selecting both the **Spatial splitting** and **Comprehensive output file** options, interpolated source oriented grid points will not be included in the comprehensive output files.

4.25 Source exclusion

The source exclusion options enable a subset of the sources defined in a model file to be excluded from the model run, thereby reducing the model run time. Only sources that do not significantly contribute to modelled concentrations should be considered for exclusion. These are likely include sources that are far away from the receptor(s) and/or sources with very low emission rates.

4.25.1 By distance from receptors

When model output is only required at a small number of receptors, e.g. as part of a validation study in which modelled concentrations are compared against monitoring data, it may not be necessary to explicitly model sources in the model file that are far away from all receptors. To exclude such sources, tick the **By distance from receptors** check box under the **Source exclusion** section of the **Setup** screen and click the **Data...** button to bring up the screen shown in **Figure 4.63**.

The **Exclusion distance from receptors** section of this screen is used to select which source types are to be considered for exclusion and, for each selected source type, the inclusion radius (m). As an example, if the inclusion radius for point sources is set to 1000 m, then any point sources that do not fall within 1000 m of *any* receptor (specified point) will be excluded from the model run. In ADMS-Urban, it is recommended to leave grid sources unticked so that the emissions from any excluded explicit sources will be aggregated into the grid source. For source types with a finite size, a source will only be excluded if no part of it falls within the inclusion regions.

The radio buttons in the **Show in Mapper** section of the screen can be used to show the inclusion regions for a particular source type in the Mapper.

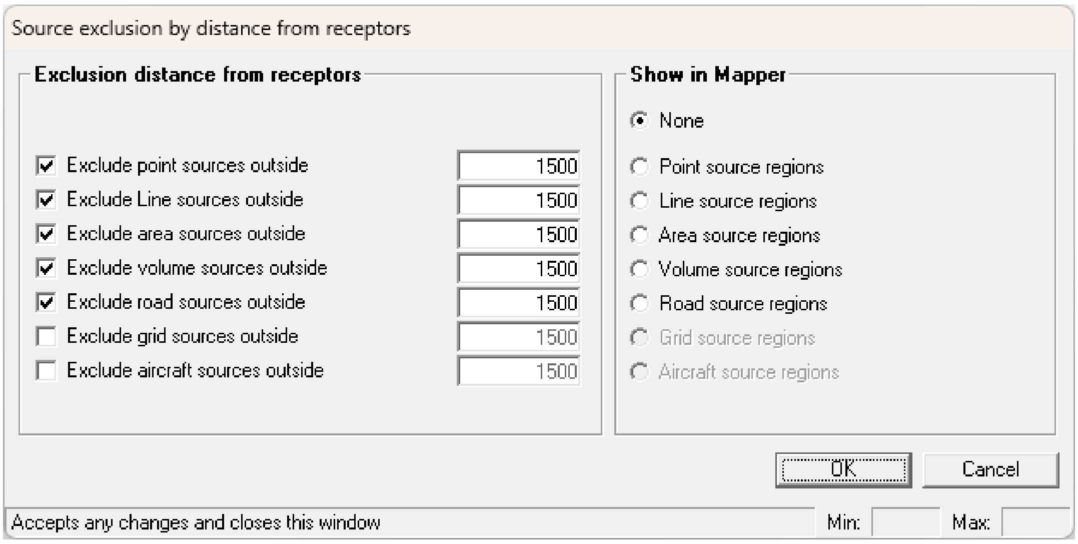


Figure 4.63 – The **Source exclusion by distance from receptors** screen.

4.25.2 By emission rate

To exclude sources in the model file from the model run based on their emission rate, tick the **By emission rate** check box under the **Source exclusion** section of the **Setup** screen and click the **Data…** button to bring up the screen shown in **Figure 4.64**.

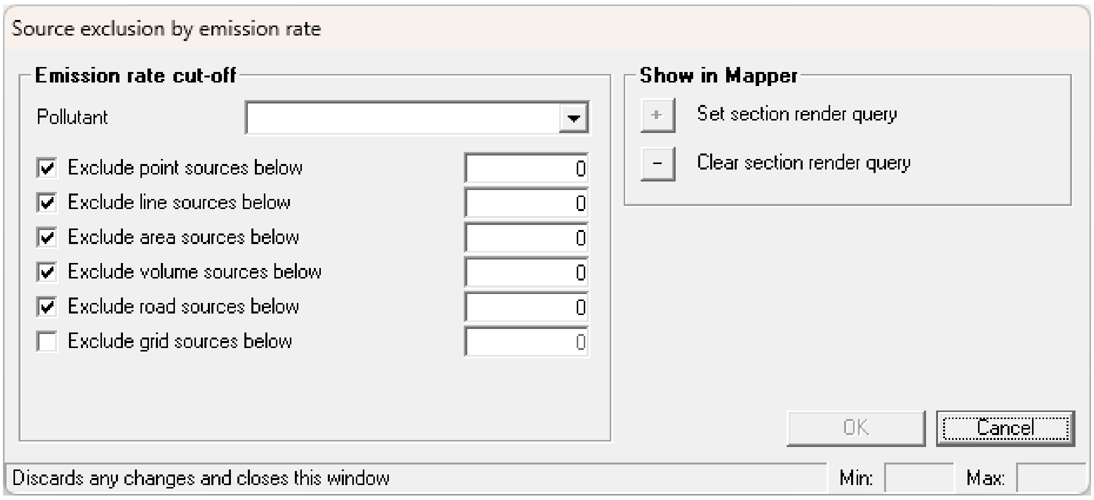


Figure 4.64 – The **Source exclusion by emission rate** screen.

In the **Emission rate cut-off** section, the top drop-down list is used to select which pollutant the cut-off emission rates refer to. Below this, the user selects which source types are to be considered for exclusion and, for each selected source type, the cut-off emission rate (in the same units as those used when defining the emission rates for a given source type). As an example, if the pollutant is set to NOx and the cut-off emission rate for point sources is set to 1 g/s, then any point sources that emit less than 1 g/s of NOx (or do not have an emission rate defined for NOx at all) will be excluded from the model run. In ADMS-Urban, it is recommended to leave grid sources unticked so that the emissions from any excluded explicit sources will be aggregated into the grid source.

The **Set section render query** button in the **Show in Mapper** section of the screen can be used to hide the sources that will be excluded from the run in the Mapper. Clicking the **Clear section render query** button will make all sources visible again.

4.25.3 Restrictions

- The **Source exclusion by distance from receptors** option can only be used when **Specified points** output is selected in the **Grids** screen.

- The **Source exclusion by distance from receptors** option cannot be used with the **Spatial splitting** option.

4.26 Extend output files

The output data in the short-term *.bld*, *.dep*, *.gst*, and *.levels.gst* files are usually restricted to the first 24 lines of meteorological data in order to prevent the output files from becoming too large. This limit can be removed by using an additional input file option.

To extend the short-term output files for a run, first create an additional input file or edit the existing additional input file (refer to Section 3.1.8) so that the **Extend output files** option (keyword LONGOP) is enabled, as shown in **Figure 4.65**.

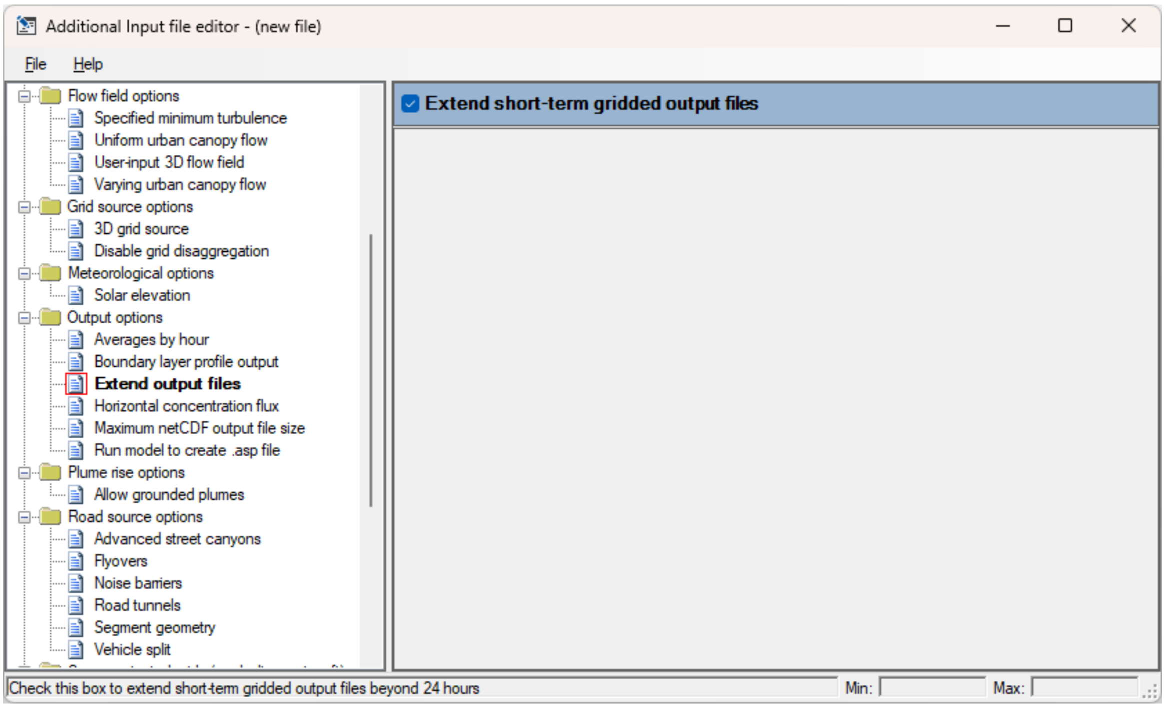


Figure 4.65 – The **Extend short-term gridded output files** option enabled in the **Additional Input file editor**

BEWARE: If running a large number of lines of meteorological data (for example one year of data), the output files may become very large, particularly the .gst *and* .levels.gst *files.*

Data for every hour of a long-term run can be obtained in a more compact format using the Comprehensive Output File option, described in Section 3.6.4.

4.27 Horizontal concentration flux

Most air quality monitors measure pollutant concentrations that can be compared directly against standard ADMS-Urban concentration output. However, monitors such as the Directional Passive Air Sampler (DPAS) work by pivoting in the wind to collect directional samples, measuring instead accumulated horizontal concentration *flux* from a number of different wind sectors (Solera Garcia et al., 2017). In order to be able to compare against data from this type of monitor, it is possible to request accumulated horizontal concentration flux output in ADMS-Urban via an *.uai* file option.

To enable this option, create a new additional input file or edit an existing additional input file (refer to Section 3.1.8) so that the **Horizontal concentration flux** section (keyword HORIZONTALFLUX) is enabled as shown in **Figure 4.66**. Choose the **Wind sector size** used by the directional monitor (available options are 30° or 10°) and the mid-point of one of the wind sectors (**Wind sector mid-point**) so that the model can calculate the boundaries between the different wind sectors.

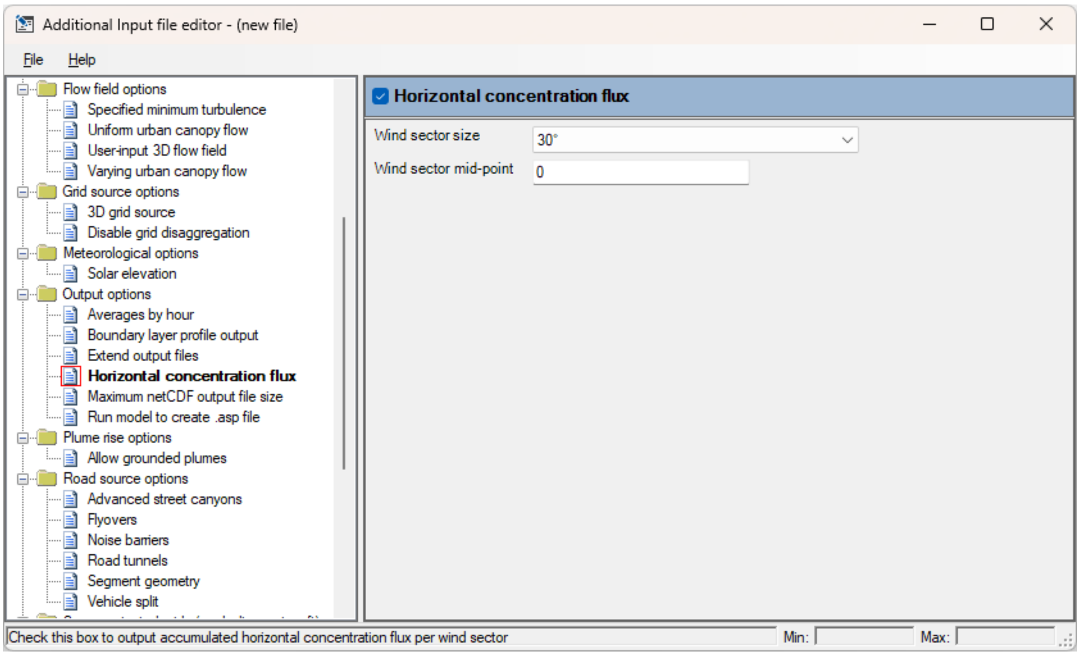


Figure 4.66 – The **Horizontal concentration flux** option in the **Additional Input file editor**.

When this option is selected, the model will create a new output file with extension *.flx*, described in Section 6.1.25, in the same directory (and with the same file stem) as the *.upl* file. This file will contain the accumulated horizontal concentration flux:

- at each specified point defined in the **Specified points** table and/or **Specified points file** from the **Grids** screen (i.e. no flux output is given at gridded output points);
- for each long-term (LT) pollutant with an averaging time of 1 hour or less that is selected for output in the **Output** screen;
- for the 'All sources' group only (which must therefore be ticked in the **Output** screen if **Groups** output is selected), or for the selected source if **Source** output is selected.

For a given receptor, the accumulated flux for a given met line and pollutant is calculated as $3600 \times C \times U$, where C is the hour-average concentration at that receptor (including any background concentration), U is the local horizontal wind speed (including any complex terrain effects but not including any buildings effects) and 3600 is the number of seconds in an hour. This value is assigned to the appropriate wind sector based on the local wind direction. An accumulated flux of zero is assigned to all other wind sectors for that met line. The output value in the *.flx* file (Section 6.1.25) is then the sum of these hourly values over all valid hours. Note that met lines for which reverse flow was encountered at that receptor are discounted from the total for that receptor.

4.28 Create *.asp* file of output points

Depending on the options chosen in the **Grids** screen, the final set of output points at which concentration values are given by the model can be a combination of standard grid points, source-oriented grid points and/or user-defined specified points. It can sometimes be useful to use the same set of output points in one model file as another separate model file. However, any changes in source inclusion and/or geometry between the two model setups are likely to affect the positioning of the source-oriented grid points. In order to overcome this difficulty, an option exists to run the model for a given setup such that only a single output *.asp* file (plus the *.log* and possibly *.wng* file) is created, containing a list of all the output points that would be used in a full model run. This *.asp* file can then be used in a separate model setup using the **Specified points file** option, thus guaranteeing that the same set of output points will be used in both model runs.

By using the **Run model to create .asp file** option, it is also possible to create an *.asp* file containing only the output points located within a specified distance from the edges of road sources. To ensure the *.asp* file is properly populated, an output grid source must be defined that encompasses all relevant road sources for which output points are required.

In order to use this option, create an additional input file, or edit the existing additional input file (refer to Section 3.1.8), so that the **Run model to create .asp file** option (keyword CREATEASP) is enabled, as shown in **Figure 4.67**. From the dropdown menu, select either the **All standard output points** or the **Roadside points only** option to create the *.asp* file, as shown in **Figure 4.67**.

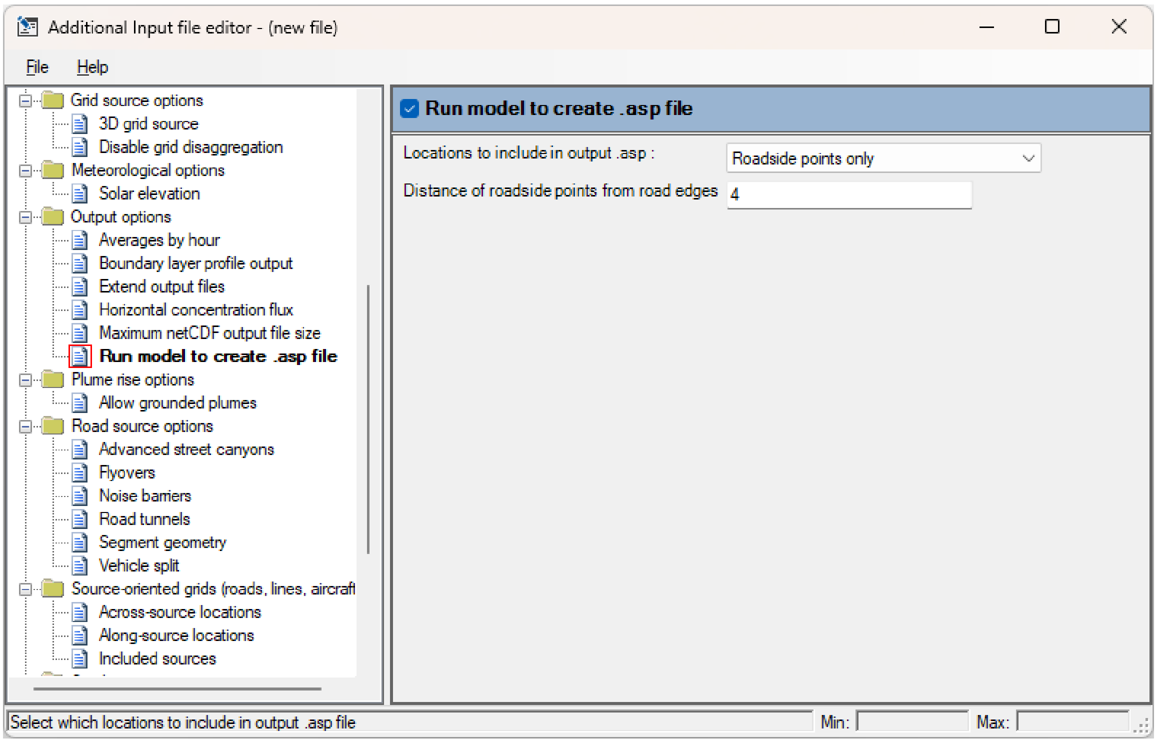


Figure 4.67 – The **Run model to create .asp file** section enabled in the **Additional Input file editor**

All other output files will not be created when running the model using this option, in order keep the run time down to a minimum.

If ***All standard output points*** *is selected, the list of output points in the created .asp file will not include any interpolation points inserted between pairs of source-oriented grid points along road and line sources.*

Section 6.1.24 describes the format of the created *.asp* files.

4.28.1 Restrictions

The *.asp* file can only be created if there are fewer than:

- 10 million standard grid points
- 10 million source-oriented grid points for road, line and aircraft sources

4.29 Unit emission rate

By default, in ADMS-Urban the source emission rates are not normalised. When using ADMS-Urban in combination with ADMS-EIT, it becomes necessary to use unit emission rates. This option allows for modelling with unit emission rates without having to set the emission rate to one manually. If time-varying emission factors are provided, they will be disregarded when this option is selected.

For road sources when vehicle counts are not supplied, the traffic induced turbulence will still be calculated using the input emission rates rather than the unit emission rates.

To enable this option, create a new additional input file or edit an existing additional input file (refer to Section 3.1.8) so that the **Unit emission rate** option (keyword UNITEMISSIONRATE) is enabled as shown in **Figure 4.68**.

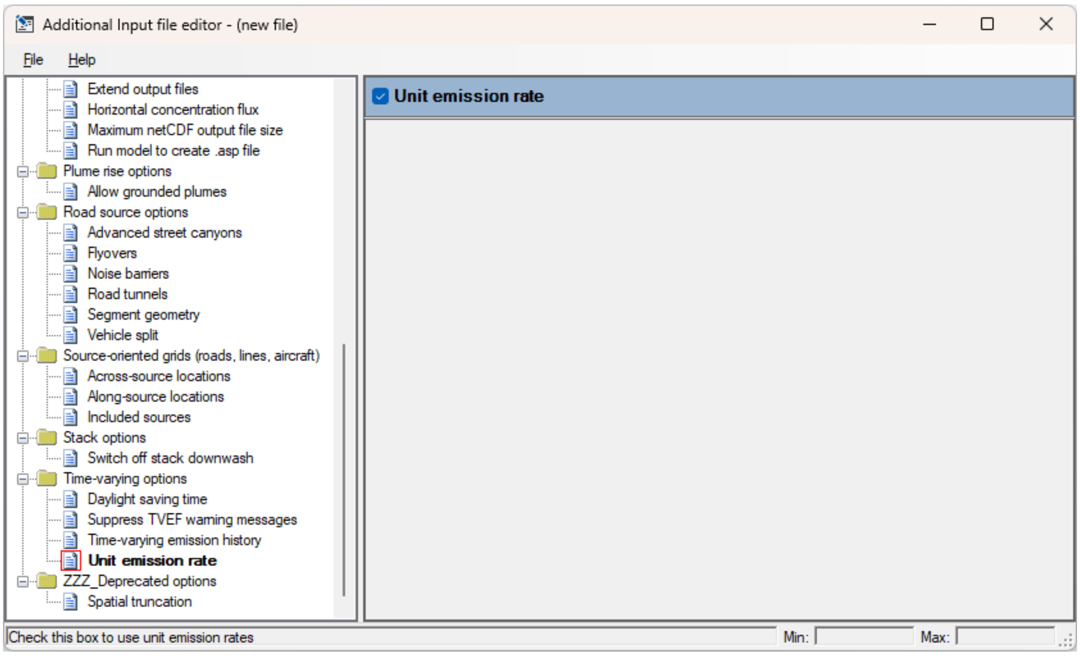


Figure 4.68 – The **Unit emission rate** option in the **Additional Input file editor**.

SECTION 5 Import and Export

Sections 3 and 4 of this User Guide have described how to set up ADMS-Urban model runs when there is a small amount of source data involved by typing data into boxes in the interface. However, if you have a large amount of source data, entering data in this manner can be time consuming. ADMS-Urban has an import facility allowing for source data to be imported into the ADMS-Urban interface from a set of comma-separated variable files. In addition to source data, source group, pollutant, and buildings data can also be imported in this way. An export facility is also provided, allowing source, pollutant and buildings data to be exported from the ADMS-Urban interface to comma-separated variable files.

Another CERC product, EMIT, an **EM**issions **I**nventory **T**oolkit, is a convenient, alternative way to prepare data for import into ADMS-Urban. EMIT is a flexible software tool for organising pollutant sources and their corresponding emissions. Further information about EMIT can be found at https://www.cerc.co.uk/environmental-software/EMIT-tool.html.

The Mapper also includes an option to facilitate the export of source data within a non-ADMS layer, such as a shape file, to these comma-separated variable files for import into ADMS-Urban. Please refer to the Mapper User Guide for full details, which can be accessed from the **Help** menu of the Mapper.

A description of the files used with the import and export facility is given in Section 5.1. Section 5.2 provides details on importing data and Section 5.3 on exporting data.

5.1 File formats

The import and export facility uses a set of comma-separated variable files each containing different aspects of the source, pollutant and buildings data. Each of these files has the same file stem and an extension indicating its contents:

- The *.spt* file contains the main properties of the sources including the efflux parameters and the source type, i.e. most of the information entered into the **Source** screen of the ADMS-Urban interface.
- The *.vgt* file contains the vertex information for all sources.
- The *.eit* file contains the pollutant emissions for all sources, i.e. the pollutant species data on the **Emissions** screen.
- The *.tft* file contains the traffic flow information for road sources.
- The *.gpt* file contains the group membership information for all sources.
- The *.ptt* file contains information about pollutant properties, i.e. the information contained in the **Palette of pollutants**
- The *.bpt* file contains information about buildings.

The first line of the *.spt* file is a version string; this is followed by a line indicating the traffic dataset, then a header row, and finally the data. All of the other files consist of a version string followed by a header row and then the data. Comment lines can be added to these files after the version line. These must start with the "!" character. The columns of data can be listed in any order and additional columns can be entered but will not be used during the import process. Full descriptions of the formats of each of these files are given in Sections 5.1.1 to 5.1.7. A facility to create a set of templates for these files is described in Section 5.1.8.

The version strings specified below (Sections 5.1.1 to 5.1.7) represent the most recent format files. However, it is also possible to import a set of legacy comma-separated variable files into ADMS-Urban that were suitable for previous versions of the model.

5.1.1 .spt file

The *.spt* file contains information about the sources, including the source type and efflux properties. The first line of the *.spt* file must contain the version information `SPTVersion2`. Following the version line is a line indicating the traffic dataset; this is of the form `Traffic dataset name | <Traffic dataset>` where `<Traffic dataset>` is replaced by the name of the traffic data set, see **Table 3.2** for details of valid traffic dataset names. So to specify that the EFT v13.1 2 vehicle category dataset should be used, the line would be `Traffic dataset name | EFT v13.1 (2 VC)`.

The header line details and data required are described in **Table 5.1**. A separate row of data must be entered for each source.

Many of the parameters in the .spt file only apply for certain source types, or if certain model options are in use. For parameters which are not required, leave blank or fill in 'n/a' or 'na'.

Header	Description
`Source name`	Source name.
`Use VAR file`	Not used in ADMS-Urban. Enter `No`.
`Specific heat capacity (J/kg/K)`	Specific heat capacity of the release in Joules per kilogram per Kelvin. The ADMS-Urban default value is 1012.
`Molecular mass (g)`	Molecular mass of the release in grams. The ADMS-Urban default value is 28.966.
`Temperature or density?`	Is the release temperature or density specified, or is it at ambient conditions? Enter the keyword `Temperature` (or `t` or `temp`), `Density` (or `d`, `den` or `rho`) or `Ambient` (or `a` or `amb`).
`Temperature (Degrees C) / Density (kg/m3)`	Temperature or density of the release in appropriate units.
`Actual or NTP?`	Is the efflux specified at the temperature or at NTP? Enter the keyword `Actual` (or `a` or `act`) or `NTP` (or `n`)
`Efflux type keyword`	There are 2 options for specifying the efflux. Enter the keyword `Velocity` (or `vel`) or `Volume` (or `vol`)

`Velocity (m/s) /`
`Volume flux (m3/s) /`
`Momentum flux (m4/s2) /`
`Mass flux (kg/s)`
	Efflux in appropriate units. Only velocity or volume flow rate are allowed for ADMS-Urban.

`Heat release rate (MW)`	Not used in ADMS-Urban.
`Source Type`	The type of the source. Enter the keyword `Point` (or `p`), `Line` (or `l`), `Area` (or `a`), `Volume` (or `v`), `Road` (or `r`) or `Grid` (or `g`).
`Height (m)`	Height of the source, in metres.
`Diameter (m)`	Diameter of a point source, in metres.

`Line width (m) /`
`Volume depth (m) /`
`Road width (m) /`
`Grid depth (m)`
	Width of a line or road source, or vertical extent of a volume or grid cell, in metres. All grid cells must have the same depth.

`Canyon height (m)`	Height of canyon walls for a road source, in metres.
`Angle 1 (deg)`	Not used in ADMS-Urban.
`Angle 2 (deg)`	Not used in ADMS-Urban.
`Mixing ratio (kg/kg)`	Not used in ADMS-Urban.
`Traffic flows used`	Is this a road source with emissions calculated from traffic flows? Enter `Yes` or `No`.
`Traffic flow year`	Year of traffic flow emissions; see **Table 3.2** for valid years for each traffic flow dataset.
`Traffic flow road type`	Road type for traffic flow emissions; see **Table 3.2** for valid road types for each traffic flow dataset.
`Gradient`	Gradient of the road, as a positive percentage. Only used if using a compatible traffic flow dataset.
`Main building`	The selection of main building. Enter the keyword `(Main)`, `(None)`, `(Auto)` or the name of a building.
`Comments`	Further comments may be added here, which will not be included in the model file.

Table 5.1 – Data to be entered in the *.spt* file.

5.1.2 .vgt file

The *.vgt* file contains vertex information for all of the sources. The first line of the *.vgt* file must contain the version information `VGTVersion2`. The header line details and data required are described in **Table 5.2**. The numbers of vertices required for each source type are as follows:

- Point sources require 1 vertex be entered.
- Line and road sources can have between 2 and 51 vertices.
- Area and volume sources can have between 3 and 50 vertices describing a convex polygon.
- Grid cells must have exactly four vertices describing a square.

Vertex information for area, volume and grid sources must be entered in either clockwise or anticlockwise order around the source.

To make import faster, always keep vertices for a source in a continuous block and order the sources the same as in the .spt file.

Header	**Description**
`Source name`	Source name.
`X (m)`	X coordinate of the vertex, in metres.
`Y (m)`	Y coordinate of the vertex, in metres.

Table 5.2 – Data to be entered in the *.vgt* file.

5.1.3 .eit file

The *.eit* file contains pollutant emissions information for the sources. For road sources set to have emissions calculated from traffic flows, any emission rates entered for calculated pollutants will be ignored and the calculated emission rates will be used instead. The first line of the *.eit* file must contain the version information `EITVersion1`. The header line details and data required are described in **Table 5.3**. Separate lines should be entered for each pollutant emitted from each source.

To make import faster, data for each source should be kept together and the sources should be entered in the same order as in the .spt *file.*

Header	**Description**
`Source name`	Source name.
`Pollutant name`	Pollutant name.
`Emission rate`	Emission rate of the pollutant in g/s for point sources, g/m/s for line sources, $g/m^2/s$ for area sources, $g/m^3/s$ for volume sources, g/km/s for road sources and $g/m^2/s$ for grid cells.
`Comments`	Further comments may be added here, which will not be included in the model file.

Table 5.3 – Data to be entered in the *.eit* file.

5.1.4 .tft file

The *.tft* file contains traffic flow information for road sources with emissions calculated from traffic flows. The first line of the *.tft* file must contain the version information `TFTVersion2`. The header line details and data required are described in **Table 5.3**. Separate lines are entered for each traffic flow for each source.

To make import faster, data for each source must be kept together and the sources should be entered in the same order as in the .spt *file.*

Header	Description
`Source name`	Source name.
`Vehicle category`	Vehicle category for this traffic flow, see **Table 3.6** for the valid vehicle categories for each dataset.
`Average speed (km/hr)`	Average vehicle speed for this traffic flow, in kilometres per hour.
`Vehicle count (vehicles/hour)`	Average vehicle count for this traffic flow, in vehicles per hour.
`Percent uphill`	Percentage of the vehicles for this traffic flow that are travelling uphill. Only used if using a compatible traffic flow dataset.

Table 5.4 – Data to be entered in the *.tft* file.

5.1.5 .gpt file

The *.gpt* file contains group membership information for all sources. The first line of the *.gpt* file must contain the version information `GPTVersion1`. The header line details and data required are described in **Table 5.5**. Each group is represented by a column and one row of data should be entered for each source.

To make import faster, sources should be entered in the same order as in the .spt *file.*

Header	Description
`Source name`	Source name.
`Source type`	Source type, either `Point` (or `p`), `Line` (or `l`), `Area` (or `a`), `Volume` (or `v`), `Road` (or `r`) or `Grid` (or `g`).
`Group|<Group name>`	Is the source a member of this group? Enter the keyword `Yes` or `No`.

Table 5.5 – Data to be entered in the *.gpt* file. Where `<Group name>` should be replaced with the name of the group, so for a group called 'IndustrialSite' the header for that column would be `Group|IndustrialSite`.

5.1.6 .ptt file

The *.ptt* file contains property data for all pollutants. The first line of the *.ptt* file must contain the version information `PTTVersion2`. The header line details and data required are described in **Table 5.6**. For gaseous pollutants a single line of data must be entered for each pollutant. For particulate pollutants separate lines of data should be entered for each particulate component; in the second and subsequent lines enter `n/a` in all columns except for the pollutant name, deposition velocity, terminal velocity, particle diameter, particle density and mass fraction.

Header	Description
`Pollutant name`	Name of the pollutant.
`Pollutant type`	Type of pollutant. Enter the keyword `Gas` (or `g` or `gaseous`) or `Particulate` (or `p` or `particulate`).
`Deposition vel. known`	Is the deposition velocity known? Enter the keyword `Yes` or `No`.
`Terminal vel. known`	For particulate pollutants, is the terminal velocity known? Enter the keyword `Yes` or `No`.
`Washout coeff known`	Is the washout coefficient known? Enter keyword `Yes` or `No`.
`Washout coeff.`	Washout coefficient independent of precipitation rate.
`Washout coeff. A`	Parameter A in expression for washout coefficient dependent on precipitation rate.
`Washout coeff. B`	Parameter B in expression for washout coefficient dependent on precipitation rate.
`Conversion factor (ug/m3 -> ppb)`	For gaseous pollutants, the conversion factor from ug/m3 to parts per billion by volume (ppb).
`Nature of gas`	For gaseous pollutants, the nature of the gas. Enter the keyword `Non-reactive` (or `n`, `nr` or `n-r`), `Reactive` (or `r`) or `Inert` (or `i`).
`Deposition velocity (m/s)`	Rate at which material is deposited to the ground due to diffusion in metres per second.
`Terminal velocity (m/s)`	For particulate pollutants, the rate at which material is deposited to the ground due to gravitational settling in metres per second.
`Particle diameter (m)`	For particulate pollutants, the diameter of the particles in metres.
`Particle density (kg/m3)`	For particulate pollutants, the density of the particles in kg/m^3.
`Mass fraction`	For particulate pollutants, the mass fraction of this diameter/density combination. The total mass fraction for each pollutant must add up to 1.

Table 5.6 – Data to be entered in the *.ptt* file.

5.1.7 .bpt file

The *.bpt* file contains data for all buildings. The first line of the *.bpt* file must contain the version information `BPTVersion1`. The header line details and data required are described in **Table 5.7**. Each building is represented by a separate row of data.

Header	Description
`Name`	Building name.
`Shape`	Shape of the buildings. Enter the keyword `Rectangular` (or `r`) or `Circular` (or `c`).
`X (m)`	X coordinate of the building centre, in metres.
`Y (m)`	Y coordinate of the building centre, in metres.
`Height (m)`	Height of the building, in metres.
`Length (m) / Diameter (m)`	Length of a rectangular building or diameter of a circular building, in metres.
`Width (m)`	Width of a rectangular building, in metres.
`Angle (Degrees)`	Angle the length of a rectangular building makes with north, measured clockwise in degrees.

Table 5.7 – Data to be entered in the *.bpt* file.

5.1.8 Import templates

A set of blank template files can be created by selecting **Create import templates** from the **Utilities** menu of the ADMS-Urban interface. This option creates a blank set of template files with the appropriate headers in a location of the user's choosing. These files can be edited in a spreadsheet package or text editor and then imported into ADMS-Urban using the **Import** option from the **File** menu. You will find that the source properties template (*.spt*) contains a convenient list of traffic datasets supported by the current interface for reference when filling the file with data.

5.2 Import

To import source, pollutant and/or buildings data from a set of comma separated variable files, select **Import** from the **File** menu to start the **Import wizard**, shown in **Figure 5.1**. Alternatively, drag and drop an *.spt* file into the ADMS-Urban interface to start the **Import wizard**. The **Import wizard** consists of a series of screens guiding you through the import process; some of these screens only appear if certain import options are selected. Navigate through the screens using the **Next >** and **< Previous** buttons. Sections 5.2.1 to 5.2.8 describe each of the screens of the **Import wizard**. Section 5.2.9 gives advice about importing sources with more vertices than the allowed maximum.

5.2.1 Select files to import

The first screen of the **Import wizard**, shown in **Figure 5.1**, is used to select the files to be imported.

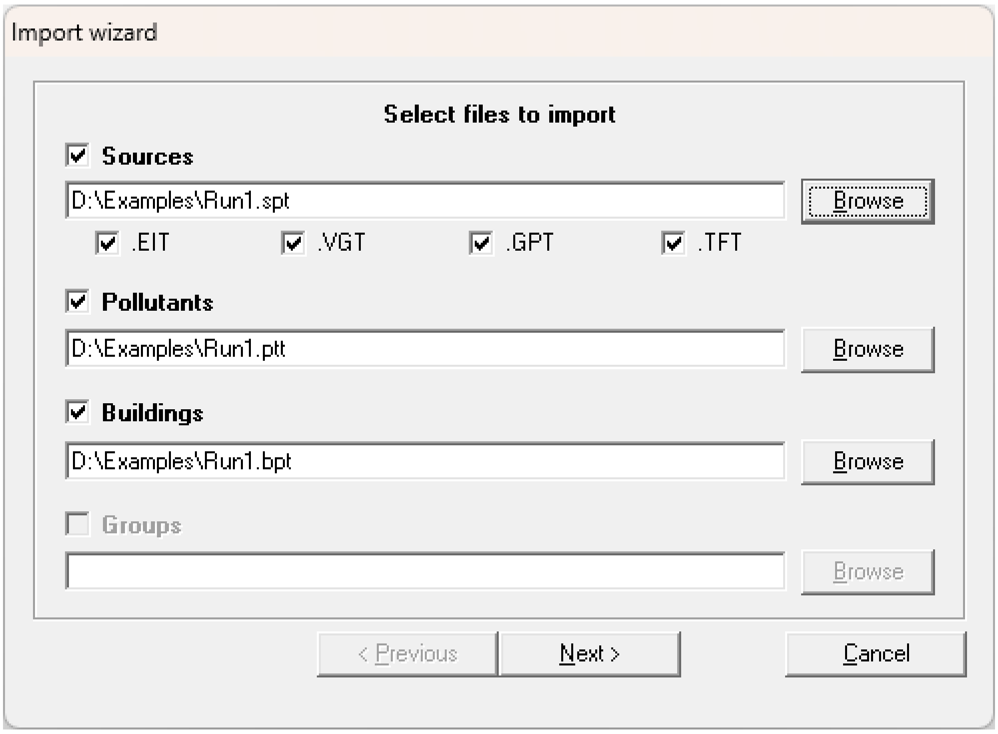


Figure 5.1 –Select files to import screen of the **Import wizard**.

There are four main file types which can be selected for import:

- **Sources**, (*.spt*) – select this option to import sources. When an *.spt* file is selected the other files corresponding to this are also automatically selected for import. The presence of an *.eit*, *.vgt*, *.gpt* or *.tft* file is indicated by the check boxes below, these can be deselected to prevent importing the data from that file.
- **Pollutants**, (*.ppt*) – select this option to import pollutants. By default, the *.ppt* file associated with the *.spt* file will be used; however you can browse to select an alternative *.ppt* file.

- **Buildings**, (*.bpt*) – select this option to import buildings. As with *.ppt* files, the *.bpt* file associated with the *.spt* file is used by default, but an alternative file can be selected.
- **Groups**, (*.gpt*) – select this option to import group membership information for sources already defined in the ADMS-Urban interface. This option is only available if **Sources** is not selected. If **Sources** is selected the availability of group membership information for the sources being imported is indicated by the **.GPT** checkbox.

Once the files to be imported have been selected, click **Next >** to move onto the next screen of the **Import wizard**.

5.2.2 Filter sources by type

If sources are selected for import, the next screen of the **Import wizard** is the **Filter sources by type** screen, shown in **Figure 5.2**.

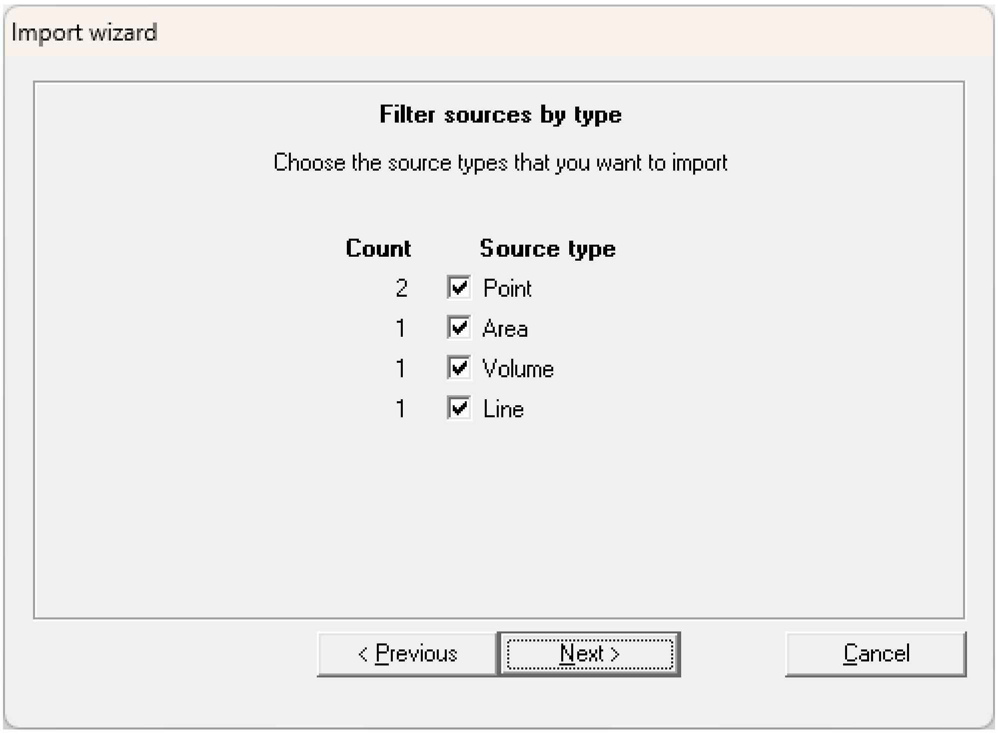


Figure 5.2 – **Filter sources by type** screen of the **Import wizard**.

This screen shows how many sources of each source type have been found in the *.spt* file. Deselect the checkbox against a source type if no sources of that type are to be imported. Any source types which are greyed out cannot be imported into ADMS-Urban.

Once the required source types are selected, click **Next >** to move to the **Select sources** screen.

5.2.3 Select sources

The **Select sources** screen allows for selection of individual sources to be imported, shown in **Figure 5.3**.

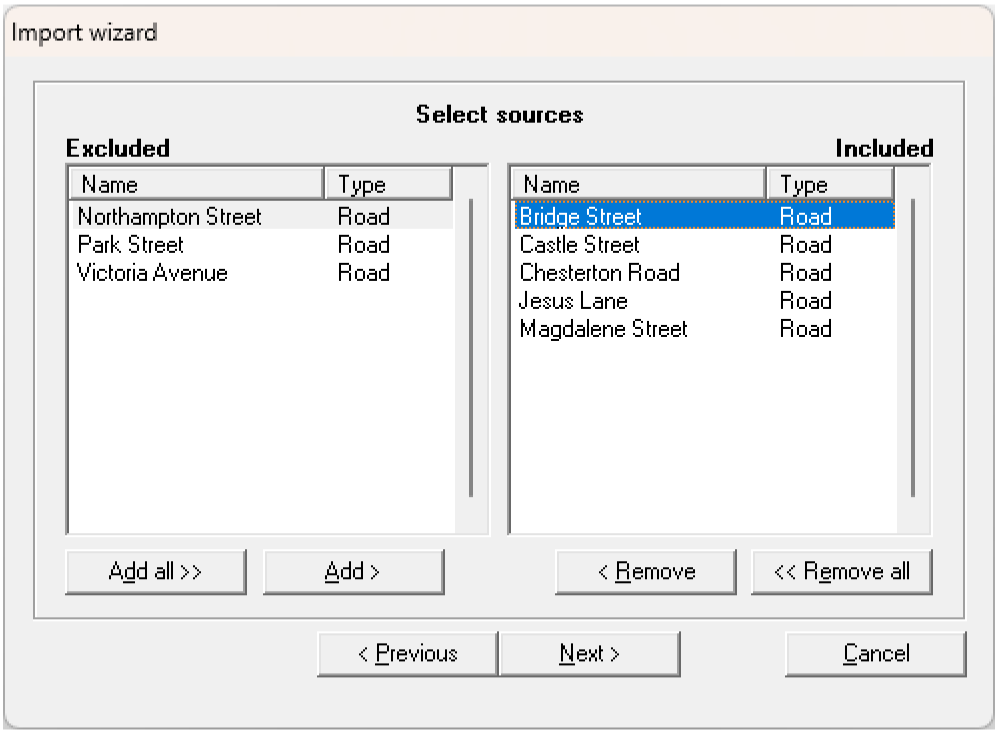


Figure 5.3 – **Select sources** screen of the **Import wizard**.

Sources listed on the right hand side will be imported whereas sources on the left will not be imported. By default, all sources from the *.spt* file will be selected for import. The **< Remove** and **<< Remove All** buttons can be used to exclude sources from the import, and the **Add >** and **Add All >>** buttons can be used to include sources in the import. Sources can also be **Included**/**Excluded** for import by double-clicking on the source name or dragging them to the appropriate side. The sources can be ordered by clicking on the **Name** and **Type** headers.

Once the required sources are selected for import, click the **Next >** button.

5.2.4 Source settings

If sources are being imported and some of the sources selected for import have the same name as sources already defined in the ADMS-Urban interface, or grid source cells are selected for import and a grid source is already defined in the ADMS-Urban interface, the **Source settings** screen will appear, shown in **Figure 5.4**.

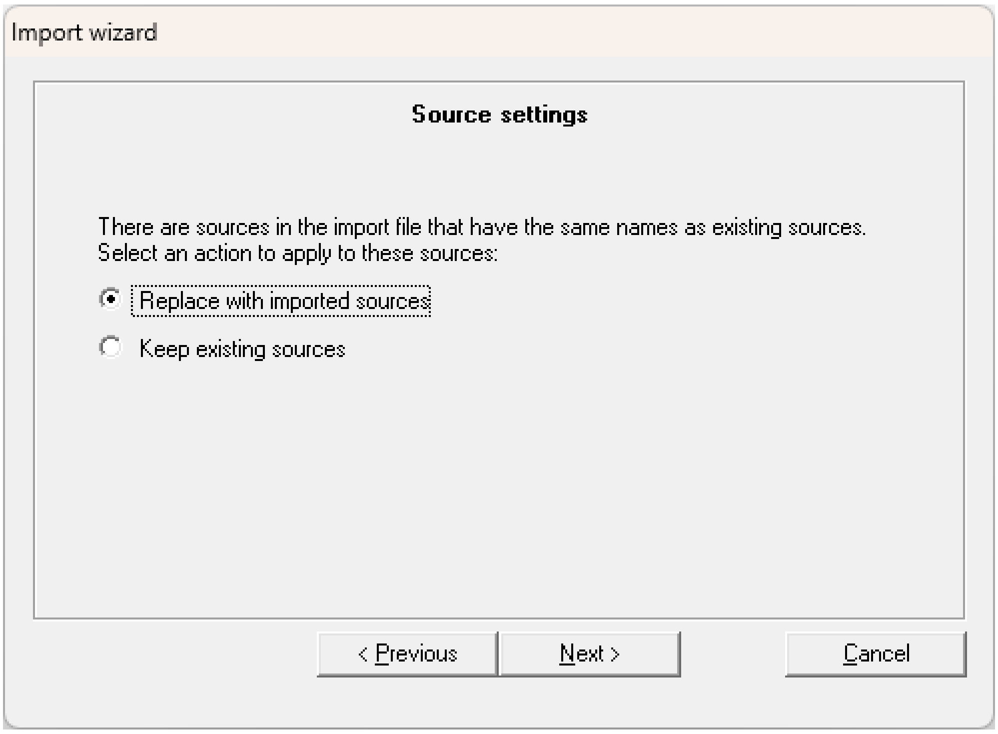


Figure 5.4 – **Source settings** screen of the **Import wizard**.

This screen has two options:

- **Replace with imported sources** – select this option to replace the existing source definitions with those contained in the import files. Any other sources without matching names already defined in the ADMS-Urban interface will be kept.
- **Keep existing sources** – select this option to keep the source definitions already in the ADMS-Urban interface. Any other sources without matching names will still be imported.

If grid source cells are selected for import and a grid source is already defined in the interface, a warning will appear that the existing grid source will be overwritten. To prevent the existing grid source from being overwritten, click on **< Previous** and remove all grid source cells from inclusion for import. Alternatively go back to the filter screen (Section 5.2.2) and deselect grid source cells from the import.

Click **Next >** to move to the next screen of the **Import wizard**.

5.2.5 Pollutant settings

If pollutants are being imported, the **Pollutant settings** screen will appear, shown in **Figure 5.5**.

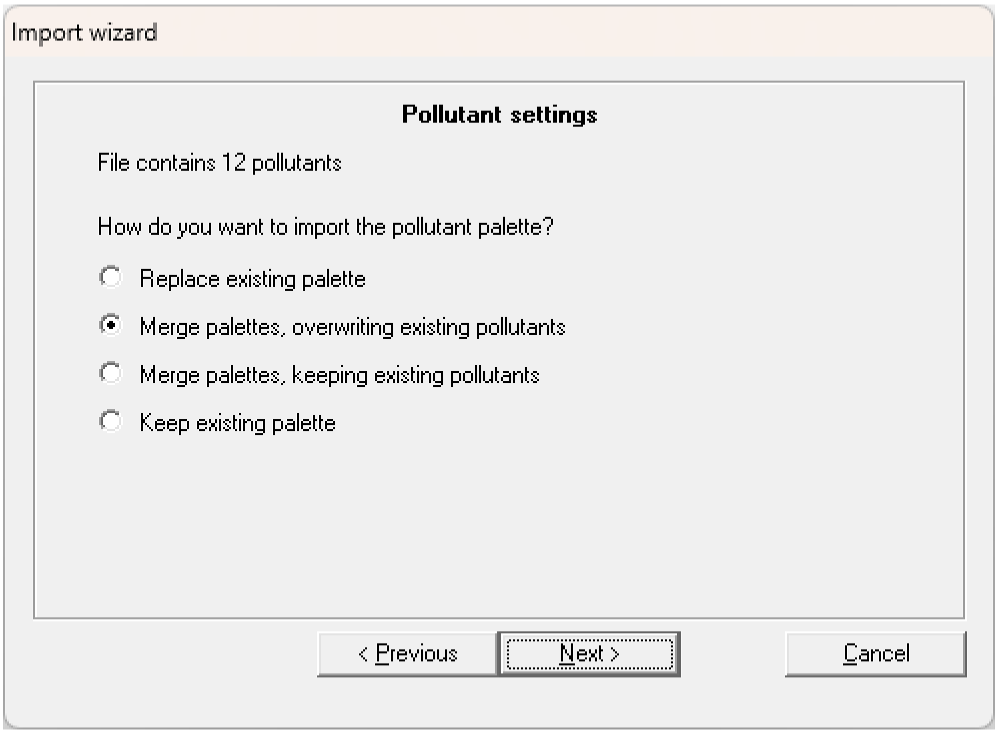


Figure 5.5 – **Pollutant settings** screen of the **Import wizard**.

This screen has four options:

- **Replace existing palette** – all pollutants from the *.ptt* file will be imported, all existing pollutants will be deleted. If `NOx`, `NO2`, `NO`, `O3`, `VOC`, `SO2`, `PM2.5`, or `PM10` are not included in the *.ptt* file to be imported, the existing version of these pollutants will be retained.
- **Merge palettes, overwriting existing pollutants** – all pollutants from the *.ptt* file will be imported, existing pollutants will be kept providing their names don't match any in the *.ptt* file.
- **Merge palettes, keeping existing pollutants** – all existing pollutants will be kept, any pollutants not already in the palette will be imported.
- **Keep existing palette** – no pollutants will be imported. This is equivalent to not ticking pollutants on the import filter screen (Section 5.2.1).

Click **Next >** to move to the next screen of the **Import wizard**.

5.2.6 Building settings

If buildings are being imported and some buildings are already defined in the ADMS-Urban interface, the **Building settings** screen will appear, shown in **Figure 5.6**.

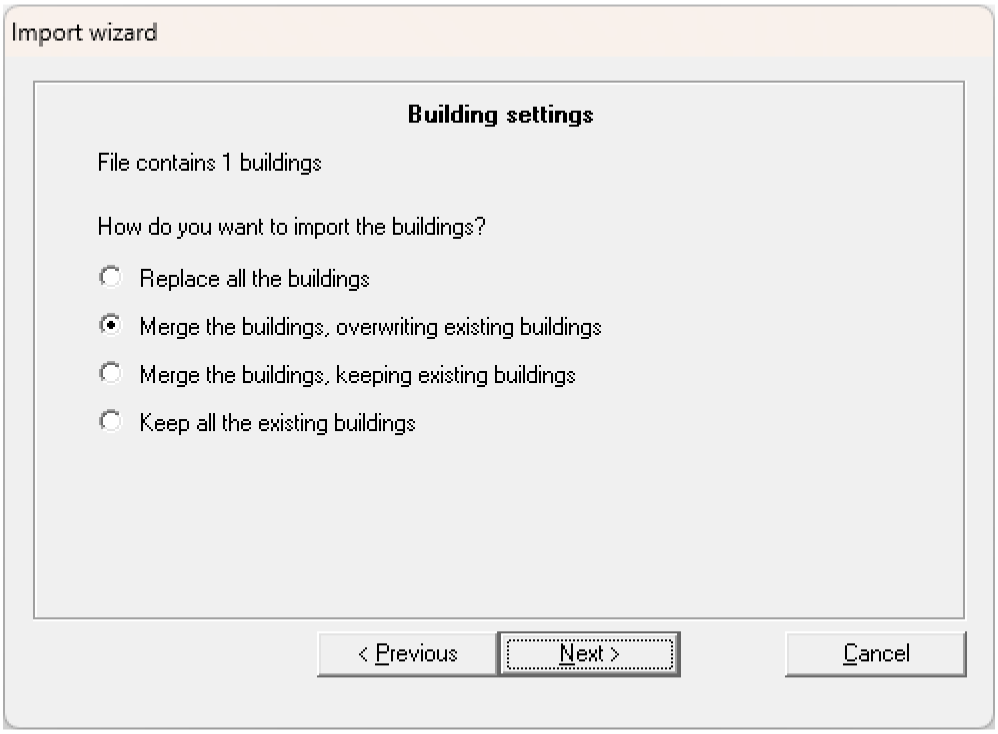


Figure 5.6 – **Building settings** screen of the **Import wizard**.

This screen has four options:

- **Replace all the buildings** – all buildings from the *.bpt* file will be imported, all existing buildings will be deleted
- **Merge the buildings, overwriting existing buildings** – all buildings from the *.bpt* file will be imported, any existing buildings will be kept providing their names don't match any in the *.bpt* file
- **Merge the buildings, keeping existing buildings** – all existing buildings will be kept, any buildings whose names don't match an existing building will be imported
- **Keep all the existing buildings** – no buildings will be imported

Click **Next >** to move to the next screen of the **Import wizard.**

5.2.7 Group settings

If group membership information is being imported without other source data, the **Group settings** screen will appear, shown in **Figure 5.7**.

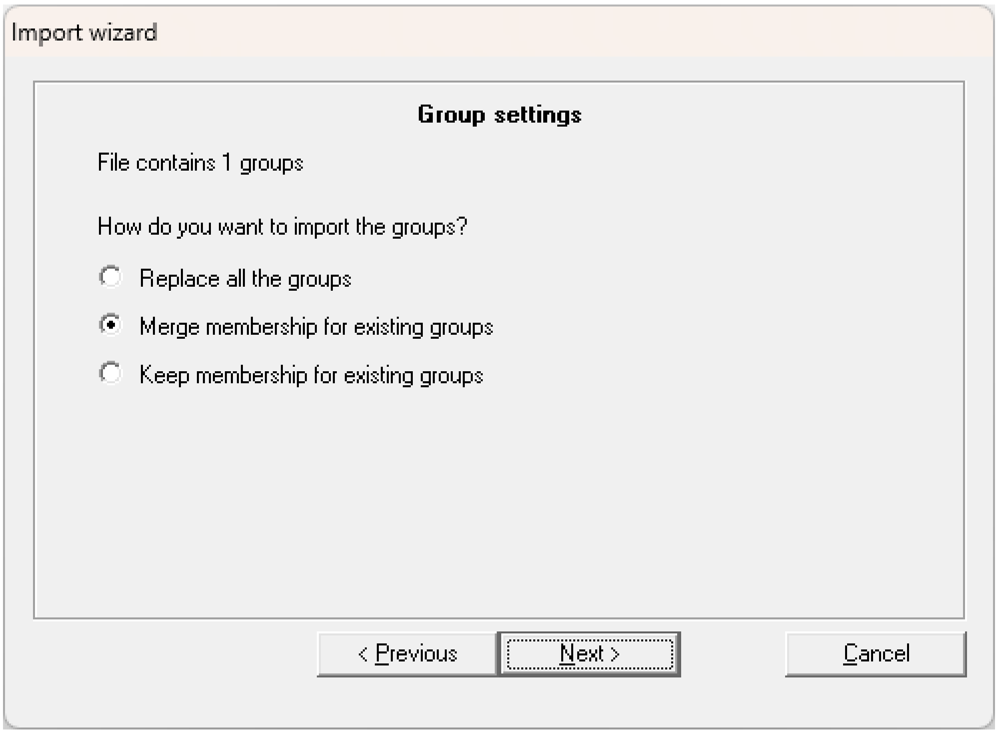


Figure 5.7 – **Group settings** screen of the **Import wizard**.

This screen has three options:

- **Replace all the groups** – all existing group membership information will be deleted and replaced with that in the *.gpt* file
- **Merge membership for existing groups** – all group membership information will be imported, for existing groups not in the *.gpt* file group membership information will be kept
- **Keep membership for existing groups** – all existing group membership information will be kept, group membership will be imported for new groups contained in the *.gpt* file

Click **Next >** to move to the next screen of the **Import wizard**.

5.2.8 Check import

The final screen of the **Import wizard** is the **Check import** screen, shown in **Figure 5.8**.

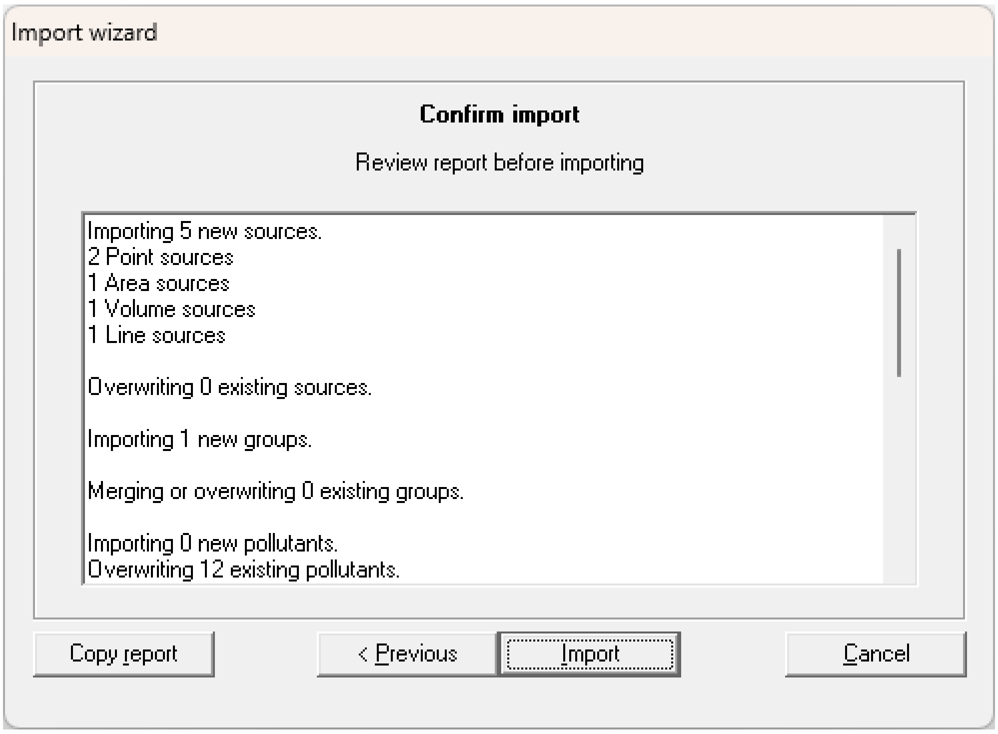


Figure 5.8 – **Check import** screen of the **Import wizard**.

This screen provides a report indicating the numbers of sources, pollutants, buildings and/or groups which will be imported or overwritten along with other information messages. The **Copy report** button can be used to copy the report data to the clipboard. To import the data, click on **Import** or click **< Previous** to alter what should be imported.

5.2.9 Importing sources with too many vertices

The maximum number of vertices that can be added to road or line sources via the interface is 51, and for area or volume sources is 50. Sources with more than this number of vertices can be imported from comma separated variable files. However, the user must reduce the number of vertices to no more than the maximum number for all sources before the model file can be saved. This can either be done manually using the **Delete** button in the relevant **Geometry** table(s), or alternatively, there is a *Simplify* (or *Simplify Layer*) feature available in the Mapper that automates the reduction of vertices for a given source (or all sources of a particular type). Please refer to the Mapper User Guide for full details, which can be accessed from the **Help** menu of the Mapper.

5.3 Export

It is also possible to export source, group, pollutant and building data from ADMS-Urban into a set of comma-separated variable files. You can use this method to transfer data to other model files, or to manipulate the data in a spreadsheet package before re-importing.

To export data from the ADMS-Urban interface select **Export** from the **File** menu to open the **Export** screen, shown in **Figure 5.9**.

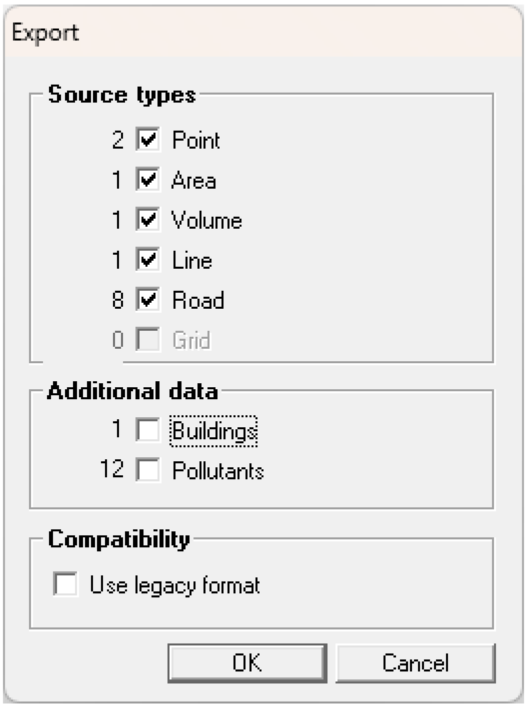


Figure 5.9 – **Export** screen.

The **Export** screen allows for the selection of the **Source types** to be exported along with options to export the **Buildings** and **Pollutants** data. Data about source groups are automatically exported alongside the corresponding sources.

If the **Use legacy format** option is ticked, the comma-separated variable files will be generated using an older format.

Once the items to be exported have been selected and the format chosen, click **OK** to open the **Save** dialogue. Enter a directory and filename stem then click **Save**. A set of comma-separated variable files will be created; refer to Section 5.1 for details of which files will be created based on the options selected.

SECTION 6 Model Output

A variety of output data are produced by ADMS-Urban, which are dependent on the way the run has been set up. This section describes each possible output and presents the tools available in ADMS-Urban to visualise the results from a run.

Section 6.1 describes the different types of files produced by ADMS-Urban. Sections 6.2 to 6.4 describe the elements of the **Results** menu of the ADMS-Urban interface, as shown in **Figure 6.1**.

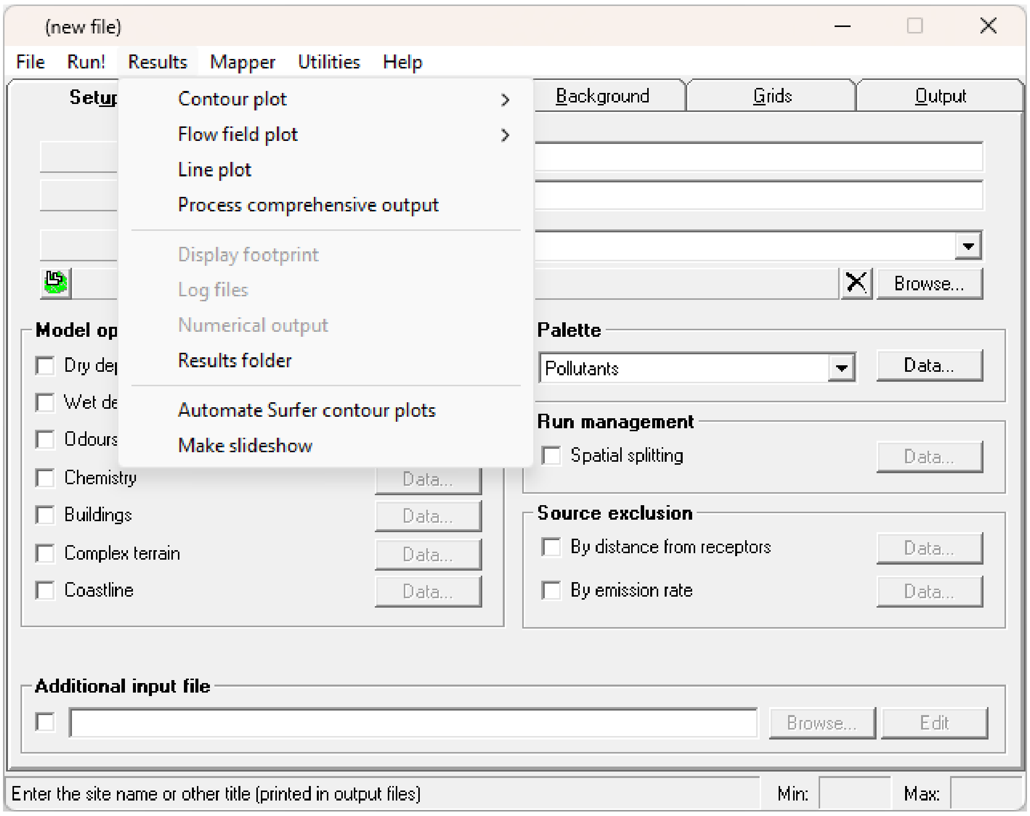


Figure 6.1 – Results menu.

The first three elements of the menu are used for visualising the results produced by ADMS-Urban in different ways: contour plots of concentrations (**Contour plot**) (Section 6.2.1), contour/vector plots of the flow field (**Flow field plot**) (Section 6.2.2), and plots of plume centreline properties (**Line Plot**) (Section 6.3).

The fourth element (**Process comprehensive output**) is for examining the output contained in a comprehensive output file. More details of this option are given in the *Comprehensive Output File Processor User Guide*, which can be opened from the **Help** menu.

The fifth option (**Display footprint**) can be used specifically for visualising source-apportionment results produced by an ADMS-Urban run using the '*Output per source*' option (Section 6.4).

The sixth option (**Log files**) is for visualising the error, warning and log files produced during the run (Section 6.5).

The seventh option (**Numerical output**) launches a window listing all files in the folder containing the model file. A particular output file can then be selected and opened in the preferred application (Section 6.6).

The eighth option (**Results folder**) opens the folder of the currently loaded model file in an Explorer window.

The last two options on the **Results** menu start up Excel templates containing macros to automate the creation of contour plots in Surfer and ‘slideshows’ of *.gif* images. For more details on these items refer to the *Surfer Automation and Slideshow Creator User Guide*, which can be found in the *<install_path>\Documents* directory.

6.1 Output files

During an ADMS-Urban model run, a number of output files are created. These files contain the results of the run and are stored in the same directory as the corresponding model file (*.upl*). The number of output files produced depends on the run configuration, such as the type of input file and the selected options.

All output files for a model run have the same file stem as the model file stem and are associated with a different extension that indicates the type of output data contained in the file. For instance, should the model file be *example.upl*, the output files could be as follows:

example.!01, example.asp, example.bef, example.bld, example.bwk, example.flx, example.dep, example.glt, example.gst, example.i01, example.j01, example.k01, example.levels.glt, example.levels.gst, example.log, example.max, example.mop, example.nc, example.out.met, example.plt, example.prc, example.pst, example.pro, example.rds example.sst, example.slt, example.t01, example.tlt, example.w01, example.wlt, example.zlt, example.zst

Additionally, if the **Spatial splitting** option is being used to divide the modelling domain into smaller regions, and all regions are to be run together (i.e. ‘Every region’ mode is selected), the output file names will also include the region number between the file stem and the file extension, e.g. *example.00001.glt*.

Each of the possible output files is described below.

The majority of the ADMS-Urban output files are in a text-based format. As an alternative to using the ADMS-Urban results utilities, the user can view them directly with a text editor such as Notepad.

6.1.1 .err, .wng and .log files

The *.err* (<u>err</u>or) file is a text file produced if an error occurs during the run that causes the model to stop. Any errors must be corrected before the model can run the model file successfully. In verification mode, the file may contain several errors. Each error must be corrected before the model can run the *.upl* file successfully.

The *.wng* (<u>w</u>ar<u>n</u>in<u>g</u>) file contains any warning messages produced during the run. These warnings do not cause the model to stop. The user should check that for each of these warnings the required behaviour is being followed, and make adjustments to model input if necessary.

The *.log* file contains a <u>log</u> of the model run detailing the input parameters, including all ancillary files used and output to be calculated. It also details the date and time of the start and end of the run, any program messages, and a summary of the meteorological data (including the number of lines of data the model was able to use).

6.1.2 .glt, .levels.glt and .plt files

The *.glt* file (gridded, long-term) contains output values resulting from long-term average calculations for each of the selected source groups. The *.glt* file contains the concentration, percentiles and exceedences output on the lowest grid level along with the deposition output, which is always calculated at ground level. The *.glt* file also contains the values at the source-oriented grid points, if these options are selected. The possible output variables are listed in **Table 6.1**.

The *.levels.glt* file contains output values resulting from the long-term average calculations for each of the selected source groups, for each output height of the output grid. A separate column of output is given in the *.levels.glt* file for each of the output grid heights. Note that certain output, like deposition, is not given in the *.levels.glt* file as it is not height dependent. The *.levels.glt* file is only created if there is more than one height in the output grid.

The *.plt* file (points, long-term) is equivalent to the *.glt* file except that output values are given for every specified point defined in the **Grids** screen.

Variable	Description
Conc	mean concentration
If percentiles selected:	
P###.##	percentile (e.g. P99.9)
If exceedences selected:	
EXpaM	number of exceedences per annum of each concentration threshold value
If dry and/or wet deposition modelled:	
Dry	long-term average dry deposition
Wet	long-term average wet deposition
Tot	long-term average total deposition
If long-term averages by hour selected:	
Conc##	mean concentration of hour ## (e.g. Conc01)

Table 6.1 – Variables contained in a *.glt* file.

6.1.3 .gst, .levels.gst and .pst files

The *.gst* file (gridded, short-term) contains output values resulting from short-term calculations for each modelled source group at every point on the lowest level of the output grid for each line of meteorological data. The *.gst* file also contains the values at the source-oriented grid points, if these options are selected. The possible output variables are listed in **Table 6.2**.

Output values are usually given for the first 24 lines of meteorological data only, to prevent the output file becoming too large. However, the file can be extended if required, as described in Section 4.26.

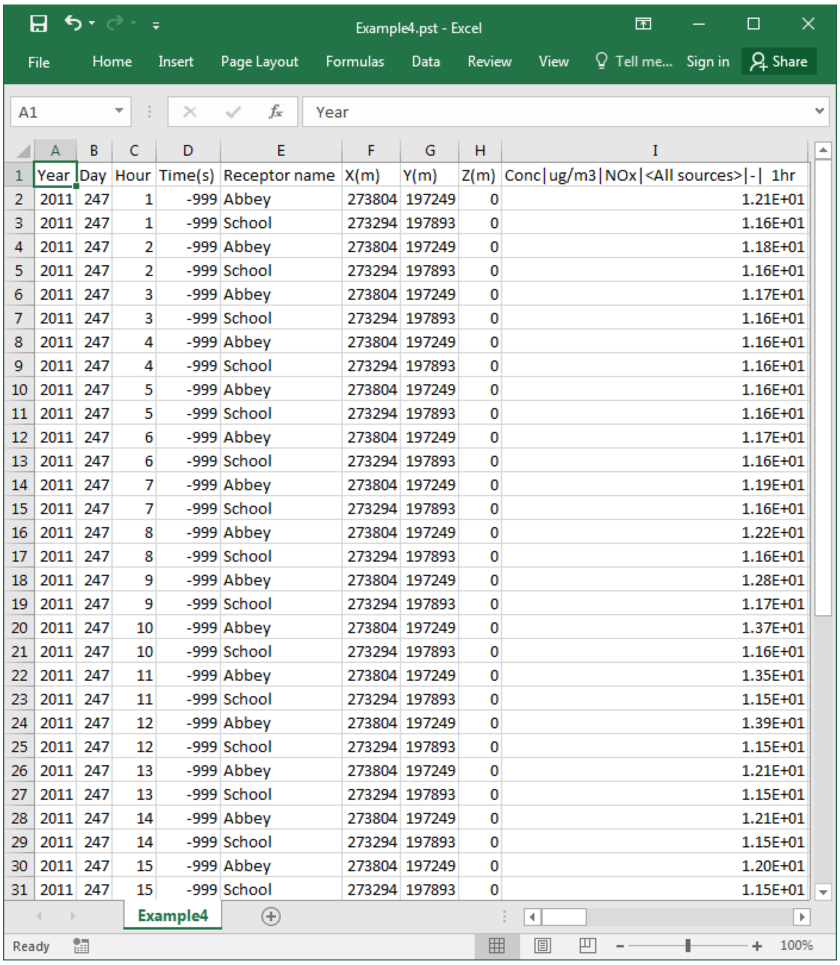
The *.levels.gst* file contains output values resulting from short-term calculations for each of the selected source groups, for each line of meteorological data and for each height of the output grid. A separate column of output is given in the *.levels.gst* file for each of the output grid heights. Note that certain output, like deposition, is not given in the *.levels.gst* file as it is not height dependent. The *.levels.gst* file is only created if there is more than one height in the output grid.

The *.pst* file (points, short-term) is equivalent to the *.gst* file except that output values are given for every specified point defined in the **Grids** screen. Output values are given for every line of meteorological data by default in the *.pst* file.

Examples of *.pst* and *.plt* files (viewed in Microsoft Excel) are illustrated in **Figures 6.2** and **6.3**. Gridded *.glt* and *.gst* files are similar to these files but with a value for every grid point.

Variable	Description
Conc	mean concentration
If dry and/or wet deposition modelled:	
Dry	dry deposition
Wet	wet deposition
Tot	total deposition

Table 6.2 − Variables contained in a *.gst* file.



<table>
<tr><th>Year</th><th>Day</th><th>Hour</th><th>Time(s)</th><th>Receptor name</th><th>X(m)</th><th>Y(m)</th><th>Z(m)</th><th>Conc|ug/m3|NOx|<All sources>|-| 1hr</th></tr>
<tr><td>2011</td><td>247</td><td>1</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.21E+01</td></tr>
<tr><td>2011</td><td>247</td><td>1</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>2</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.18E+01</td></tr>
<tr><td>2011</td><td>247</td><td>2</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>3</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.17E+01</td></tr>
<tr><td>2011</td><td>247</td><td>3</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>4</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>4</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>5</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>5</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>6</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.17E+01</td></tr>
<tr><td>2011</td><td>247</td><td>6</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>7</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.19E+01</td></tr>
<tr><td>2011</td><td>247</td><td>7</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>8</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.22E+01</td></tr>
<tr><td>2011</td><td>247</td><td>8</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>9</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.28E+01</td></tr>
<tr><td>2011</td><td>247</td><td>9</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.17E+01</td></tr>
<tr><td>2011</td><td>247</td><td>10</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.37E+01</td></tr>
<tr><td>2011</td><td>247</td><td>10</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.16E+01</td></tr>
<tr><td>2011</td><td>247</td><td>11</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.35E+01</td></tr>
<tr><td>2011</td><td>247</td><td>11</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.15E+01</td></tr>
<tr><td>2011</td><td>247</td><td>12</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.39E+01</td></tr>
<tr><td>2011</td><td>247</td><td>12</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.15E+01</td></tr>
<tr><td>2011</td><td>247</td><td>13</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.21E+01</td></tr>
<tr><td>2011</td><td>247</td><td>13</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.15E+01</td></tr>
<tr><td>2011</td><td>247</td><td>14</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.21E+01</td></tr>
<tr><td>2011</td><td>247</td><td>14</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.15E+01</td></tr>
<tr><td>2011</td><td>247</td><td>15</td><td>-999</td><td>Abbey</td><td>273804</td><td>197249</td><td>0</td><td>1.20E+01</td></tr>
<tr><td>2011</td><td>247</td><td>15</td><td>-999</td><td>School</td><td>273294</td><td>197893</td><td>0</td><td>1.15E+01</td></tr>
</table>

Figure 6.2 − An example of a *.pst* file.

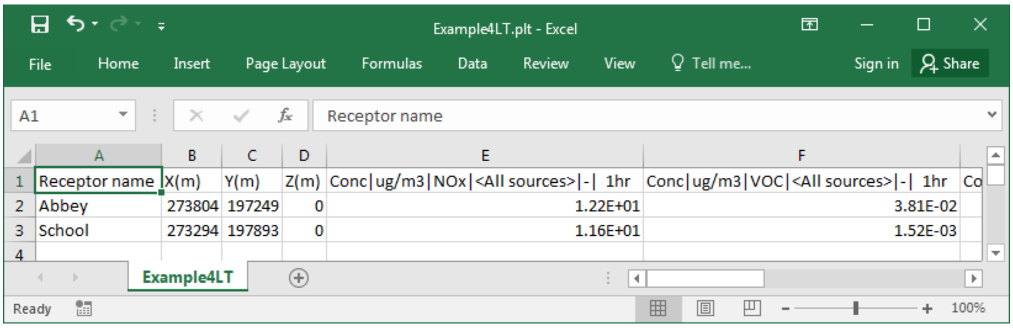


Figure 6.3 − An example of a *.plt* file.

6.1.4 .sst and .slt files

The *.sst* (source short-term) and/or *.slt* (source long-term) files are only produced when the **Output per source** option is selected. These files are similar in format to the *.pst* and *.plt* files except that there is one line of data for each source in the model file (and for each line of meteorological data for short-term output), and the output values of concentration, any deposition and (for long-term output) any percentiles and exceedences are given for each specified point defined in the **Grids** screen. The source type is also included in these files. Section 6.4 describes how data in these files can be plotted in the Mapper, where the sources themselves are colour-coded to indicate their individual contribution to the concentration at a given receptor.

6.1.5 .max file

The *.max* file (maximum concentrations) contains the maximum value of the mean concentration and each of the percentiles selected, the position of the maximum values and, in the case of the percentiles, the line of meteorological data on which the maximum values occur. It is created when a long-term average calculation is made, with or without percentiles.

In addition, following each maximum percentile value, the concentrations exceeding that value at that point are listed, alongside the line number of the meteorological data record for which they occur. For example, when using one year of data (8760 lines), there will be 2% of 8760 lines (176) where the 98th percentile value is exceeded. This output is only given for the 98th percentile and above.

6.1.6 .mop file

The *.mop* file (meteorological input and output parameters) contains the meteorological input parameters plus the calculated meteorological output values for each line of data in the meteorological data file. The frequency of occurrence of each condition is also reported. The input parameters are in the left-hand columns of the file, and the calculated output parameters in the right-hand columns.

6.1.7 .out.met file

The *out.met* file contains a copy of the input meteorological data, cut down to the subset being used, with the effects of the choice of wind sector size taken into account.

This is done as follows:

- For wind not in sectors, the data in the *.out.met* file is identical to the input meteorological data file.
- For wind in sectors of less than 15^o, the data in the *.out.met* file is identical to the input meteorological data, except that the wind directions are altered to take into account the cycling applied.
- For wind in sectors of equal to or greater than 15^o, for each line of input meteorological data there are five lines of data in the *.out.met* file. These lines are a copy of the input meteorological data except that the wind directions are altered to give the five directions used to represent that sector and the frequency is divided equally between the five lines.

6.1.8 .!01 files

The *.!01, .!02,* etc. files are files used by the **Line Plot** utility of ADMS-Urban (see Section 6.3). They are only produced for short-term, single point source calculations with gridded output. The number in the extension refers to the line of meteorological data to which the results correspond.

6.1.9 .i01 files

The *.i01, .i02* etc. files contain the ground level plume centreline concentration as a function of distance downstream. These files are only produced for short-term, single point source calculations with gridded output.

6.1.10 .j01 files

The *.j01, .j02* etc. files contain the dry, wet and total deposition as a function of distance downstream. These files are only produced for short-term, single point source calculations with gridded output.

6.1.11 .k01 files

The *.k01, .k02* etc. files contain the plume centreline, mean height, transverse and vertical spread, plus PINV (the fraction of material above the boundary layer) as a function of distance downstream. These files are only produced for short-term, single point source calculations with gridded output.

6.1.12 .nc file

The *.nc* file is a netCDF format file containing the concentration and dry or wet deposition (if selected) data for each unique pollutant and averaging time combination selected for long-term output entered in the **Output** screen of the ADMS-Urban interface. For averaging times greater than one hour, the averaging time used in creating this file is one hour. Percentile or exceedence data are not output to the netCDF file. As well as containing the concentration (and deposition) data, the netCDF file also contains basic meteorology relating to each of the lines in the meteorological data file. The *.nc* file is created if the **Comprehensive output file** option is selected on the **Output** screen of the ADMS-Urban interface.

The *.nc* file is in netCDF format with the dimensions, variables and attributes as described in **Tables 6.3** to **6.5**.

Name	**Description**
nPoints_XY	Number of lowest level grid points, intelligent grid points, nested grid points and all specified points (XY output points)
nPoints_XYZ	Total number of all output points (XYZ output points)
nMetLines	Number of met. lines
nGroups	Number of groups
nDatasets	Number of datasets, with one dataset for each long-term pollutant, for each averaging time less than or equal to one hour, for concentration, dry deposition (if selected) and wet deposition (if selected)
OutputOptionsUsed	*Number of possible output options (=3)*
NumberOfTypesOfOutputPoint	*Number of types of output point (=5)*
MetLineStringLength	*Maximum length of the met line description (=11)*
DatasetNameStringLength	*Maximum length of the dataset names (=44)*
GroupStringLength	*Maximum length of the group names (=30)*
PointNameStringLength	*Maximum length of an output point name (=44)*
OutputOptionsStringLength	*Maximum length of an output option name (=14)*
OutputPointTypesLength	*Maximum length of an output point type name (=16)*

Table 6.3 – Dimensions in the *.nc* file. The dimensions in italics are dimensions required as part of the netCDF format and are the same for all files.

Name	Dimensions	Description
PointX_XY	nPoints_XY	X-coordinate of XY output points
PointY_XY	nPoints_XY	Y-coordinate of XY output points
PointName_XY	nPoints_XY, *PointNameStringLength*	Name of output point for XY output points • "Grid, grid" for standard grid points • "Grid, nested, <source>" for nested grid points, where <source> is the name of one of the sources associated with this nested grid • "Grid, intelligent" for intelligent grid points* • "Receptor, *name*" for specified points, where *name* is the name entered by the user for the point
PointX_XYZ	nPoints_XYZ	X-coordinate of XYZ output points
PointY_XYZ	nPoints_XYZ	Y-coordinate of XYZ output points
PointZ_XYZ	nPoints_XYZ	Z-coordinate of XYZ output points
PointName_XYZ	nPointsXYZ	Name of output point for XYZ output points, see PointName_XY
Output_Points_Type	NumberOfTypesOfOutputPoint, OutputPointTypesLength	Description of the types of output point
Number_Of_Output_ Points_Of_Each_Type	NumberOfTypesOfOutputPoint	Number of output points of each type
Met_Line	nMetLines, *MetLineStringLength*	Date of *.met* line or *.met* line number for each *.met* line
Met_Freq	nMetLines	Frequency for each *.met* line
Met_Uat10m	nMetLines	10 m wind speed for each *.met* line
Met_Phi	nMetLines	Wind direction for each *.met* line
Met_H_over_LMO	nMetLines	Boundary layer height divided by Monin-Obukhov length for each met. line
Output_Options_ Description	OutputOptionsUsed, OutputOptionsStringLength	Names of the output options
Output_Options_Used	OutputOptionsUsed	Indicator of each output option is used (1=yes, 0=no)
Group	nGroups, *GroupStringLength*	Name of each output group
DatasetNames	nDatasets, *DatasetNameStringLength*	Descriptive name for each dataset
Dataset#	nMetLines, nGroups, nPoints	Datasets containing concentration or deposition data for each long-term pollutant and averaging time less than or equal to one hour.

Table 6.4 – Variables in the *.nc* file. Dimensions are listed with the fastest varying last and the string length dimension is indicated in italics for character variables. For the datasets (Dataset#) nPoints is either nPoints_XYZ for concentration datasets or nPoints_XY for deposition datasets. *Interpolated intelligent grid points are not included in this file.

Associated with	Name	Description
Global	File_type	Indicator that this is a comprehensive output file
Global	File_version	Version number of the comprehensive output file format (currently 2.0)
Global	Model_name	Name of the model which created the file
Global	Model_Version	Version number of the model which created the file
Global	Model_Build_Number	Build number of the model which created the file
Global	Model_Release_Date	Build date of the model that created the file
Global	Date_Created	The date the file was created
Global	Time_Created	The time the file was created
Global	Site_name	The site name entered in the interface
Global	Project_name	The project name entered in the interface
Global	Is_Data_Hourly_ Sequential	Indicates if **Met. data are hourly sequential** is selected in the interface
Global	Grid_Description	Description of the output grid
Variables	Units	Units of the variable
Dataset#	Dataset_Name	Name of the dataset
Dataset#	Pollutant_Name	Name of the pollutant for the dataset
Dataset#	Output_Type	Type of output for the dataset (concentration, dry deposition or wet deposition)
Dataset#	Av_Time_s	Averaging time for the dataset in seconds
Dataset#	Units	Units of the dataset
Dataset#	ugm3_to_ppb_conv_ factor	$\mu g/m^3$ to ppb conversion factor for the pollutant associated with the dataset.

Table 6.5 – Global and variable attributes in the *.nc* file.

6.1.13 .pro file

The *.pro* file contains the boundary layer properties output for a series of heights for each meteorological data line. The *.pro* file is produced if the **Boundary layer profile output** option is enabled using an additional input file; refer to Section 4.23 for more details. The possible output variables are listed in **Table 6.6**.

Variable	Description
Met line no.	Met line number
Year	Year
Day	Julian day
Hour	Hour
z(m)	Height of output
z/h	Height of output as fraction of boundary layer height
U(z) (m/s)	Mean wind speed
Phi(z) (deg)	Wind direction
Temp(z) (C)	Temperature (Celsius)
SigU(z) (m/s)	Longitudinal turbulence
SigV(z) (m/s)	Transverse turbulence
SigW(z) (m/s)	Vertical turbulence
DUDZ(z) ((m/s)/m)	Gradient of mean wind speed
Rlv(z) (m)	Transverse turbulence length scale
Rlw(z) (m)	Vertical turbulence length scale
TL(z) (s)	Lagrangian time scale
Eps(z) (m2/s3)	Energy dissipation rate
Pot Temp(z) (K)	Potential temperature (Kelvin)
Press(z) (mbar)	Pressure
N(z) (1/s)	Buoyancy frequency

Table 6.6 – Variables contained in a *.pro* file.

6.1.14 Buildings: .bef, .bld and .bwk files

These files are related to the **Buildings** module.

The *.bef* file (effective building) contains the coordinates of the effective building for each source and line of meteorological data. This file can be easily plotted in the Mapper using the **Add layer** option.

The *.bld* file (buildings) contains the dimensions of buildings input to the model as well as some parameters calculated during the run (including the region affected by the presence of buildings, the cavity and cavity concentration). Output values are usually given for the first 24 lines of meteorological data only, in order to prevent the file from becoming too large. However, the file can be extended if required, as described in Section 4.26.

The *.bwk* file (building wake) contains the cavity and wake height as a function of downstream distance from the upwind edge of the effective building. It is only produced for single (point) source short-term runs with buildings.

6.1.15 Deposition: .dep file

The *.dep* file (deposition) is related to the deposition option. It contains the values of deposition variables (e.g. terminal velocity) input to the model as well as those calculated during the run.

Output values are usually given for the first 24 lines of meteorological data only, to prevent the output file from becoming too large. However, the file can be extended if required, as described in Section 4.26.

6.1.16 Spatial splitting: .prc file

The *.prc* file (per region counts) is related to the spatial splitting option. Created only in verification mode and when spatial splitting is set to every region, the file contains the region name, region number, the number of output points per region and the number of sources being modelled (including the buffer region) for each source type per region.

6.1.17 Urban canopy flow field output: .W01 files

The *.W01*, *.W02*, etc, files are related to the **Urban canopy flow** option and are only created for short-term calculations with an output grid if flow field output is selected. The *.W01*, *.W02*, etc. files contain flow field data at the locations defined for the output grid. These files are only created for the first 24 lines of met. data.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **U(m/s)**: Component of wind velocity in west-east direction
- **V(m/s)**: Component of wind velocity in south-north direction
- **W(m/s)**: Vertical component of wind velocity (relative to sea level)
- **Ux(m/s)**: Longitudinal component of wind velocity (i.e. in the direction of the upstream wind; the direction specified for the wind in the meteorological data)
- **Uy(m/s)**: Transverse component of wind velocity (i.e. perpendicular to the direction of the upstream wind)
- **Angle(deg)**: Wind direction (degrees measured anticlockwise, 0^o = wind from west)
- **Magnitude of horizontal wind (m/s)**: Magnitude of horizontal wind velocity ($=\sqrt{U^2+V^2}$)

6.1.18 Urban canopy flow field output: .wlt file

The *.wlt* file is related to the **Urban canopy flow** option and is only created for long-term calculations with an output grid if flow field output is selected. The *.wlt* file contains flow field data, averaged over all the lines of met. data, at the locations defined for the output grid.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **U(m/s)**: Component of mean wind velocity in west-east direction
- **V(m/s)**: Component of mean wind velocity in south-north direction
- **W(m/s)**: Vertical component of mean wind velocity (relative to sea level)
- **Mean horizontal wind speed (m/s)**: Mean value of the horizontal wind speed ($=\overline{\sqrt{U^2+V^2}}$)
- **Magnitude of mean horizontal wind vector (m/s)**: Magnitude of the mean horizontal wind vector ($=\sqrt{\overline{U}^2+\overline{V}^2}$)
- **Angle(deg)**: Direction of the mean horizontal wind vector (degrees measured anticlockwise, 0^o = wind from west).
- **Magnitude of mean horizontal wind vector (m/s)**: Magnitude of the mean horizontal wind vector (Note that this column appears twice in the file)

6.1.19 Urban canopy flow field output: .T01 files

The *.T01*, *.T02*, etc, files are related to the **Urban canopy flow** option and are only created for short-term calculations with an output grid if flow field output is selected. The *.T01*, *.T02*, etc. files contain turbulence data at the locations defined for the output grid. These files are only created for the first 24 lines of met. data.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **Sig-U(m/s)**: Longitudinal turbulence σ_u
- **Sig-V(m/s)**: Transverse turbulence σ_v
- **Sig-W(m/s)**: Vertical turbulence σ_w

6.1.20 Urban canopy flow field output: .tlt file

The *.tlt* file is related to the **Urban canopy flow** option and is only created for long-term calculations with an output grid if flow field output is selected. The *.tlt* file contains turbulence data, averaged over all the lines of met. data, at the locations defined for the output grid.

Variables are:

- **X(m)**: X coordinate of output point

- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **Sig-U(m/s)**: Mean longitudinal turbulence σ_u
- **Sig-V(m/s)**: Mean transverse turbulence σ_v
- **Sig-W(m/s)**: Mean vertical turbulence σ_w

6.1.21 Urban canopy flow field output: .zst file

The *.zst* file is related to the **Urban canopy flow** option and is only created for short-term calculations with specified points if flow field output is selected. The *.zst* file contains flow field and turbulence data at the specified points for each line of met. data.

Variables are:

- **Year**: Year of the current line of met. data
- **Day**: Julian day number of the current line of met. data
- **Hour**: Hour of the current line of met. data
- **Receptor name**: Name of the specified point
- **X(m)**: X coordinate of specified point
- **Y(m)**: Y coordinate of specified point
- **Z(m)**: Height above terrain of specified point
- **U(m/s)**: Component of wind velocity in west-east direction
- **V(m/s)**: Component of wind velocity in south-north direction
- **W(m/s)**: Vertical component of wind velocity (relative to sea level)
- **Ux(m/s)**: Longitudinal component of wind velocity (i.e. in the direction of the upstream wind; the direction specified for the wind in the meteorological data)
- **Uy(m/s)**: Transverse component of wind velocity (i.e. perpendicular to the direction of the upstream wind)
- **Angle(deg)**: Wind direction (degrees measured anticlockwise, 0^o = wind from west)
- **Magnitude of horizontal wind (m/s)**: Magnitude of horizontal wind velocity ($=\sqrt{U^2+V^2}$)
- **Sig-U(m/s)**: Longitudinal turbulence σ_u
- **Sig-V(m/s)**: Transverse turbulence σ_v
- **Sig-W(m/s)**: Vertical turbulence σ_w

6.1.22 Urban canopy flow field output: .zlt file

The *.zlt* file is related to the **Urban canopy flow** option and is only created for long-term calculations with specified points if flow field output is selected. The *.zlt* file contains flow field and turbulence data, averaged over all of the lines of met. data, at each of the specified points.

Variables are:

- **Receptor name**: Name of the specified point
- **X(m)**: X coordinate of specified point
- **Y(m)**: Y coordinate of specified point
- **Z(m)**: Height above terrain of specified point
- **U(m/s)**: Component of mean wind velocity in west-east direction
- **V(m/s)**: Component of mean wind velocity in south-north direction
- **W(m/s)**: Vertical component of mean wind velocity (relative to sea level)
- **Mean horizontal wind speed (m/s)**: Mean value of the horizontal wind speed ($= \overline{\sqrt{U^2 + V^2}}$)
- **Magnitude of mean horizontal wind vector (m/s)**: Magnitude of the mean horizontal wind vector ($= \sqrt{\overline{U}^2 + \overline{V}^2}$)
- **Sig-U(m/s)**: Mean longitudinal turbulence σ_u
- **Sig-V(m/s)**: Mean transverse turbulence σ_v
- **Sig-W(m/s)**: Mean vertical turbulence σ_w

6.1.23 .rds file

The *.rds* file (roads geometry) contains, for each road source for each segment of the source, the start and end coordinates of: the road centreline; the left and right edges of the road carriageway; the left and right canyon wall edges (advanced street canyon sources only), and; the relevant building height. The *.rds* file is produced if the **Segment geometry** option is enabled, and the relevant check box ticked, in the additional input file; refer to Section 4.6 for more details.

6.1.24 .asp file

An *.asp* file (additional specified points) is created if the model is run using the **Run model to create .asp file** additional input file option (see Section 4.28). The option allows for the creation of two different files described below.

6.1.24.1 All standard output points

This *.asp* file contains a list of all the output points that would be used in a full model run for a given model setup. Each line contains the point name, X coordinate, Y coordinate and Z coordinate. For specified points, the point names will be the same as those specified in the **Grids** screen. For standard and source-oriented grid points, the point names are automatically assigned. For each output point type present in the original model file, the points in the *.asp* file will be listed in the following order:

- Standard grid points (point names are prefixed with '|G|'), followed by
- specified points (point names as given in the **Grids** screen), followed by
- source-oriented grid points for point, area and volume sources (point names are prefixed with '|N|'), followed by
- source-oriented grid points for road and line sources (point names are prefixed with '|I|').

6.1.24.2 Roadside points only

This *.asp* file contains a list of all the output points that are located within a specified distance from the edges of road sources. Each line includes the road name—appended with a _L or _R suffix to indicate whether the point lies on the left or right side of the road—followed by the X, Y, and Z coordinates.

6.1.25 .flx file

A *.flx* file is created if the model is run using the **Horizontal concentration flux** additional input file option (see Section 4.27). For each receptor, the accumulated flux for a specific meteorological line and pollutant is calculated by multiplying the hourly average concentration (including background levels) by the local horizontal wind speed (accounting for terrain but not buildings), and then by 3600 (seconds in an hour). This value is assigned to the wind sector matching the local wind direction; all other sectors are assigned zero for that hour. The *.flx* file contains the total of these hourly values across all valid hours.

The *.flx* file includes the following variables:

- **Receptor name**: Name of specified point
- **X(m)**: X coordinate of specified point
- **Y(m)**: Y coordinate of specified point
- **Z(m)**: Z coordinate of specified point
- **Total valid hours**: Total number of valid hours used in the accumulated flux calculation for this receptor.
- **Num valid hrs (###.# deg)**: Number of valid hours when the wind at this receptor was blowing from a given sector, for each wind sector. The value in brackets gives the mid-point of the wind sector, e.g. 270.0°. The sum of these values over all wind sectors will equal the value in the 'Total valid hours' column.
- **Mean horizontal wind speed (m/s) (###.# deg)**: Mean horizontal wind speed (m/s) over hours when the wind at this receptor was blowing from a given sector, for each wind sector.
- **AccHFlux (###.# deg)**: Accumulated horizontal concentration flux over hours

when the wind at this receptor was blowing from a given sector, for each wind sector. A set of columns are given per relevant pollutant. This has units of [conc]·m, where [conc] is the user-selected output units for that pollutant. Note that it is also possible to calculate an indicative average concentration over hours when the wind was blowing from a given sector by dividing this value by the corresponding ‘Mean horizontal wind speed’ value × the corresponding ‘Num valid hrs’ value × 3600.

6.2 Contour and flow field plots

This section outlines how to produce contour and vector plots from gridded ADMS-Urban output data using the **2-D Output Plotter** utility, launched from the ADMS-Urban interface via the **Results** menu. It is possible to produce contour and vector plots in either the Mapper or Surfer (if installed). Selecting one of these brings up the screen shown in **Figure 6.4**.

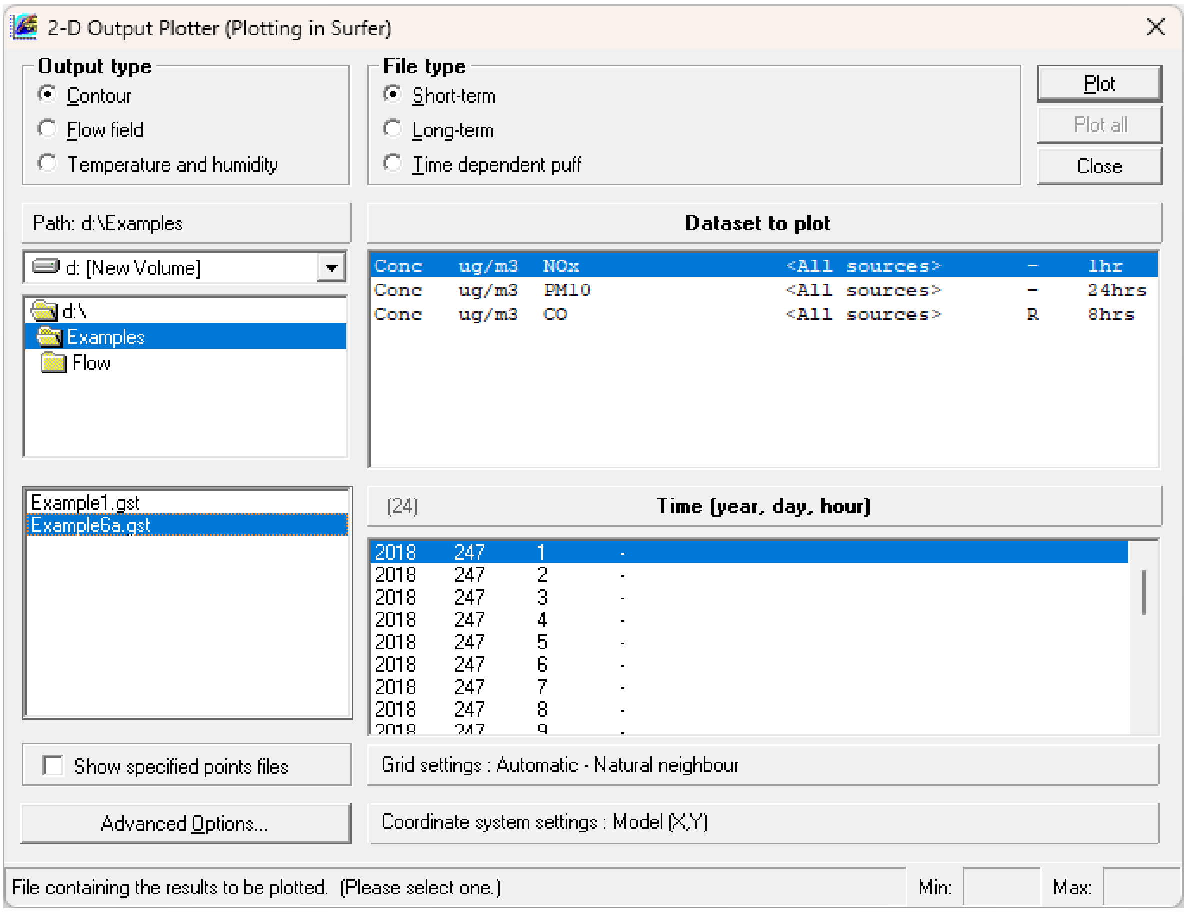


Figure 6.4 – The **2-D Output Plotter** screen.

The **2-D Output plotter** allows for plotting many different types of gridded output from ADMS-Urban. The **Output type** should first be selected:

- **Contour**: for plotting contour plots of data output to the *.gst* or *.glt* file: concentration and deposition.
- **Flow field**: for plotting contour or vector plots of flow field output from the **Urban canopy flow** options.
- **Temperature and humidity**: this option relates to output from the Temperature and Humidity model extension to ADMS-Urban and is not relevant for standard output.

6.2.1 Contour

When **Contour** is selected as the **Output type** the **2-D Contour plotter** will look as shown in **Figure 6.4**. Here are the steps to follow in order to create a contour plot of data:

Step 1 Choose the **File type** from **Short term** and **Long term**. This defines the types of files listed in the lower left hand box. If **Short term** is chosen, all the files with the extension *.gst* or *.levels.gst* (in the chosen directory) are listed and if **Long term** is chosen, all the files with the extension *.glt* or *.levels.glt* are listed.

Checking the ***Show specified points files*** *box also allows for specified points files (*.pst, .plt*) to be plotted, providing they contain sufficient points.*

Step 2 Select the appropriate folder and click on the name of the file containing the data to plot. Right-clicking on **Path:** allows the path of a file or directory to be pasted in. Files or directories can also be dragged from Explorer onto **Path:** to use that file or directory.

Step 3 Click on the dataset to plot (**Dataset to Plot** box), and if relevant also click on the time for which the data are to be plotted (**Time (year, day, hour)** box).

The **Dataset to Plot** box shows a list of all the variables that can be used for the plot. Each record in the list shows the type of output (e.g. 'Conc' for concentration), the units of output (e.g. 'ppm'), the pollutant name (e.g. 'CO'), the source or group name (e.g. 'Power Station'), whether it is a rolling average ('R'), maximum daily output ('M'), both ('RM') or neither ('-'), and the averaging time (e.g. '1hr'). For *.levels.glt* and *.levels.gst* files, an additional vertical height column is displayed (e.g. 'z=10.0m', as shown in **Figure 6.5**).

For short-term runs only, a **Time (year, day, hour)** box is also displayed. The box displays the time (year, day, hour) as given in the meteorological data irrespective of whether these data are entered from a file or by hand. If no time information is given in the meteorological data file, numbers corresponding to each consecutive meteorological condition are used. For example, numbers 1 to 7, corresponding to meteorological conditions A to G respectively, are listed when the example file *R91a-g.met* is used.

Step 4 Click on the **Advanced Options...** button to set some properties of the plot (optional – refer to Section 6.2.3 for details).

Step 5 Click on **Plot** to plot the selected data in the Mapper / Surfer.

If the selected file is one of a set of output files produced using the ***Spatial splitting*** *option in* ***Every region*** *mode (refer to Section 4.24), it is also possible to click on* ***Plot all*** *to plot the same dataset across all files (regions) in the set simultaneously.*

The ADMS-Urban output file is converted to a grid file (.grd*) for the Mapper / Surfer to plot. Save this to an appropriate location. Once the drawing has been saved, the* .grd *file can be deleted as it is no longer required.*

The **Close** button closes the **2-D Output Plotter**.

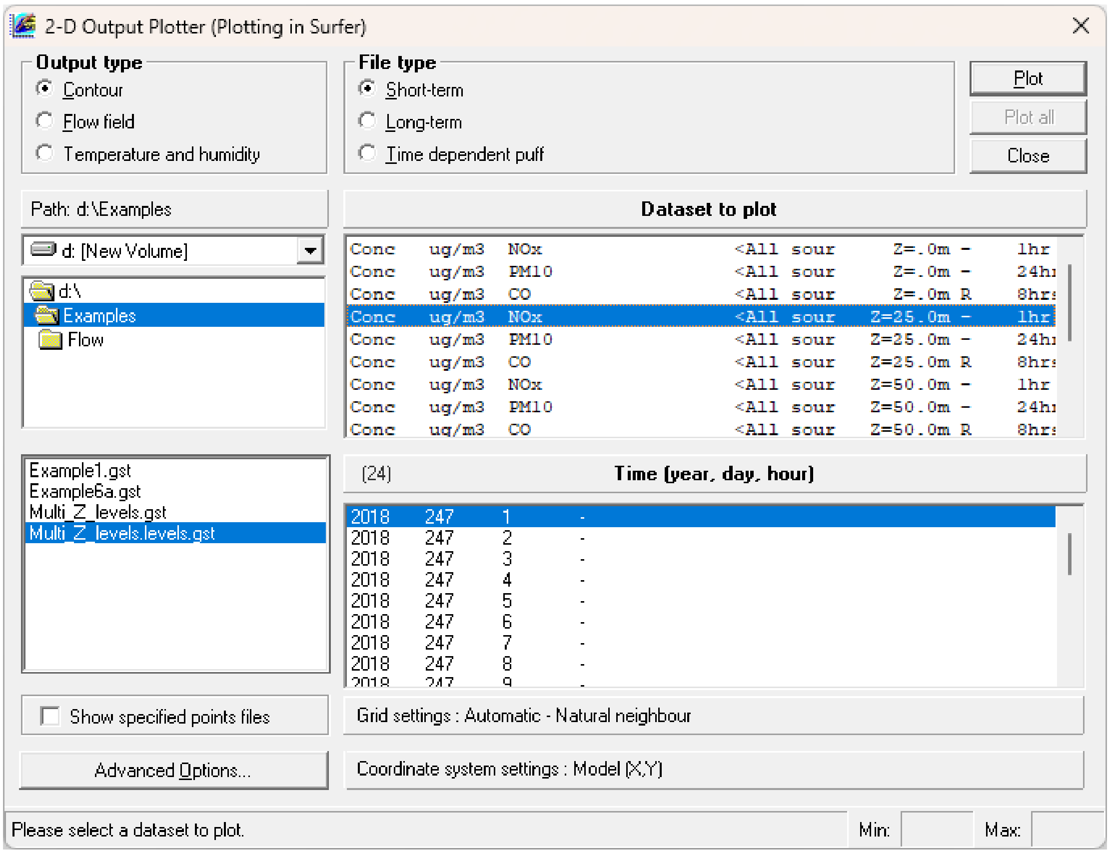


Figure 6.5 – The **2-D Output Plotter** screen showing output at multiple z-levels.

If you are creating many Surfer contour plots then you may wish to use the Surfer Automation tool. Details of this tool can be found in the Surfer Automation and Slideshow Creator Tools User Guide (SurferAutomationandSlideshowCreator_UserGuide.pdf) located in the <install_path>\Documents *directory.*

6.2.2 Flow field

The **Urban canopy flow** option includes the ability to output flow fields. The flow field output files contain the wind and turbulence fields:

- for each line of meteorological data if short-term output is selected. These output files are described in Sections 6.1.17 and 6.1.19.
- as averages of these quantities over all meteorological data lines if long-term is selected. These output files are described in Sections 6.1.18 and 6.1.20.

These files can be plotted by selecting **Flow field** in the **2-D Output plotter**, shown in **Figure 6.6**.

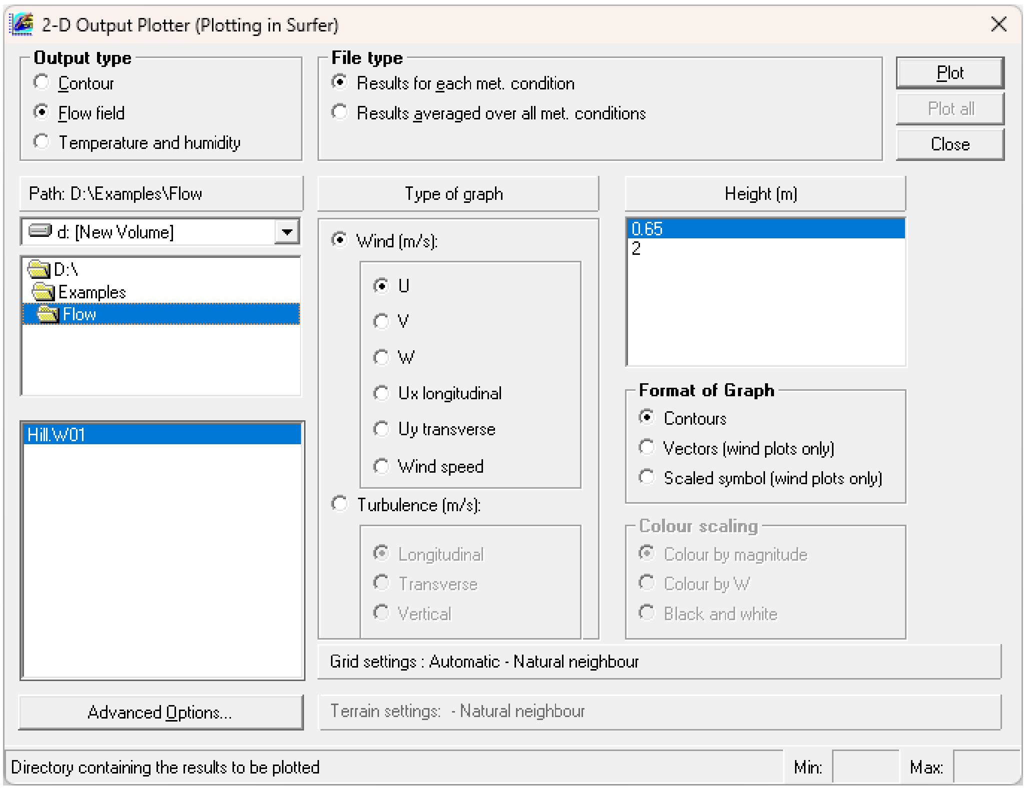


Figure 6.6 – The **2-D Output plotter** utility with **Flow field** selected.

In order to create a flow field plot, proceed as follows:

Step 1 Select whether to plot **Results for each met. condition** (short-term) or **Results averaged over all met. conditions** (long-term).

Step 2 In the **Type of graph** box, select the appropriate data to plot (see further on for a description of all variables available).

Step 3 Select whether a contour plot, a vector plot or a scaled symbol plot is to be made, from the **Format of Graph** box.

Step 4 If making a **Vectors (wind plots only)** plot, select the colour scaling to use from the **Colour scaling** box.

Step 5 From the boxes on the left, select the folder and then the file that contains the data to plot. Right-clicking on **Path:** allows the path of a file or directory to be pasted in. Files or directories can also be dragged from Explorer onto **Path:** to use that file or directory.

Step 6 Choose the height from the **Height (m)** box.

Note that these heights are measured from the surface of the terrain.

Step 7 Click on **Plot** to display the plot in the Mapper / Surfer.

*If the selected file is one of a set of output files produced using the **Spatial splitting** option in **Every region** mode (refer to Section 4.24), it is also possible to click on **Plot all** to plot the same dataset across all files (regions) in the set simultaneously.*

An example of a contour plot is shown in **Figure 6.7** and an example of a vector plot in **Figure 6.8**.

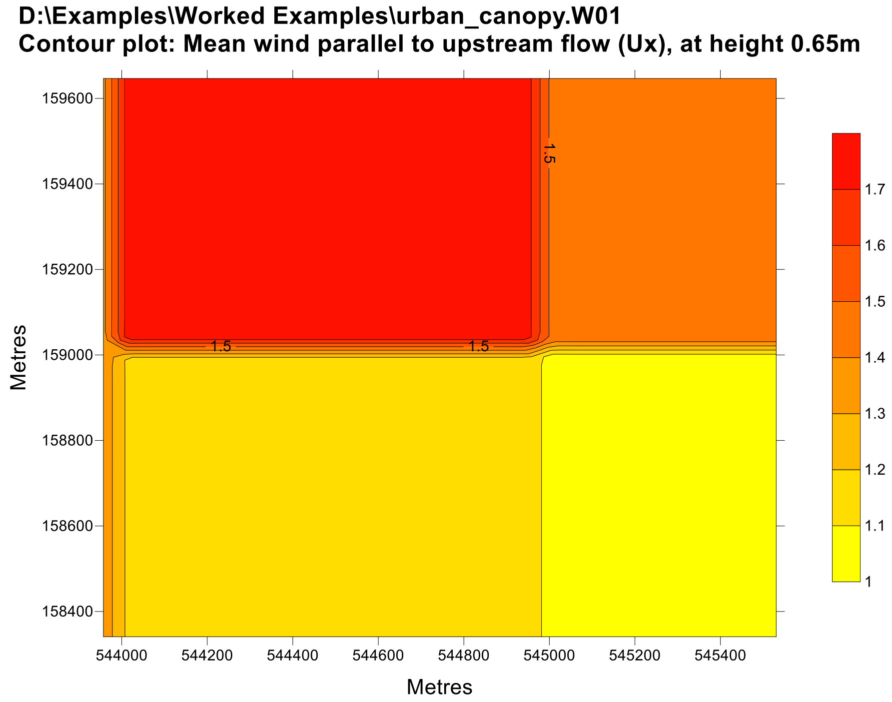


Figure 6.7 – Example flow field contour plot.

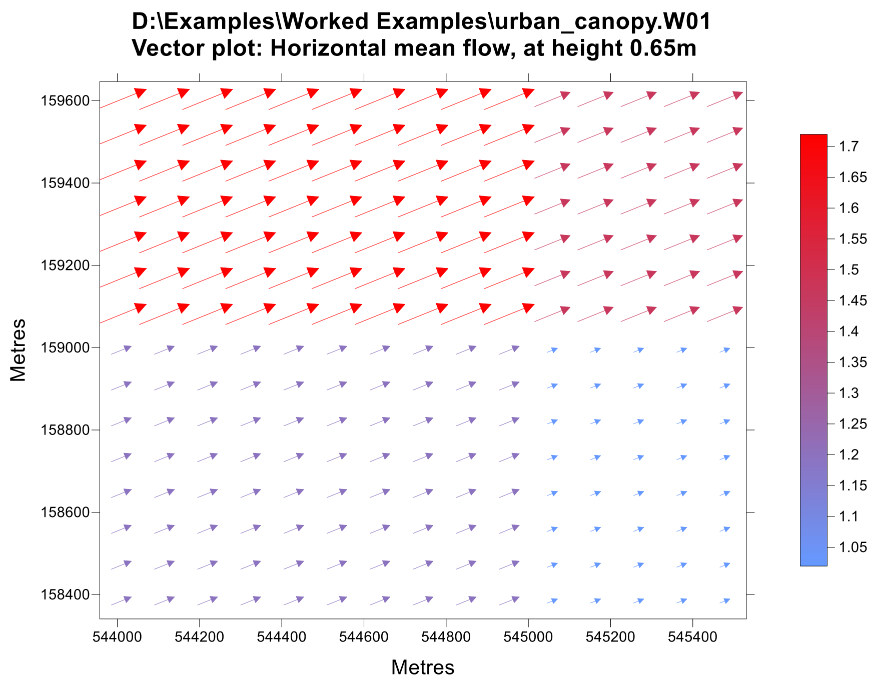


Figure 6.8 – Example flow field vector plot with the colour scaled by vertical velocity.

Type of graph

This section lists all the possible variables that can plotted:

Wind (ms^{-1})

1. **U**: component of wind velocity in west-east direction.

2. **V**: component of wind velocity in south-north direction.
3. **W**: vertical component of wind velocity (relative to sea level).
4. **Ux longitudinal**: longitudinal component of wind velocity, i.e. in the direction of the upstream wind (the wind direction specified in the meteorological data). Short-term output only.
5. **Uy transverse**: transverse component of wind velocity, i.e. perpendicular to the direction of the upstream wind. Short-term output only.
6. **Mean Speed**: Mean value of the horizontal wind speed. Long-term output only.
7. **Mean vector magnitude**: Magnitude of the mean horizontal wind vector (refer to Section 6.1.18 for definition). Long-term output only.
8. **Wind Speed**: horizontal wind velocity.

Turbulence (ms^{-1})

9. **Longitudinal**: longitudinal turbulence parameter (σ_u).
10. **Transverse**: transverse turbulence parameter (σ_v).
11. **Vertical**: vertical turbulence parameter (σ_w).

Vector plot

The horizontal wind, i.e. the vectors (U,V), can also be displayed as a vector plot by selecting **Wind (ms^{-1})** in the **Type of graph** box and **Vectors (wind plots only)** in the **Format of Graph** box. A choice of colour scaling is then available in the **Colour scaling** box:

- **Colour by magnitude**: the vectors are coloured by the horizontal wind speed
- **Colour by W**: the vectors are coloured by the vertical velocity
- **Black and white**: the vectors are not coloured

An example of a vector plot is shown in **Figure 6.8**.

Scaled symbol plot

The horizontal wind, i.e. the vectors (U,V), can also be displayed as a scaled symbol plot by selecting **Wind (ms^{-1})** in the **Type of graph** box and **Scaled symbol (wind plots only)** in the **Format of Graph** box. The scaled symbol plot is similar to the vector plot but plots arrows at each of the locations in the output file rather than gridding the data as is carried out in the vector plot. An example plot is shown in **Figure 6.9**

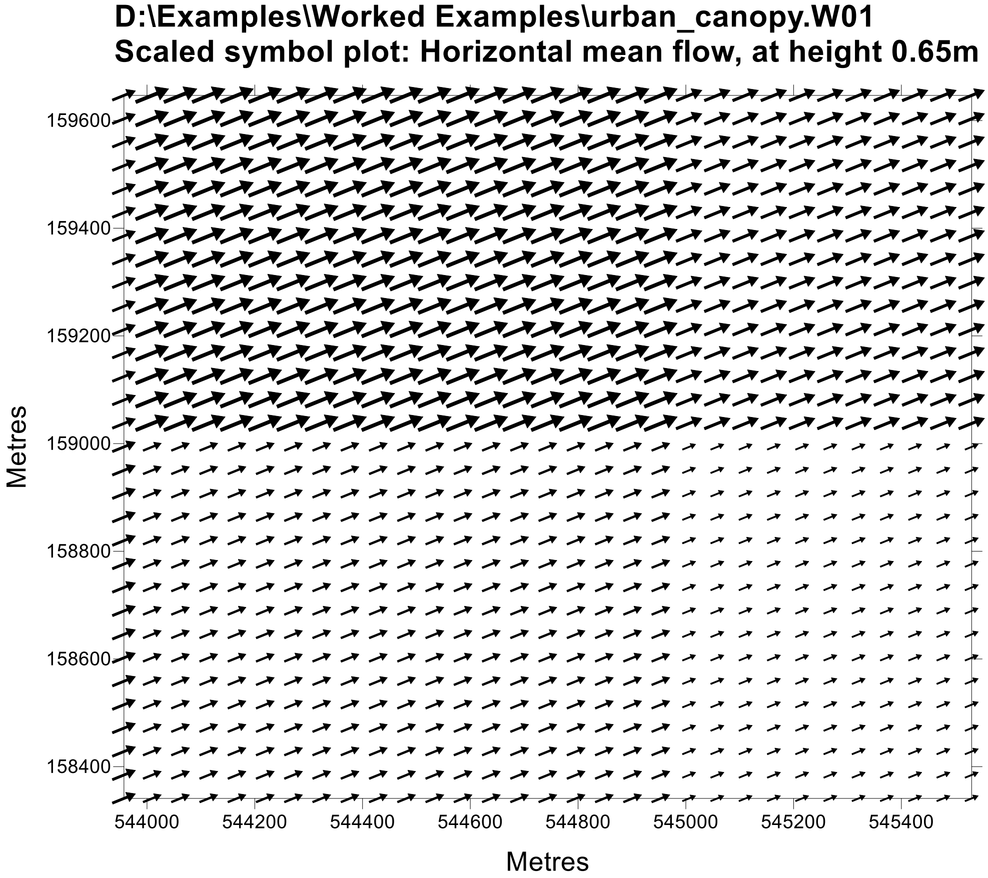


Figure 6.9 − Example scaled symbol plot.

6.2.3 Advanced options

The **Advanced Options...** button gives the user a number of options to enhance the plot. Clicking on it brings up the **Advanced Options** screen shown in **Figure 6.10**. From that screen, you can set the contour levels, the coordinate system of the grid, the gridding options, and whether to overlay the terrain file.

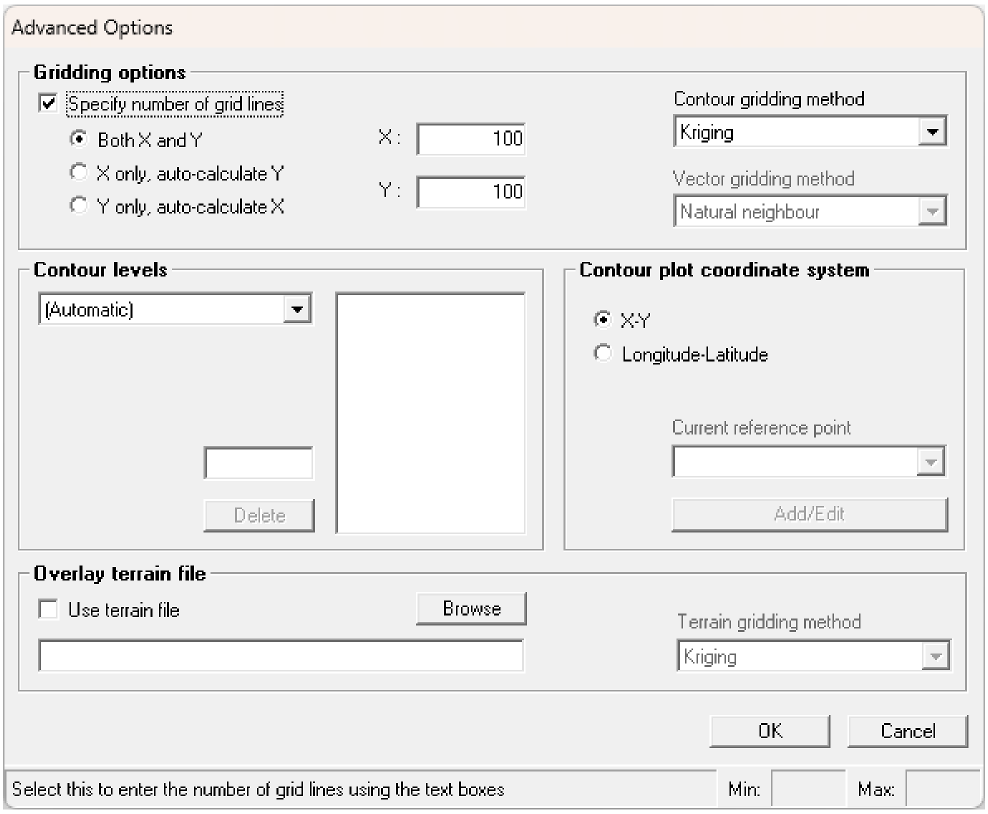


Figure 6.10 – The **Advanced Contour Options** screen.

When you have finished with the **Advanced Options** screen, click on the **OK** button to return to the **2-D Output Plotter** main screen. Once the required information has been selected in this screen, click on **Plot** to create the plot.

Some of the options on the ***Advanced Options*** *screen are only available for some plot types.*

Further help with using Surfer is given in Appendix E.

Gridding options

A number of different gridding methods for a contour plot can be selected from the **Contour Gridding method** drop-down list. The available gridding methods will depend on whether you are plotting in Surfer or the Mapper. The Surfer gridding methods are described in the Surfer user guide.

To specify the grid resolution, tick the **Specify number of grid lines** box. Select the appropriate option to specify the number of grid lines in both the X and Y directions, or in one direction only. If the number of lines is specified in one direction only, the contouring interface will ensure that the resolution is the same in the X and Y directions. Enter the number of grid lines in the **X:** and/or **Y:** boxes.

If the ***Specify number of grid lines*** *option is not selected, the* ***2-D Output Plotter*** *will choose the resolution based on the resolution of the results to be plotted.*

Contour levels

To choose the contour levels to plot, click on **(User-specified)** from the drop-down list. For each level you want to plot (up to a maximum of 40 levels), type the level value in the small box above the **Delete** button and press the space bar to enter it into the larger box. If you want to delete any level, click on that number in the larger box and click on the **Delete** button.

Contour plot coordinate system

To convert your output from X-Y coordinates to longitude-latitude coordinates, select **Longitude-Latitude** and then click on **Add/Edit**. You must supply the X-Y and longitude-latitude coordinates of a reference point, as shown in **Figure 6.11** (left). Longitude and latitude should typically be given to 6 decimal places.

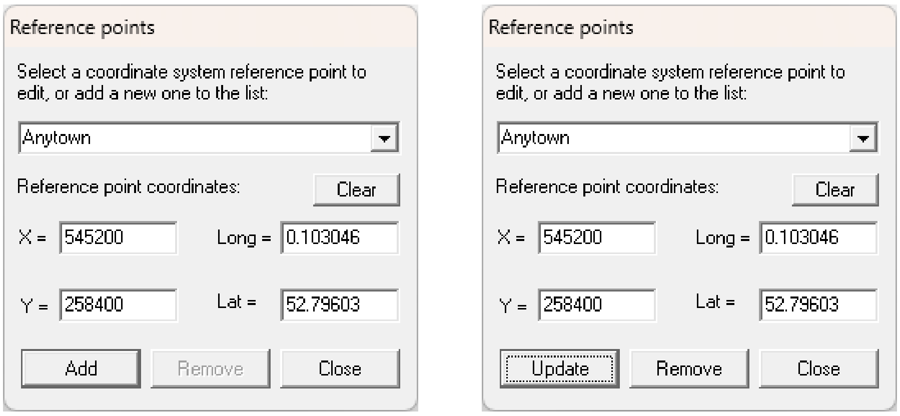


Figure 6.11 − Left: entering coordinates of a datum point. Right: revising and updating coordinates of a datum point.

In the UK, the X-Y coordinates will typically be national grid (OSGB) references.

Click on **Add** to store the coordinate data and on **Close** to leave the screen. Datum information can be deleted by clicking on **Remove** or revised by entering the new data and then clicking on **Update** as in **Figure 6.11** (right).

If you have entered several datum points, because you are studying sites in different locations that use different local co-ordinate systems, choose the appropriate datum point from the drop down list.

Overlay terrain file

If the effects of hills have been included in the model run, it may be appropriate to overlay the plot on the terrain data. To do this, check the **Use terrain file** box, then click on the **Browse** button to locate the *.ter* file. The gridding method for the terrain plot can be chosen from the **Terrain gridding method** list. You should ensure that the terrain file is the right one for the plot being created, i.e. the plot is contained within the terrain contours.

The **Grid settings** and the **Coordinate system** or **Terrain settings** selected on the **Advanced Options** screen are displayed at the bottom of the main screen, as shown at the bottom of **Figure 6.6**.

6.3 Line plots (ADMS Line Plotter)

ADMS-Urban has a built-in line plot utility called **ADMS Line Plotter**. This can only be used with model runs where short-term gridded output for a single point source is selected. Concentration and plume dispersion data are plotted along a line of points extending in the downstream direction from the source, i.e. along the 'plume centreline', as far as the downstream edge of the output grid. This utility is only available for point sources, because the plume 'centreline' is not well-defined for other source types.

The **ADMS Line Plotter** uses a specific type of file to produce the plots. The files have the extension *.!n* where *n* is a two-digit number corresponding to the line of meteorological data of the *.met* file used in the model run (see Section 6.1.8). These files contain the same information as the *.i01, .j01 and .k01* files but they have been formatted so that they can be read by the graphical interface. *They are not intended to be read and/or interpreted by the user.*

To launch the **ADMS Line Plotter**, select **Results**, **Line Plot** from the main ADMS-Urban interface. This brings up the screen shown in **Figure 6.12**.

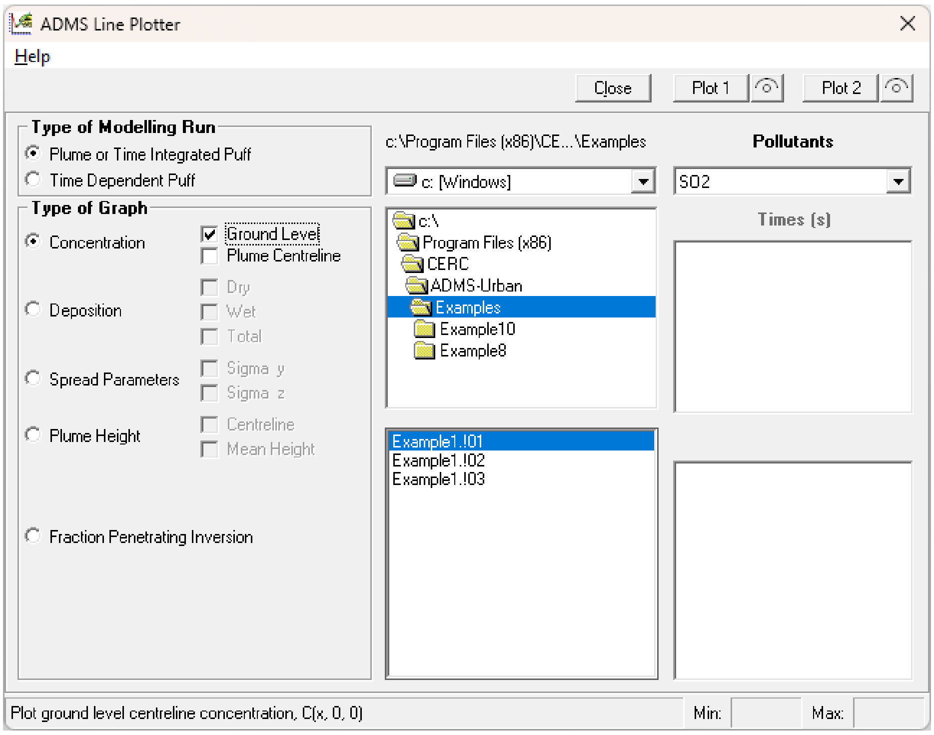


Figure 6.12 − The **ADMS Line Plotter** utility screen.

6.3.1 Main interface

The utility offers two separate windows for plotting selected data, accessible via the **Plot 1** and **Plot 2** buttons (see further). The **Close** button closes the utility. It is possible to paste the path to the folder containing the files directly into the interface, or open the folder in Explorer by right-clicking the path next to the **Pollutants** label. In the file display box, you can refresh the file list by right-clicking and selecting the **Refresh** option.

To use the **ADMS Line Plotter**, proceed as follows:

Step 1 In the **Type of Modelling Run** box ensure that the **Plume or Time Integrated Puff** option is selected. The *.!n* files for the current path are displayed in the lower box.

Step 2 Select the appropriate folder and click on the file containing the data to plot.

*Up to ten output files from the list displayed in the screen can be plotted at the same time. Use the **SHIFT** and **CTRL** keys to select which files to plot.*

Step 3 In the **Type of Graph** area, choose the variable to plot (see below).

More than one variable can be plotted at the same time.

Step 4 Click on the **Plot 1** button. A plot window will automatically open (see **Figure 6.13**).

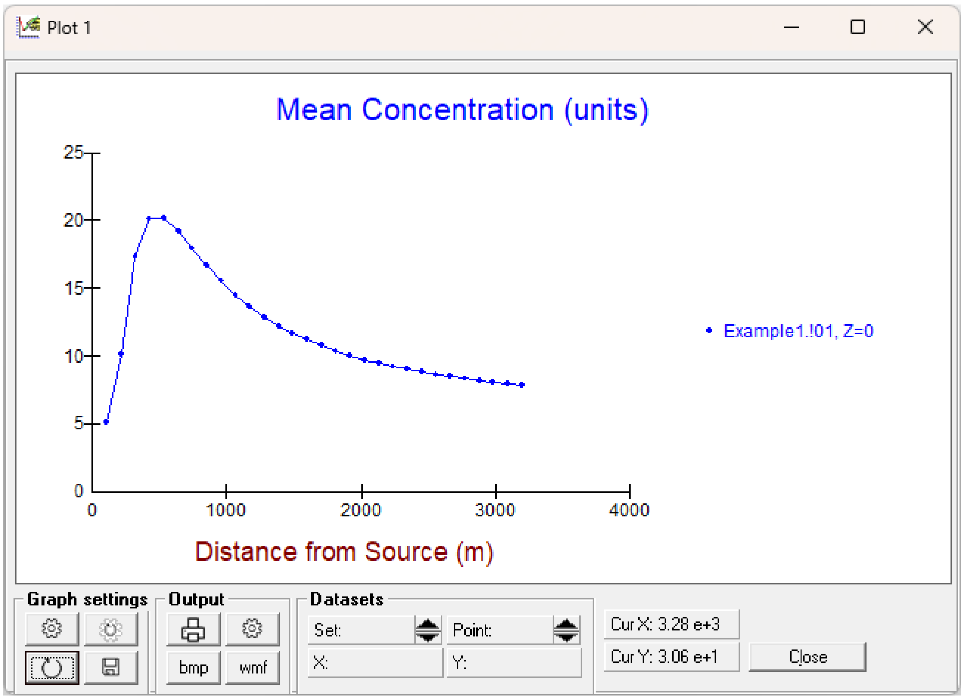


Figure 6.13 – Example of plot created with the **ADMS Line Plotter** utility. Plot 1 window.

Step 5 Go back to the main screen and select a different file containing the data to plot. Leave **Type of Modelling Run** and **Type of Graph** options unaltered.

Step 6 Click on the **Plot 2** button. A new plot window will automatically open (see **Figure 6.14**).

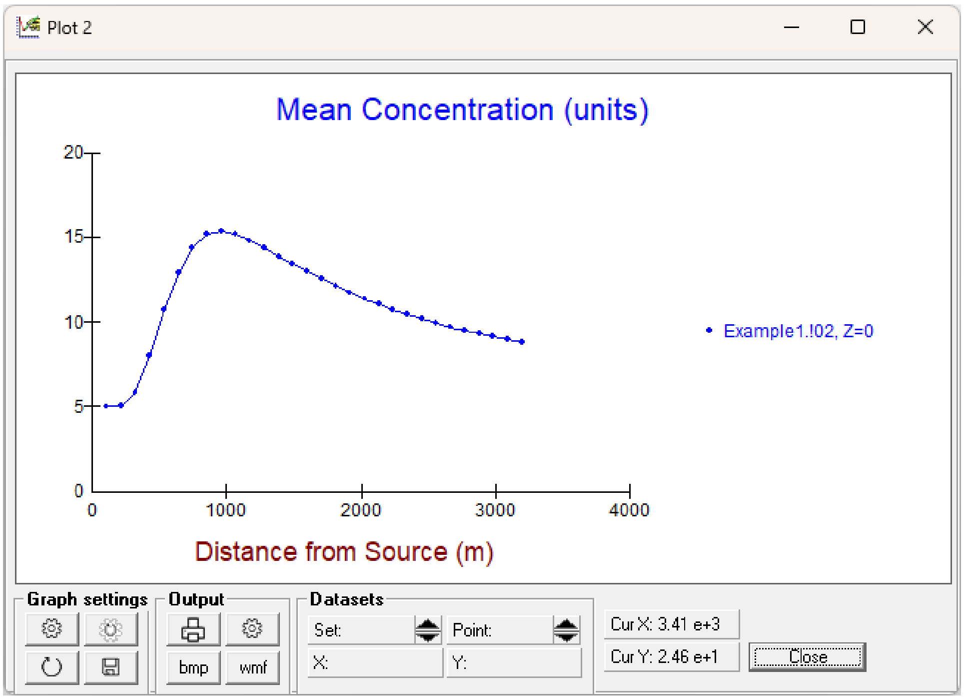


Figure 6.14 – Example of plot created with the **ADMS Line Plotter** utility. Plot 2 window.

Types of graph

This section lists all the possible variables that can be plotted.

1. **Concentration** (**Ground Level**): ground-level concentration beneath the plume centreline.

   Concentration (**Plume Centreline**): plume centreline concentration.

   If there is an inversion at the top of the boundary layer that splits the plume in two, the plume centreline concentration displayed is for the part of the plume below the inversion. When the **Buildings** option is selected, the centreline concentration of the elevated plume is given.

2. **Deposition** (**Dry**, **Wet**, **Total**): ground-level deposition fluxes below the plume centreline.

   These output options should only be used if the **Wet deposition** and/or **Dry deposition** options were selected in the **Setup** screen of the ADMS-Urban interface.

3. **Spread Parameters** (**Sigma y**): lateral plume spread

   Spread Parameters (**Sigma z**): vertical plume spread

   If there is an inversion at the top of the boundary layer that splits the plume in two, this output is for the part of the plume below the inversion. When buildings are also modelled these outputs apply to the elevated plume only.

4. **Plume Height** (**Centreline**): height of the plume centreline,

   Plume Height (**Mean Height**): mean height of the plume.

If there is an inversion at the top of the boundary layer that splits the plume in two, the output is for the part of the plume below the inversion. If the **Buildings** option is selected, centreline and mean plume heights are given for the elevated plume only.

5. **Fraction Penetrating Inversion**: fraction of the plume that has penetrated above the top of the boundary layer.

6.3.2 Graph display features

A typical example of a graph produced after clicking either the **Plot 1** or **Plot 2** button is shown in **Figure 6.15**. In this section we describe other aspects of the display.

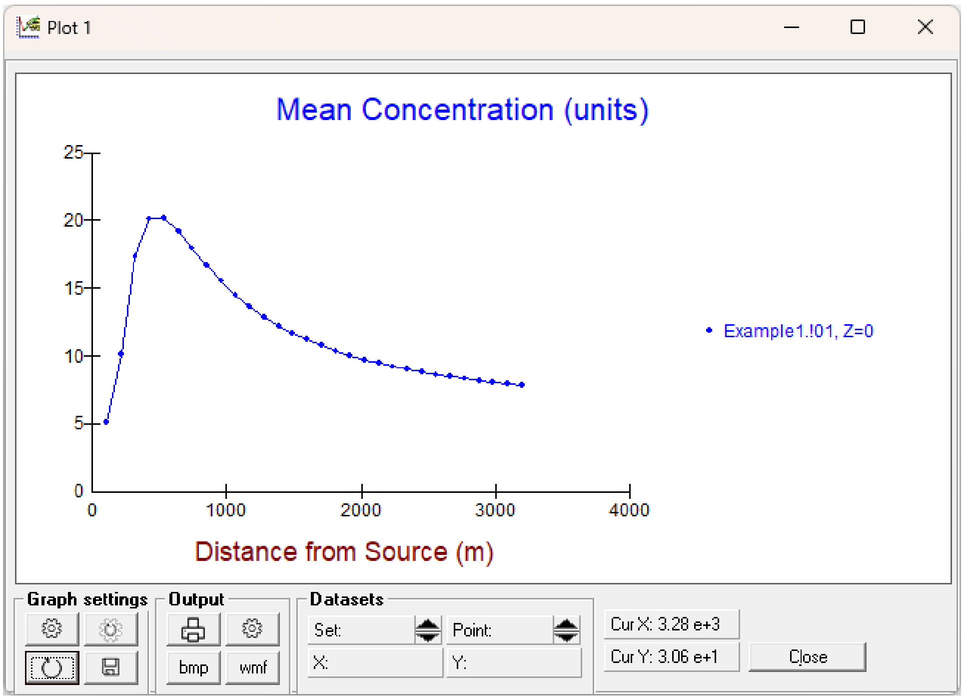
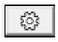


Figure 6.15 – Example of plot created with the **ADMS Line Plotter** utility.

Datasets

The **Datasets** box below the graph gives the actual values plotted on the independent (*x*) and dependent (*y*) axes. The set and point being displayed can be changed using the up and down arrows ( ). The graph can also respond to mouse clicks – if you position your cursor over a point and click the left mouse button, the corresponding data value will be displayed in the **Datasets** box. Alternatively, as you move the cursor over the graph, the **Cur X** and **Cur Y** panels below the graph to the right will display the current cursor location.

Graph setting

The graph settings will be reset to default values whenever the plot window is closed and reopened. However, there is an option to save the current settings as the new defaults using the icon. The button can be used to refresh the settings by re-reading from the *.ini* file, and the button can be used to restore the factory default settings. The appearance of the plot can be altered by clicking on the wheel icon  , prompting the Graph Design dialogue box shown in **Figure 6.16**.

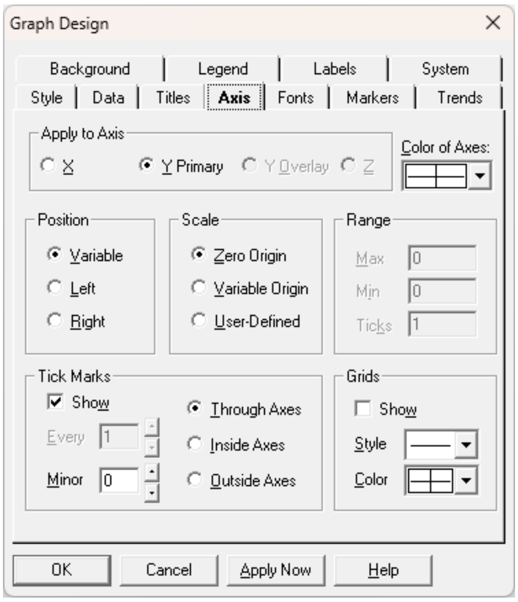


Figure 6.16 − The **Graph Design** box of the **ADMS Line Plotter** utility.

The **Graph Design** dialogue box has 11 options, the purpose of which can be briefly summarised as follows.

1. **Background**: allows the user to alter the background colour of the graph and the style of borders drawn around titles, the legend and the graph.
2. **Legend**: allows the user to select the legend text, size and position.
3. **Labels**: allows the user to turn axis labels and data point labels on and off, and change their format.
4. **System**: allows the user to print a graph and save a graph (data and/or graph layout) to file.
5. **Style**: allows the user to select how the data are to be displayed on the graph (e.g. just points or with lines joining the points), and change the scale of the axes to a log scale.
6. **Data**: allows the user to add or remove data points and control the range of data points plotted.
7. **Titles**: allows the user to write/edit titles and change the orientation of titles. The limit of 80 characters in the title is set by the graphics software.
8. **Axis**: allows the user to alter the minimum/maximum values of axes and the number of ticks. To do this, choose the axis in the **Apply to Axis** box. Click on **User-Defined** in the **Scale** box and enter the appropriate minimum/maximum values and number of ticks in the **Range** box. Click on **Apply Now**. If the graph is correct, click on **OK**. Labels and their orientation are set on the **Labels** screen.
9. **Fonts**: allows the user to set the font of the graph title, axes labels and legends.
10. **Markers**: allows the user to alter the symbol used in the graph to denote the data points, change the size and colour of these symbols, and adjust the colour and

thickness of the line and whether it is dashed or solid.

11. **Trends**: allows the user to add limit lines, statistical lines and fit a curve to data points.

Output

Once happy with the appearance of a graph, it can be copied to the clipboard for pasting into a separate application, or printed.

The buttons in the **Output** box below the graph can be used to copy the graph to the clipboard as a bitmap (**bmp**) file or as a Windows metafile (**wmf**); the latter is a vector format and may therefore be better for including in documents if using a compatible application (e.g. Microsoft Word).

The  button prints the graph using the printer settings specified in the **Graphics print settings** screen (see **Figure 6.17**), accessed via the wheel icon . This screen allows you to set some of the commonly used printing options, including the printer, the size and position of the graph on the page and the overall page title.

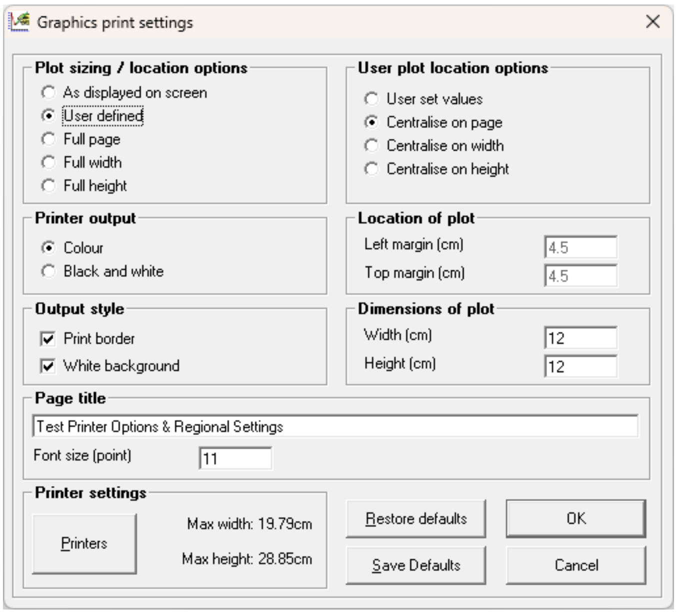


Figure 6.17 – **Graphics print settings** screen.

6.4 Displaying footprints

After performing a run with the **Output per source** option selected, the **Results**, **Display footprint** menu option becomes available. This option facilitates the plotting of the data within the *.sst*/*.slt* files in the Mapper. These files contain the short-term/long-term contribution of each source to the concentration and/or deposition at each specified point in the model file (see Section 6.1.4 for further details). Selecting this menu option brings up the **Choose footprints to display** screen, as shown in **Figure 6.18**.

*All table columns in the **Choose footprints to display** screen are resizable.*

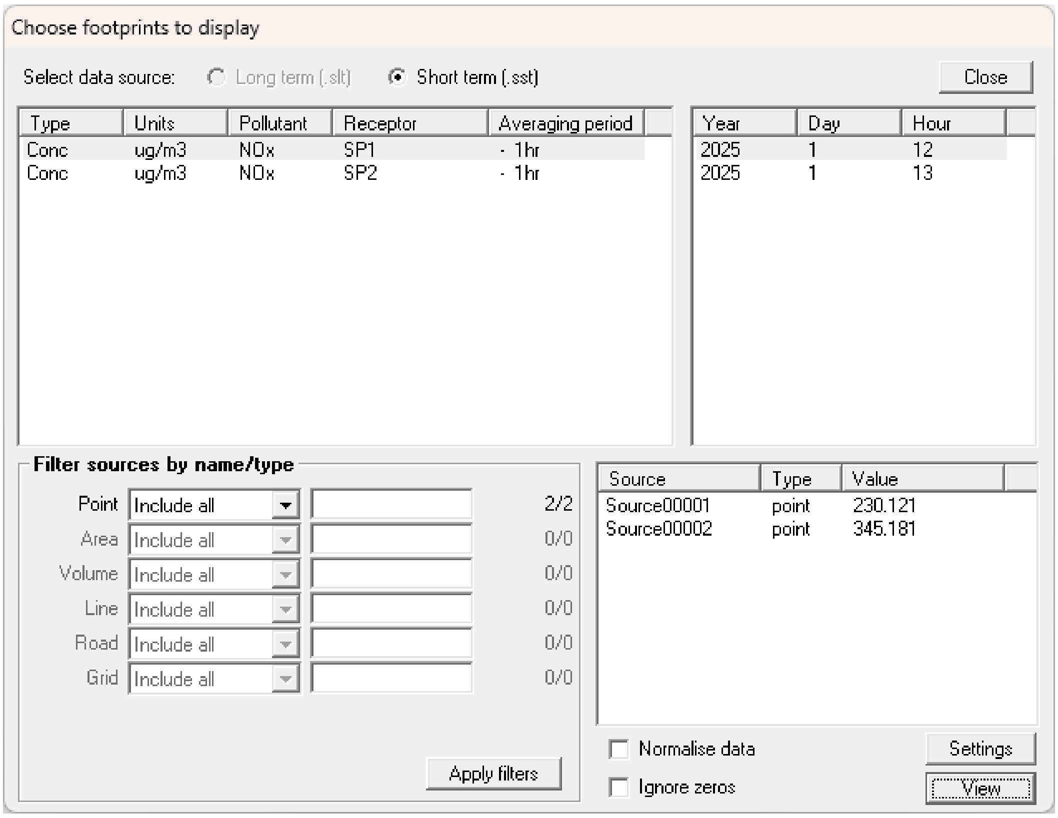


Figure 6.18 − The **Choose footprints to display** screen.

Below are the steps to follow in order to create a plot from the data:

Step 1 Select whether to plot the data from a **Long term (.slt)** or **Short-term (.sst)** file using the radio buttons at the top of the screen

Step 2 Select the dataset to plot from the top left table and, for short term output only, the time for which the data are to be plotted from top right table.

The top left table shows a list of all the variables that can be used for the plot. Each record in the list shows the type of output (e.g. 'Conc' for concentration), the units of output (e.g. 'ppm'), the pollutant name (e.g. 'CO'), the receptor name (e.g. 'School'), whether it is a rolling average ('R'), maximum daily output ('M'), both ('RM') or neither ('-'), and the averaging time (e.g. '1hr'). Right-click options exist that allow you to filter the data in this table in order to quickly locate the dataset of interest.

For short-term runs only, the top right table displays the time (year, day, hour) as given in the meteorological data irrespective of whether these data are entered from a file or by hand. If no time information is given in the meteorological data file, numbers corresponding to each consecutive meteorological condition are used. For example, numbers 1 to 7, corresponding to meteorological conditions A to G respectively, are listed when the example file *R91a-g.met* is used. Again, right-click options exist that allow you to quickly filter the data in this table.

Step 3 Use the drop-down lists in the **Filter sources by name/type** box to select which sources from the model file should be plotted. The list of included sources will appear in the bottom right table after clicking **Apply filers**.

There are four filter options available for each source type:

- **Include all** will include all sources of that source type.
- **Include matches** will include all source of that source type that match the pattern entered in the adjacent field. Wildcards can be used, so, for example, entering *Rail** will include all sources whose name starts with the string *'Rail'*.
- **Exclude matches** will exclude all sources of that source type that match the pattern entered in the adjacent field. Wildcards can be used, so, for example, entering *Rail** will exclude all sources whose name starts with the string *'Rail'*.
- **Exclude all** will exclude all sources of that source type.

Step 4 Tick the **Normalise data** checkbox if you wish to normalise the data for each source by an appropriate source dimension. By default: point sources are not normalised; road and line sources are normalised by their length; area, volume and grid sources are normalised by their (plan-view) area. Different normalisations are available by clicking the **Settings** button.

Using a road network as an example, normalising the data by the source lengths will likely show up busy roads lying close to and upwind of the receptor, regardless of their length.

Step 5 Tick the **Ignore zeros** checkbox if you want to remove from the plot any sources that don't contribute at all to the concentration/deposition at the selected receptor.

This can improve the distinction between sources with non-zero contributions in the plot by narrowing the colour scale range.

Step 6 Click the **View** button to plot the selected data in the Mapper. An example is shown in **Figure 6.19**.

Each included source will be colour-coded to indicate to what degree it contributes to the concentration at the selected receptor, as indicated by the blue square. The default colour scales can be modified if desired; please refer to the *Mapper User Guide* for full details, which can be accessed from the **Help** menu of the Mapper.

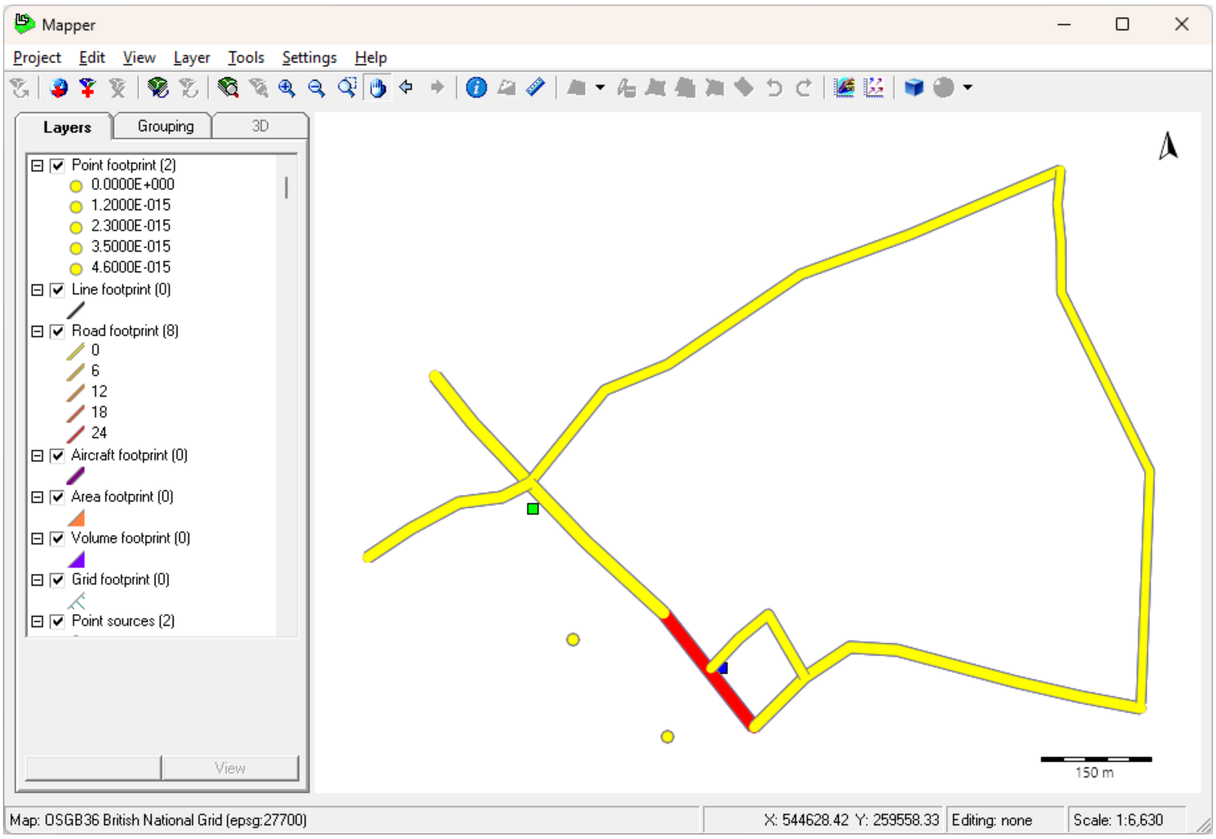


Figure 6.19 – An example plot of *.slt* data in the **Mapper**.

Click **Close** to close the **Choose footprints to display** screen and return control to the main ADMS-Urban interface. If open, the Mapper will remain open and data displayed unchanged.

6.5 Log files

The reporting files produced during a model or verification run for the currently open model file can be viewed from the ADMS-Urban interface at any time by selecting **Log files** from the **Results** menu. This will bring up the file verification results screen, as shown in **Figure 6.20**.

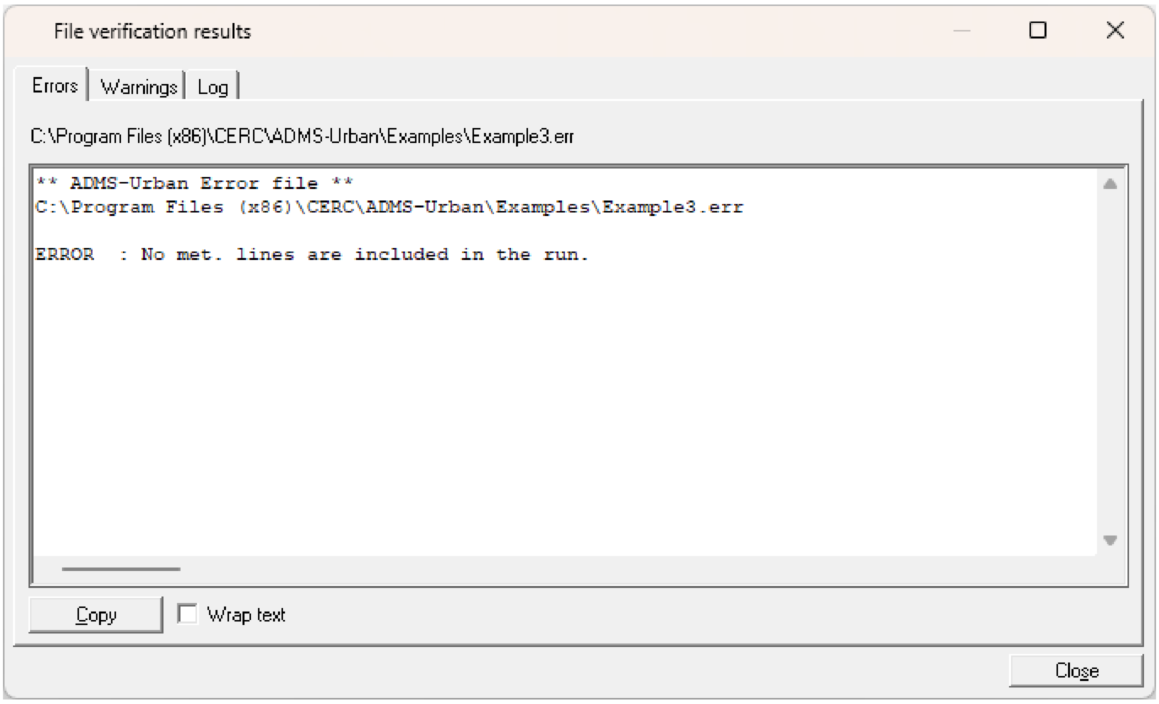


Figure 6.20 – The **File verification results** screen.

This window shows each of the three different reporting files in separate tabs (these files are described in more detail in Section 6.1.1). If a file does not exist for the current model file (for instance, if there were no errors found in verification) then the corresponding tab will not appear.

On each tab the horizontal and vertical scroll bars can be used to view the information outside the current visible area. Alternatively, checking the **Wrap text** box will cause the text in each tab to be wrapped (each line broken to fit horizontally in the window) so that only vertical scrolling is required. The **Copy** button can be used to copy all of the text in the current tab to the clipboard.

6.6 Viewing numerical data

Most of the numerical output files created by ADMS-Urban are comma-separated text files. Input and output files can be accessed from the ADMS-Urban interface (**View** buttons for the input files; **Results, Numerical output** menu option for the output files), and viewed in WordPad, Notepad, Microsoft Excel or some other application of the user's choice. The application used to view files is set in the preferences (see Section 6.6.1).

An alternative way of opening the output files in Microsoft Excel is outlined in Section 6.6.3.

6.6.1 Choosing application to view output

To change the preferred viewing application for input data and output files, select **File**, **Preferences**, **Viewing options** from the menu bar of the ADMS-Urban interface. This will bring up the **File Viewing Preferences** screen shown in **Figure 6.21**.

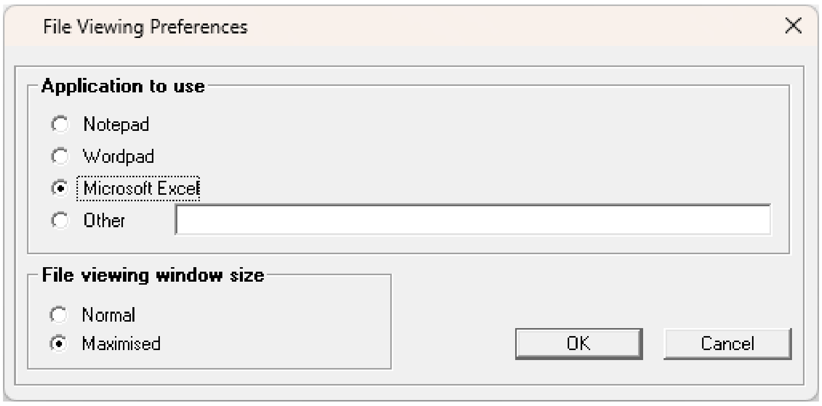


Figure 6.21 − The **File Viewing Preferences** window.

Application to use allows the user to select which application to use to **View** files. If an application other than Notepad, WordPad or Microsoft Excel is to be used, then select **Other** and enter the path of the application.

File viewing window size defines the size of the viewing window when it is opened. Choices are either **Normal** (size of the window last used) or **Maximised** (full screen).

6.6.2 Viewing output from the ADMS-Urban interface

The output files associated with the model file that is currently open in the interface can be viewed using the **Results**, **Numerical output** menu option. A screen will appear showing all of the output files for the current model file. After selecting a file and clicking **Open**, the file will be opened with the application selected in the **File Viewing Preferences**, as described in Section 6.6.1.

If the file is being opened in Microsoft Excel, then it will automatically be opened as a comma-separated file.

6.6.3 Use of Microsoft Excel to view numerical output

The numerical output files can be opened using Microsoft Excel without using the ADMS-Urban interface, proceeding as follows. These instructions relate to Microsoft Excel 2019.

Step 1 Start Microsoft Excel.

Step 2 Select **File > Open** from the ribbon, and browse through the directories to locate the relevant file, making sure that **All files (*.*)** is selected in the lower right drop-down menu. You can also type, e.g. '*.gst' in the **File name** box and press Enter to see just those files that end with the extension *.gst* in the current directory.

Step 3 Click on the file and then click **Open**. This starts the Microsoft Excel **Text Import Wizard** shown in **Figure 6.22**.

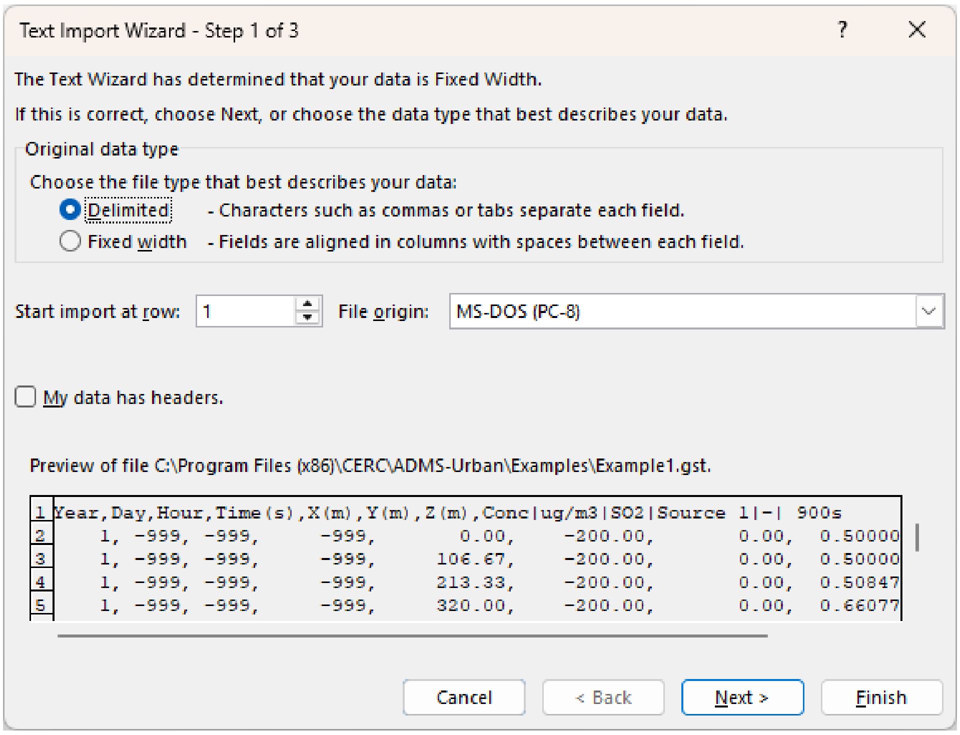


Figure 6.22 − Step 1 of the Microsoft Excel **Text Import Wizard**.

Step 4 ADMS-Urban output files are comma-delimited so they can be imported easily into many standard packages.

In **Figure 6.22**, select the **Delimited** option and click on **Next >** to move to the next step, shown in **Figure 6.23**.

Double-click on the option to select it and to move directly to the next step.

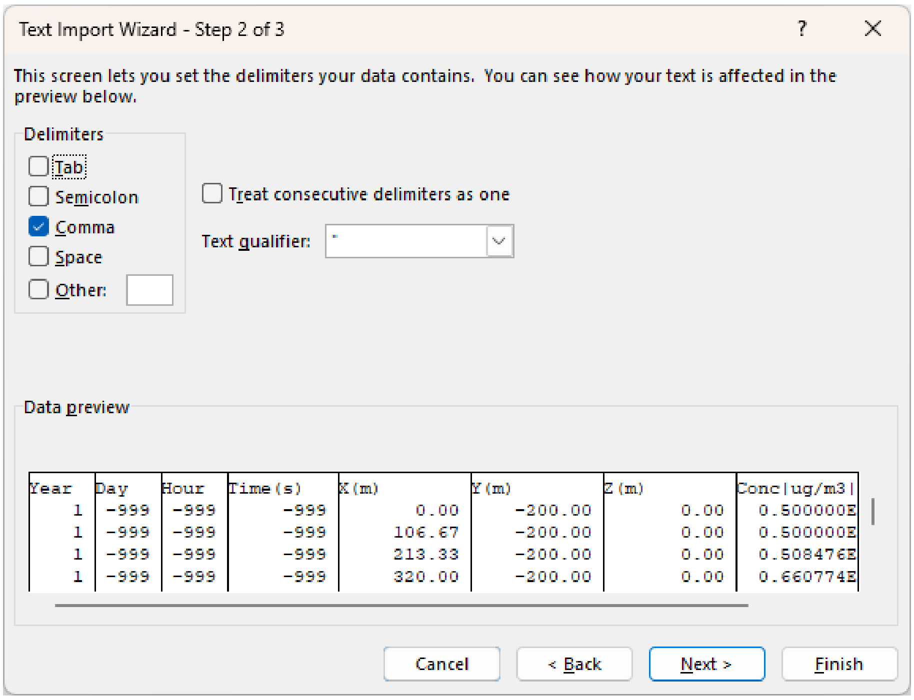


Figure 6.23 − Step 2 of the Microsoft Excel **Text Import Wizard**.

Step 5 In the **Delimiters** box, select only the **Comma** check box and click on **Next >** to move to the last step, shown in **Figure 6.24**.

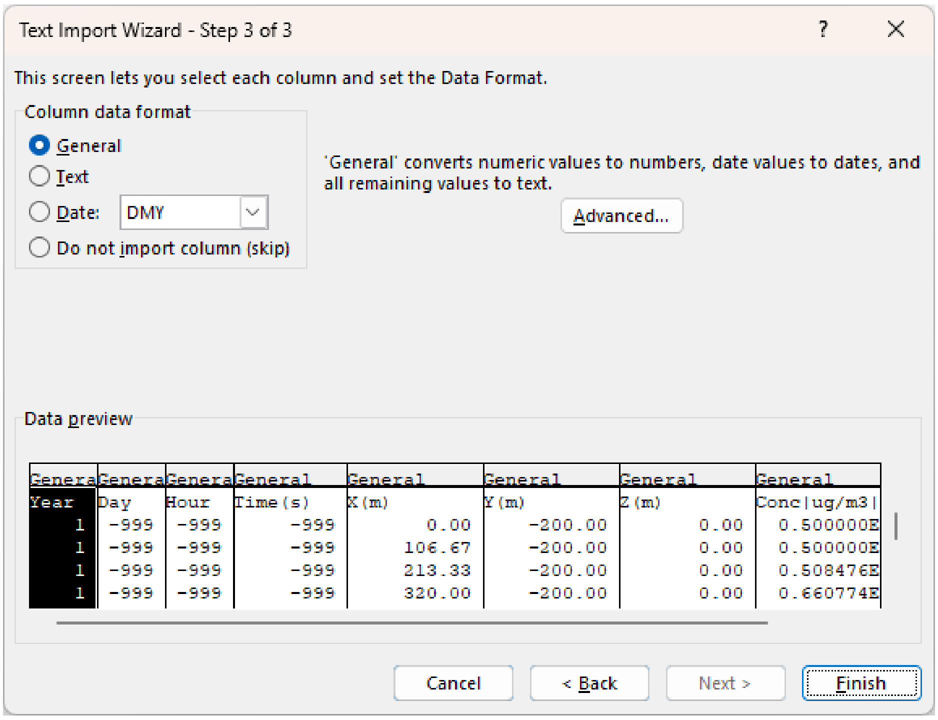
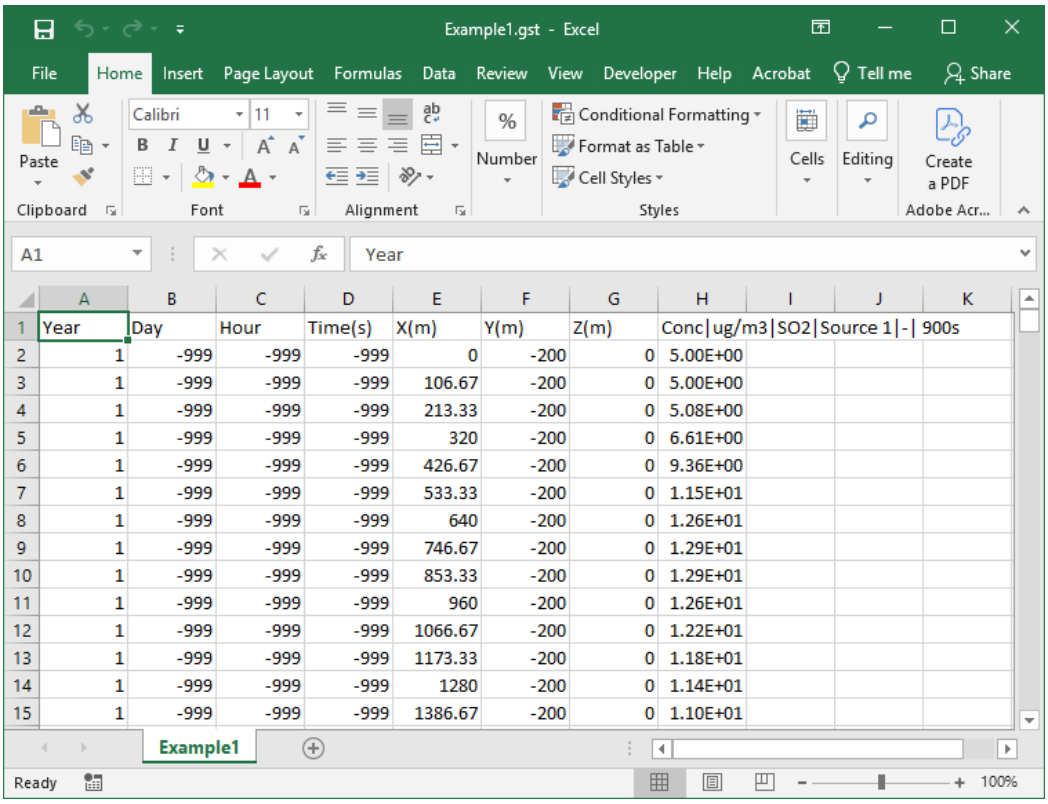


Figure 6.24 − Step 3 of the Microsoft Excel **Text Import Wizard**.

Step 6 In the **Data Preview** box, ensure that the file has been properly separated and click on **Finish** to import the data.

Imported data in Microsoft Excel are shown in **Figure 6.25**.

If the column headings are not visible because the columns are too narrow, select all the columns containing data and use the **Format** button in the **Cells** section of the **Home** tab and select **Auto Fit Selection** to increase the column widths to fit the longest entry in each column.



Year	Day	Hour	Time(s)	X(m)	Y(m)	Z(m)	Conc|ug/m3|SO2|Source 1|-| 900s
1	-999	-999	-999	0	-200	0	5.00E+00
1	-999	-999	-999	106.67	-200	0	5.00E+00
1	-999	-999	-999	213.33	-200	0	5.08E+00
1	-999	-999	-999	320	-200	0	6.61E+00
1	-999	-999	-999	426.67	-200	0	9.36E+00
1	-999	-999	-999	533.33	-200	0	1.15E+01
1	-999	-999	-999	640	-200	0	1.26E+01
1	-999	-999	-999	746.67	-200	0	1.29E+01
1	-999	-999	-999	853.33	-200	0	1.29E+01
1	-999	-999	-999	960	-200	0	1.26E+01
1	-999	-999	-999	1066.67	-200	0	1.22E+01
1	-999	-999	-999	1173.33	-200	0	1.18E+01
1	-999	-999	-999	1280	-200	0	1.14E+01
1	-999	-999	-999	1386.67	-200	0	1.10E+01

Figure 6.25 – ADMS-Urban output data imported into Microsoft Excel.

6.6.4 Use of Microsoft Excel to create a time series graph

Microsoft Excel can also be used to produce a time series graph of short-term concentration, etc., at each of the specified output points. To produce such a graph, follow these steps, again these instructions relate to Microsoft Excel 2019:

Step 1 Open a *.pst* output file in Microsoft Excel as described in Section 6.6.3.

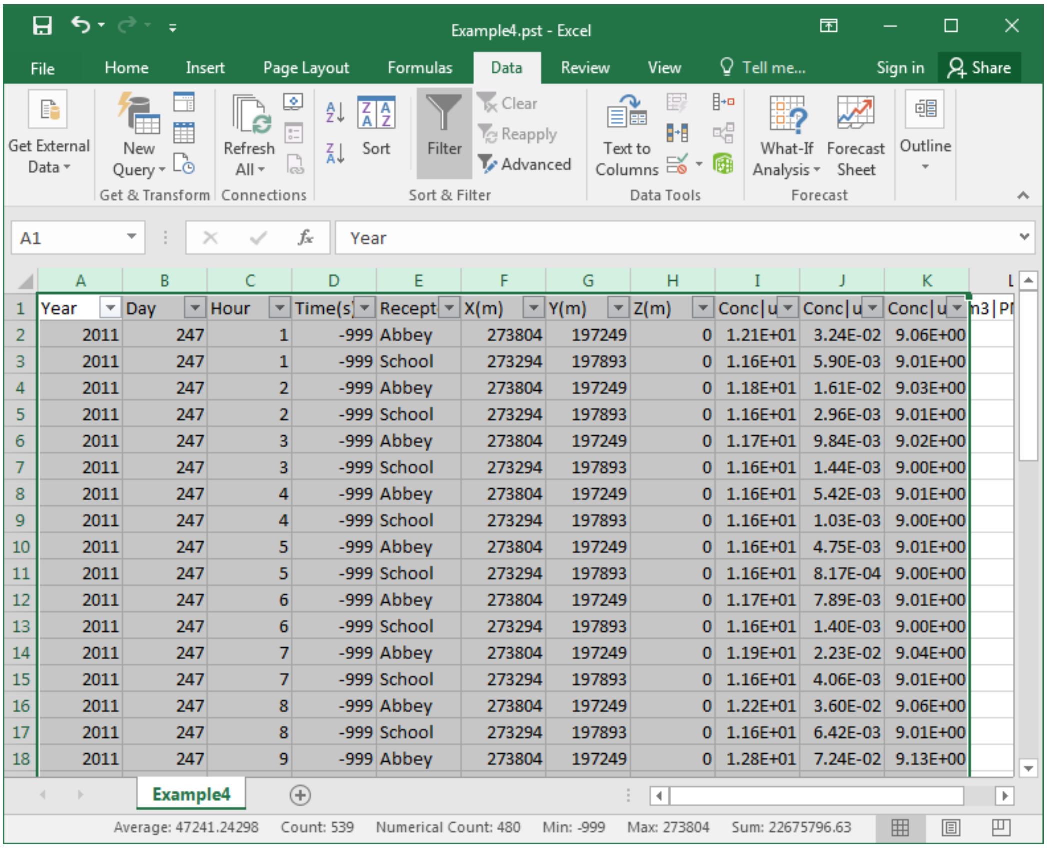
Step 2 Select all columns containing data (e.g. columns A:K in **Figure 6.26**) and choose **Filter** from the **Sort & Filter** section of the **Data** tab.

Filters are activated and represented by the arrow box in each of the heading cells.

Step 3 Filter the data as required (e.g. all values for a specific receptor) by clicking on the appropriate arrows. For example, the ‘Abbey’ value as shown in **Figure 6.27**.

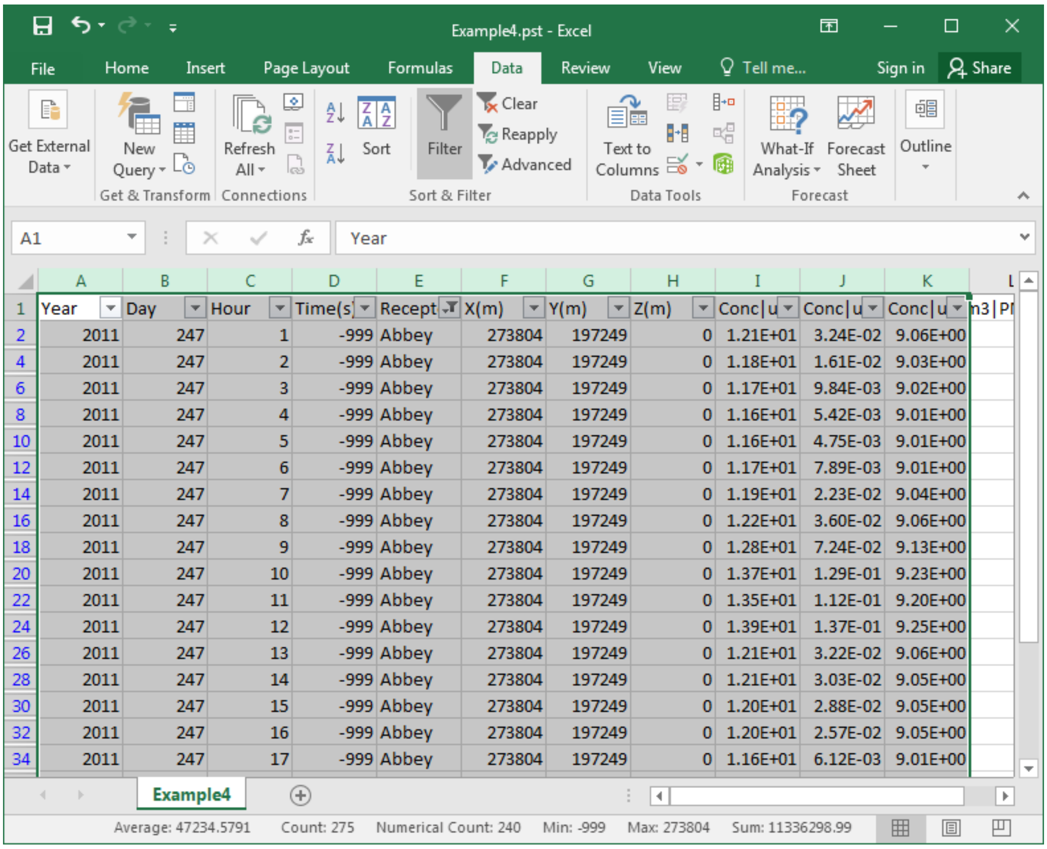
Step 4 Select the columns to be plotted by clicking on these columns while holding down the **Control** key. Then choose the chart type you want from the **Charts** section of the **Insert** tab.

Step 5 The chart will appear and can be customized by first selecting the chart and then using the subtabs that appear under the **Chart Tools** tab.



	Year	Day	Hour	Time(s	Recept	X(m)	Y(m)	Z(m)	Conc|u	Conc|u	Conc|u	n3|PI
2	2011	247	1	-999	Abbey	273804	197249	0	1.21E+01	3.24E-02	9.06E+00	
3	2011	247	1	-999	School	273294	197893	0	1.16E+01	5.90E-03	9.01E+00	
4	2011	247	2	-999	Abbey	273804	197249	0	1.18E+01	1.61E-02	9.03E+00	
5	2011	247	2	-999	School	273294	197893	0	1.16E+01	2.96E-03	9.01E+00	
6	2011	247	3	-999	Abbey	273804	197249	0	1.17E+01	9.84E-03	9.02E+00	
7	2011	247	3	-999	School	273294	197893	0	1.16E+01	1.44E-03	9.00E+00	
8	2011	247	4	-999	Abbey	273804	197249	0	1.16E+01	5.42E-03	9.01E+00	
9	2011	247	4	-999	School	273294	197893	0	1.16E+01	1.03E-03	9.00E+00	
10	2011	247	5	-999	Abbey	273804	197249	0	1.16E+01	4.75E-03	9.01E+00	
11	2011	247	5	-999	School	273294	197893	0	1.16E+01	8.17E-04	9.00E+00	
12	2011	247	6	-999	Abbey	273804	197249	0	1.17E+01	7.89E-03	9.01E+00	
13	2011	247	6	-999	School	273294	197893	0	1.16E+01	1.40E-03	9.00E+00	
14	2011	247	7	-999	Abbey	273804	197249	0	1.19E+01	2.23E-02	9.04E+00	
15	2011	247	7	-999	School	273294	197893	0	1.16E+01	4.06E-03	9.01E+00	
16	2011	247	8	-999	Abbey	273804	197249	0	1.22E+01	3.60E-02	9.06E+00	
17	2011	247	8	-999	School	273294	197893	0	1.16E+01	6.42E-03	9.01E+00	
18	2011	247	9	-999	Abbey	273804	197249	0	1.28E+01	7.24E-02	9.13E+00	

Figure 6.26 − Filtering ADMS-Urban output files in Excel.



	Year	Day	Hour	Time(s	Recept	X(m)	Y(m)	Z(m)	Conc|u	Conc|u	Conc|u	n3|PI
2	2011	247	1	-999	Abbey	273804	197249	0	1.21E+01	3.24E-02	9.06E+00	
4	2011	247	2	-999	Abbey	273804	197249	0	1.18E+01	1.61E-02	9.03E+00	
6	2011	247	3	-999	Abbey	273804	197249	0	1.17E+01	9.84E-03	9.02E+00	
8	2011	247	4	-999	Abbey	273804	197249	0	1.16E+01	5.42E-03	9.01E+00	
10	2011	247	5	-999	Abbey	273804	197249	0	1.16E+01	4.75E-03	9.01E+00	
12	2011	247	6	-999	Abbey	273804	197249	0	1.17E+01	7.89E-03	9.01E+00	
14	2011	247	7	-999	Abbey	273804	197249	0	1.19E+01	2.23E-02	9.04E+00	
16	2011	247	8	-999	Abbey	273804	197249	0	1.22E+01	3.60E-02	9.06E+00	
18	2011	247	9	-999	Abbey	273804	197249	0	1.28E+01	7.24E-02	9.13E+00	
20	2011	247	10	-999	Abbey	273804	197249	0	1.37E+01	1.29E-01	9.23E+00	
22	2011	247	11	-999	Abbey	273804	197249	0	1.35E+01	1.12E-01	9.20E+00	
24	2011	247	12	-999	Abbey	273804	197249	0	1.39E+01	1.37E-01	9.25E+00	
26	2011	247	13	-999	Abbey	273804	197249	0	1.21E+01	3.22E-02	9.06E+00	
28	2011	247	14	-999	Abbey	273804	197249	0	1.21E+01	3.03E-02	9.05E+00	
30	2011	247	15	-999	Abbey	273804	197249	0	1.20E+01	2.88E-02	9.05E+00	
32	2011	247	16	-999	Abbey	273804	197249	0	1.20E+01	2.57E-02	9.05E+00	
34	2011	247	17	-999	Abbey	273804	197249	0	1.16E+01	6.12E-03	9.01E+00	

Figure 6.27 − Filtering ADMS-Urban output files in Excel, filter on the receptor name.

SECTION 7 Utilities

This section describes the **Mapper** and the utilities listed in the **Utilities** menu of the ADMS-Urban interface (**Figure 7.1**), i.e.:

1. **View a Wind Rose**
2. **Visualise Input in Surfer…**
3. **Create/Run batch file**
4. **Convert met. data**
5. **Create terrain file**
6. **Create ASP grid**
7. **Create import templates**
8. **Create advanced canyon template**
9. **Download terrain data**

These utilities are divided into visualisation tools (1 and 2), input data processing tools (3, 4, 5 and 6), an option for creating template files for Import (7, refer to Section 5.1.8 for more details) and an option for creating a template input file for the advanced street canyon modelling option (8, refer to Section 4.2.2 for more details). A link is also provided to a website where SRTM[1] terrain data can be downloaded (9). This terrain data can be converted into a format suitable for use in ADMS-Urban. There are also options to **Start Excel** and **Start Surfer** which launch Microsoft Excel and the Surfer application respectively. Help and tips on the use of Surfer are provided in Appendix E.

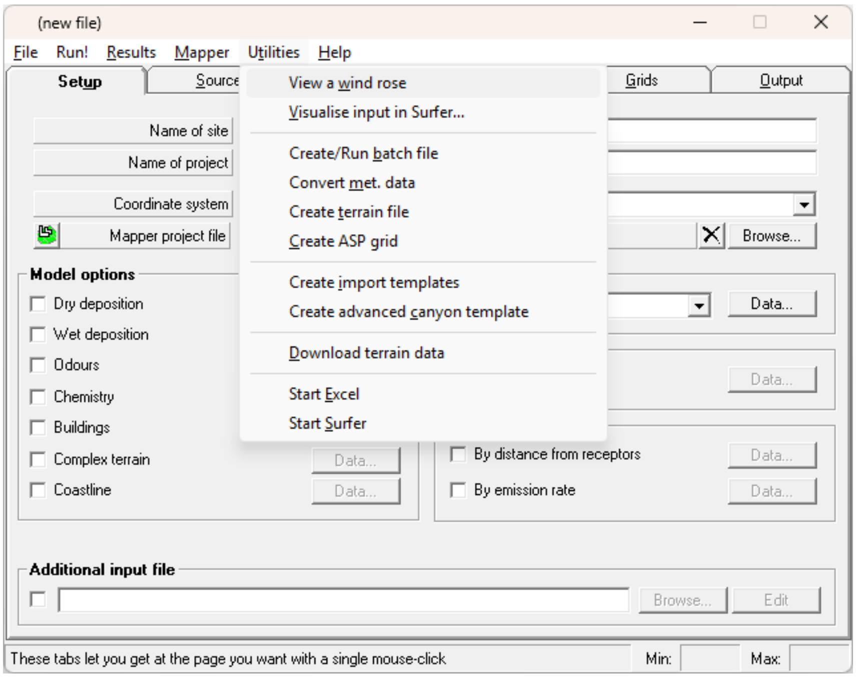


Figure 7.1 – The **Utilities** menu.

[1] Shuttle Radar Topography Mission data on a near global scale from 56º S to 60º N.

7.1 Mapper

The **Mapper** is an integrated mapping tool for displaying and editing source data, building, coastline and receptor locations and viewing results. It is launched via the **Mapper** menu item and illustrated in **Figure 7.2**. Please refer to the *Mapper User Guide* for full details on using the Mapper, which can be accessed from the **Help** menu of the Mapper.

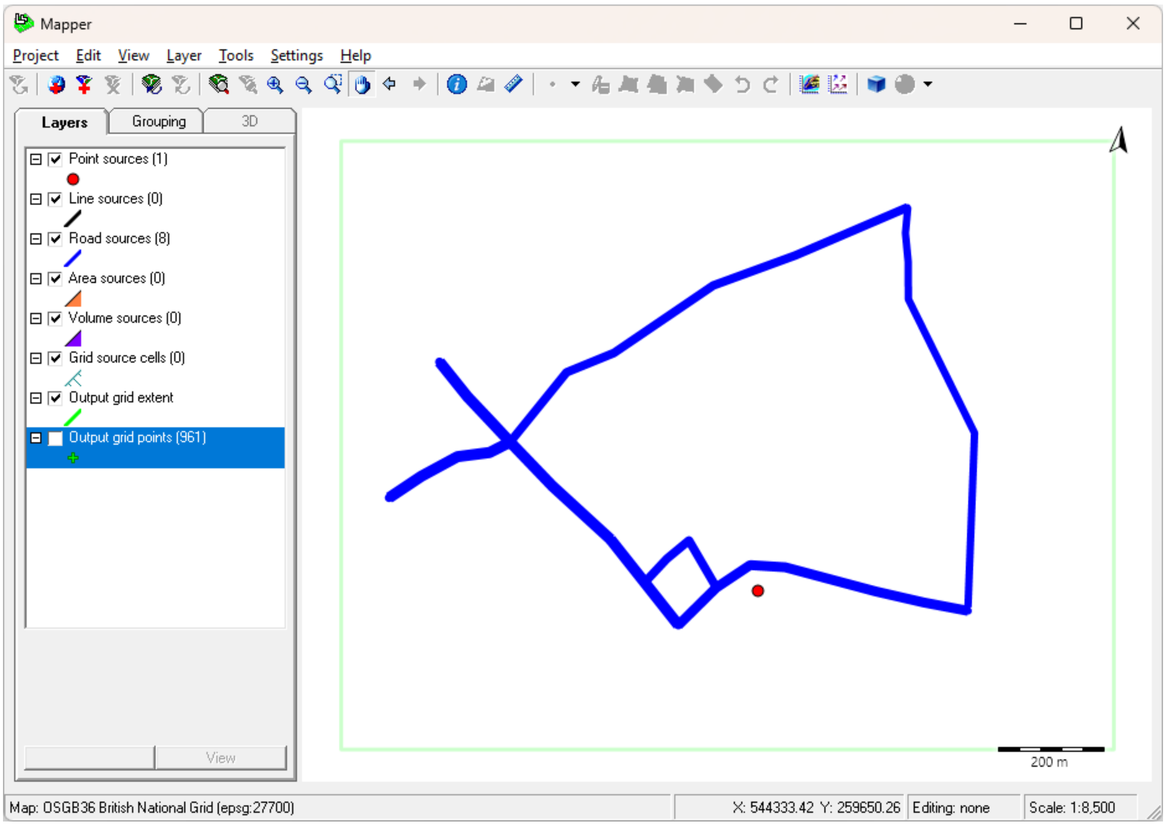


Figure 7.2 – The **Mapper** interface.

Some of the useful features in the **Mapper** allow the user to:

- create new sources by drawing;
- create buildings and receptor points by drawing;
- edit geometry of existing sources, buildings, and receptor points;
- automatically simplify source geometry by removing redundant vertices, either per source or over all source of a certain type;
- view receptor point (*.asp*), terrain (*.ter*) and surface roughness (*.ruf*) files;
- create contour plots;
- create canyon and canopy files using the dedicated tool;
- colour code sources to indicate their contribution to a receptor's concentration;
- create terrain files, e.g from OS Terrain 50 or SRTM data;
- export sources or buildings in various formats, including shape files, MapInfo MIF files and KML files;

- automate the generation of comma-separated variable files, for direct import into ADMS-Urban, from source data contained within non-ADMS layers, e.g. shape files;
- create images for inclusion in reports showing contour plots, positions of sources and buildings;
- view map image tiles, shape files, MapInfo files, AutoCAD DXF files, etc.

7.2 Viewing a wind rose

The **Wind rose viewer** utility allows the data in a meteorological data file (*.met*) to be visualised graphically. The visualisations can be viewed on the screen, printed, copied to another Windows application, or exported to a file. An example of a copied image is shown below in **Figure 7.3**.

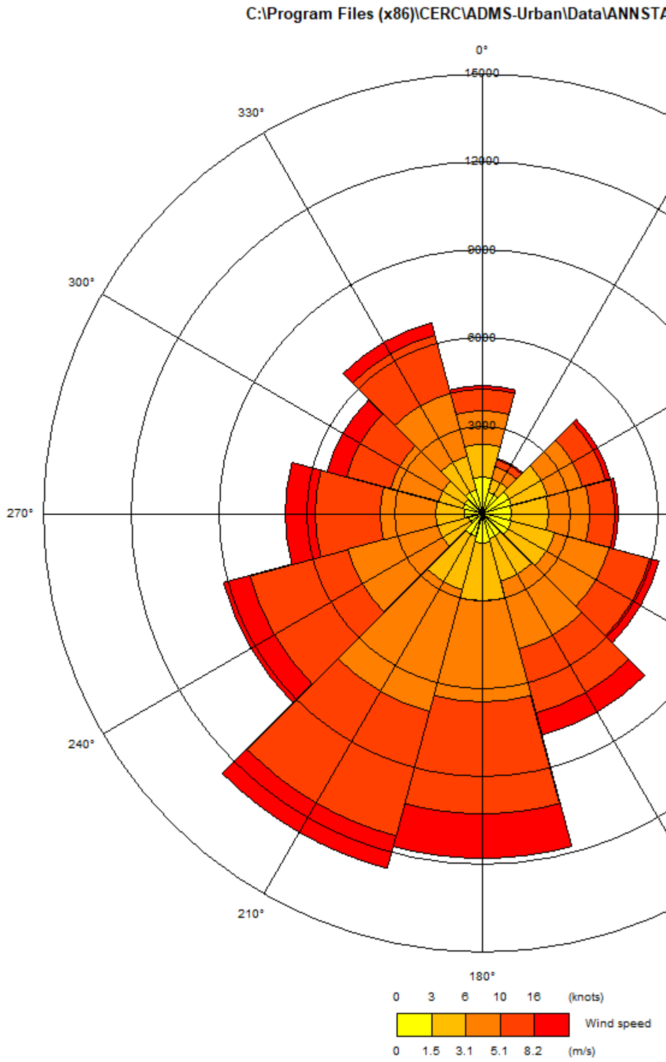


Figure 7.3 – Example of a wind rose produced with the ADMS-Urban **Wind rose viewer** utility.

7.2.1 Categories for the data

To produce these visualisations, the data in the meteorological file is divided into categories by wind speed and wind direction. This information is displayed in a polar plot, in which the angle of an element represents the wind direction, the radial distance from the centre represents the frequency of occurrence and the colour represents the wind speed.

For statistical meteorological data, the frequency is taken from the data in the file. For sequential meteorological data, the frequency of each line is taken to be 1.

The wind speed categories are those used by the UK Met Office in their statistical meteorological data, namely 0-3 knots, 3-6 knots, 6-10 knots, 10-16 knots and 16 knots or above. Meteorological lines with wind speeds of less than 0.01 m/s are not included in the plot.

The **Wind rose viewer displays** the number of **Lines used** and the number of **Lines skipped**. Lines may be skipped if they contain invalid wind speed/direction data, have wind speeds which are too low or are outside of the defined subset period.

7.2.2 Using the wind rose utility

The wind rose utility is accessed from the **Utilities**, **View a wind rose** command or from the **Wind rose** button on the **Meteorology** screen and is shown in **Figure 7.4**.

---

*If the **Wind rose** button on the **Meteorology** screen is used the meteorological data file currently selected in the interface will automatically be loaded into the **Wind rose viewer**. Launching from the **Wind rose** button will allow the selected sector size and Met. Data subset to be taken into account.*

Opening a meteorological file (.met)

To open a meteorological file, go to the **File, Open…** command and browse for the *.met* file.

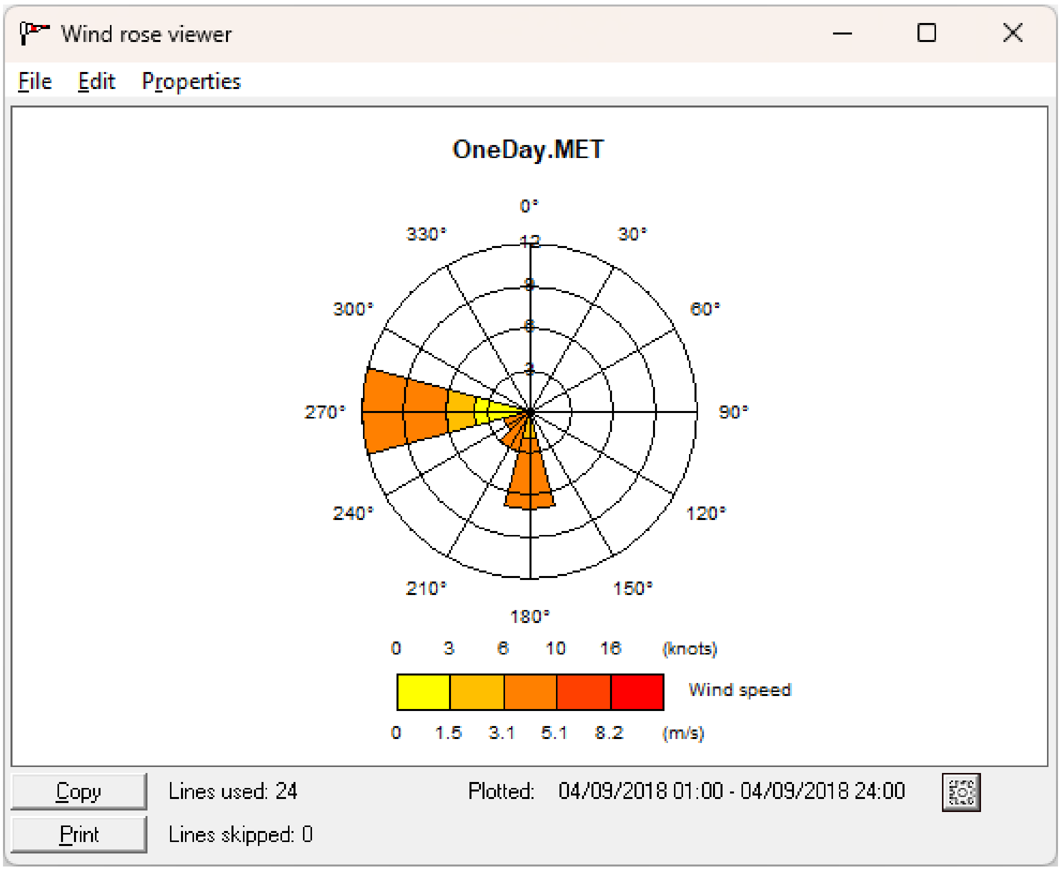


Figure 7.4 – Main screen of the **Wind rose viewer** Utility.

It is also possible to open a boundary layer profile output (.pro*) file that contains only one output height; see Section 4.23 for more details. In particular, this allows you to view the wind rose at e.g. 10 m at the dispersion site, which may differ from the wind rose at the met. site depending on the options chosen in the* **Meteorology** *screen.*

Customising the appearance of the wind rose

The appearance of the wind rose can be altered using the various options from the **Properties** menu:

- The colours used in the wind rose can be set by selecting the **Colour for highest wind speed…** and **Colour for lowest wind speed…** options.
- The title can be shown or removed by selecting the appropriate value in the **Title** option: **None**, **File path**, **File Name**, **File stem** or **User defined…** .
- The fonts used in the wind rose display can be altered as follows: choose **Label Font…** to alter the font for the wind speeds and angles, and choose **Title Font** to alter the font for the title.
- There are several different visualisation styles for the wind rose. These can be selected as follows: choose **Arc segments** to display each wind sector as a pie slice, choose **Polygon** to join points along neighbouring wind sector centrelines to form the sides of a polygon, or choose **Triangles** to display each wind sector as a triangle.
- The angle interval into which the wind speeds are "binned" for display can be altered by choosing one of the available values under the **Intervals** option: **10° intervals**, **22.5° intervals**, **30° intervals** or **User defined**. Note that changing the binning interval can take several seconds.
- It is possible to plot a date and time subset of the data by clicking on the gear icon  (see **Figure 7.4**).

Printing the wind rose

To print the wind rose, click the **Print** button or choose **File, Print** from the menu. To use a different printer or to change the settings of the current printer choose **File, Print Setup…** from the menu. The wind rose will occupy a complete printed page.

Copying the wind rose to the clipboard

The wind rose can be copied to the clipboard by clicking the **Copy** button or by choosing **Edit, Copy** from the menu. It can then be pasted into another Windows application, such as Microsoft Word.

Exporting the wind rose to a file

The wind rose image can be exported as an enhanced metafile file (*.emf*) by choosing **File**, **Export Picture…** from the menu. This file may then be imported into other Windows applications, such as Microsoft Word. The imported image may be resized without reducing the resolution quality.

7.3 Visualising input data in Surfer

ADMS-Urban includes a visualisation utility, which is accessed from the **Utilities**, **Visualise Input in Surfer…** menu (see illustration in **Figure 7.5**).

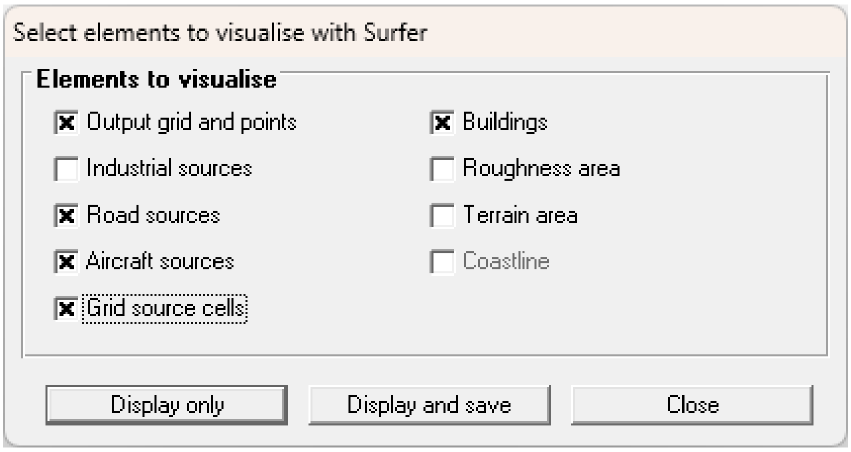


Figure 7.5 – Dialogue box to select the elements to visualise.

This utility allows the user to visualise the positions of various features and their relation to one another. This provides a very useful mechanism for checking the model setup before clicking on the **Run!** command.

The output is produced in Surfer and can be saved as a Surfer file (*.srf*) and imported into report documents. These files can also be combined with ADMS-Urban output contour plots to create a Surfer overlay.

The features that can be visualised are:

- the output grid as specified in the **Grids** tab,
- specified points as specified in the **Grids** tab,
- source locations as specified in the **Source** tab,
- building positions, dimensions and orientations as specified in the **Buildings** screen,
- the area covered by the surface roughness file (*.ruf*) as specified in the **Complex terrain** screen,
- the area covered by the terrain file (*.ter*) as specified in the **Complex terrain** screen,
- the location of the orientation of the coastline as specified in the **Coastline** screen.

Only the features *currently defined in the interface (or loaded model file)* can be visualised. By default, all the available features are selected. You can narrow the selection by unchecking the box(es) of the features you do not want to visualise.

If it appears after the first visualisation that, say, a source position has been specified incorrectly, then return to the main ADMS-Urban interface, make the appropriate changes and then click on **Utilities**, **Visualise input in Surfer…** again until the input data are as required.

7.3.1 Display and/or save the visualisation

The user can choose **Display only** to make a quick check of the data at any stage during the creation of a model file or **Display and Save** to save the output as a Surfer file (*.srf*).

Clicking **Display only** displays the visualisation in Surfer. This visualisation can be modified and saved afterwards in Surfer if needed.

On clicking **Display and save**, the user is prompted for a Surfer file name (*.srf*). The visualisation will then be created in Surfer and saved using the file name given. A number of so-called "post files" (*.dat*) will also be created, based on the Surfer file name provided.

For guidance on customising the Surfer visualisation, please refer to Appendix E.

7.3.2 Layout of visualised features

A legend is provided as a key for all the elements in the visualisation. This legend should appear below the map. If the default paper size in Surfer is not A4, the legend may be incorrectly positioned. The mouse can be used to select the legend and move it to a new position.

Output grid (grid and specified points)

Regular and variable Cartesian grids are shown on the visualisation as a grid of green lines with the X and Y coordinates shown along each axis.

If specified points are also chosen as output, these are shown as green crosses with their labels plotted to the right. An example of a regular Cartesian grid with two specified points is shown in **Figure 7.6**.

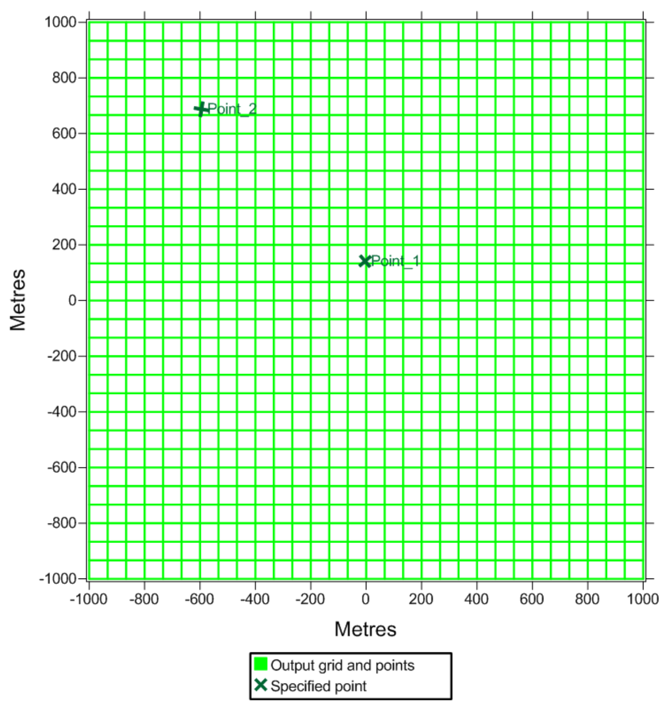


Figure 7.6 – A regular Cartesian output grid and two specified points.

Sources

Each type of source has a different symbol and is labelled with the name assigned to it in the interface, as detailed below and shown in **Figure 7.7**.

- Point sources are shown as five-pointed red stars, labelled to the right.
- Area and volume sources are shown as solid red polygons, labelled at the centre. Their height is not indicated.
- Line sources are shown as solid lines, labelled at the centre. The width of these lines is the width given to the line sources in the interface.
- Road sources are shown as solid violet lines. All road sources are drawn with the same width i.e. road widths entered in the ADMS-Urban interface are not displayed.
- Grid sources are shown as blue lines.

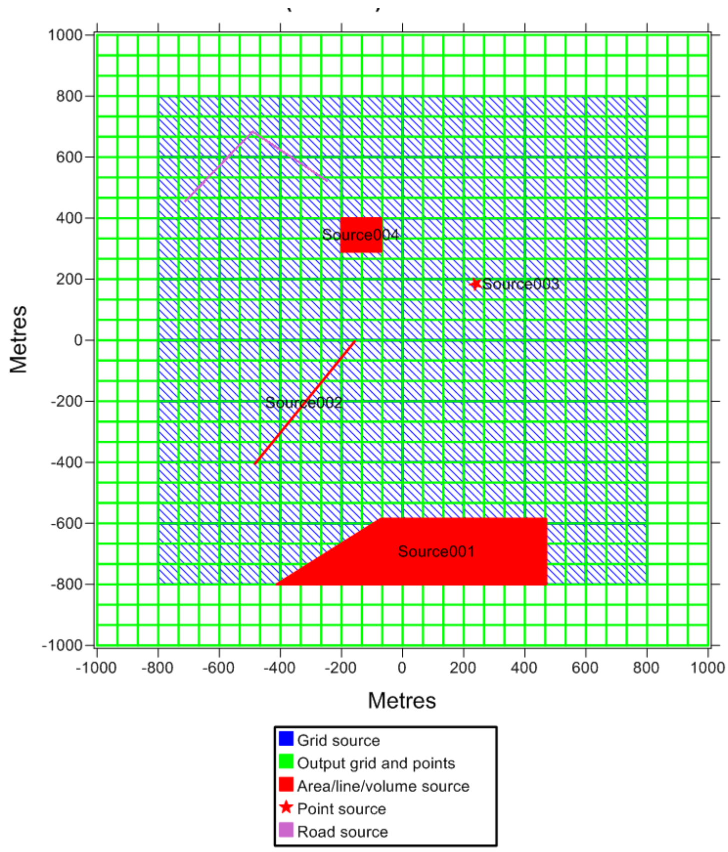


Figure 7.7 – Different types of source available in ADMS-Urban.

Buildings

Buildings are represented as grey areas labelled at the centre with the names assigned to them in the interface. The main building has an asterisk (*) appended to its name. The height of the buildings is not indicated. This is shown in **Figure 7.8** below.

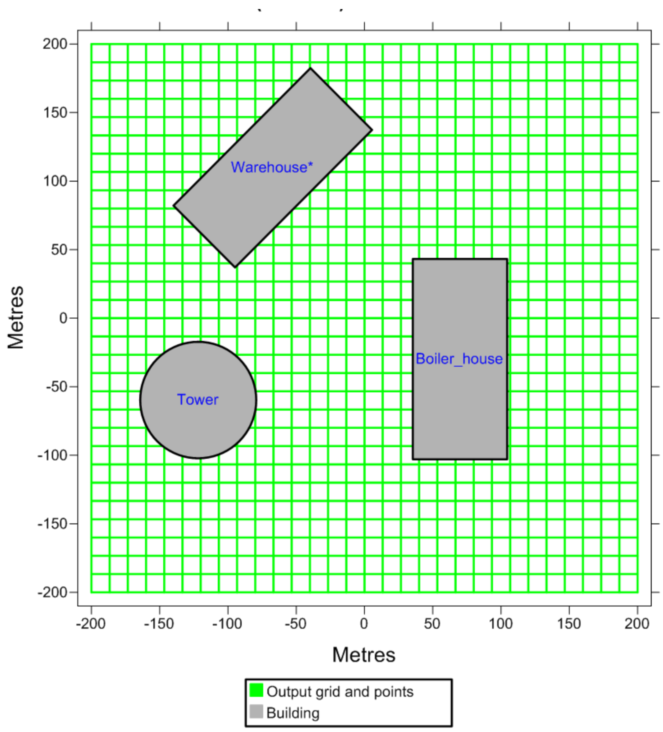


Figure 7.8 – Rectangular and circular buildings.

Roughness area

The area covered by the roughness file is represented as an open blue rectangle (**Figure 7.9**).

Terrain area

The area covered by the terrain file is drawn as an open brown rectangle (**Figure 7.9**).

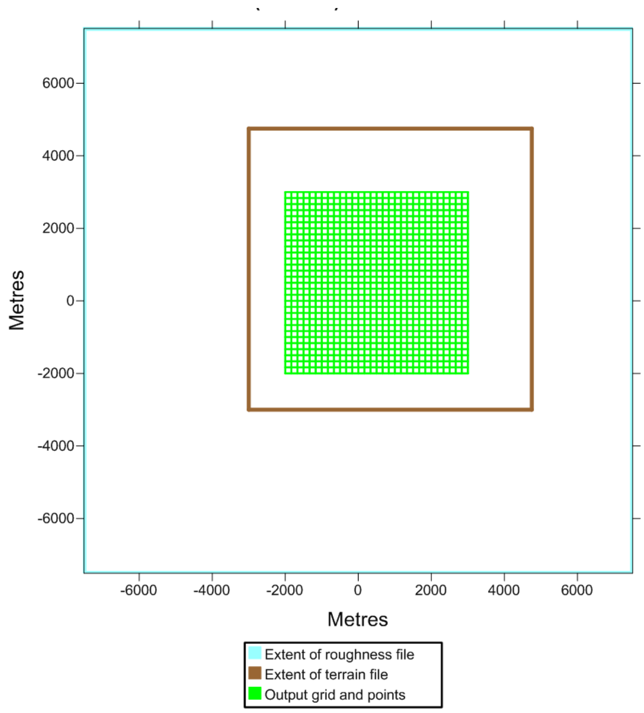


Figure 7.9 – Output grid and extent of terrain and roughness data.

Coastline

The coastline (position and orientation) is represented as a blue line labelled with the word "Coastline" at its centre. The coastline is defined as an infinitely long line, but only the portion of this line, if any, that falls inside the output grid is shown. The coastline can only be shown for scenarios with a regular, Cartesian, output grid. An example is shown in **Figure 7.10**.

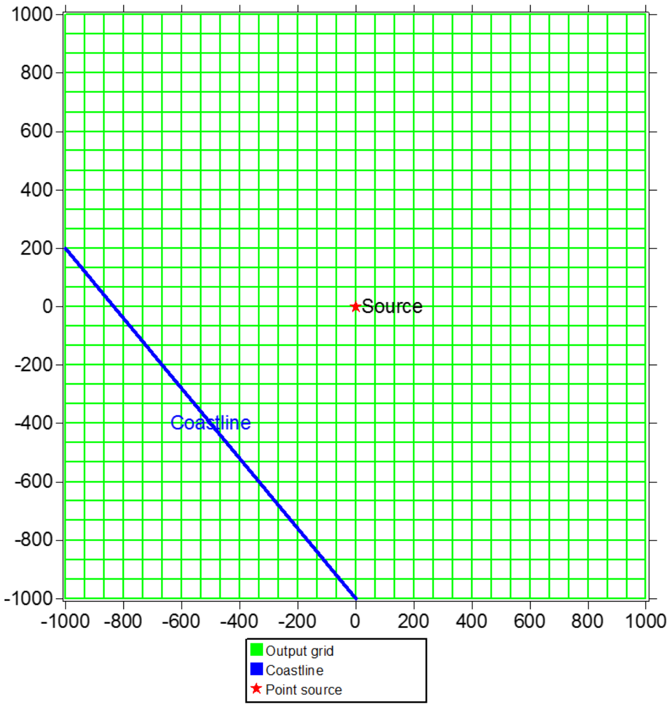


Figure 7.10 – A regular Cartesian grid, a source and a coastline.

7.4 Creating and running a batch file

It is often useful to be able to run several files consecutively, without opening each model file in the ADMS-Urban interface and using the **Run!** menu option. This can be done using a batch file. A batch file is a plain text file with the extension *.bat*, containing DOS commands.

7.4.1 Format of batch files

The most common syntax for running ADMS-Urban is:

```
<model path name> <file path name> /e2
```

where <model path name> is the full path name of the ADMS-Urban executable file (*ADMSUrbanModel.exe* located in the model installation directory), enclosed in inverted commas ("),

<file path name> is the full path name of the model file (*.upl*) you wish to run, enclosed in inverted commas ("),

/e2 is an option to cancel the prompt window at the end of the model run.

For example, if ADMS-Urban is installed in the directory *C:\Program Files (x86)\CERC\ADMS-Urb* and the model file to run is *D:\MyRun\Test.upl*, the command will be:

```
"C:\Program Files (x86)\CERC\ADMS-Urb\ADMSUrbanModel.exe" "D:\MyRun\Test.upl" /e2
```

Repeat this line for each model file you are running. The text in a *.bat* file is not case-sensitive. To start the model run(s), double-click on the *.bat* file in Explorer.

If using the **Spatial splitting** *option in* **Every Region** *mode, add the flag* /regionX *to the command to run a specific region, where* X *should be replaced by the region number. Repeat the full command for each region to be run.*

7.4.2 Batch File Creator utility

Batch files may also be created and run using the **Batch File Creator** utility of ADMS-Urban.

To launch it, select the **Utilities, Create/Run batch file** menu command. It opens the screen shown in **Figure 7.11**.

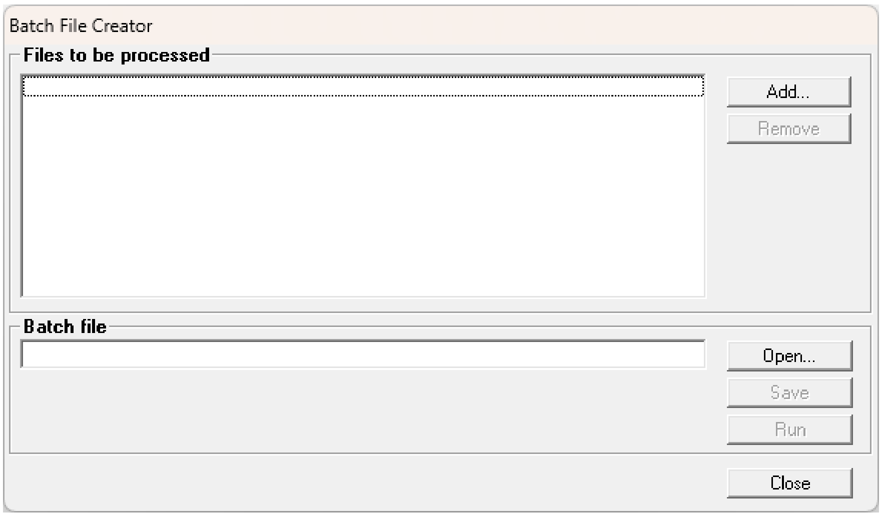


Figure 7.11 – **Batch File Creator** utility.

To set up and run a batch file, proceed as follows.

Step 1 Click on **Add...** and browse for the model file (*.upl*) to be run. The path and name of the selected file is added to the **Files to be processed** box.

Repeat until all the files you wish to run are included in the list (**Figure 7.12**). Files may be removed from the list with the **Remove** button.

Multiple model files can be selected together in the browse window and added. In this case, they are listed alphabetically.

*For any model file using the **Spatial splitting** option in **Every Region** mode, the batch file creator will automatically generate one run command per region upon saving the batch file.*

Step 2 Save the batch file by clicking on **Save**. The file is saved with the extension *.bat*.

The batch file must always be saved prior to running.

Step 3 Run the batch file immediately (**Run**) or exit the utility (**Close**).

The model files are run in the order in which they are listed.

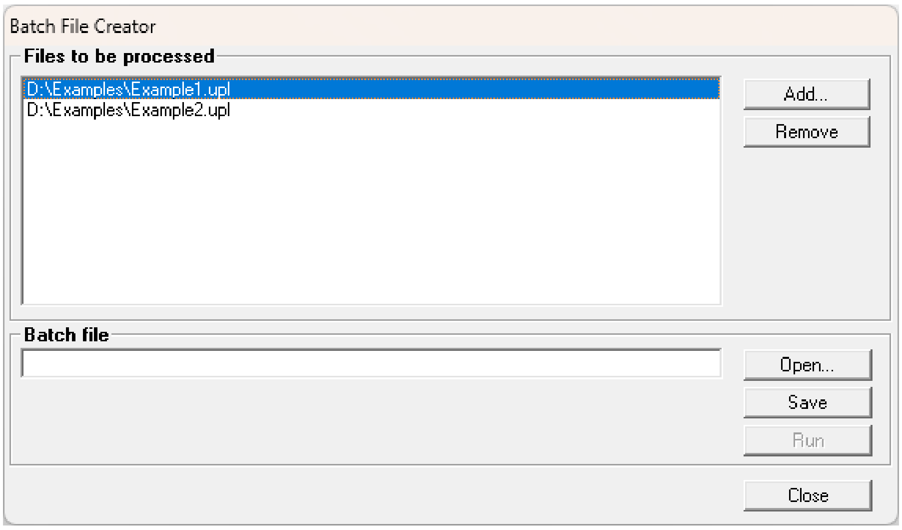


Figure 7.12 – Selection of files to be included in the batch file.

To open and edit or run a batch file which you have created earlier, click **Open...** and browse for the batch file. The model files to be run will be listed in the **Files to be processed** box.

7.5 Converting meteorological data

The **Convert met. data** utility (**Figure 7.13**) converts two types of meteorological data used in the United States into the ADMS-Urban format. The two accepted types of data are:

1. NOAA/NCDC files (*.dat*) created using the HUSWO program to extract data from a NOAA/NCDC Hourly Weather Observations archive.
2. Surface data files (*.sfc*) used as input files for the American AERMOD model.

In addition it is possible to incorporate upper air files with the NOAA/NCDC files, or to incorporate an upper air file into an existing ADMS-Urban format meteorological data file (*.met*). There are two accepted formats of upper air files:

1. FSL Rawinsonde data format from NOAA.
2. Upper air data format used as input to Stage 2 of the American AERMET meteorological processor.

The meteorological converter will ***not*** *handle blank lines or missing data. Input data must be complete otherwise output data will be incorrect.*

Some examples of input files are supplied with ADMS-Urban as well as the corresponding meteorological output files produced. See the files *<install_path>\Examples\ex_dat.** and *<install_path>\Examples\ex_sfc.**.

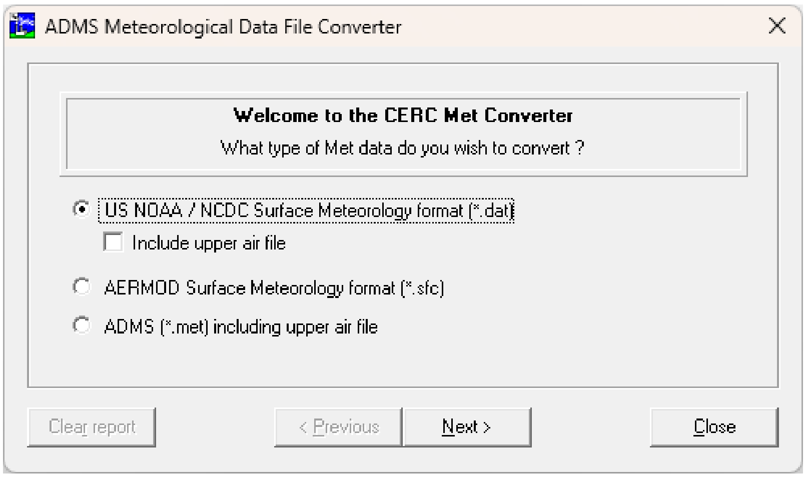


Figure 7.13 – **Convert met. data** utility.

7.5.1 Running the utility

To launch the utility, select the **Utilities**, **Convert met. data** menu command to access to the screen shown in **Figure 7.13**. Use the **< Previous** and **Next >** buttons to navigate between the screens.

Step 1 Select the type of meteorological data you want to convert.

Step 2 Indicate successively the path and name of the file to convert, the upper air file (if one is being used), the meteorological output file and the report file.

By default, the output and report files are located in the same directory as the input file. They use the same generic name with different extensions (.met *and* .rpt *respectively).*

Step 3 Once you are ready, click on **Convert** in the screen shown in **Figure 7.14**. The converted meteorological data are written to the output file (*.met*) and a report file (*.rpt*) is created with information on the conversion.

Beware that any previously existing files with the same name will be overwritten when you click on ***Convert****, without warning message.*

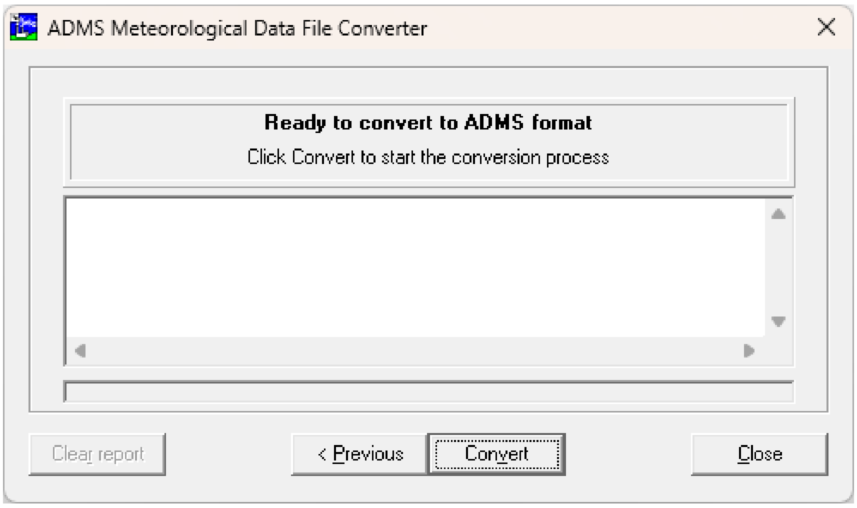


Figure 7.14 – **Convert met. data** utility, **Ready to convert to ADMS format** screen.

Step 4 If the upper air file supplied does not contain a longitude value then a screen to enter this will appear (**Figure 7.15**).

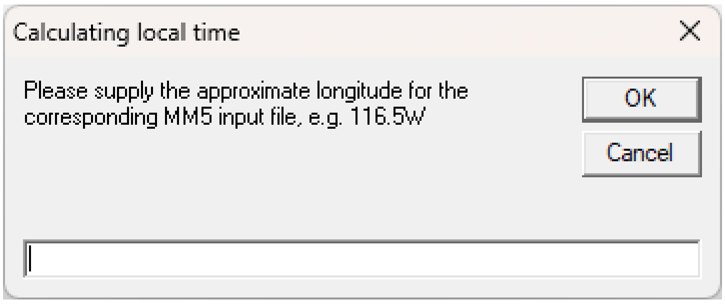


Figure 7.15 – **Convert met. data** utility, **Calculating local time** screen.

Step 5 During the conversion, data are written to the screen and to the report file (*.rpt*) (see further for a full description). When the conversion is finished (successfully or not), a processing completed screen appears, click **OK** to return to the **Convert met. data** utility (**Figure 7.16**).

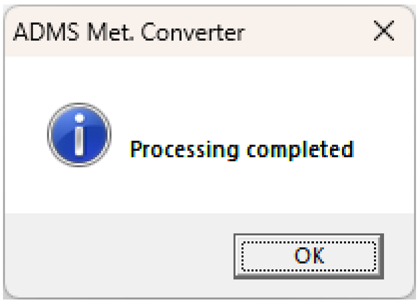


Figure 7.16 – **Convert met. data** utility, processing completed screen.

Step 6 Click on **Close** to exit the utility. Alternatively, click on **< Previous** to return to the earlier screens to convert another file. **Clear report** can be used to clear the report data shown on the screen.

7.5.2 Format of the output meteorological data file (*.met*)

Conversion from NOAA/NCDC* .dat *files

In order to successfully convert the NOAA/NCDC *.dat* files into ADMS-Urban format, a minimum number of variables must be present in the file, given in metric units. These are:

- the station identifier (automatically included when the *.dat* file was created),
- the time (automatically included when the *.dat* file was created),
- the sky cover,
- the dry bulb temperature,
- the relative humidity,
- the station pressure,
- the wind direction,
- the wind speed, and
- the hourly precipitation.

Additional variables present in the file will not be included in the output *.met* file produced by the conversion utility.

The header line of the output *.met* file contains the station identifier. Then follows a list of the variables included in the *.met* file. Then the following variables are listed:

- the year (`YEAR`),
- the month (`MONTH`),
- the day of the month (`DATE`),
- the Julian day (`DAY`),
- the hour of the day (`HOUR`),
- the wind speed (`U`),

- the wind direction (PHI),
- the cloud cover (CL),
- the surface temperature (T0C),
- the relative humidity (RHUM), and
- the precipitation rate (P).

If an upper air file is included in the conversion the buoyancy frequency above the boundary layer (NU) will also be included in the *.met* file. A full description of the ADMS-Urban meteorological data is given in Section 9.1.

The month and the day of the month are given for information only, and are not used by ADMS-Urban.

Conversion from AERMOD .sfc *files*

The header line of the *.sfc* file is reproduced at the start of the output *.met* file. Then follows a list of the variables included in the *.met* file. Then the following variables are listed:

- the year (YEAR),
- the month (MONTH),
- the day of the month (DATE),
- the Julian day (DAY),
- the hour of the day (HOUR),
- the surface heat flux (FTHETA0),
- the reciprocal of the Monin-Obukhov length (RECIPLMO),
- the surface albedo (ALBEDO (M)),
- the wind speed (U),
- the wind direction (PHI), and
- the surface temperature (T0C).

A full description of the ADMS-Urban meteorological data is given in Section 9.1.

The month and the day of the month are given for information only, and are not used by ADMS-Urban.

Conversion from ADMS .met files

The output *.met* file is a copy of the input *.met* file with the addition of the buoyancy frequency above the boundary layer (NU). If the input *.met* file already contains the buoyancy frequency above the boundary layer the conversion will not take place.

7.5.3 Format of the report file (*.rpt*)

Conversion from NOAA/NCDC **.dat** *files*

The report file lists the path and name of the NOAA/NCDC input file, the meteorological output file and the report file, and reproduces the station identifier from the input file.

If an upper air file is included then the path and name of the upper air file is also included in the report.

Conversion from AERMOD **.sfc** *files*

The report file lists the path and name of the AERMOD input file, the meteorological output file and the report file. It reproduces the header line from the input file, and provides the range of roughness lengths Z_0, the heights of the wind measurements Z_u and the Monin-Obukhov length L_{MO} of the input file. Z_0 and Z_u are not included in ADMS-Urban *.met* files, since they have to be given as single representative values in the **Meteorology** tab of the ADMS-Urban interface. The range of values of L_{MO} provides useful information when using the ADMS-Urban option to restrict the values of L_{MO} to avoid unrealistically stable boundary layers.

Conversion from NOAA/NCDC **.dat** *files*

The report file lists the path and name of the meteorological input file, the upper air file, the meteorological output file and the report file.

7.6 Making a terrain file

To run the **Complex terrain** model option, ADMS-Urban requires data to be input describing the topography surrounding the dispersion site. Section 4.16 describes the **Complex terrain** model option in detail and includes a description of the format of ADMS-Urban terrain files.

Terrain files can be created using the Mapper, refer to Section 6.1 of the Mapper User Guide for details. The Mapper can be used with a variety of input data files including OS Terrain 50 and SRTM data.

ADMS-Urban includes a utility called the **ADMS Terrain Converter** (see **Figure 7.17**) to convert the following formats of digital terrain data into ADMS-Urban format:

1. OS Landform PROFILE NTF (Great Britain only),
2. OS Landform PANORAMA NTF (Great Britain only),
3. IGN XYZ (France only),
4. OS Northern Ireland DTM (Northern Ireland only),
5. OS Northern Ireland XYZ (Northern Ireland only),
6. OS Ireland XYZ (Eire only).

For options (1) and (2) listed above, the **ADMS Terrain Converter** also informs the user which map tiles they need to purchase or download in order to include their chosen area of complex terrain in an ADMS-Urban calculation.

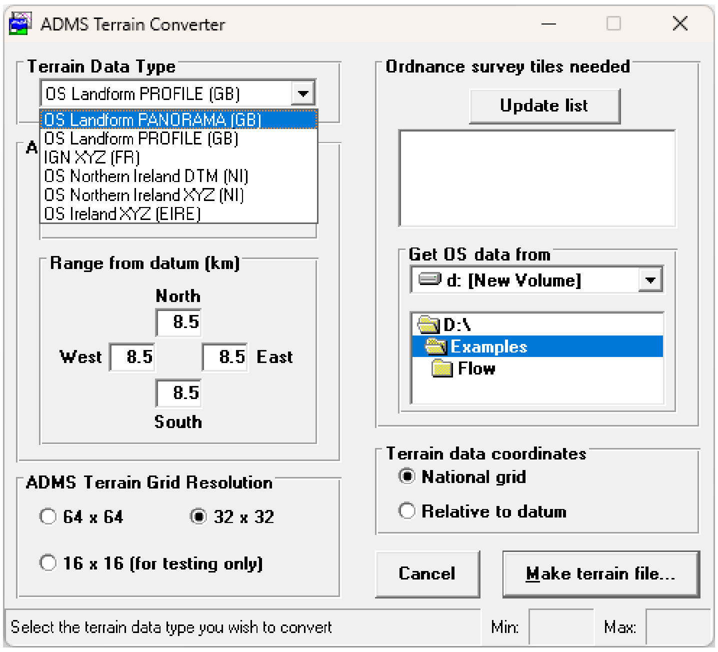


Figure 7.17 – Main screen for **ADMS Terrain Converter**.

To start the utility, click on **Utilities**, **Create terrain file** from the menu bar at the top of the main ADMS-Urban interface. This will open the screen shown in **Figure 7.17**. To change between different digital data input formats, use the drop-down list in the top-left corner of the screen.

For more information regarding the file formats listed above and contact details for suppliers of these data, please refer to Appendix F.

7.6.1 Creating a terrain file from OS Landform NTF (GB) data

Ordnance Survey (OS) data for England, Scotland and Wales are available in NTF tiles. **Landform PANORAMA** NTF data are supplied in 20 km × 20 km tiles with 50-m resolution. **Landform PROFILE** NTF data are supplied in 5 km × 5 km tiles with 10-m resolution. These tiles use the National Grid coordinate system, where the whole region is divided into 100 km × 100 km 2-lettered squares (see **Figure 7.18**).

Each 2-lettered square is made up of 25 smaller squares each with an area of 20 km × 20 km. Each 20 km × 20 km square is further divided into 16 smaller squares, each with an area of 5 km × 5 km. The coordinate (0,0) is the bottom left-hand corner of the square SV.

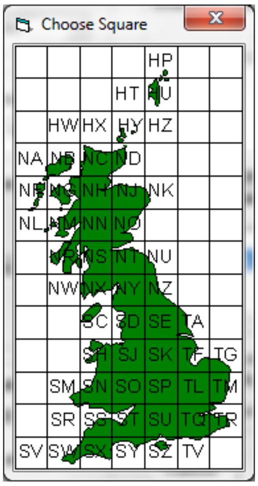


Figure 7.18 – Reference map of UK showing National Grid 100 km squares.

Area to cover

In the **Grid reference of datum point** box, enter the datum point for the area of interest in 5-figure National Grid Coordinates accompanied by a 2-letter code. Click on the following in order:

- **SV**: this is a default National Grid Reference. Double-click on this box to display the National Grid map shown in **Figure 7.18**. Click on the grid square that contains the source(s) for your modelling scenario, and the corresponding letters will automatically be entered in this box. Alternatively, if you know them, type the letters directly into the box.
- **00000,00000**: the first is the default Easting, the second the default Northing. Note that this grid reference is relative to the south-west corner of each lettered square. Enter the Easting and Northing of the datum, e.g. the primary source position, in metres. An accuracy of 100 m may be sufficient, except for near-field calculations when 10 m resolution is required. The minimum value is 0 m, while the maximum is 99999 m.
- **Range from datum (km)**: in each box, enter the distance from the primary source,

in the specified direction, to be included in the terrain file.

- **ADMS Terrain Grid Resolution**: the following resolutions for the terrain grid are available:

  64 × 64 for accurate results in larger domains
  32 × 32 for accurate results
  16 × 16 to be used for testing only

  These grid sizes refer to the resolution of the terrain data in the *.ter* file; the resolutions listed in Section 4.16 refer to the ADMS-Urban internal calculation grid. For best results users might select a high resolution for both the terrain data and the internal calculation grid but this will increase model run times.

Ordnance survey tiles needed

Once the **Area to cover** box has been completed, click on the **Update list** button. The Ordnance Survey tiles required to make the terrain file for the chosen area will be listed in the box.

Then use the **Get OS data from** box to specify where the Ordnance Survey data files are stored. If the Ordnance Survey tile data have already been saved on the hard disk, they can be accessed by specifying drive *C:*. When loading data from disk, choose the drive, e.g. *D:*, from which the Ordnance Survey tile data are to be loaded.

In the **Terrain data coordinates** box, choose either the **National grid** option or the **Relative to datum** option to specify in which format the coordinates are to be in the created terrain file. If **National grid** is selected, the data will be in 6-figure numbers relative to the origin at SV 000000, 000000. If **Relative to datum** is selected, the data will be in metres relative to the datum.

When the required information has been input, click on the **Make terrain file...** button. You will be prompted to name the terrain file to be created and specify where it is to be stored.

When you click on the **OK** button, the program will start loading Ordnance Survey data and will display a progress window (**Figure 7.19**).

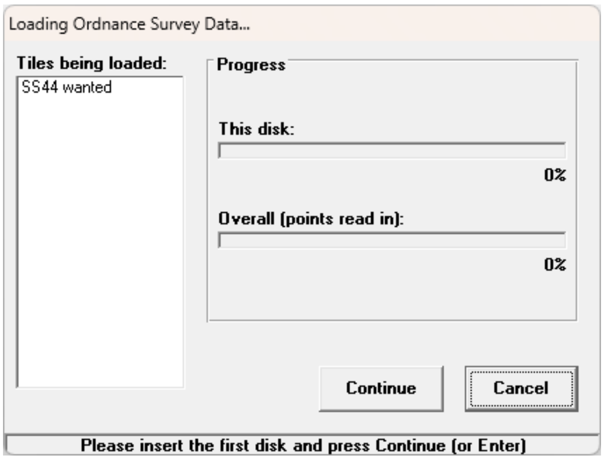


Figure 7.19 – The **Loading Ordnance Survey Data…** progress window.

The name of the file being loaded is shown at the bottom of the screen. A 'beep' will sound after the successful loading of each file, and the message shown in **Figure 7.20** will appear on the screen when the terrain file is complete.

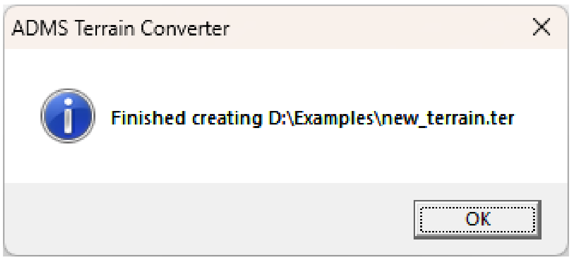


Figure 7.20 – Message to show terrain file has been successfully created.

7.6.2 Creating a terrain file from IGN XYZ (FR) data

Selecting the area to cover

When choosing the area that you wish to cover with ADMS-Urban terrain data, you must think about the boundaries of your IGN data.

If the area you wish to cover overlaps a coastline, and therefore contains missing data over the sea, then proceed as normal. The **ADMS Terrain Converter** will prompt you to identify the missing data as sea and will then set the elevation of the missing data points to be zero. If this is not the case, then the area you choose to cover should lie within the area covered by your IGN data. The **ADMS Terrain Converter** will detect any missing data, for instance at international boundaries, and will prompt you either to select your area manually, or to use the **ADMS Terrain Converter** facility to re-calculate the extent of the area to cover to avoid missing data at the edges.

Creating the terrain file

To convert French IGN XYZ data to ADMS-Urban terrain data, follow the steps below:

Step 1 Select **IGN XYZ (FR)** from the drop-down list in the top left corner of the **ADMS Terrain Converter** interface.

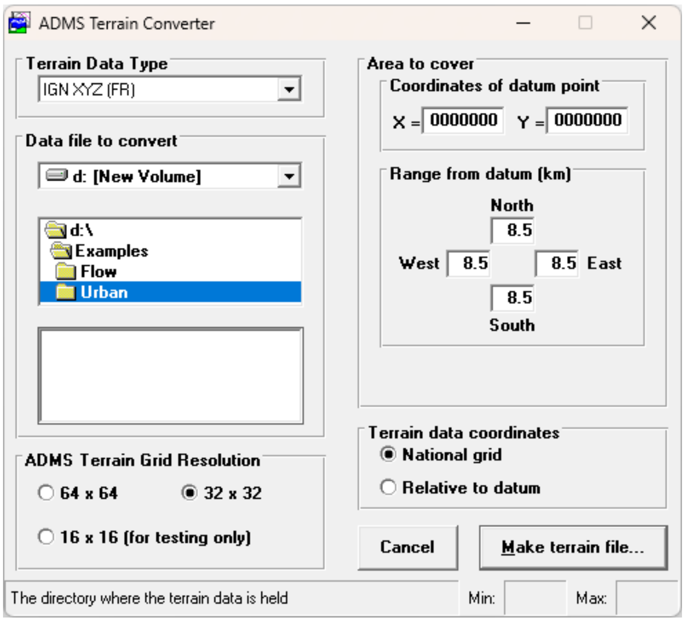


Figure 7.21 – **ADMS Terrain Converter** screen with **IGN XYZ (FR)** data selected.

Step 2 In **Data file to convert**, select the drive, directory and file name of the IGN terrain file that you wish to convert to ADMS-Urban format.

Step 3 Select the **ADMS Terrain Grid Resolution** you require. The following resolutions for the terrain grid are available:

64 × 64 for accurate results in larger domains
32 × 32 for accurate results
16 × 16 to be used for testing only

These grid sizes refer to the resolution of the terrain data in the *.ter* file whereas the resolutions listed in Section 4.16 refer to the ADMS-Urban internal calculation grid. For best results users might select a high resolution for both the terrain data and the internal calculation grid but this will increase model run times.

Employing the finest internal calculation grid significantly increases the run time for each problem, but also increases the accuracy of the solution.

Step 4 In **Area to cover**, enter the coordinates of the datum point, in metres. This may be any point.

Step 5 Also in **Area to cover**, enter the range from the datum point (in kilometres) that you wish the ADMS-Urban terrain grid to cover.

Step 6 Select the **Terrain data coordinates**. If you choose **National grid** then the coordinates in the ADMS-Urban terrain file will be the same as the coordinate system of the IGN terrain data. If you choose **Relative to datum** then the coordinate system used in the ADMS-Urban terrain file will have the origin at the datum point.

Step 7 You are now ready to make the ADMS-Urban terrain file. Click the **Make the terrain file…** button.

Step 8 Specify a directory and file name for the newly created ADMS-Urban terrain file.

Step 9 When the ADMS-Urban terrain file has been successfully created, the **ADMS Terrain Converter** will beep and display the following message:

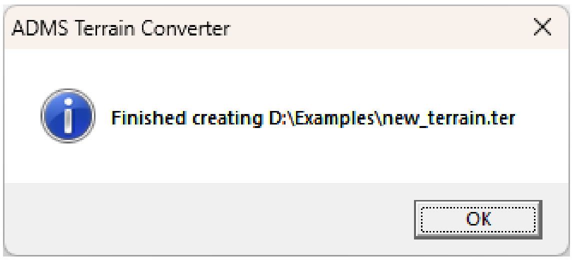


Figure 7.22 – Message to show terrain file has been successfully created.

7.6.3 Creating a file from OS Northern Ireland DTM (NI) data

Northern Ireland terrain data are supplied as 20 km × 20 km DTM data tiles in the Irish National Grid format, which is shown in **Figure 7.23**.

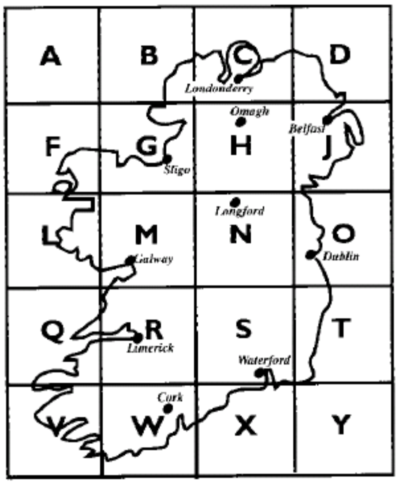


Figure 7.23 – The Irish National Grid.

Each lettered grid square covers an area of 100 km × 100 km; the point in the bottom left corner of grid square V has coordinates (0,0).

If a source is positioned in the centre of the H i.e. at 50,000 m, 50,000 m, the actual 6-figure National Grid coordinate is (250000, 350000). It is up to you to find the coordinates of the south-west and north-east corner of the 20 km × 20 km square area around the source that you are considering. These coordinates should be supplied to OS-NI when ordering data.

This option is different to the OS Landform options in that it does not tell you which tiles you need to obtain for a certain area of terrain, but rather you add all the NI terrain data you have to an index. When you choose a distance from the source point in every direction that you want to include, the utility will either create that terrain file from the data you have in the index or tell you if you do not have the required amount of data.

To convert Northern Irish DTM data to ADMS-Urban terrain data, follow these steps:

Step 1 Select **OS Northern Ireland DTM (NI)** from the drop-down list in the top left-hand corner of the **ADMS Terrain Converter** interface.

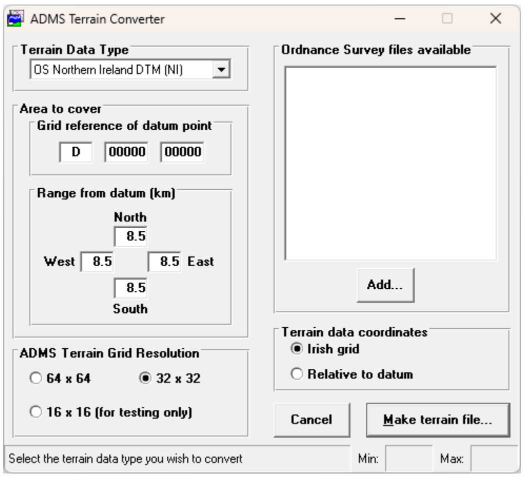


Figure 7.24 – The **ADMS Terrain Converter** screen for Northern Ireland DTM data.

Step 2 Once you have obtained the data you require, it must be added to an index in the utility. Firstly, ensure the data is copied onto the hard disk of your machine. Click on the **Add** button and browse for your files. Locate all the NI DTM terrain data and add each file that you have in turn.

The utility will build the index progressively as each new file is added. All the data you add will be shown in the **Ordnance Survey files available** box. The data need only be added once and the utility will always look for the data in the same place each time it is used.

Step 3 Enter the letter of the grid square e.g. **D** within which your source or datum point is positioned. Then enter the coordinates (easting then northing) of that point within the chosen square (0-99999 m).

Step 4 The next step is to enter the distance in kilometres from the point that you want to include in the terrain file in each direction.

Step 5 The user must then choose the grid size required. This must be 16 × 16, 32 × 32 or 64 × 64. Increasing the grid resolution gives a more accurate representation of the terrain but can increase ADMS-Urban run times. A 32 × 32 grid resolution is typical, but if the domain is large or if the terrain is particularly complex, a grid resolution of 64 × 64 should be selected. A 16 × 16 grid resolution should only be used for testing purposes.

Step 6 Choose whether the resultant terrain file should give coordinates in the Irish National Grid Coordinate system or relative to the datum. The default is the former.

Step 7 When all of the above steps have been completed, click on **Make terrain file...**. You will be prompted to name and save the new file in a directory of your choice.

Step 8 Once you have chosen the name and location for the file and selected **Save** one of two things may happen:

1) You have the necessary data to complete the file in the index and the

terrain file is completed. If this is the case the utility will indicate that it has found the data. It will then show the window from **Figure 7.25** to say that the terrain file has been successfully created.

Figure 7.25 – Message to show terrain file has been successfully created.

2) The data required to make the specified terrain file are not available. In this case a window such as that shown in **Figure 7.26** will appear detailing what data are missing. The green dots refer to areas for which you do have data; blue dots refer to known areas of sea (the OS data only records these very close to the coast) and the red dots refer to areas where no data are available i.e. in the sea or where you need to buy more data. **Figure 7.26** shows that the area of terrain chosen extends across into the Republic of Ireland and was therefore not included in the index.

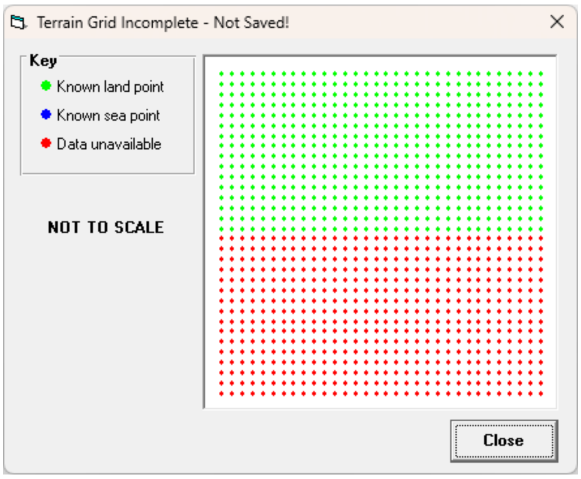


Figure 7.26 – Data not available for the terrain file.

If this window appears then the terrain file has not been created. You are required to change the extent of the area you want to cover in the interface so that it covers the available data, or obtain more data from OS-NI. (It is important to note that when the file has not been completed, the window does not draw the area to scale. It will always be a square of the same number of dots so images may appear compressed or elongated.)

7.6.4 Creating a terrain file from OS Northern Ireland XYZ (NI) data

Terrain data for Northern Ireland are also available in XYZ format, which lists the X, Y and Z coordinates of each data point separated by commas. The Z coordinates are in decimetres and there is a header line at the start of the file. Coordinates that are in the sea are given the value -1000. Files of this type should have the extension *.txt*.

To create an ADMS-Urban terrain file from this type of data, first select **OS Northern Ireland XYZ (NI)** from the drop-down list in the top left hand corner of the **ADMS Terrain Converter** interface and then follow Steps 2 to 9 from the instructions for French IGN XYZ data (see Section 7.6.2).

7.6.5 Creating a terrain file from OS Ireland XYZ (EIRE) data

Data for the Republic of Ireland are based on the same National Grid as data for Northern Ireland (see **Figure 7.23**). Data files have the extension *.dtm* but are in XYZ format. This is a simple text file that lists the X, Y and Z coordinates separated by spaces. The Z coordinates are in metres and there is no header. Coordinates that are in the sea are given the value 0.

To create an ADMS-Urban terrain file from this type of data, first select **OS Ireland XYZ (EIRE)** from the drop-down list in the top left hand corner of the **ADMS Terrain Converter** interface and then follow Steps 2 to 9 from the instructions for French IGN XYZ data (see Section 7.6.2).

7.7 Create ASP grid

The **ASP grid creator** utility **Figure 7.27** generates specified points in a user-defined grid and adds them to a specified points file (*.asp*); refer to Section 3.5.4 for more details on .asp files.

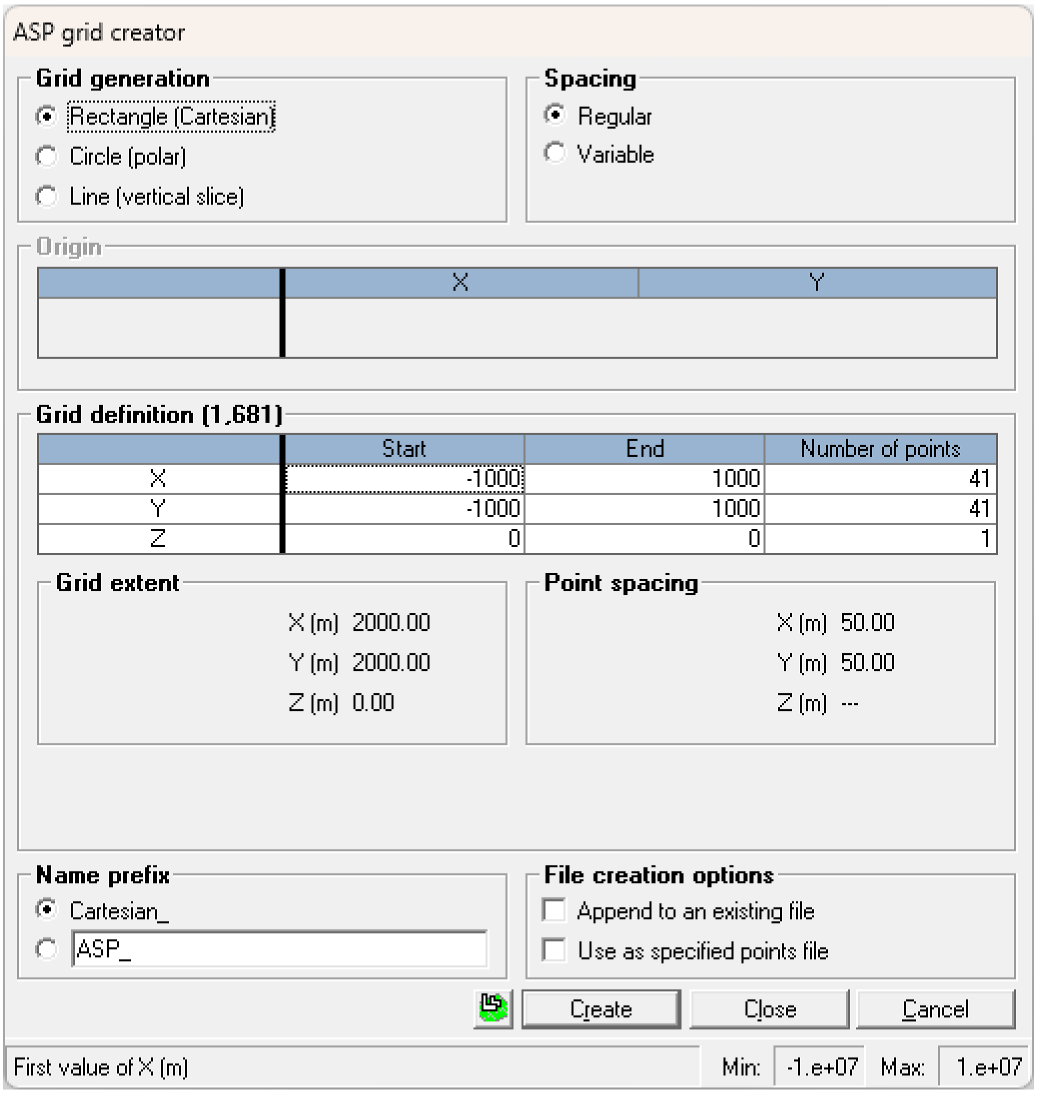


Figure 7.27 – ASP grid creator

The ASP grid creator utility creates specified points in a rectangular grid aligned with the coordinate system (Section 7.7.1), a circular grid around an origin (Section 7.7.2) or a rectangular grid around a specified line (Section 7.7.3). The grid may be viewed and the origin and/or grid extents edited in the **Mapper**. The created output points can be saved to a new specified points file (*.asp*) or appended to an existing one (Section 7.7.4).

To launch the utility, select the **Utilities**, **Create ASP grid** menu item to access the screen shown in **Figure 7.27**.

7.7.1 Rectangle (Cartesian) grids

To create a rectangular grid aligned with the coordinate system select **Rectangle (Cartesian)** in the **Grid generation** section. Two types of grid can be created, either **Regular** or **Variable**.

Regular grid

For a regular grid the **Start**, **End** and **Number of points** must be entered for each of the **X**, **Y** and **Z** directions, **Figure 7.28**. The **Grid extent** and **Point spacing** for each direction is automatically calculated from these parameters.

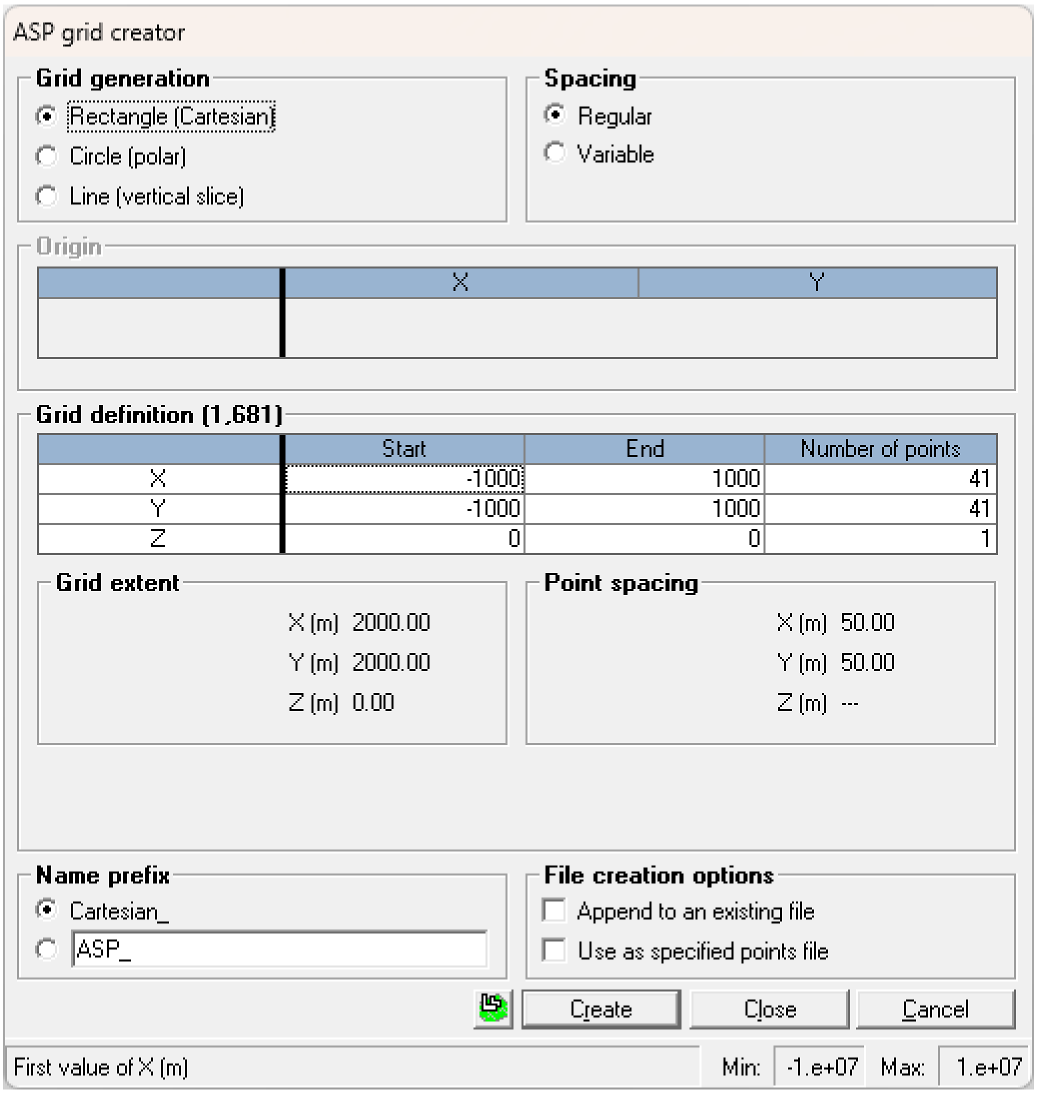


Figure 7.28 – **ASP grid creator** – **Regular Rectangle (Cartesian)** grid.

The **Mapper** displays two layers, the horizontal extent in the **ASP grid extent** layer and the points to be generated in the **ASP grid points** layer. The **ASP grid extent** layer can be redrawn , edited  or moved  to alter the **Start** and **End X** and **Y** values.

Variable grid

For a variable grid values need to be entered for each of the **X**, **Y** and **Z** directions, **Figure 7.29**. To enter a value click on the small text box in that column, type the value then press the **SPACE** bar. To remove points, click on them in the larger, upper text box, and press the **DELETE** key. A right-click menu provides a convenient way to delete many points at the same time. Points will be calculated on a grid of points at the specified **X**, **Y** and **Z** values.

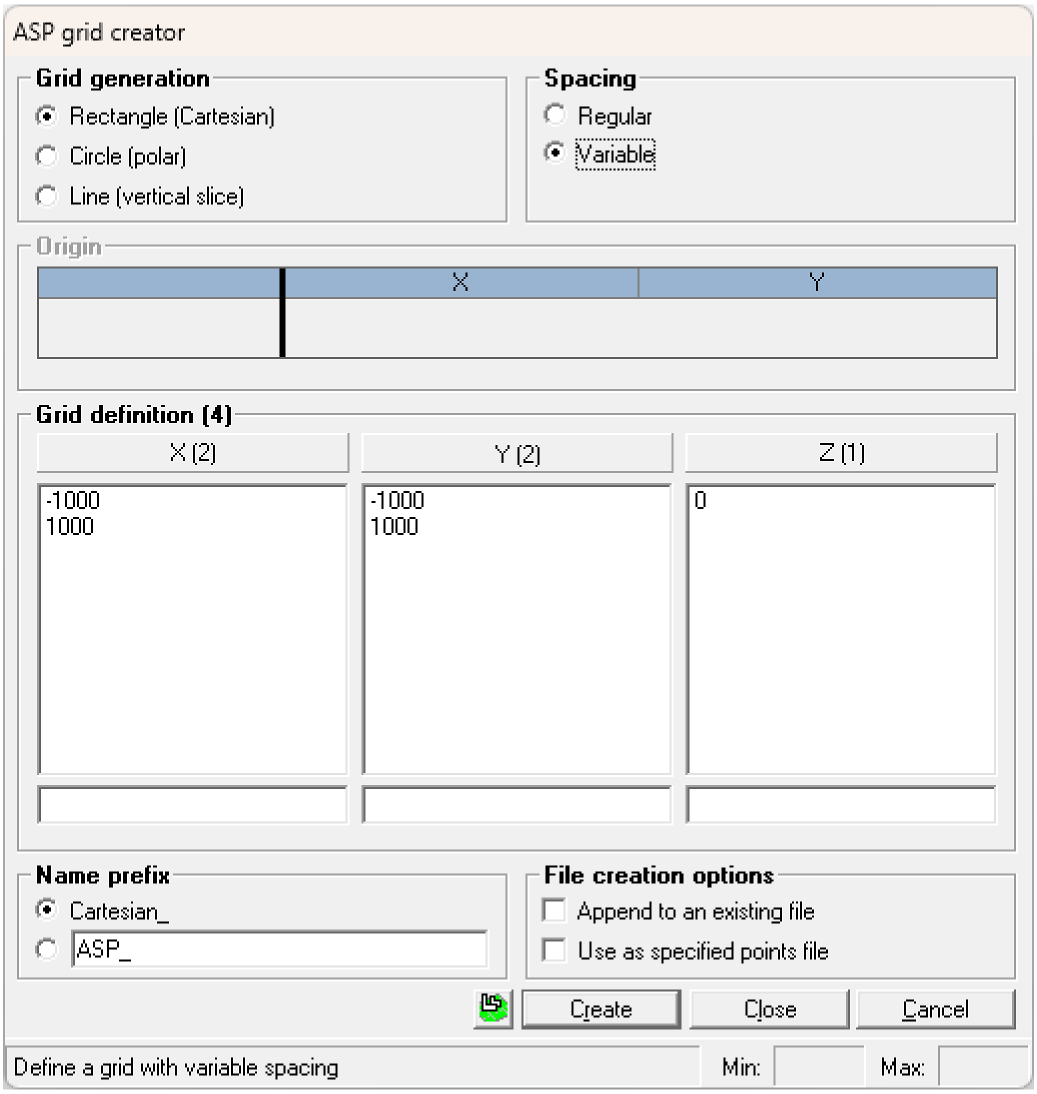


Figure 7.29 – ASP grid creator – Variable Rectangle (Cartesian) grid.

The **Mapper** displays two layers, the horizontal extent in the **ASP grid extent** layer and the points to be generated in the **ASP grid points** layer. The **ASP grid extent** layer can be redrawn , edited  or moved  to alter the **X** and **Y** values. The minimum and maximum **X** and **Y** values will be updated to the new values, and all other intermediate **X** and **Y** values recalculated to maintain the same relative position.

7.7.2 Circle (polar) grids

To create a circular grid around an origin select **Circle (polar)** in the **Grid generation** section. Two types of grid can be created, either **Regular** or **Variable**.

Regular grid

For a regular grid the **Origin X** and **Y** coordinates and the **Start**, **End** and **Number of points** must be entered for each of **Radius**, **Theta** and **Z**, **Figure 7.30**. The **Grid extent** and **Point spacing** for each of these is automatically calculated from these parameters. The **Theta** values are measured anti-clockwise from the positive X axis, if the **Start** value is higher than the **End** value the span of points will cross the 0° line.

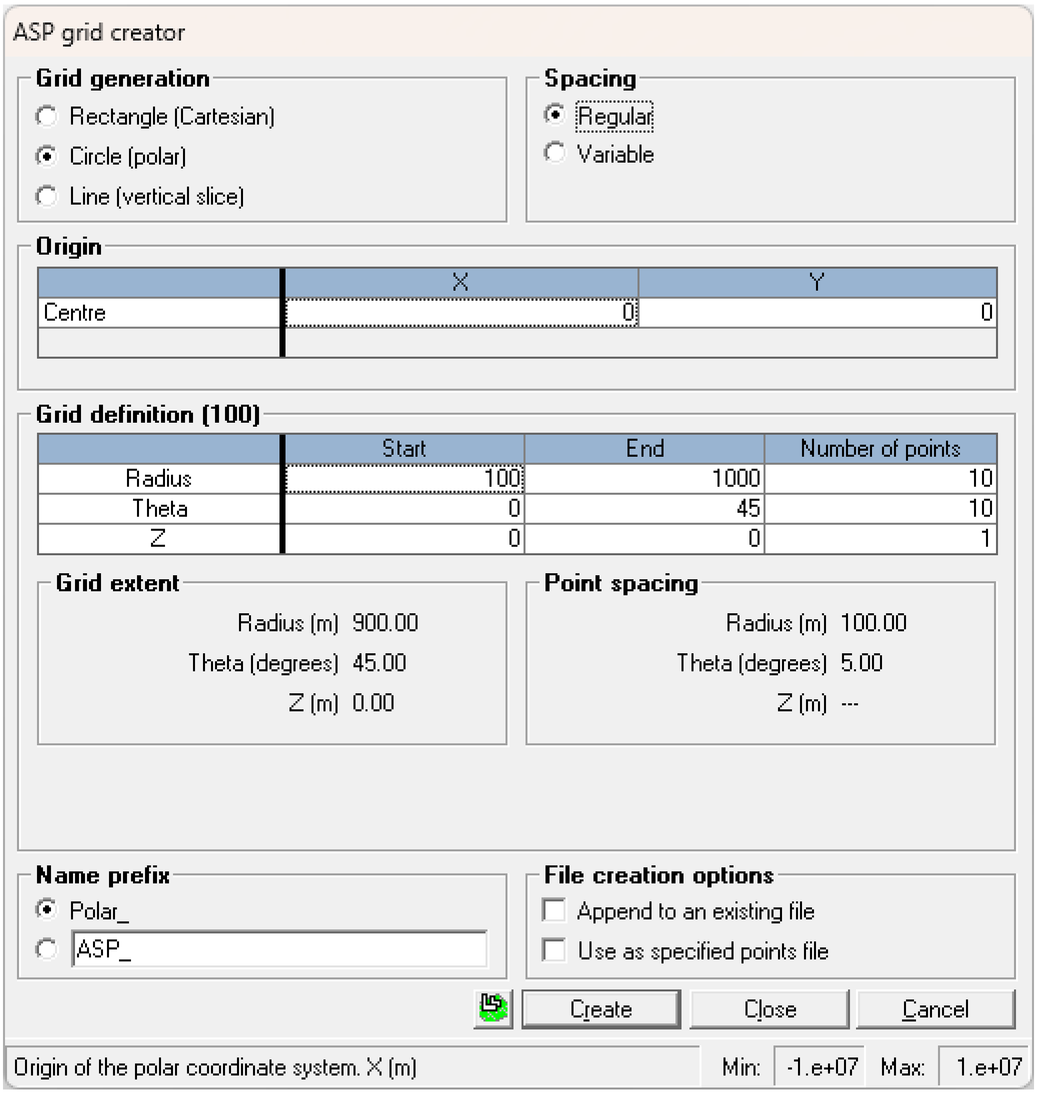


Figure 7.30 – **ASP grid creator** – **Regular Circle (Polar)** grid.

The **Mapper** displays two layers, the horizontal extent in the **ASP grid extent** layer and the points to be generated in the **ASP grid points** layer. The **ASP grid extent** layer can be redrawn , edited  or moved  to alter the **Origin** and **End Radius** values.

Variable grid

For a variable grid, in addition to the **Origin** coordinates values need to be entered for each of **Radius**, **Theta** and **Z**, **Figure 7.31**. To enter a value click on the small text box in that column, type the value then press the **SPACE** bar. To remove points, click on them in the larger, upper text box, and press the **DELETE** key. A right-click menu provides a convenient way to delete many points at the same time. Points will be calculated on a grid of points at the specified **Radius**, **Theta** and **Z** values.

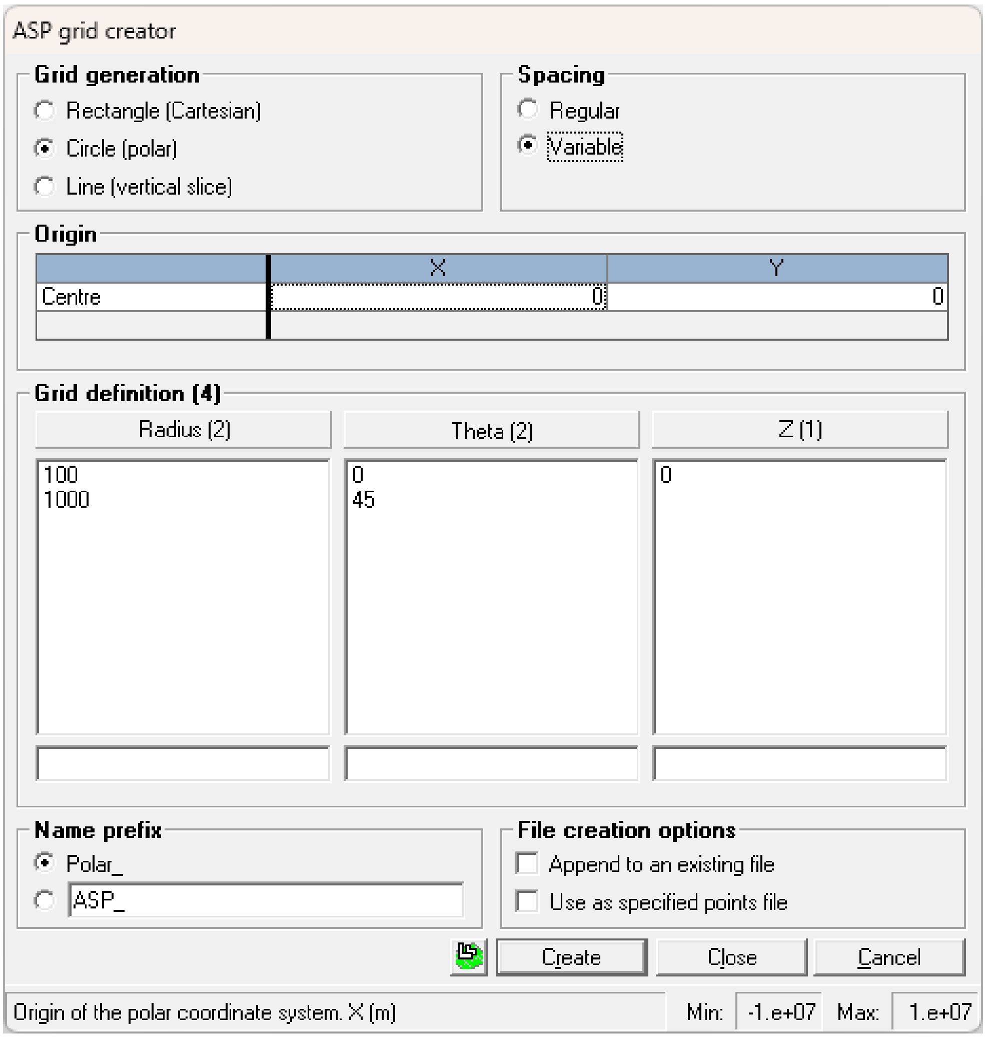


Figure 7.31 – **ASP grid creator** – **Variable Circle (polar)** grid.

The **Mapper** displays two layers, the horizontal extent in the **ASP grid extent** layer and the points to be generated in the **ASP grid points** layer. The **ASP grid extent** layer can be redrawn , edited  or moved  to alter the **Origin** and maximum **Radius** value, with all other **Radius** values recalculated to maintain the same relative position.

7.7.3 Line (vertical slice) grids

To create a rectangular grid aligned with a line select **Line (vertical slice)** in the **Grid generation** section. The line is defined in the **Origin** section as **Start** and **End X** and **Y** values. The **Mapper** displays the line in the **ASP grid extent** layer and can be used to redraw , edit  and move  the line.

The grid points are shown in the **Mapper** in the **ASP grid points** layer. The horizontal locations of the grid points are defined relative to the line in terms of their **Fractional position** and **Transverse distance.**

- **Fractional position** corresponds to the relative position along the line, 0 being the **Start** of the line and 1 being the **End** of the line. Values less than 0 or greater than 1 can be entered to extend the grid of points beyond the line drawn.
- **Transverse distance** corresponds to the perpendicular distance away from the line. Positive values represent to the left of the line when going from the **Start** of the line to the **End.**

Two types of grid can be created, either **Regular** or **Variable.**

Regular grid

For a regular grid in addition to the **Origin** the **Start**, **End** and **Number of points** must be entered for each of the **Fractional position**, **Transverse distance** and **Z**, **Figure 7.32**. The **Grid extent** and **Point spacing** for each of these is automatically calculated from these parameters.

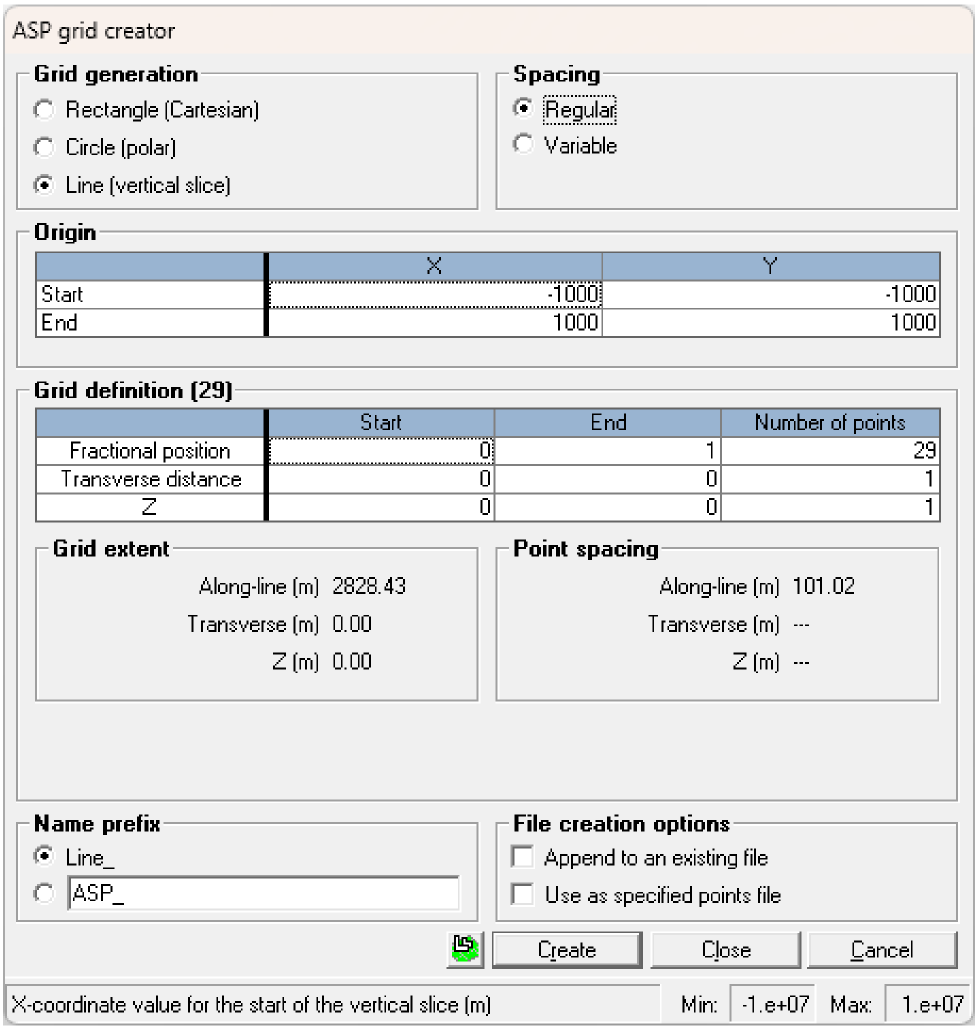


Figure 7.32 – **ASP grid creator** – **Regular Line (vertical slice)** grid.

Variable grid

For a variable grid in addition to the **Origin** values need to be entered for each of the **Fractional position**, **Transverse distance** and **Z**, **Figure 7.33**. To enter a value click on the small text box in that column, type the value then press the **SPACE** bar. To remove points, click on them in the larger, upper text box, and press the **DELETE** key. A right-click menu provides a convenient way to delete many points at the same time. Points will be calculated on a grid of points at the specified **Fractional position**, **Transverse distance** and **Z** values.

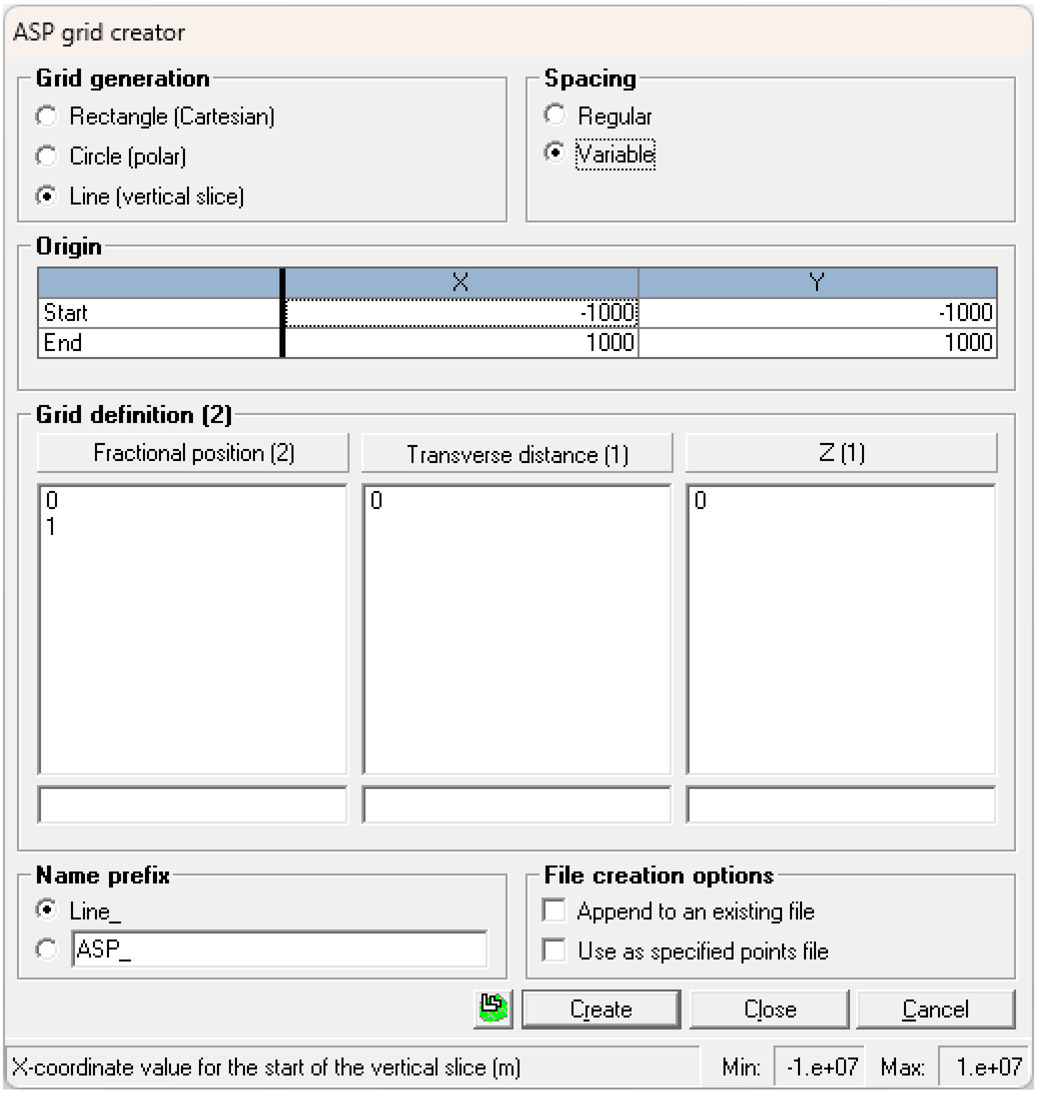


Figure 7.33 – **ASP grid creator** – **Variable Line (vertical slice)** grid.

7.7.4 Saving to a specified points file

In the **Name prefix** section, you may input an identifying prefix for the points in the specified points file. Choose between the default option for that grid type or enter a user specified prefix. This is helpful if you have more than one set of specified points in the file.

The **File creation options** section allows you to specify how the points will be output to a specified point file. Select **Append to an existing file** to add the points to an existing specified points (*.asp*) file, otherwise a new specified point file will be created. To set the current model file to use the file you are saving, check **Use as specified points file**.

Click **Create** and choose a filename, or select an existing file if appending, and then click **Save** to create/append to the chosen file.

SECTION 8 Worked Examples

In this section, worked examples to guide you through setting up some basic ADMS-Urban model runs and presenting their results are described. It is recommended that you work through these examples when starting to use ADMS-Urban. Note that a complete set of the resulting ADMS-Urban input and output files can be found in the *<install_path>\Examples* directory.

Some of the examples cover the use of ADMS-Urban with the Mapper. Further details regarding using the Mapper can be found in the Mapper User Guide which can be accessed from the **Help** menu of the Mapper.

You can launch ADMS-Urban in several different ways:

- double-click on the icon for the shortcut created earlier (see Section 2);
- use the Windows **Start** menu and select **Programs**, **ADMS-Urban**; or
- go to the main ADMS-Urban directory *<install_path>* and double-click on the file *ADMS-Urb.exe*.

It is strongly recommended to create an additional directory for setting up and running these examples, in order to keep them separate from the examples provided in <install_path>\Examples directory supplied with the model.

8.1 Example 1: Modelling a single elevated point source

In this example you will carry out a simple modelling run. In particular, you will learn:

- how to launch ADMS-Urban in stand-alone mode;
- about the ADMS-Urban model interface;
- how to add a new point source;
- how to enter source data such as volume flow rate, source diameter, source height, etc.;
- how to enter emission rates of different pollutants;
- how to select a meteorological input data file;
- how to edit a meteorological data file using a desktop editor;
- how to enter a constant background concentration;
- how to manually define an output grid;
- how to make an X-Y plot of the wind aligned ground level concentration; and
- about the location and magnitude of the maximum concentration for elevated point

sources under different weather conditions.

8.1.1 Setting up the run

Step 1 Start ADMS-Urban or click on **File**, **New**, if open, to create a new input file.

Step 2 Enter the **Name of site** and the **Name of project**.

These are optional, but meaningful titles should be entered as they help to differentiate between model runs. The titles will be written out to the log file of the model run.

Step 3 Move to the **Source** screen by clicking on the **Source** tab at the top of the ADMS-Urban window. From the source type drop-down list select **Industrial sources**.

Step 4 Click on the **New** button to add a source to the table.

Step 5 Enter a new name for the source.

Step 6 Click with the mouse on the data input section of the source table. You can change the default data for the source by typing in new values in highlighted cells. Notice that the source type says ‘Point’ for point sources. Leave the source height at 50 m. Change the source diameter to 3.5 m, the exit velocity to 25 m/s, and the temperature to 65.5 °C. Leave the coordinates of the source as (0,0).

Information about each input parameter appears in the help bar at the bottom of the interface as you select each one.

Step 7 Click on the **Emissions...** button and then click **New** to enter pollutant emissions for the source. By default, the pollutant shown is NO_X. Change the pollutant to SO_2 by clicking on the **Pollutant name** list and selecting it from there.

Step 8 Change the emission rate to 8.5 g/s. Click on the **OK** button to return to the Source screen.

Step 9 Move to the **Meteorology** screen by clicking on the tab at the top of the ADMS-Urban window.

Step 10 For the **Dispersion site Surface roughness (m)** select the value for **Agricultural areas (min.)** (0.2 m).

Step 11 Select the **From file** option in the **Met. data** box.

Step 12 Click on the **Browse...** button, and select the file *nsc.met* from the supplied *<install_path>/Data* directory. This file contains three lines of meteorological data representing convective, neutral and stable conditions, respectively.

Step 13 Untick the **Met. data in sectors of (degrees)** and **Met. data are hourly sequential** check boxes.

Meteorological input data

The file metdemo.met includes a list of all the meteorological parameters that can be used in a meteorology file and explains how to create a new file. The number of parameters in each line of data is specified at the beginning of the meteorology file, with the names of those parameters (in block capitals) listed below. The data values must be comma-separated.

Step 14 Move to the **Background** screen by clicking on the tab at the top of the ADMS-Urban window.

Step 15 Check the **Enter by hand** option.

Step 16 Click on the table and enter a background concentration of 5μg/m^3 for SO_2. (Change the units from ppb to μg/m^3 by clicking on 'ppb' in the **Units** column and selecting μg/m^3 from the list.)

Step 17 Move to the **Grids** screen by clicking the tab at the top of the ADMS-Urban window.

Step 18 If it is not already selected, choose **Gridded** output with **Regular spacing**.

Step 19 Enter (0,-200) as the lower left-hand coordinate and (3200, 3200) as the upper right-hand coordinate of the grid. Leave the number of points in the x and y directions at the default value of 31.

All coordinates are in metres. Recall that in this example the source is located at (0, 0).

Step 20 For the z-value, leave the **Minimum (m)** and **Maximum (m)** at 0, and the number of points as 1.

Step 21 Move to the **Output** screen by clicking on the tab at the top of the ADMS-Urban window.

Step 22 Click on the **New** button and choose SO_2 as the pollutant from the list.

Step 23 Select a short-term average (by selecting '**ST**' in the **Short/Long** column), with an averaging time of 15 minutes, and output units of μg/m^3. Check that there is a tick in the **Include** column for SO_2, so that SO_2 concentrations will be calculated for this model run.

Step 24 At the bottom of the screen, choose **Source** output to calculate output for one source only, and, if necessary, put a tick in the **Include** column next to the new source.

Step 25 From the **File** menu choose **Save As...**, enter a new file name, e.g. *example1.upl*, and browse to the directory where you would like to save the file. It is not recommended that files are saved in the model installation directory. Click **OK** to save the file.

Step 26 Run the model by choosing **Run!** from the menu bar.

8.1.2 Viewing line plotting output results

Step 1 From the **Results** menu choose **Line plot**

Step 2 Select the three *example1* output files (there is one per line of meteorological data).

Step 3 Choose SO_2 from the **Pollutants** list.

Step 4 Choose **Concentration**, **Ground Level** in the **Type of Graph** box.

Step 5 Click on the **Plot 1** button.

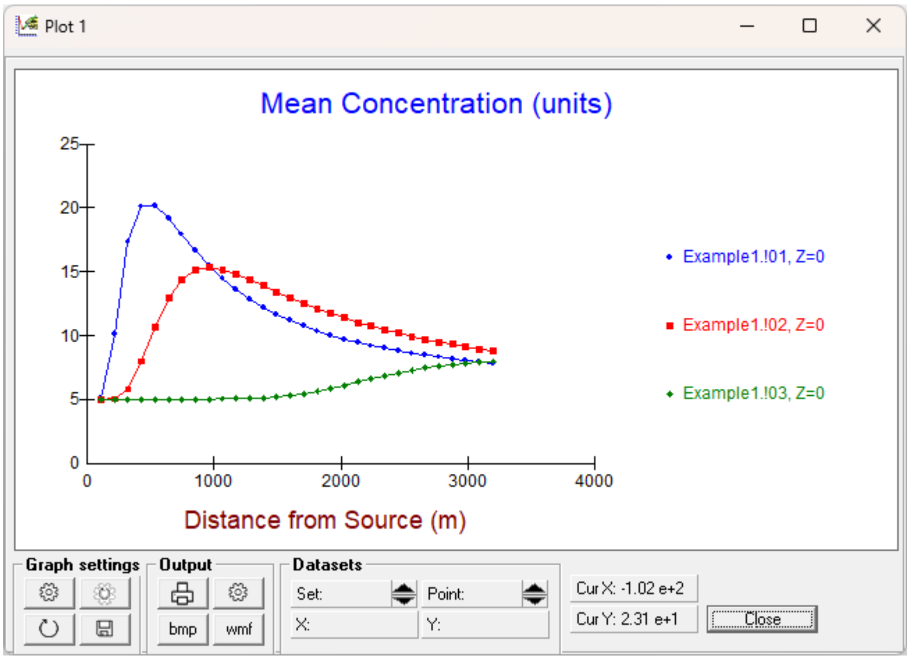


Figure 8.1 – Ground level concentration from an elevated point source.

The three lines show the ground level concentration from an elevated source, plus background concentration, under convective, neutral and stable conditions, respectively (**Figure 8.1**).

Worst case weather conditions for elevated source

We can see from the graph above that the highest ground level concentration for an elevated source occurs under convective conditions. This is due to strong mixing in the vertical direction. Under neutral conditions, the maximum ground level concentration is lower in magnitude and further downwind. Under stable conditions, the ground level concentration due to the source is very low. The plume spreads less quickly, and reaches the ground much further downstream.

Step 6 Click on the **Close** button.

Step 7 Close ADMS-Urban by selecting **Exit** from the **File** menu.

8.2 Example 2: Multiple point sources

In this example you will use the Mapper to enter point sources into ADMS-Urban, display the point sources and create a concentration contour plot. In particular, you will learn

- how to load a base map in the Mapper
- how to enter a point source using the Mapper
- how to create groups of sources and calculate the contribution of each group to the overall air quality
- how to add a new pollutant to the default list
- how to enter deposition parameters for different pollutants
- how to enter meteorological data manually
- how to produce a contour plot of the concentration in the Mapper

8.2.1 Opening the Mapper and adding a map tile

Step 1 Start ADMS-Urban or click on **File**, **New**, if open, to create a new input file.

Step 2 Enter a sensible **Name of site** and **Name of project**.

Step 3 Change the **Coordinate system** to *OSGB 1936 British National Grid (epsg:27700)*.

Step 4 Select **Mapper** from the menu to launch the Mapper, **Figure 8.2**.

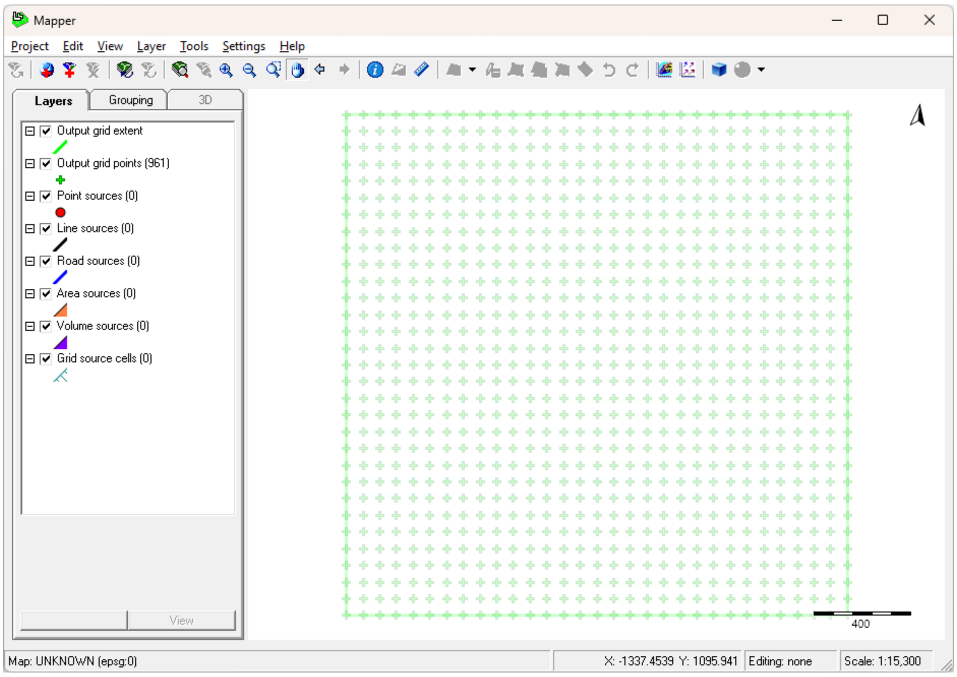


Figure 8.2 – The **Mapper**.

Step 5 Click on the **Add Layer** button, .

Step 6 Browse to the location of the map tile you wish to add and click **Open**.

For the purposes of this example, we will be using SS79R.tif which can be found in the <*install_path*>*Data* directory.

The coordinates are set using the world file (*SS79R.tfw*) which is associated with the *.tif* file. These are used automatically provided the *.tfw* file is in the same directory as the *.tif* file.

Step 7 The map tile should appear in the Mapper window.

8.2.2 Using the Mapper to specify an output grid

Step 1 Select the **Output grid extent** layer, and then the **Add feature** tool, .

Step 2 Click and drag on the map to draw a rectangle to represent the output grid.

Step 3 The values for the **Minimum (m)** and **Maximum (m)** grid extent for **x**, **y** will be filled in on the **Grids** screen of the ADMS-Urban interface.

Step 4 Return to the Mapper by clicking on the Mapper title bar.

Step 5 Click on the **Refresh Layers** button, , to refresh the view. The output grid extent should now be visible on the map.

---

*If the output grid is not visible this may be because it is behind the map added earlier. In the list of layers on the left hand side of the Mapper (**Layers** tab), drag the map to the bottom of the list.*

Details on changing the colour schemes of layers can be found in the Mapper User Guide.

8.2.3 Adding the point sources using the Mapper

Step 1 Select the **Zoom In** tool and zoom in to the area where you want to place a refinery.

Step 2 Select the **Point Sources** layer and then the **Add Feature** tool, , and click on the location where you wish to add the point source

Step 3 Enter the name of the source (Refinery) in the ADMS-Urban industrial source table

Step 4 Enter the same source details as for Example 1. Note that the location of the source has already been entered in National Grid coordinates.

Step 5 Enter the same emissions as in Example 1 (8.5 g/s of SO_2).

Step 6 Return to the Mapper by clicking on the Mapper title bar. Note that if you click on the map area a new point source will be added, as the **Add Point Source** tool is still active.

Step 7 The point source you added should be visible in the map window, if not click on the **Refresh Layers** button to refresh the view.

Step 8 Repeat Steps 2 to 7 for the following sources:

Paint Shop: source height 27.5 m, diameter 2.15 m, volume flow rate

13.56 m^3/s, temperature 45 °C

To make the volume flow rate editable for a particular source, either right-click on the ***Velocity*** *or* ***Volume flux*** *entry in the source table and select* ***Use volume flux****, or set the* ***Vel or Vol*** *entry to* ***Vol.****.*

Factory: source height 33 m, diameter 1 m, exit velocity 22 m/s, temperature 35 °C

8.2.4 Setting up the rest of the run in ADMS-Urban

Step 1 In ADMS-Urban, click on the **Emissions...** button in the **Source** screen to define the emissions for the new sources.

Step 2 Click on the **Pollutants...** button to define a new pollutant.

Step 3 Click on the **New** button and add a new pollutant called Methane.

Step 4 In the upper table, enter a **Conversion factor ug/m³ → ppb** of 1.5.

Step 5 In the lower table, enter a deposition velocity of 0.01 m/s. (Note that the deposition parameters are only used if you are modelling deposition, i.e. if **Dry deposition** or **Wet deposition** are selected on the **Setup** screen.)

Step 6 Click on the **OK** button to return to the **Emissions** screen. Click **New** and choose Methane from the new list available.

Step 7 Enter emission rates of 100 g/s of Methane for the Refinery, 12.5 g/s for the Paint Shop and 20 g/s for the Factory.

You can use the ***< Back*** *and* ***Next >*** *buttons to move between sources.*

Step 8 Click on the **OK** button to return to the **Source** screen.

Step 9 Select the **Groups** option on the **Source** screen.

Step 10 Click on the **New** button, and make a new group called ‘Industrial Estate’.

Step 11 Add the sources Paint Shop and Factory to this group by putting a tick next to them by double-clicking in the **In This Group** column of the **Complete list of sources:** box.

Step 12 Create a second group called ‘Refinery’, and add the Refinery source to this group.

Step 13 Move to the **Output** screen, click on **New** and choose Methane from the pollutants list.

Step 14 Calculate a short-term average with a sampling time of 1 hour. Change the output units to **ppb**. If necessary, put a tick in the **Include** column for Methane.

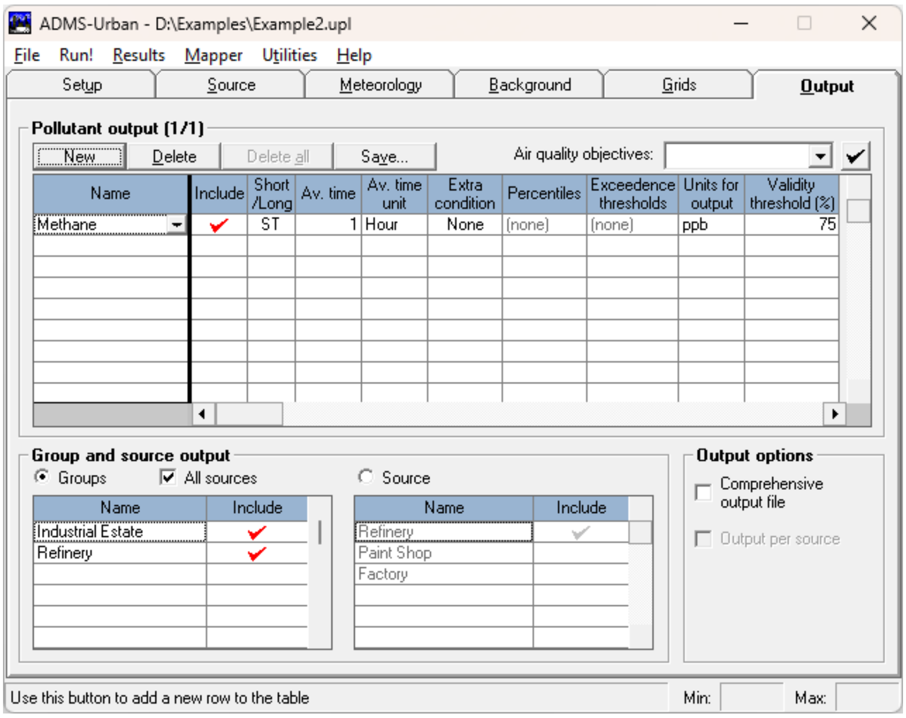


Figure 8.3 – Selecting output.

Step 15 If it is not already selected, select the **Groups** output option, and include both user-defined groups in the calculation by putting a tick in the **Include** column next to each of the sources in the **Groups** table. Also ensure there is a tick next to the **All sources** option.

Step 16 Move to the **Meteorology** screen.

Step 17 Select the **Enter on Screen** option and click on the **Data...** button to bring up the **Meteorological Data** window.

Step 18 Click on the **New** button to add a new set of meteorological data.

Step 19 Enter the following data in the table: wind speed 5 m/s, wind angle 255º, year 2020, Julian day number 137, local time 14 hours, cloud cover 7 oktas. Remove the check from the **boundary layer height, h (m)** box in the **Met. parameters to be entered** list. Click **OK** to return to the **Meteorology** screen.

Step 20 Untick the **Met. data in sectors of (degrees)** check box.

Step 21 From the **File** menu in ADMS-Urban, choose **Save As**.

Step 22 Enter the file name *Example2.upl*.

Step 23 Press **Run!** on the menu bar.

8.2.5 Viewing contour output in the Mapper

Step 1 When the calculation has been completed, return to the Mapper.

Step 2 Click on the **Contour** button, .

Step 3 In the **Contour Plotting** screen, select **Short-term** near the top of the window and use the left sections to browse to the correct directory and select the output file *example2.gst* from the list.

Step 4 In the **Dataset to plot** box, choose Methane concentration from the source group <All sources>.

Step 5 Click on **Advanced Options...** to bring up the **Advanced Contour Options** screen, **Figure 8.4**.

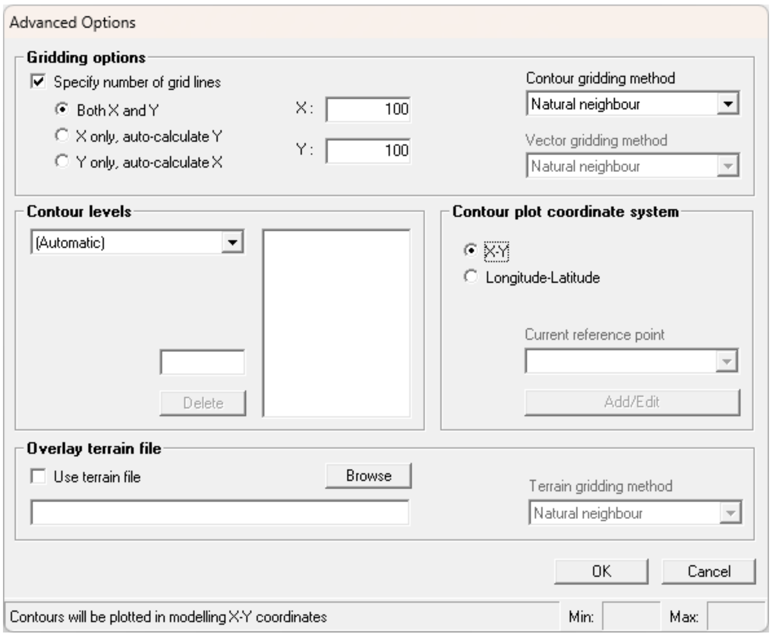


Figure 8.4 – **Advanced Contour Options** screen.

Step 6 Check the **Specify number of grid lines** option and specify 100 for each direction. Click on **OK** to return to the ADMS Contour Plotter.

Step 7 Click on the **Plot** button and **Save Surfer Grid File As...** screen is displayed (**Figure 8.5**).

Step 8 Browse to an appropriate location (it is best to save the file with your other ADMS-Urban input/output files) and click **Save**.

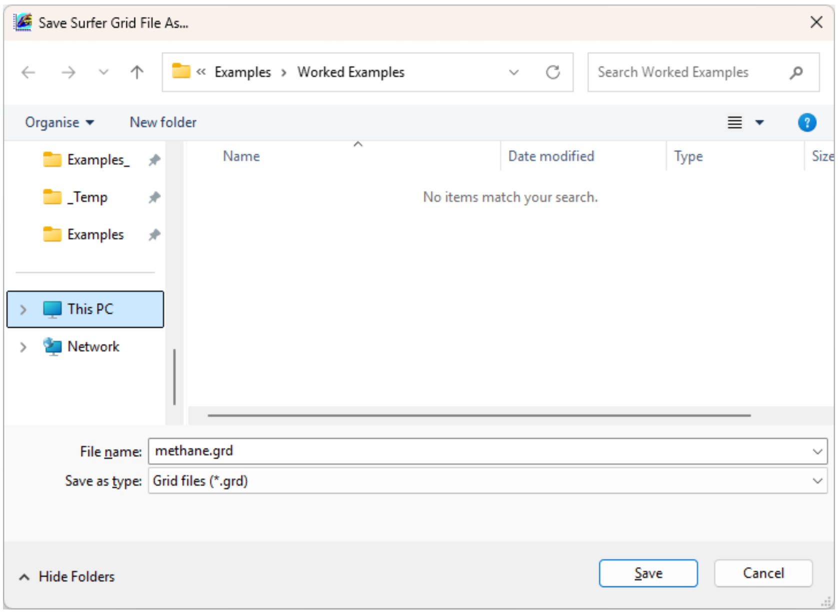


Figure 8.5 – Mapper **Save Surfer Grid File As...** specification screen.

Step 9 The pollutant concentrations contour will now be displayed in the Mapper (again, you may have to drag the base map layer to the bottom of the list to be able to see the contours).

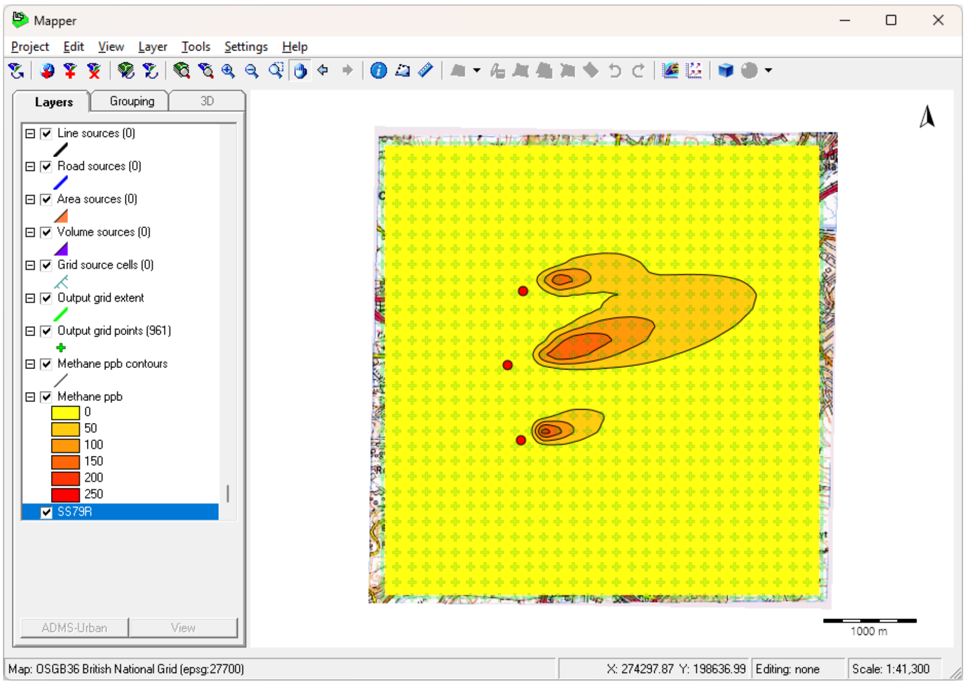


Figure 8.6 – Pollutant concentrations displayed in the Mapper.

Step 10 You can edit the colour scheme of the contour plot, see the Mapper User Guide for more details.

Step 11 Repeat Steps 2 to 10 for the Industrial Estate and Refinery groups.

8.3 Example 3: Area sources

In this example you will enter an area source into ADMS-Urban using the Mapper, export source data to an Emissions Inventory database, and create multiple contour plots showing pollutant concentrations from a ground level source under convective, neutral and stable weather conditions.

Specifically, you will learn:

- how to enter an area source using the Mapper; and
- about the location and magnitude of the maximum concentration for ground level sources under different weather conditions.

8.3.1 Adding an area source using the Mapper

Step 1 Start ADMS-Urban or click on **File**, **New**, if open, to create a new input file.

Step 2 Enter a sensible **Name of site** and **Name of project**.

Step 3 Change the **Coordinate system** to *OSGB 1936 British National Grid (epsg:27700).*

Step 4 Start the Mapper and add the *SS79R.tif* map file as in Example 2.

Step 5 As in Example 2, use the Mapper to specify an output grid, note that the wind direction will be from a bearing of 270°, i.e. from west.

Step 6 Select the Area sources layer and then the **Add feature** tool, , and draw an area source. The area source must have between three and fifty vertices, and be convex. Double-click to finish the area source, then click on the **Save edits** button, .

Step 7 Enter a name for the source in the ADMS-Urban industrial source table. Change the height to 0 m (i.e. ground-level). Change the exit velocity to 0 m/s – this indicates that the release is passive, so 'plume rise' will not be modelled. Enter an emission of particulates (PM_{10}) equal to $1.0x10^{-4}$ $g/m^2/s$.

When modelling line, area or volume sources, emission rates must be specified in g/m/s, $g/m^2/s$ or $g/m^3/s$, respectively.

Step 8 Return to the Mapper by clicking on the title bar and if necessary refresh the view using the **Refresh layers** button to view your area source. As in the previous example, you may have to drag the base map layer to the bottom of the list to be able to see the area source.

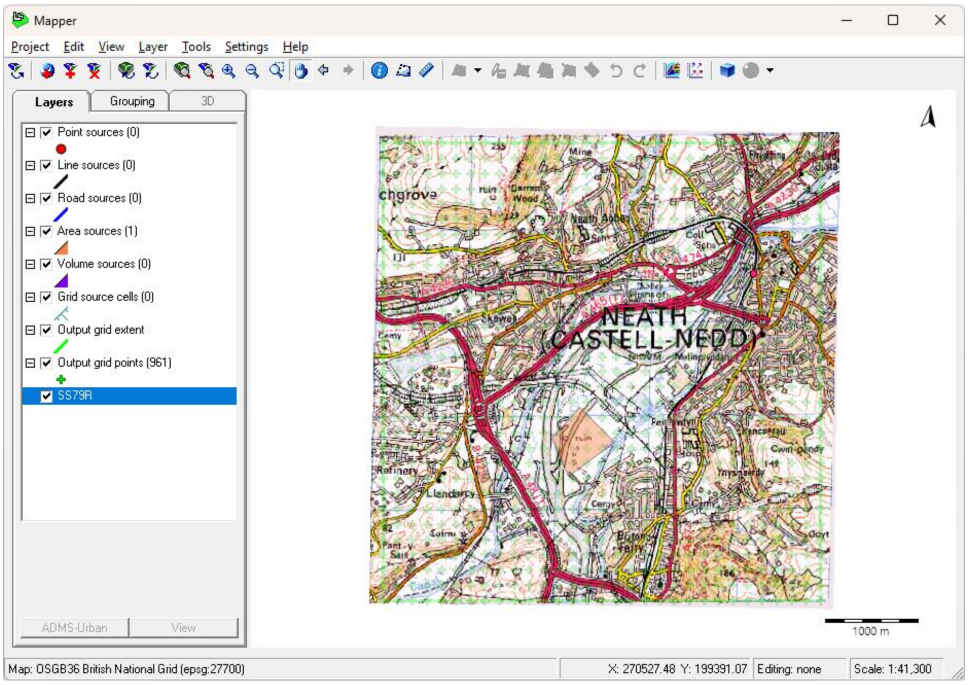


Figure 8.7 – Area source displayed in the Mapper

8.3.2 Meteorological data and output options

Step 1 Move to the **Meteorology** screen in the ADMS-Urban.

Step 2 Choose the **From file** option.

Step 3 **Browse** to find the file *nsc.met*, as in Example 1. Untick the **Met. data in sectors of (degrees)** and **Met. data are hourly sequential** check boxes.

Step 4 In ADMS-Urban, move to the **Output** screen. Click on **New** and choose PM_{10} from the **Pollutants** drop-down list. Calculate a short-term average with a sampling time of 1 hour. If necessary, put a tick in the **Include** column for PM_{10}.

Step 5 From the **File** menu in ADMS-Urban, choose **Save As**.

Step 6 Enter the file name *Example3.upl*.

Step 7 Press **Run!** on the menu bar.

8.3.3 Making a contour map

Step 1 When the calculation has been completed, return to the Mapper.

Step 2 Click on the **Contours** button.

Step 3 Select the output file *Example3.gst*.

Step 4 Select the PM_{10} concentration for the source group ‘All sources’.

Step 5 Select data set 1 in the **Time** box – this corresponds to unstable conditions (stability category B).

Step 6 Click on the **Plot** button.

Step 7 Browse to an appropriate directory, enter a file name for the grid file and click on the **Save** button.

Step 8 Change the name of the filled contour layer to indicate that it represents unstable conditions by double clicking on the layer to bring up the **Raster** box and altering the **Caption** on the **General** section. Also change the **Transparency** to 70 %.

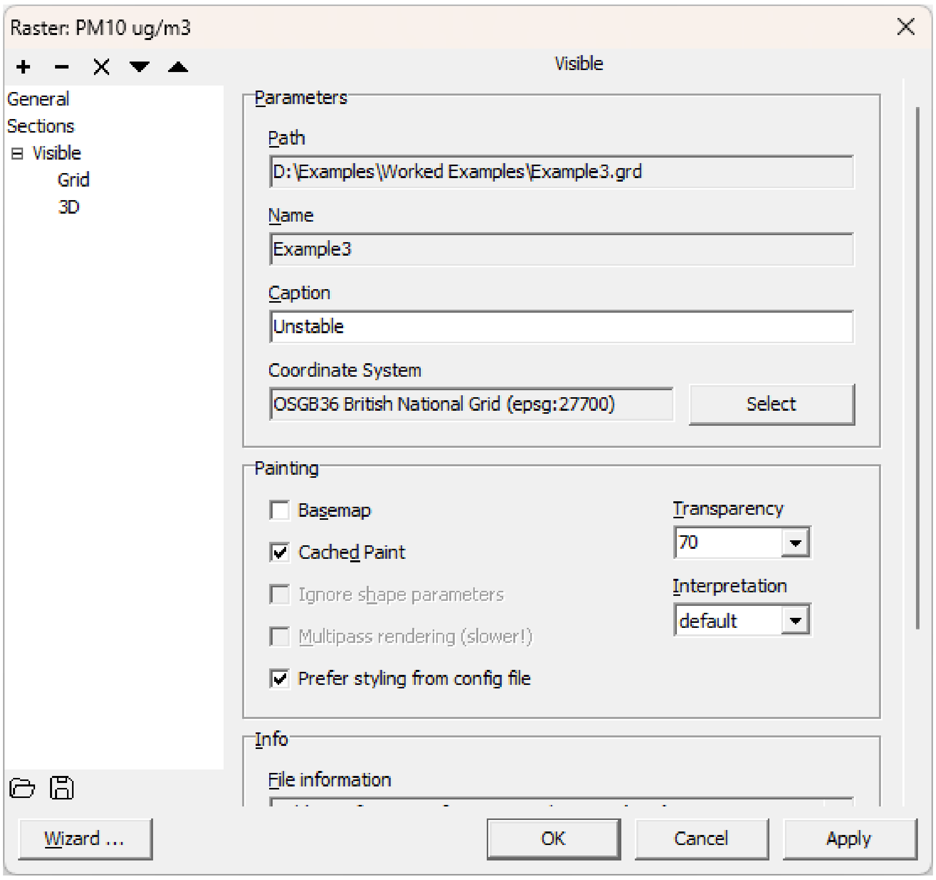


Figure 8.8 – Mapper **Raster** screen.

Step 9 Repeat Steps 2 to 8 for data sets 2 and 3 (neutral and stable conditions).

The resulting plots are shown below. Each plot has been set to use the same colour scheme; for details on how to do this, see the *Mapper User Guide*.

Worst case weather conditions for ground-level source

We can see that the worst case occurs under stable conditions. The plume is at ground level and disperses slowly. Under neutral and convective conditions, the pollutant disperses more rapidly resulting in lower concentrations.

Unstable conditions

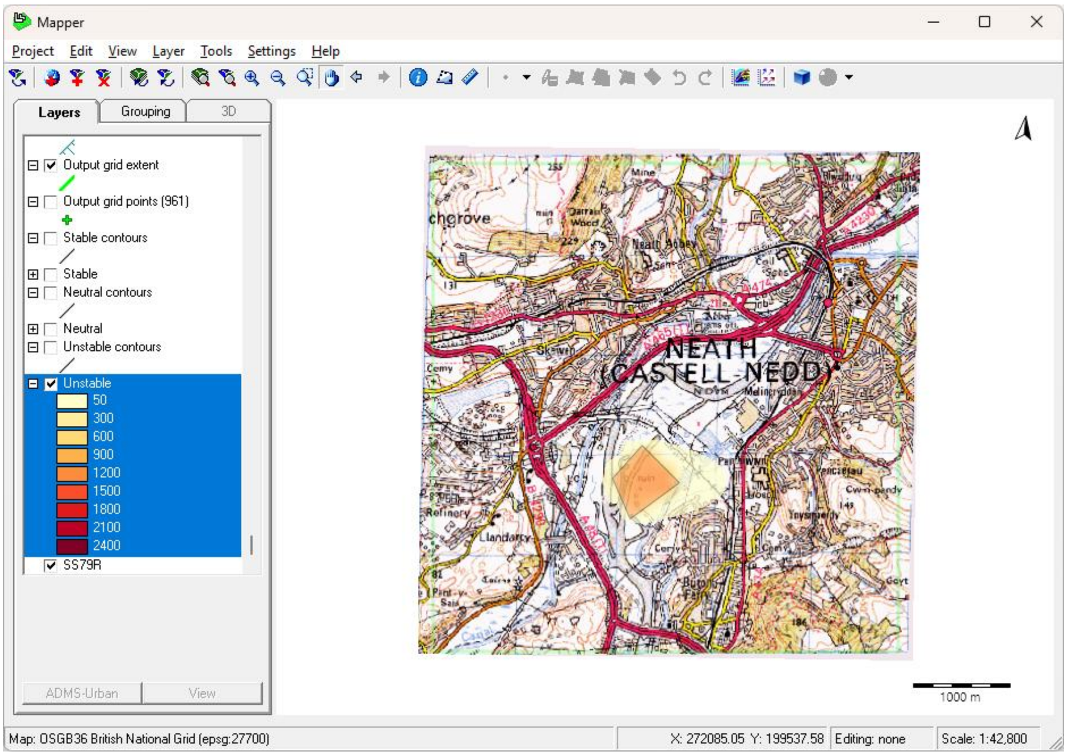


Figure 8.9 – Concentration due to area source in unstable conditions.

Neutral conditions

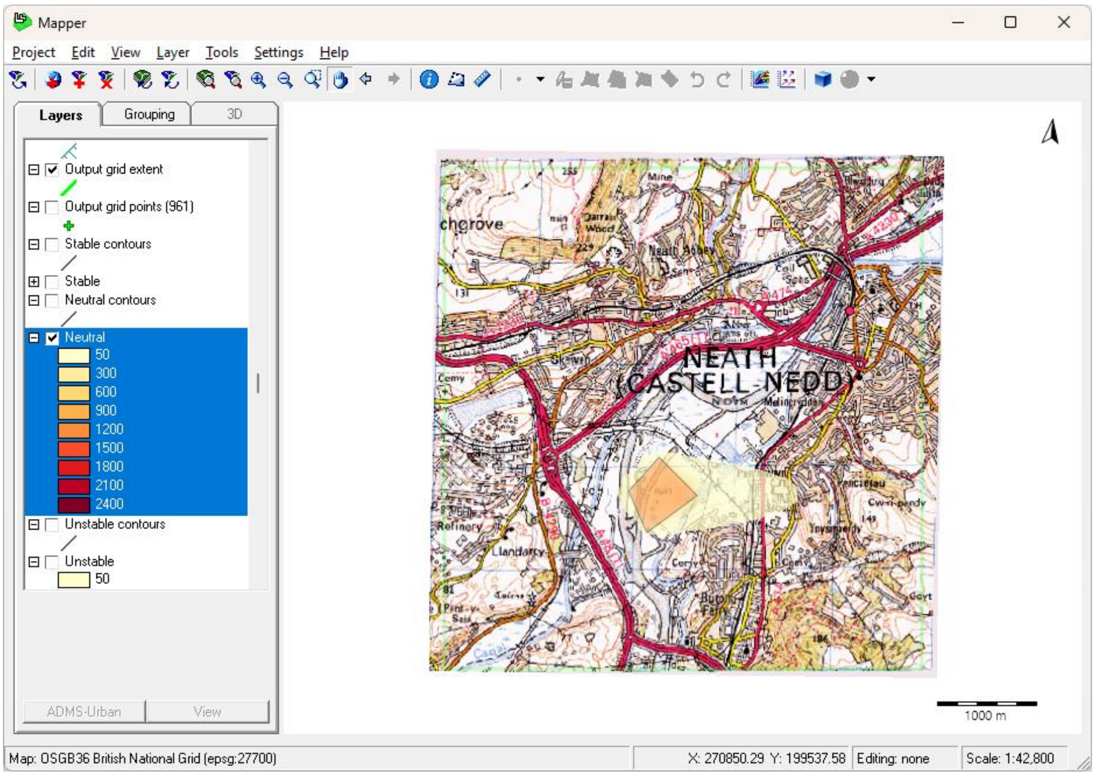


Figure 8.10 – Concentration due to area source in neutral conditions.

Stable conditions

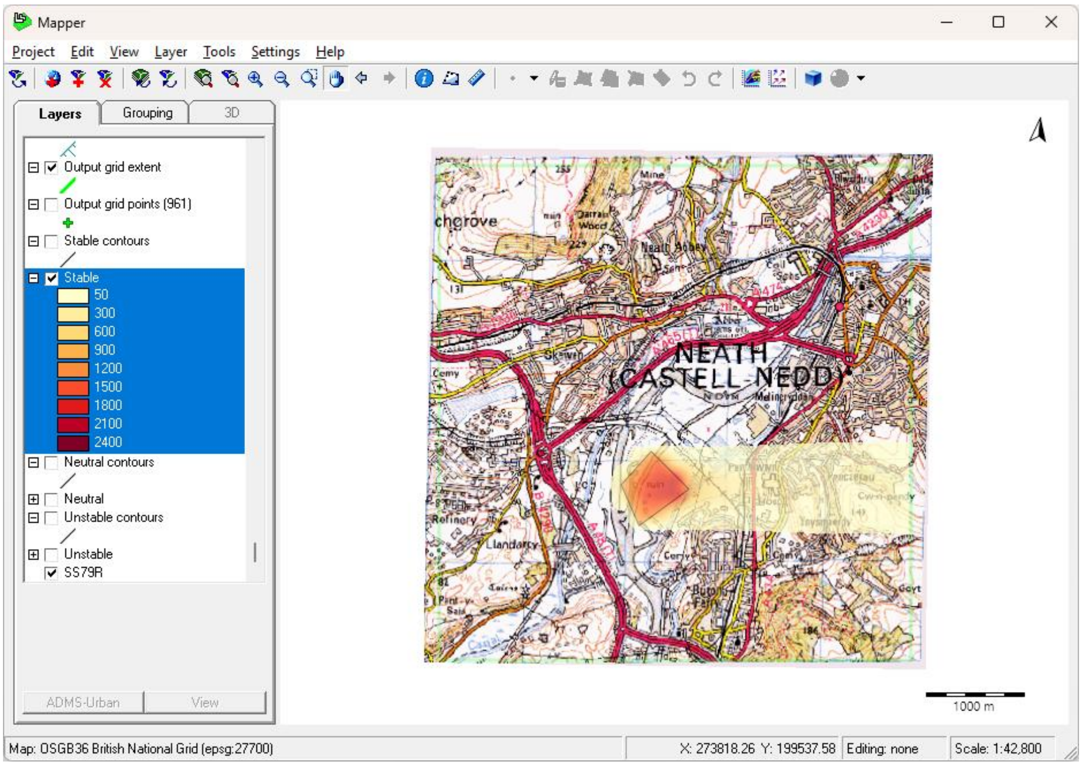


Figure 8.11 – Concentration due to area source in stable conditions.

8.4 Example 4: Road sources

In this example you will model a road source in ADMS-Urban, using traffic data to calculate the emissions, and plot a time series graph of concentrations in Microsoft Excel.

Specifically, you will learn:

- how to enter a road source using the Mapper;
- how to use traffic data to calculate emissions from a road;
- how to model hourly variation of emissions;
- how to add a receptor point using the Mapper; and
- how to plot a time series graph in Microsoft Excel.

8.4.1 Adding a road source using the Mapper

Step 1 Start ADMS-Urban or click on **File**, **New**, if open, to create a new input file.

Step 2 Enter a sensible **Name of site** and **Name of project a**nd change the **Coordinate system** to *OSGB 1936 British National Grid (epsg:27700)*.

Step 3 Move to the **Grids** screen and select **Specified points**.

Step 4 Start the Mapper and load the *SS79R.tif* base map as in previous examples.

Step 5 Click on the **Road Sources** layer and then on the **Add feature** button, , and draw a road source in the view. Double click (or press **Enter**) to end the road, then click on the **Save edits** button, .

Step 6 Enter the name of the road in the road source table:

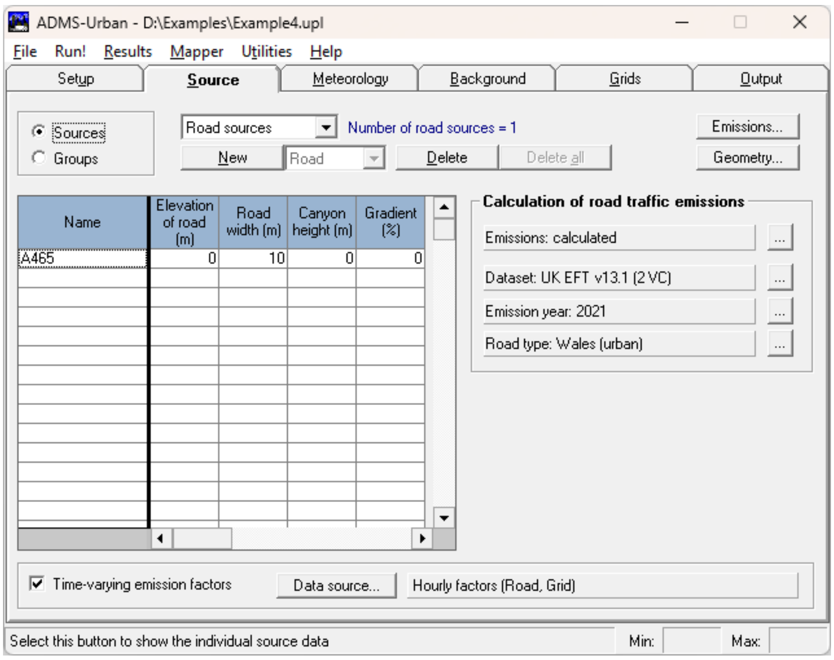


Figure 8.12 – ADMS-Urban road source table.

Step 7 Enter the **Elevation of road (m)** above ground level, the **Road width (m)**, the **Canyon height (m)** (the average height of the buildings along the road) and the **Gradient (%)** of the road into the road source table as shown in **Figure 8.12**. ADMS-Urban will decide whether to run the street canyon model option using the canyon height information (refer to Section 3.2.1).

The road width can be measured from the map using the Mapper measuring tool.

Step 8 In the **Calculation of road traffic emissions** section, ensure that **Emissions** is set to *calculated* and that the **Dataset** being used is *EFT v13.1 (2 VC)*. Set the **Emission year** for all sources to be *2021* via the '...' button next to that field. Select a **Road type** via the '...' button next to that field; here we have selected *Wales (urban)*.

Step 9 To see the road source you have defined in the Mapper, return to the Mapper and click on the **Refresh layers** button. As in the previous example, you may have to drag the base map layer to the bottom of the list to be able to see the road source.

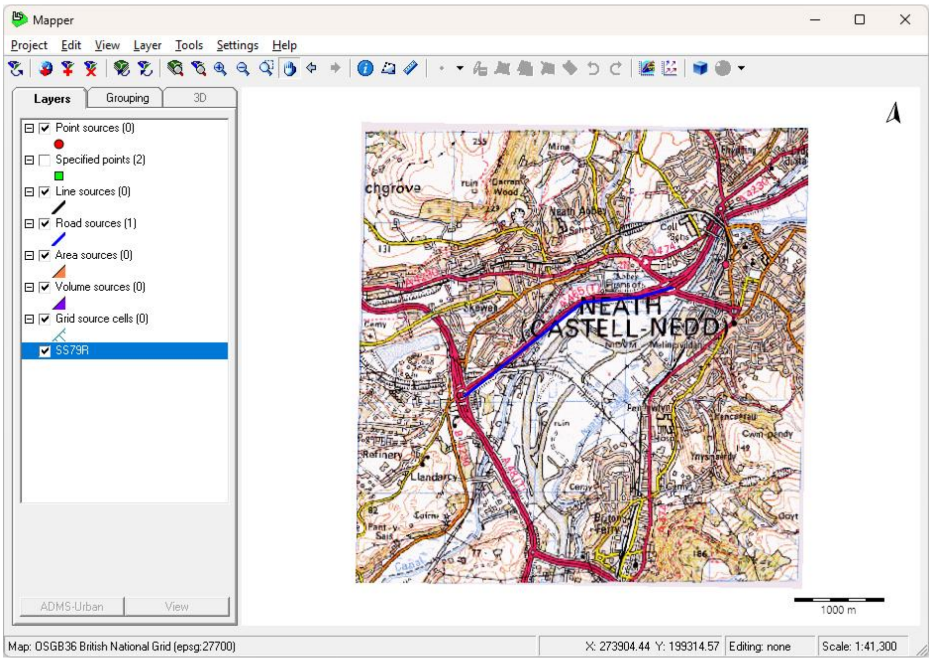


Figure 8.13 – Road source displayed in Mapper.

Step 10 Return to the ADMS-Urban interface. Check the locations of the vertices of the road by clicking on the **Geometry...** button.

Step 11 Click on the **OK** button to leave the **Road geometry** screen.

8.4.2 Calculating traffic emissions data and entering hourly traffic variation

When you are modelling the diurnal variation in traffic flows, you must enter information on the average emission rate in the **Emissions** screen – either by entering an average emission rate directly or, as in this example, by using an emission factors dataset such as EFT v13.1 (2 VC) to calculate emissions from the average traffic flow.

Step 1 Highlight the new road source and click on the **Emissions...** button to display the **Emissions** screen.

Step 2 Check that the **Calculate emissions using traffic flows** option is selected in the **Emissions** box and that the year of the scenario is set to 2021. This tells ADMS-Urban to calculate emissions from traffic flow data using the chosen dataset of emission factors, for 2021.

Step 3 Click on the **New** button in the **Traffic flows** box to add a new row of traffic data to the traffic table.

Step 4 Select the **Vehicle category** from the list (**Light duty vehicle** or **Heavy duty vehicle**), the **Average speed**, the number of **Vehicles per hour** and (for heavy duty vehicle categories) the **Uphill %** of vehicles. The speed does not have to be the same for both vehicle categories. Enter data for 700 light duty vehicles travelling at 60 km/hr, and 100 heavy duty vehicles travelling at 50 km/hr with 50% travelling uphill. These figures correspond to average traffic flows. Notice that the emission rates in the upper table are automatically calculated.

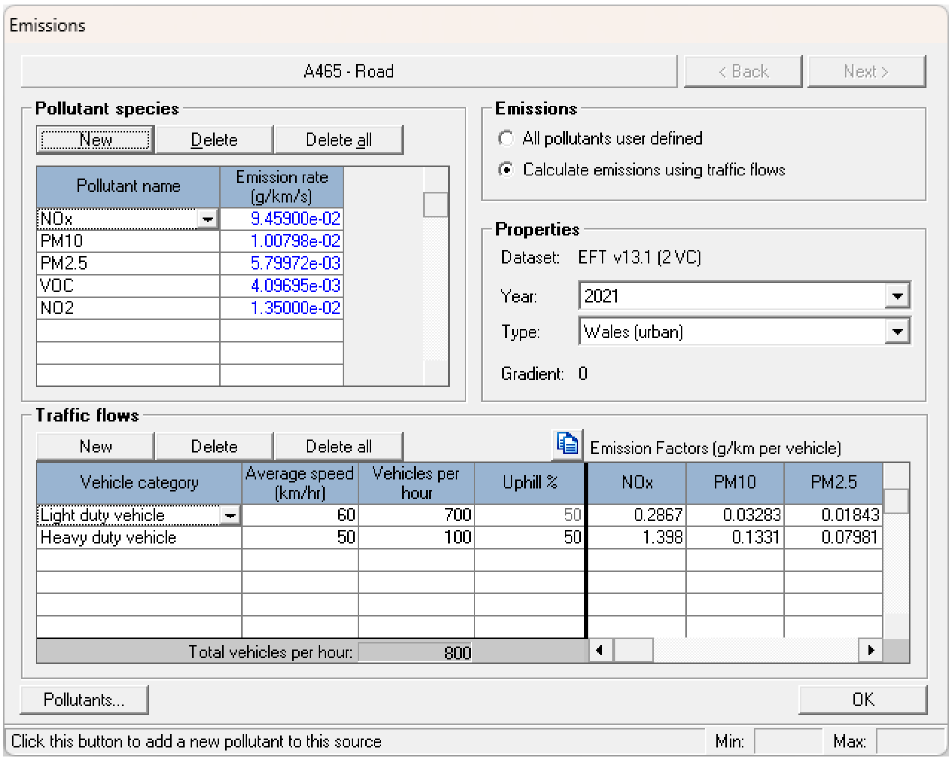


Figure 8.14 – **Emissions** screen for road source with emissions calculated from traffic flows.

Step 5 If you want to model a pollutant not included in the emission factors dataset, click on the **New** button in the **Pollutant species** box, choose a pollutant from the list and enter an emission rate manually.

Step 6 Click on the **OK** button to return to the **Source** screen.

Step 7 Place a check in the **Time varying emission factors** check box on the **Source** screen.

Step 8 Click on the **Data source...** button, and the **Time varying emission factors** sub-screen is displayed.

Step 9 Enter data corresponding to the variation in traffic from the hourly average flow shown below in **Figure 8.15**. For convenience, these factors can be imported into the table; click on the button and select the file *3DayDiurnal.csv*, which can be found in the *<install_path>\Data* directory. ADMS-Urban will multiply the emission rates shown in the **Pollutant Species** box of the **Emissions** sub-screen by the factor entered for each hour.

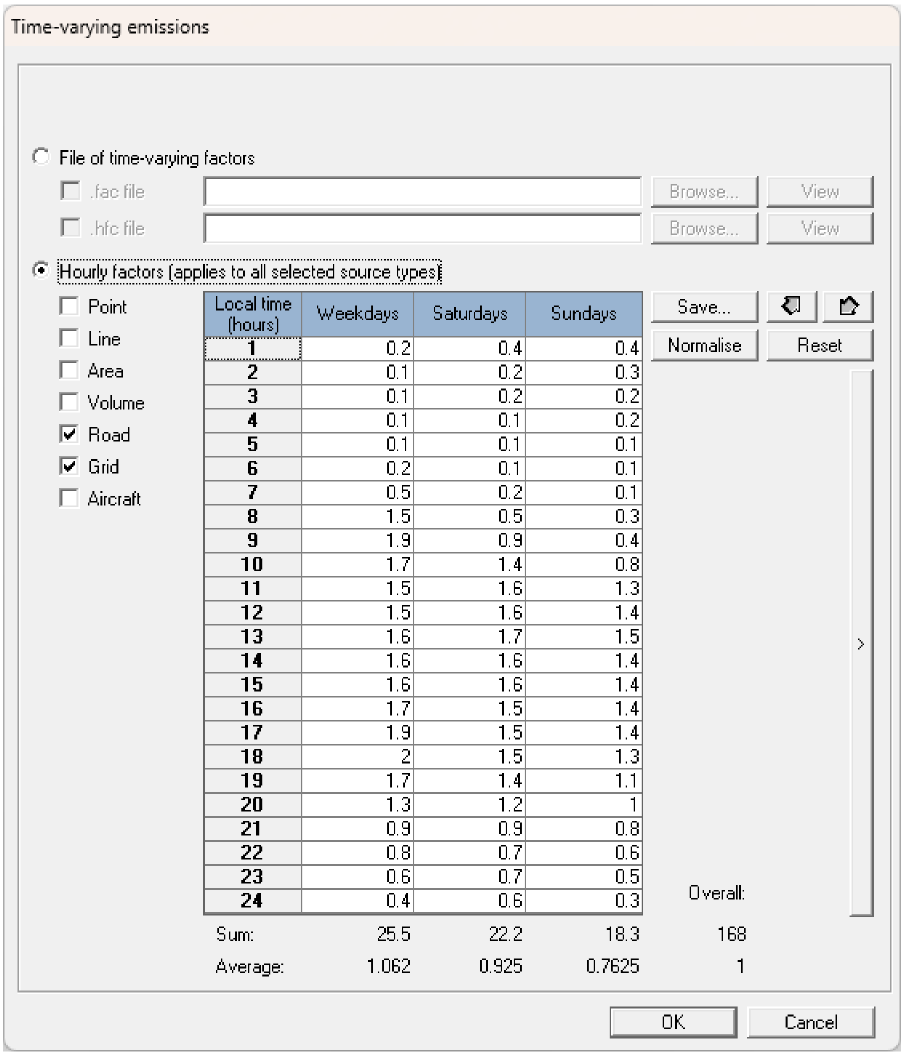


Figure 8.15 – **Time-varying emission factors** screen.

---

*ADMS-Urban automatically assigns the correct set of factors to a calculation using the data in the associated meteorological data file. If the day and hour are not included in the meteorological data, then no hourly variation is assumed and the traffic emissions for each hour are as given in the **Emissions** screen.*

Step 10 Click on the long thin **>** button to the right of the screen to view a time-series plot of the current set of emission factors, in which each profile (weekdays, Saturdays, Sundays) is plotted using a different line and/or symbol colour. Click **OK** to return to the **Source** screen.

8.4.3 Adding a new receptor point to ADMS-Urban using the Mapper

Step 1 Back in the Mapper, select the **Specified Points** layer and use the **Add Feature** tool, , to draw a new receptor, and the **Grids** screen is displayed:

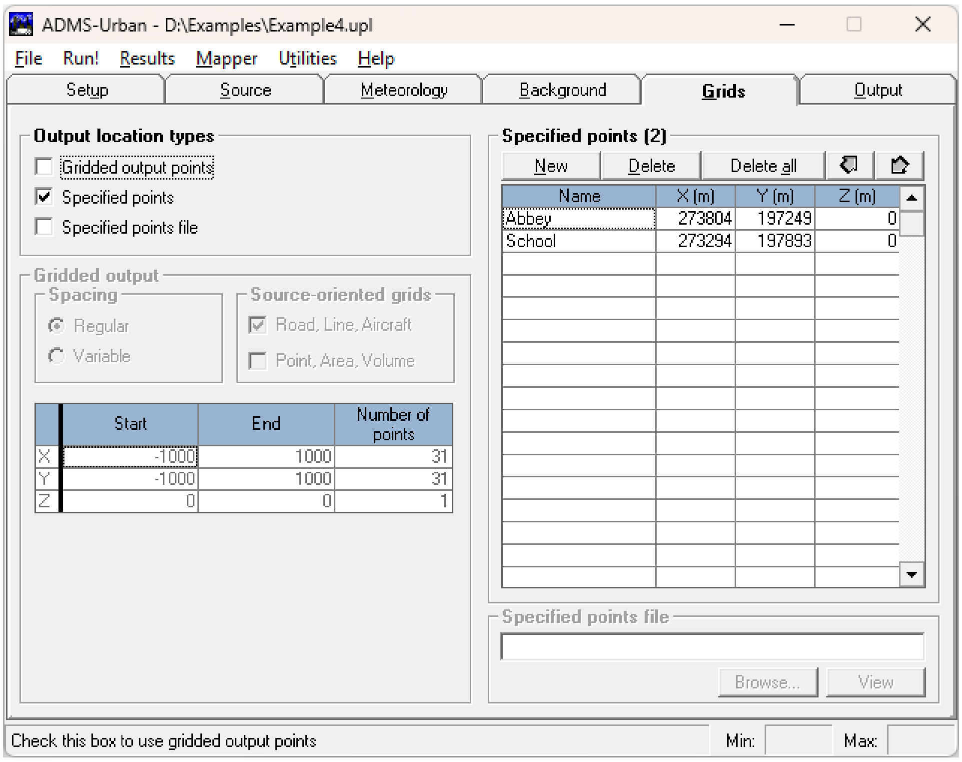


Figure 8.16 – **Grids screen** with two receptors defined.

Step 2 Enter the name of the receptor and the height of the receptor above ground level in the **Specified points** table.

Step 3 Repeat Steps 1 and 2 to add another receptor.

Locating receptors in roads

If output is required within a road, you should ensure that the receptor is actually positioned inside the road defined in ADMS-Urban as the calculated concentration can be quite different inside and just outside a road.

Step 4 Move to the **Meteorology** screen.

Step 5 Select the **From file** option.

Step 6 Select the file *oneday.met* from the *<install_path>\Data* directory. View the file in a text editor – it contains hourly sequential data for one day.

Step 7 Ensure the **Met. data in sectors of (degrees)** and **Met. data are hourly sequential** check boxes are both ticked.

Step 8 Move to the **Background** screen.

Step 9 Select the **From file** option.

Step 10 Select the file *backgrnd.bgd* from the *<install_path>\Data* directory.

View the file in a text editor – it contains one day of background concentrations, for the same day as the meteorology file *oneday.met*.

Step 11 Move to the **Output** screen.

Step 12 Click on **New** three times to add NO_X, VOC and PM_{10} to the **Pollutant output** list.

Step 13 To obtain hour-by-hour concentrations at the receptor locations, calculate a short-term one hour average for each pollutant.

Step 14 Save the model set-up as *example4.upl* and run the model.

8.4.4 Creating a time series graph in Microsoft Excel

Step 1 Start Microsoft Excel.

Step 2 Select **File > Open** from the ribbon, and browse through the directories to locate *example4.pst*, the ADMS-Urban output file containing calculated pollutant concentrations at each of the receptor locations entered in the ADMS-Urban **Grids** screen. Make sure that **All files (*.*)** is selected in the lower right drop-down menu. You can also type '*.pst' in the **File name** box and press Enter to see just those files that end with the extension *.pst* in the current directory.

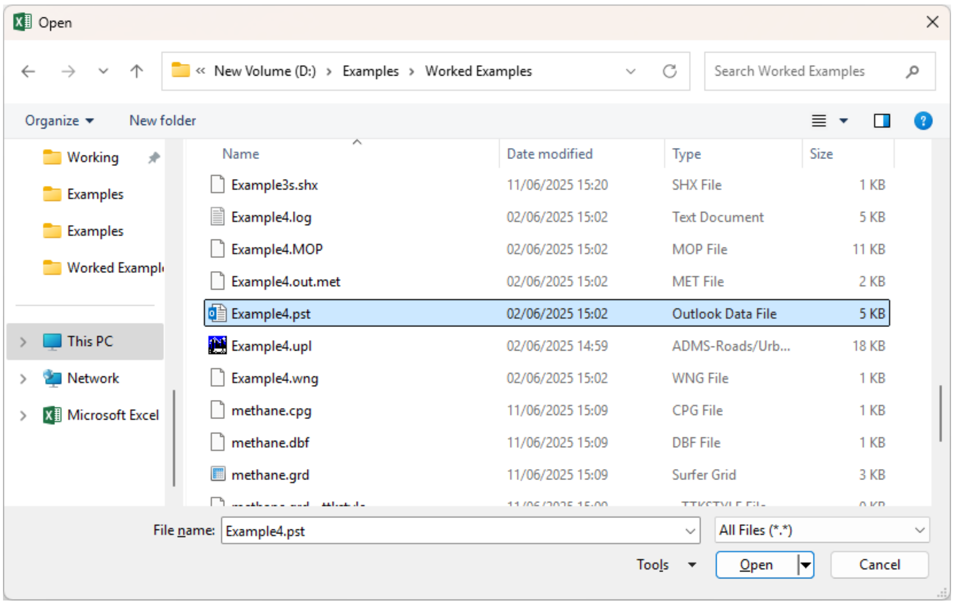


Figure 8.17 – Opening a *.pst* file in Microsoft Excel.

Step 3 Click on the file and then click **Open**. The **Text Import Wizard** is launched.

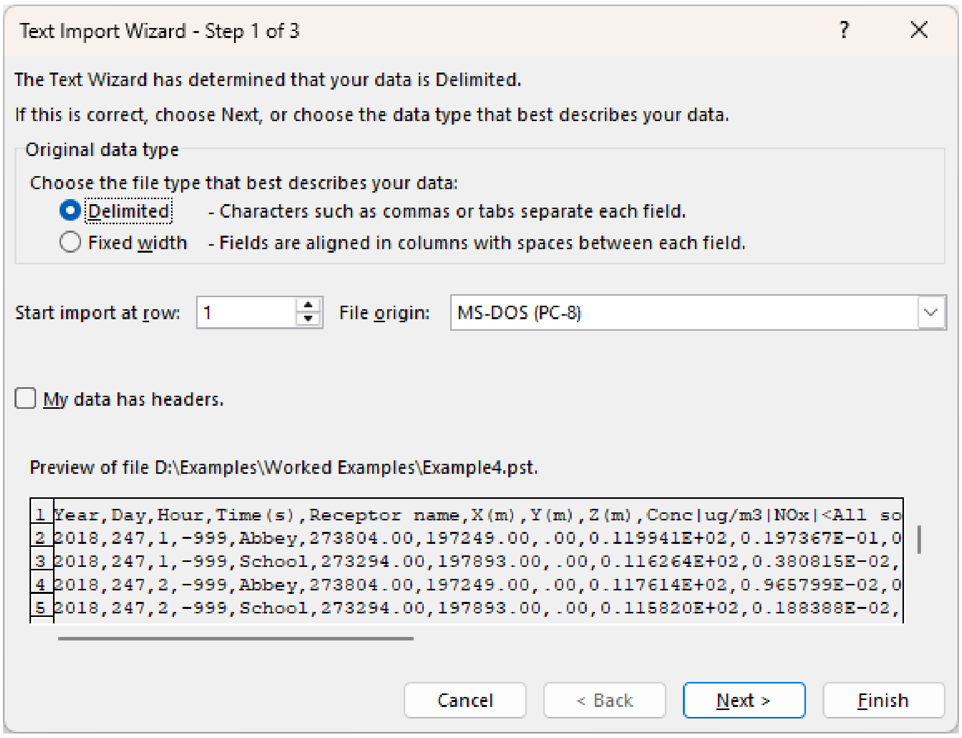


Figure 8.18 – Microsoft Excel **Text Import Wizard – Step 1 of 3**.

Step 4 In the **Original data type** box, choose the **Delimited** option, and click on the **Next >** button to move to the next step of the Wizard.

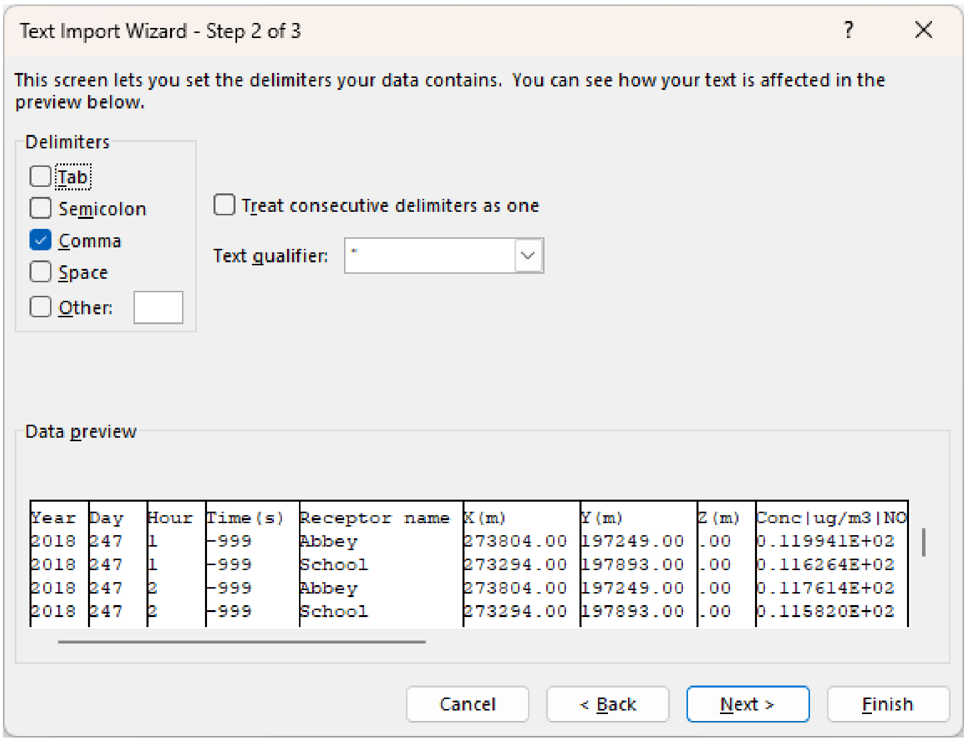


Figure 8.19 – Microsoft Excel **Text Import Wizard – Step 2 of 3**.

Step 5 In the **Delimiters** box, select only the **Comma** check box, and click on the **Next >** button to move to the next step of the Wizard.

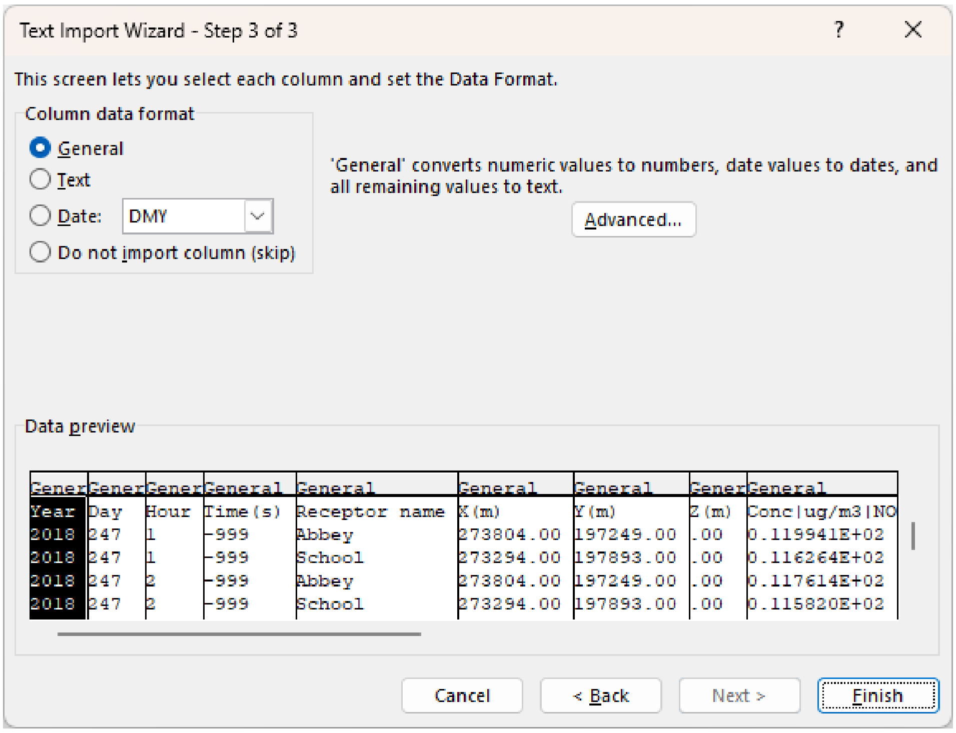


Figure 8.20 – Microsoft Excel **Text Import Wizard – Step 3 of 3**.

Step 6 Ensure that the file has been properly separated in the **Data preview** box, and click on the **Finish** button to import the data.

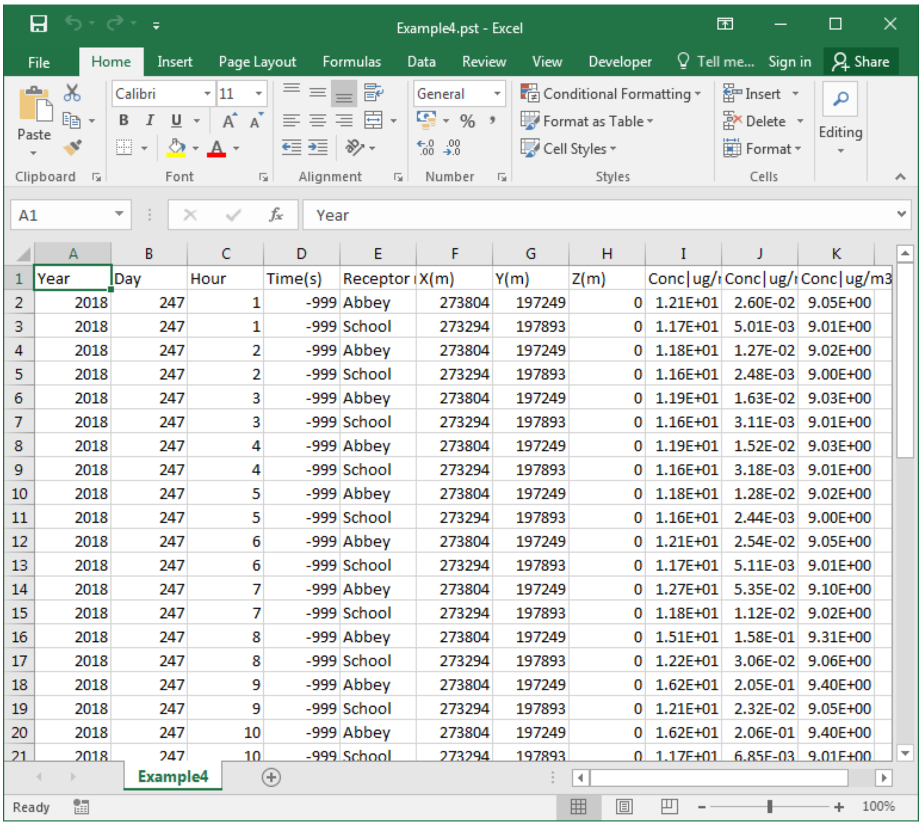


Figure 8.21 – *.pst* file displayed in Microsoft Excel.

Step 7 To filter the data by receptor name, highlight the appropriate column (in this case, Column E).

Step 8 From the **Sort & Filter** section of the **Data** tab, choose **Filter** to filter the data and select the appropriate receptor name from the list.

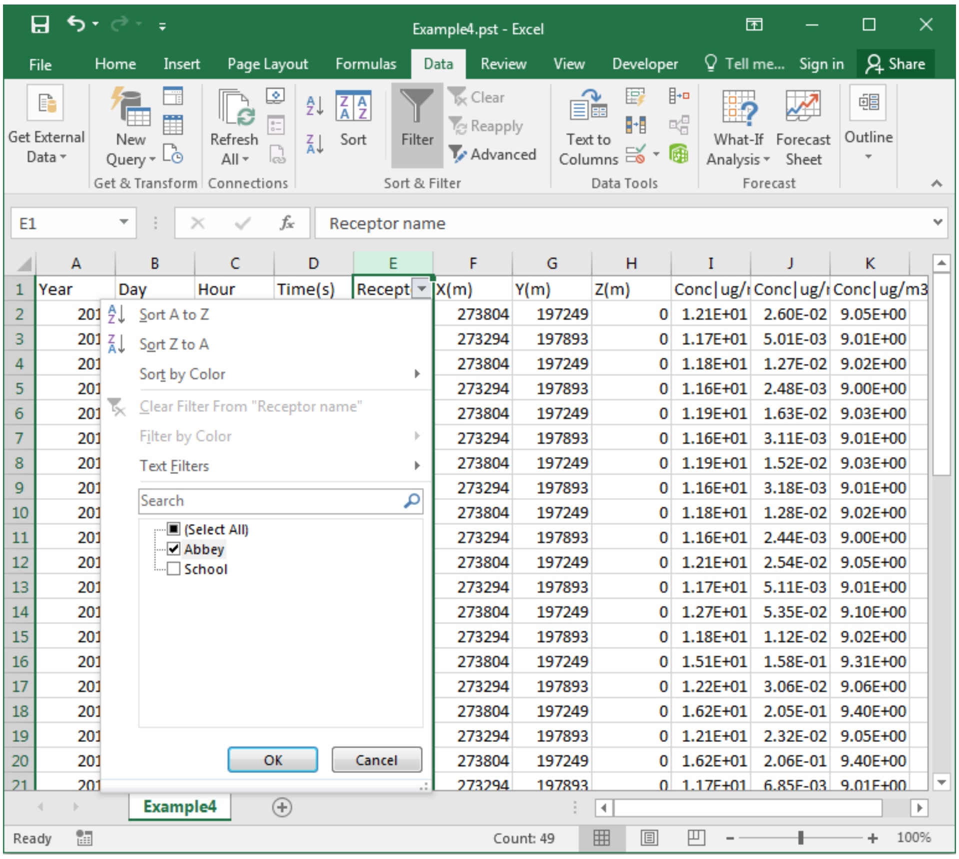


Figure 8.22 – Filtering the data.

Step 9 Highlight the hour and concentration column that you wish to plot.

*Hold the **Ctrl** key down to select non-adjacent columns.*

Step 10 To create a time series chart of concentration, on the **Charts** section of the **Insert** tab, choose a **Scatter** chart with lines. Click on the chart created and in the **Location** section of the **Chart Tools Design** tab select **Move Chart**, select the **New sheet** option and click **OK**.

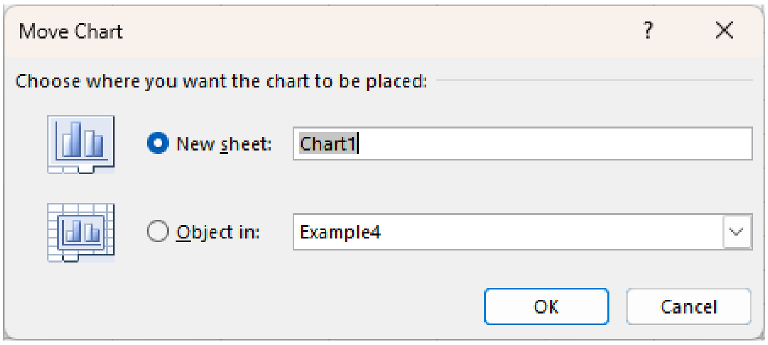


Figure 8.23 – Microsoft Excel **Move Chart** screen.

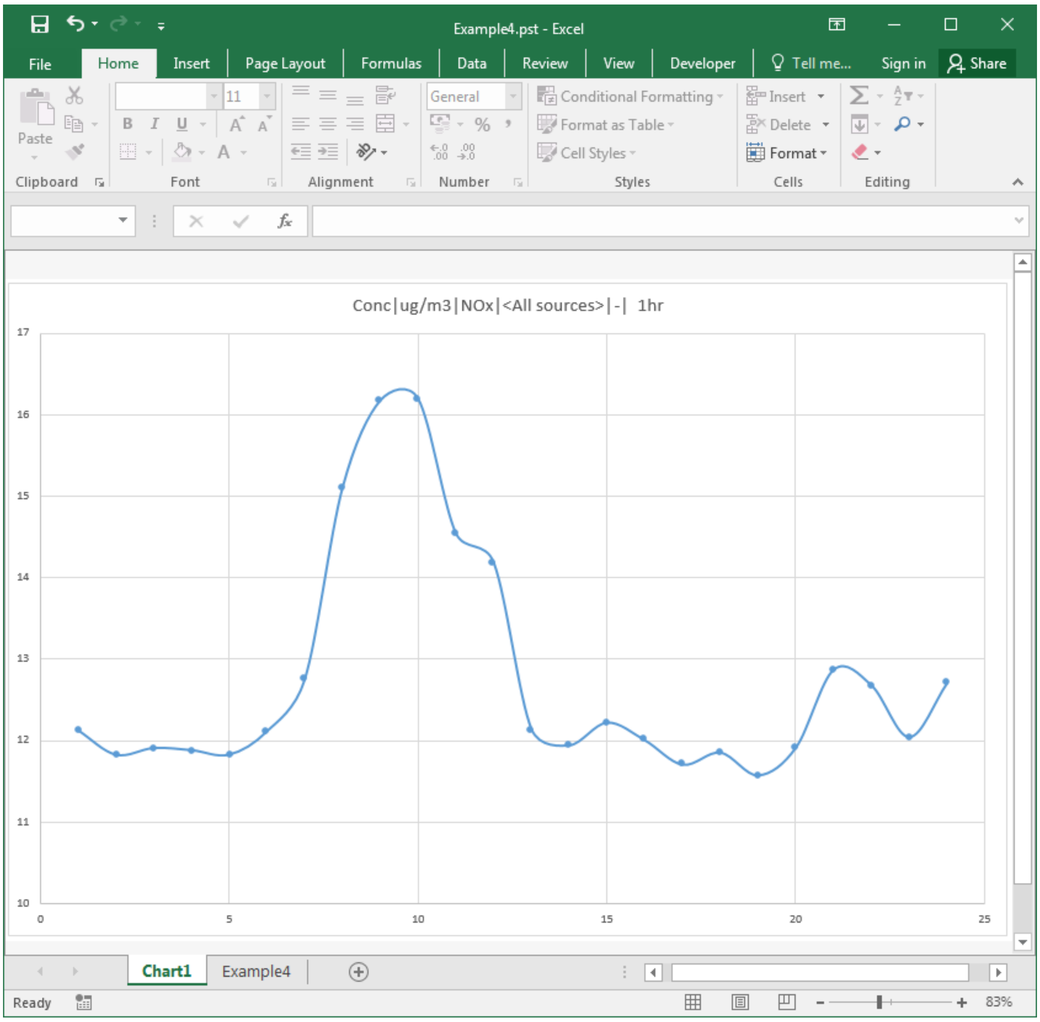


Figure 8.24 – Time series graph.

8.5 Example 5: Importing source data from SPT files

In this example you will use SPT files to import sources into ADMS-Urban. You will then use ADMS-Urban to calculate emissions using the EFT v13.1 (2 VC) emission factor dataset and export them back into SPT files to compare different years.

Specifically, you will learn:

- how to import source data into ADMS-Urban from SPT files.
- how to use ADMS-Urban to re-calculate emissions for each of the roads; and
- how to export the re-calculation emissions back to the Emissions Inventory.

More information about SPT files is given in Section 5.2.

8.5.1 SPT files overview

SPT files are comma-separated variable files that can be used to export and import sources from ADMS-Urban. There are 5 file types, which contain different types of information about each source.

.spt files contain details about the source specific properties, such as dimensions, efflux parameters, traffic flows. For road sources we are primarily concerned with road width, road elevation, and canyon height and traffic flow types.

.vgt files contain the vertices of each source. *.tft* files contain the vehicle types, counts and average speeds for the traffic flow for each source. The road types and traffic flow year are detailed in the *.spt* file.

The other two import file types are *.eit* and *.gpt* files, these aren't included for this example and won't be needed for importing. *.eit* files contain the pollutants and emission rates for each source. *.gpt* files contain the source groups.

8.5.2 Importing road source data and emissions into ADMS-Urban

Step 1 Start ADMS-Urban or click on **File**, **New**, if open, to create a new input file.

Step 2 Enter a sensible **Name of site** and **Name of project a**nd change the **Coordinate system** to *OSGB 1936 British National Grid (epsg:27700)*.

Step 3 Select **Import** from the **File** menu.

Step 4 Click **Browse** and select *example5.spt* in the *<install_path>\Data\Example5* directory. Click **Open**. Leave the checkboxes for *.vgt* and *.tft* files ticked and click **Next >**.

Step 5 Leave the checkbox ticked to import the road sources and click **Next >**.

Step 6 On the **Select sources** screen, click **Next >**. Check the report and click **Import** then **OK**.

Step 7 The **Source** screen now contains the data imported from the SPT files.

Click on the **Emissions...** button. The **Emissions** sub-screen now contains emission rates for 2021 calculated using the EFT v13.1 (2 VC) dataset.

Step 8 Use the **Next >** and **< Back** buttons to view the emissions for each road in turn. Click on **OK** to return to the **Source** screen. Select appropriate data in the **Meteorology**, **Background**, **Grids** and **Output** screens and save the file as *example5.upl*.

8.5.3 Exporting 2021 emissions to SPT files

The emission rates calculated by ADMS-Urban can now be exported to SPT files.

Step 1 In ADMS-Urban, select **Export** from the **File** menu.

Step 2 Make sure the checkbox for road source types is ticked and click **Ok**.

Step 3 Save the new *.spt* file in your working directory and name it, for example, *2021.spt*.

8.5.4 Changing emission rates to 2022 levels

You will now use ADMS-Urban to recalculate the road source emissions using emission factors appropriate for 2022.

Step 1 Go to the **Source** screen in the ADMS-Urban interface.

Step 2 In the **Calculation of road traffic emissions** box, click the '...' button for the **Emission year**. Click on 2022 and the click **OK** to recalculate the road source emissions.

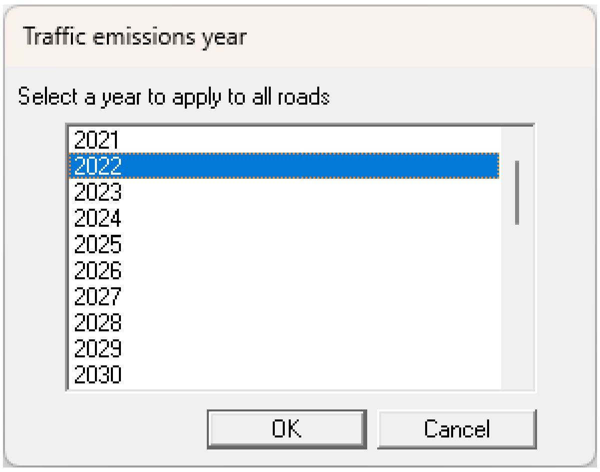


Figure 8.25 – **Traffic emissions year** screen

Step 3 The following message will then be displayed.

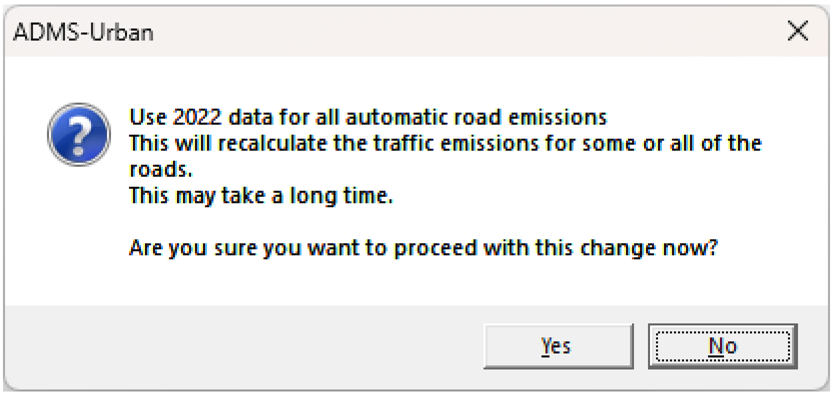


Figure 8.26 – **ADMS-Urban** recalculation warning message

Click **Yes** to continue with the calculation. The calculation will take a few seconds.

Step 4 Select one of the road sources and click on the **Emissions…** button. The **Emissions** screen now contains the emission rates for 2022. Click on **OK** to return to the **Source** screen.

Step 5 Save the file with a different filename, for example *ex5_2022.upl*.

8.5.5 Exporting back to SPT files and comparing emissions for 2021 and 2022

Step 1 In ADMS-Urban, select **Export** from the **File** menu.

Step 2 Make sure the checkbox for road source types is ticked and click **Ok**.

Step 3 Save the new *.spt* file in your working directory and name it, for example, *2022.spt*.

Step 4 Open the two *.eit* emissions files in Excel and compare the emission rate data. See Section 6.6 for more information on how to open comma-delimited files in Excel.

Step 5 Run the two ADMS-Urban model scenarios and compare the results.

8.6 Example 6: Combination of different source types

In this example you will use the sources in the SPT files to create a baseline modelling scenario for an example city. You will add a proposed new source and model the situation with and without the new source.

Specifically, you will learn

- how to create different modelling scenarios from base data; and
- about combinations of source types within the same modelling scenario.

8.6.1 Setting up the base scenario

Step 1 Start ADMS-Urban and import the network of roads created using the EFT v13.1 (2 VC) dataset from the *2021.spt* file created in Example 5.

Step 2 Start the Mapper and display the road sources using the **Refresh layers** button.

Step 3 Select a sequential meteorological data file and appropriate options in the **Meteorology** screen of ADMS-Urban.

Step 4 Select appropriate background concentrations in the **Background** screen.

Step 5 Define an output grid contour region in the Mapper and ensure the **Road, Line** option in the **Source-oriented grids** box in ADMS-Urban is checked.

Step 6 In the **Output** screen, define appropriate averaging times and percentiles for NO_X and PM_{10} (for instance, using current AQS limits, see Appendix D).

Step 7 Save the base model file as *example6a.upl* and run the model.

8.6.2 Creating a modified modelling scenario

Step 1 Use the Mapper to add a new point source near the road network to the base model file. Use the following source details: Car Respray – source height 15 m, diameter 0.5 m, exit velocity 15 m/s, temperature 50°C and NO_X emission rate of 5 g/s.

Step 2 Save and run the model with a different file name, *example6b.upl*.

Step 3 Compare the results.

8.6.3 Case studies

For further practice, you could try to solve the following problems using ADMS-Urban and the Mapper, using the source database created in Example 5 as a starting point.

1. A large power station 10 km to the south west may be having a significant effect on SO_2 levels within the city.

2. Residents have complained about the local waste incinerator. The stack is 48 m high and 2.78 m in diameter. The exit velocity is 3 m/s. Unfortunately accurate emission rates are not known. Increasing the stack height to 60 m and the exit velocity to 9 m/s has been suggested as a possible solution to the resident complaints.

3. Concerns at the levels of PM_{10} recorded at the air quality monitoring station in the pedestrian shopping precinct have led to a lengthy period of consultation between the Environmental Health Department, the Transport Planners and Busybus Bus Company. It is proposed that the Busybus vehicle fleet be retro-fitted with particulate traps. It is anticipated that this will reduce particulate emissions from the bus fleet by up to 75%. As part of the scheme, Busybus will be allowed to increase its service to the bus station from 100 to 120 vehicles/hour. You have been asked to compare the current level of particulate concentrations to those predicted when the scheme is fully operational. Hint: you will need to apply manually the 75% reduction in emissions to the figures calculated by ADMS-Urban. You could also use a set of emissions factors which allow traffic counts for buses to be specified separately, e.g. the EFT v13.1 (6 VC).

4. Detailed information about the building geometry within in the road network has been acquired, allowing you to create the input data file required by the advanced street canyon module. A copy of this file, *Adv_Canyon.csv*, is supplied in the *<install_path>\Data* directory. You have been asked to compare the effects of using the advanced street canyon module (refer to Section 4.2) as opposed to the basic street canyon module (refer to Section 3.2.1). Hint: When using the basic street canyon module, the road widths entered in the **Source** screen should refer to the building-to-building distances. Conversely, when using the advanced street canyon module, the road widths should refer to the actual road carriageway widths (i.e. kerb-to-kerb distances). It is therefore necessary to modify the road widths in the interface; assume that the road width is half the building-to-building distance for all roads within the network.

SECTION 9 Technical Summary

This section is intended to provide a summary of the mathematical and physical background to ADMS-Urban, and is essentially an abbreviated form of the Technical Specification documentation (CERC, 2025), to which readers should refer for full details.

9.1 Meteorological input and output

Section 3.3 describes the types of meteorological data that can be input to ADMS-Urban. For all the data types, each line of input data describes the meteorological conditions for one or more hours, the conditions being assumed to be fixed for any given hour. The meteorology input module reads the data and uses processing algorithms to estimate values of the various meteorological quantities required for running the dispersion model.

Data may be in the form of a chronological record. These data are termed *hourly sequential*. Other data may have a certain weight or frequency associated with each line of meteorological data. Such data are usually non-chronological and are termed *statistical* as they have been statistically analysed. It is also possible that the data are neither sequential nor statistical, but some more general collection of one or more meteorological conditions.

9.1.1 Input

The meteorological input dataset can contain a variety of input meteorological parameters.

The complete list of possible input variables is shown in **Table 9.1**. The first three columns are alternative forms of the variable names that may be used in creating meteorological data files. As ADMS-Urban matches the string to identify a variable the user may want to use short versions of the variable names (columns 1 and 3) in order to avoid making errors in typing. Names from different lists may be used in the same meteorological input file, e.g. using U for wind speed and CLOUD for cloud cover in the same file is allowed. The fourth column shows in which units the data should be entered and the fifth column whether that variable is able to be interpolated (refer to Section 3.3.6)

It is likely that the two most common types of input data will be sequential data (or data for a single hour) giving wind speed, direction, cloud cover, time of day, time of year, temperature and possibly precipitation; and statistical data, comprising wind speed, direction, surface sensible heat flux, boundary layer height (or depth) and precipitation. Such data are supplied by the Met Office for the main UK met. stations.

Note that the Pasquill-Gifford (P-G) stability categories cannot be directly input into the model (although of course values of U, ϕ, L_{mo} and h may be), nor are they output. In ADMS-Urban the boundary layer structure is characterised by the two parameters, h and L_{MO}. Values of these parameters corresponding *approximately* to the P-G categories are shown in the data file *r91a-g.met* and below in **Table 9.2**.

Short name	Long name	Abbreviated name	Units	Min value	Max value	Can Interpolate?*
WIND SPEED	WIND SPEED	U	m/s	0.0	100.0	R
UG/USTAR	GEOSTROPHIC WIND SPEED/FRICTION VELOCITY	UGSTAR	-	5.0	1000.0	N
WIND DIRN	WIND DIRECTION (DEGREES)	PHI	°	0.0	360.0	R
DIRN CHANGE	GEOSTROPHIC MINUS SURFACE WIND DIRECTION (DEGREES)	DELTAPHI	°	-60.0	60.0	N
HEAT FLUX	SENSIBLE HEAT FLUX	FTHETA0	W/m^2	-200.0	1000.0	N
1/LMO	1/MONIN-OBUKHOV LENGTH	RECIPLMO	m^{-1}	-10.0	10.0	N
BL DEPTH	BOUNDARY LAYER DEPTH	H	m	40.0	4000.0	N
CLOUD	CLOUD AMOUNT (OKTAS)	CL	oktas	0.0	8.0	R
SOLAR RAD	INCOMING SOLAR RADIATION	SOLAR RAD	W/m^2	0.0	1500.0	N
TEMPERATURE	TEMPERATURE (C)	T0C	°C	-100.0	60.0	Y
N ABOVE BL	BUOYANCY FREQUENCY ABOVE BOUNDARY LAYER	NU	s^{-1}	0.0	0.1	Y
DELTA THETA	TEMPERATURE JUMP ACROSS BOUNDARY LAYER TOP	DELTATHETA	°C	0.0	25.0	N
PRECIP	PRECIPITATION RATE (MM/HOUR)	P	mm/h	0.0	500.0	Y
SEA TEMP	SEA SURFACE TEMPERATURE (C)	TSEA	°C	-10.0	40.0	Y
DELTA T	TEMPERATURE OVER LAND MINUS SEA SURFACE TEMPERATURE	DELTAT	°C	-40.0	40.0	Y
SIGMA THETA	SIGMA THETA (DEGREES)	SIGMATHETA	°	0.0	90.0	Y
S HUMIDITY**	SPECIFIC HUMIDITY	S HUMIDITY	kg/kg	0.0	0.1	Y
R HUMIDITY**	RELATIVE HUMIDITY (PERCENT)	RHUM	%	0.0	100.0	Y
RH ABOVE BL**	RELATIVE HUMIDITY ABOVE BOUNDARY LAYER (PERCENT)	RH ABOVE BL	%	0.0	100.0***	Y
DRH/DZ**	D(RELATIVE HUMIDITY)/DZ ABOVE BOUNDARY LAYER (PERCENT/M)	DRH/DZ	%/m	-10.0	10.0	N
LAT HT FLUX**	LATENT HEAT FLUX	LAT HT FLUX	W/m^2	-100.0	1000.0	N
Z0 (M)	ROUGHNESS LENGTH (MET SITE)	Z0 (M)	m	$1.0x10^{-10}$	100.0	Y
Z0 (D)	ROUGHNESS LENGTH (DISPERSION AREA)	Z0 (D)	m	$1.0x10^{-10}$	100.0	Y
ALBEDO (M)	ALBEDO (MET SITE)	R	-	0.0	1.0	Y
ALBEDO (D)	ALBEDO (DISPERSION AREA)	ALBEDO (D)	-	0.0	1.0	Y
ALPHA (M)	MODIFIED PRIESTLEY-TAYLOR PARAMETER (MET SITE)	ALPHA	-	0.0	3.0	Y
ALPHA (D)	MODIFIED PRIESTLEY-TAYLOR PARAMETER (DISPERSION AREA)	ALPHA (D)	-	0.0	3.0	Y
HOUR	HOUR	THOUR	-	0	24	R
DAY	DAY	TDAY	-	1	366	R
YEAR	YEAR	YEAR	-	1900	2500	R
FREQUENCY	FREQUENCY	FR	-	0	-	N
FREQUENCY FOR MONTHS xx TO xx, HOURS xxxxx TO xxxxx (GMT + xxxxx)**	FREQUENCY FOR MONTHS xx TO xx, HOURS xxxxx TO xxxxx	MONTHS xx TO xx, HOURS xx TO xx	-	-	-	N

Table 9.1 − Variables that may be input into the meteorological input module. *Key for interpolation indicator: R = Required for interpolation, Y = interpolatable, N = not interpolatable. **These variables can be entered but will not be used in the model. *** Relative humidity values up to 120% can be entered via a *.met* file, but will be taken as 100%.

There is no exact correspondence between the boundary layer parameters (h, L_{MO}) and the Pasquill-Gifford categories since many different values of h and L_{MO} may correspond to one Pasquill-Gifford category.

Stability: Stable $h/L_{MO} > 1$
Neutral $-0.3 \leq h/L_{MO} \leq 1$
Convective $h/L_{MO} < -0.3$

U (m/s)	L_{MO} (m)	$1/L_{MO}$ (m^{-1})	h (m)	h/L_{MO}	P-G Category
1	-2	-0.5	1300	-650	A
2	-10	-0.1	900	-90	B
5	-100	-0.01	850	-8.5	C
5	∞	0	800	0	D
3	100	0.01	400	4	E
2	20	0.05	100	5	F
1	5	0.2	100	20	G

Table 9.2 − Values of wind speed, Monin-Obukhov length (L_{MO}) and boundary layer height (h), which may be used to represent Pasquill-Gifford categories A-G.

The meteorological data minimum requirement (for the model to run) is:

- wind speed (this would normally be a near surface wind, but could be a geostrophic wind or friction velocity – in each case the height of wind must be entered, which would typically be 10 m for the near-surface case and should be 1000 m and 0 m for the geostrophic and friction velocity cases, respectively),
- wind direction,

plus *one* of the following:

- reciprocal of Monin-Obukhov length,
- surface sensible heat flux,
- cloud cover, time of day and time of year.

With regard to the latter three options, if more than one of these is supplied, they are used in order of preference in the order they are listed above. That is, if the reciprocal of Monin-Obukhov length is supplied it will be used, otherwise the surface sensible heat flux will be used. If neither the reciprocal of the Monin-Obukhov length nor the surface sensible heat flux are specified then the cloud cover, time of day and time of year will be used.

If cloud cover, time of day and time of year are the *only* data specified in addition to wind speed and direction, then it is advisable to add temperature and boundary layer height, if a good estimate is known, to the variables in the *.met* file. If the boundary layer height is not known, the estimate for a particular hour will be improved if meteorological data for all the hours from midnight are included, as the ADMS-Urban scheme uses this information from previous hours to improve the estimate.

It is preferable to include cloud cover data if available. However, if unavailable, incoming solar radiation may be specified instead. Incoming solar radiation is measured at ground level, as measured by a radiometer, although check with the manufacturer exactly what is measured.

In general, the boundary layer height should always be entered if a better estimate is available than that calculated by the meteorological input module. The module will provide a good estimate when the site is in mid latitudes, e.g. UK, and it is either

1. daytime with sequential data stretching back at least to dawn, *or*
2. night time.

If however, neither 1 nor 2 apply and boundary layer height information is *not* available, then it is advisable to provide the model with as much information as possible (e.g. all five of the above meteorological parameters). This may help the model to estimate boundary layer height by enabling it to estimate the values of heat flux occurring prior to the current hour under consideration. In addition, specifying temperature may also help to improve the module's estimate of the boundary layer height.

Additional meteorological input data that may be required are:

- precipitation rate, if wet deposition is to be calculated (unless a constant value of washout coefficient is specified);
- sea surface temperature and near-surface temperature over land (or the difference between them), if the **Coastline** option is used (see Section 4.20);
- frequencies with which particular meteorological conditions occur, if the meteorological data are statistical.

Further input parameters

In addition to the data in the meteorological input data set, the module also requires certain data that are provided via the interface. These are:

- surface roughness,
- height of recorded wind, i.e. the height above ground at which the wind measurements were made (usually 10 m),
- whether the meteorological data are hourly sequential,
- whether the wind direction data are in sectors, and if so, the sector size. Wind direction data are often recorded to the nearest 10°, 22.5° or 30°. Inputting the appropriate sector size allows the model to account for the approximation in the wind direction. For small sector sizes (less than 15°) the model will use one of three equally spaced wind directions within the sector. For example, if the data are in 10° sectors, and the wind direction in the meteorological data file is given as 30°, the model will use 26.7°, 30° or 33.3°. For large sector sizes (equal to or greater than 15°), the model will do calculations for all of 5 equally spaced wind directions within the sector. The results for all wind directions are given equal weighting in the concentration calculations. For example, if the data are in 30° sectors and the wind direction in the meteorological data file is given as 30°, the model will use 18°, 24°, 30°, 36° and 42°. If the data are not in sectors, the wind

direction is used exactly as given in the meteorological data file.

Further optional parameters may also be specified via the interface. These include a minimum value of L_{MO} (for urban areas) and values of Priestley-Taylor parameter and surface albedo.

If the met. site is distant from the area of dispersion, the meteorological input module presents the option to modify the wind profile at the source by taking account of the surface roughness both at the met. site and the source. Different values of surface albedo, Priestley-Taylor parameter and minimum value of L_{MO} at the met. site and dispersion site may also be entered. It is also possible to enter a precipitation factor to account for differences in rainfall between the sites (see Section 3.3).

9.1.2 User-input vertical profile data

The user can input vertical profiles for the wind speed, turbulence parameters, temperature and humidity. These data are entered at a sequence of heights for each line of meteorological data. To obtain the value at a particular height, the value is interpolated from those entered using the algorithm outlined below.

The interpolation algorithm is based on a linear interpolation of the input *.prf* data and the standard ADMS-Urban profiles. Above the maximum height in the *.prf* data, values are held constant. Below the minimum height, values are reduced exponentially to zero. Essentially a profile similar in shape to the ADMS-Urban profile is fitted to the input *.prf* data. The interpolation algorithm is detailed in the ADMS Technical Specification (CERC, 2025).

Note that the data entered in the *.prf* file override the corresponding values entered in the *.met* file.

9.1.3 Meteorological data processing

The meteorological input module is called once for each hour's data and uses standard algorithms to calculate the boundary layer meteorological parameters required by the dispersion model. Full details can be found in Holtslag and van Ulden (1983) and the ADMS Technical Specification (CERC, 2025).

In processing the data, the module checks that the input data are plausible, i.e. that they lie within certain limits. While it is running, messages may be provided giving warnings and notification of errors. Where an error occurs which prevents calculations being done for that particular line, output for that line will be given as –999.

Note that ADMS-Urban cannot model dispersion in totally calm conditions. Hence, if the wind speed at 10 m is less than 0.75 m/s, the model will reset the value to 0.75 m/s. If this occurs the model will use the wind direction from the last hour which had a wind speed greater than 0.75 m/s. If no such hour exists, because it is the start of the run, then this hour will be marked as invalid.

9.1.4 Output

The variables that are output to the *.mop* file by the meteorological input module are listed in **Table 9.3**.

Variable	Description
u_*	Friction velocity (m/s)
U_g	Geostrophic wind speed (m/s)
U_g^*	Geostrophic wind speed normalised by the friction velocity
ϕ_0	Surface wind direction (angle from which wind blows in degrees measured clockwise from north, e.g. 270° is a westerly wind) (°)
ϕ_g	Geostrophic wind direction (angle from which wind blows in degrees measured clockwise from north) (°)
$\Delta\phi$	Geostrophic wind direction minus surface wind direction (°)
ϕ	Wind direction (as obtained from the meteorological input data set) (°)
ϕ_{sec}	Wind direction used by the model for this meteorological data line for long-term calculations, which may differ from the input value if the data are in sectors (°)
w_*	Convective velocity scale (if $F_{\theta_0} > 0$, $w_* = (g F_{\theta_0} h/\rho c_p T_0)^{1/3}$; if $F_{\theta_0} \le 0$, $w_* = 0$) (m/s)
F_{θ_0}	Surface heat flux (W/m^2)
K	Incoming solar radiation (W/m^2)
$1/L_{MO}$	Reciprocal of the Monin-Obukhov length (m^{-1})
h	Boundary layer height (m)
N_u	Buoyancy frequency above the boundary layer (s^{-1})
$\Delta\theta$	Temperature jump across the boundary layer top (K)
T_0^c	Near-surface temperature (°C)
P	Precipitation rate (mm/h)
ΔT	Near-surface temperature over land minus sea surface temperature (°C)
σ_θ	Standard deviation of mean wind direction (°)
q_0	Surface specific humidity (kg/kg)
λ_E	Surface latent heat flux (W/m^2)
RH_u	Relative humidity just above the boundary layer (%)
$d(RH_u)/dz$	Relative humidity lapse rate above the boundary layer (%/m)

Table 9.3 − Output variables from the meteorological input module.

The variables P, ΔT, q_0, λ_E, RH_u and $d(RH_u)/dz$ may be missing if corresponding input data are not given. The data are now available for use by the other modules, in particular the boundary layer structure module.

9.1.5 Limitations

In calculating the boundary layer parameters, it is assumed that the boundary layer is self-similar for a given value of h/L_{MO}. However, users of the model should be aware that there are some situations, such as latitudes near the equator or the poles, where the approximation can lead to significant errors. Full details are given in the ADMS Technical Specification (CERC, 2025).

9.2 Parameterisation of the boundary layer

In ADMS-Urban the boundary layer is characterised by the boundary layer height h and the Monin-Obukhov length L_{MO} and **not** by a Pasquill-Gifford stability category.

The Monin-Obukhov length is defined as

(9.1)
$$L_{MO} = \frac{-u_*^3}{\left(\dfrac{\kappa g F_{\vartheta 0}}{\rho c_p T_0}\right)}$$

in which u_* is the friction velocity at the Earth's surface, κ is the von Karman constant (0.4), g is the acceleration due to gravity, $F_{\theta 0}$ is the surface sensible heat flux, ρ and c_p are, respectively, the density and specific heat capacity of air and T_0 is the near-surface temperature.

In unstable or convective conditions, the Monin-Obukhov length is negative. The magnitude of the length is then a measure of the height above which convective turbulence, i.e. turbulent motions caused by thermal convection, is more important than mechanical turbulence, i.e. turbulence generated by friction at the Earth's surface.

In stable conditions, the Monin-Obukhov length is positive. It is then a measure of the height above which vertical turbulent motion is significantly inhibited by the stable stratification.

Figure 9.1 shows the different regions of the boundary layer in terms of the parameters h/L_{MO} and z/h where z is height above the ground. **Figure 9.2** shows the same information but with a dimensional vertical scale, z.

In the different regions of the boundary layer different mechanisms are important in generating turbulence. These are:

1. surface heating or convectively generated turbulence (the convective eddies increase in energy as they rise through the boundary layer),
2. mechanically generated turbulence due to shearing at the surface,
3. local shear, for instance at the top of the boundary layer, that can be a weak source of turbulence.

This approach to boundary layer stability, whereby the boundary layer structure is defined in terms of two variables (z/L_{MO} and z/h) supersedes the Pasquill-Gifford formulation, and differs crucially from the Pasquill formulation in allowing the variation of boundary layer properties with height to be included. However, it is difficult to make exact comparisons between the two schemes.

Figures 9.1 and **9.2** show Pasquill-Gifford stability categories corresponding *approximately* to ranges of h/L_{MO}. Note that, particularly in stable meteorological conditions, the Pasquill class is not a simple function of h/L_{MO}.

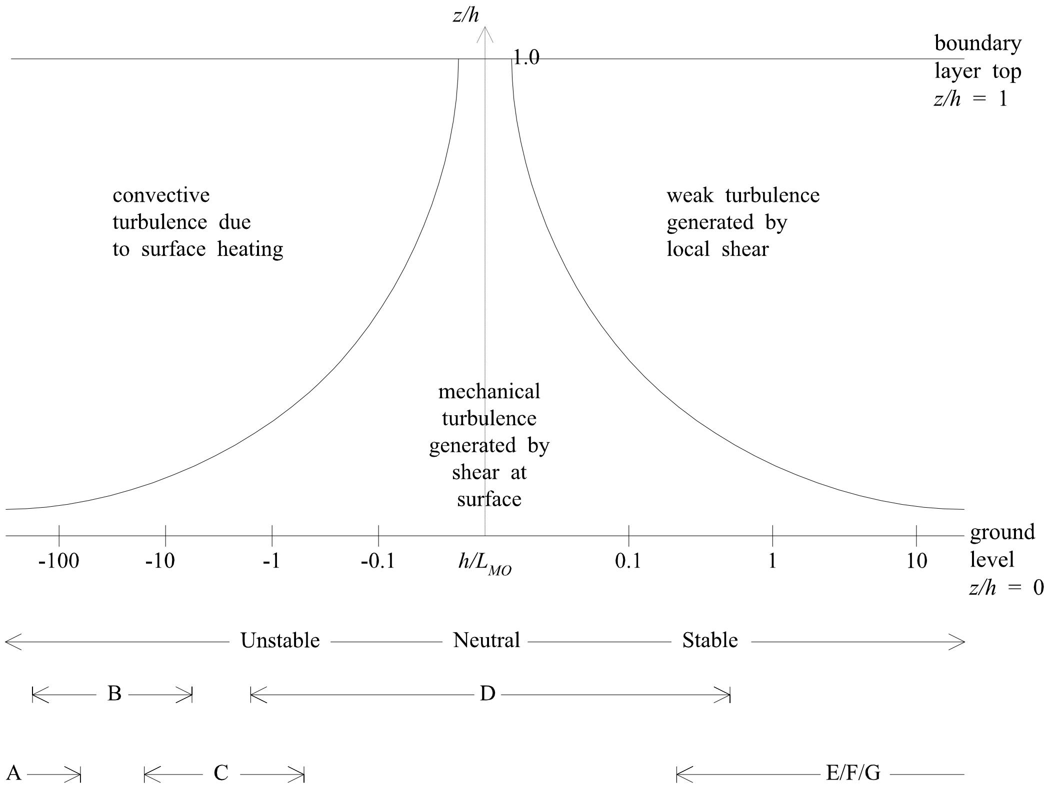


Figure 9.1 – Non-dimensional schematic representation of variation of Monin-Obukhov length with atmospheric stability.

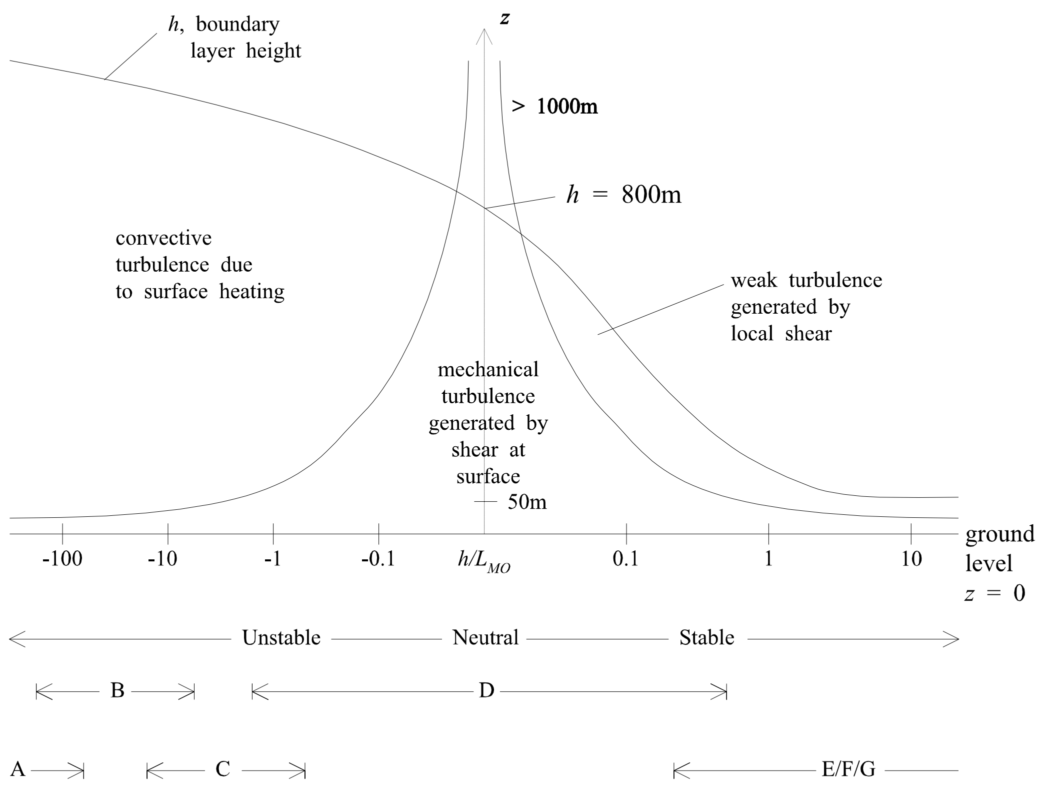


Figure 9.2 – Dimensional schematic representation of variation of Monin-Obukhov length with atmospheric stability.

9.2.1 Boundary layer structure

ADMS-Urban calculates the boundary layer variables listed in **Table 9.4** at different heights. Vertical profiles are expressed as functions of z/L_{MO} and z/h and have been derived from experimental data (Caughey and Palmer, 1979; van Ulden and Holtslag, 1985; Hunt *et al*., 1988d). These variables are used, in turn, by other modules.

Variable	Description
$U(z)$, $\frac{dU}{dz}$, $\frac{d^2U}{dz^2}$	Mean wind speed (m/s) and its first (s^{-1}) and second derivatives with height ($m^{-1}s^{-1}$)
$\sigma_u(z)$, $\sigma_v(z)$, $\sigma_w(z)$	Root-mean-square turbulent velocities (m/s)
$\Lambda_v(z)$, $\Lambda_w(z)$	Turbulent length scales (m)
$\varepsilon(z)$	Energy dissipation rate (m^2/s^3)
$T_L(z)$	Lagrangian time scale (s)
$N(z)$	Buoyancy frequency (s^{-1})
$T(z)$	Temperature (K)
$\rho(z)$	Density (kg/m^3)
$P(z)$	Pressure (mbar)

Table 9.4 − Boundary layer variables calculated by ADMS-Urban.

Note that the plume spread parameters σ_y and σ_z are calculated using these boundary layer variables and hence vary with source height and plume height. This contrasts with the approach adopted in models using Pasquill categories, in which values of σ_y and σ_z are obtained from measured profiles of σ_y and σ_z and are independent of plume height.

Turbulent velocities σ_u, σ_v and σ_w are subject to a minimum value of turbulence that ranges between 0 and 0.2 m/s, depending on the user-input minimum value of L_{MO}. This accounts for the fact that in urban areas (high minimum L_{MO}), where conditions never become very stable, there will always be some turbulence.

9.3 Dispersion over flat terrain

9.3.1 Dispersion parameters

Field experiments and research have shown that the dispersion parameters σ_y and σ_z vary with downwind distance from a point source in a way that depends on the atmospheric boundary layer height (h), the height of the source (z_s) and the height of the plume as it grows downwind. For reviews of this subject, see Hunt *et al*. (1988a), Hanna and Paine (1989) and Weil (1985). This approach is in contrast to older methods, described in the NRPB report R91 (Clarke, 1979) and used in the model ISC (U.S. E.P.A., 1995), in which the effect of the source height is not taken into account when calculating the width and depth of the plume.

There is no general theory or even generally accepted semi-empirical expression that describes the dispersion from a source at all heights within the boundary layer ($0 < z_s < h$) in all conditions of atmospheric stability and over the complete range of distances from the source to about 30 km downwind. In developing ADMS-Urban the approach adopted has been first to use formulae that have been developed and broadly accepted for specific ranges of the parameters z_s/h, h/L_{MO} (stability) and x/h (downwind distance). Interpolation formulae have then been constructed to cover the complete parameter range. The basis for these formulae is set out at length in the report by Hunt *et al*. (1988a).

9.3.2 The stable and neutral boundary layers

All the turbulence in the stable boundary layer is mechanically generated, i.e. there is no generation of turbulence due to convective motions. Usually, the level of turbulence decreases with height, as the relative effects of stratification increase, although it can be enhanced by wave motions at the top of the boundary layer. (The effect of such wave motions is not considered by ADMS-Urban.)

Concentration distribution

The distribution of the concentration profile is a Gaussian plume with reflections at the ground and the inversion layer, and ignoring any lower order reflections.

(9.2)
$$C = \frac{Q_s}{2\pi \sigma_y \sigma_z U} e^{-y^2/2\sigma_y^2} \times \left(e^{-(z-z_s)^2/2\sigma_z^2} + e^{-(z+z_s)^2/2\sigma_z^2} + e^{-(z+2h-z_s)^2/2\sigma_z^2} + e^{-(z-2h+z_s)^2/2\sigma_z^2} + e^{-(z-2h-z_s)^2/2\sigma_z^2} \right)$$

Spread parameters

The vertical dispersion parameter (σ_z) at the mean height of the plume (z_m) is linked directly to the vertical component of turbulence (σ_w) and the travel time from the source (t) by the relationship

(9.3) $$\sigma_z = \sigma_w t \left(\frac{1}{b^2} + \frac{N^2 t^2}{1 + 2Nt} \right)^{-1/2}$$

where N and σ_w are the buoyancy frequency and root-mean-square vertical turbulent velocity at z_m, respectively (Weil, 1985; Hunt, 1985). The factor b is a function of z_s/h and ensures a smooth transition between the solution for surface releases and elevated releases.

The transverse dispersion parameter (σ_y) is given by

(9.4) $$\sigma_y^2 = \sigma_{yt}^2 + \sigma_{yw}^2$$

The spreading due to turbulence (σ_{yt}) is expressed as:

(9.5) $$\sigma_{yt} = \sigma_v\, t \left(1 + \sqrt[3]{15.6}\, \frac{u_* t}{h} \right)^{-1/2}$$

With u_* the friction velocity at the Earth's surface and σ_v the root-mean-square crosswind turbulent velocity crosswind.

The spread due to variations in mean wind direction (σ_{yw}) is equal to $\sigma_\theta x$. The parameter σ_θ is the standard deviation of the mean wind direction. Either it is specified as a measured meteorological input parameter in degrees or an effective σ_θ is calculated by the meteorological data processor using the following expression:

(9.6) $$\sigma_\theta = 0.065 \sqrt{\frac{7T}{U_{10}}}$$

Here T is the averaging time in hours and U_{10} is the mean wind speed at height 10 m.

Near-field to far-field transition

In neutral conditions the part of the plume that does not have sufficient momentum or buoyancy to penetrate the top of the boundary layer is effectively confined within the boundary layer, because material reaching the top of the layer is reflected downwards. Sufficiently far from the source, after parts of the plume have been reflected at the ground and at the top of the boundary layer, the vertical variation in concentration of the pollutant is so small as to be negligible. This occurs approximately at the downwind distance where $\sigma_z \approx h$ (in fact, where $\sigma_z = 1.5\, h$). Downwind of this point the plume is considered to grow horizontally as a vertical wedge as if from a uniform vertical line source extending from 0 to h, rather than as a cone, so the variation with z in equation (9.2) is ignored.

9.3.3 The convective boundary layer

Field experiments of diffusion from elevated sources in the convective boundary layer (Briggs, 1985) have confirmed earlier laboratory and computational studies (Lamb, 1982) that the form of the vertical profiles of concentration are skewed and significantly non-Gaussian, thus changing the distribution of concentration compared with an assumed Gaussian distribution. Near the ground this is important for modelling processes such as wet or dry deposition. It is also very important for evaluating maximum ground-level concentrations from elevated releases. Ignoring the non-Gaussian effects can lead to under-estimates of ground-level concentration from elevated sources. Models that use non-Gaussian profiles are referred to as "new generation" models. New generation models include the Hybrid Plume Diffusion Model (HPDM) of Hanna and Paine (1989), the Almanac code of National Power (Moore and Lee, 1982), the CTDM code of the U.S. E.P.A. (Perry, 1991) and AERMOD (Cimorelli *et al*., 2004).

Concentration distribution

In the convective boundary layer (CBL) the probability distribution of the vertical velocity and, hence, the concentration distribution is **non-Gaussian**, or skewed. The non-Gaussian distribution ensures that, for elevated sources, the height within the plume at which the concentration is a maximum descends as the plume moves downwind, while the plume mean height ascends. After the height of the maximum concentration reaches the ground it can rise again.

Spread parameters

The transverse dispersion parameter (σ_y) is calculated in three parts, the first for dispersion due to convection (σ_{yc}), the second due to mechanically driven turbulence (σ_{yn}), and the third ($\sigma_{yw} = \sigma_\theta x$) is included to allow for the variation in the wind direction:

(9.7)
$$\sigma_{yc} = \sigma_{vc}\, t \left(1 + \frac{t}{h}\sqrt[3]{0.75 w_*}\right)^{-1/2}$$

(9.8)
$$\sigma_{yn} = \sigma_{vn}\, t \left(1 + \frac{t}{h}\sqrt[3]{15.6 u_*}\right)^{-1/2}$$

σ_{vc} and σ_{vn} are the root-mean-square horizontal velocities due to convection and mechanically driven turbulence, respectively. σ_θ is as defined in equation (9.6).

The total transverse spread is therefore given by

(9.9)
$$\sigma_y^2 = \sigma_{yc}^2 + \sigma_{yn}^2 + \sigma_{yw}^2$$

For the vertical spread σ_z, the skewed nature of the probability distribution function for the vertical turbulent velocity σ_w leads to the definition of σ_{w+} and σ_{w-} for the upwards and downwards velocities, respectively. Then $\sigma_{z+} = \sigma_{w+}\, t$ and $\sigma_{z-} = \sigma_{w-}\, t$ are defined and used in the calculation of C_{CBL}. A more detailed description is given in the ADMS Technical Specification (CERC, 2025), P10/01 & P12/01 ("Plume/puff spread and mean concentration module specifications").

The formulae above (equations (9.7) and (9.8)) reduce to forms *suitable* for neutral conditions as the meteorological conditions tend towards neutral conditions, but the actual forms are not the same as the ADMS-Urban Gaussian plume formula for the neutral boundary layer. A smooth transition from the convective boundary layer solution to the neutral boundary layer solution is used.

Near-field to far-field transition

The transition to a far-field model, where the source is equivalent to a vertical line source extending from 0 to *h* and of uniform strength, is assumed to occur when $\sigma_z = 1.5\ h$.

9.4 Plume rise

The plume rise module predicts the rise in trajectory and enhanced dilution of a continuous emission of gaseous material that is hot or has momentum. It takes into account the effects of plume buoyancy and momentum in generating plume rise, and includes penetration of inversions.

The underlying theory is based on an integral model, in which integral conservation equations are solved for the fluxes of mass, momentum and heat. The plume is assumed to be of circular cross-section. The plume properties such as velocity, density, etc., are uniform in any cross-section (i.e. a "top-hat" profile) but vary along the plume centreline. Entrainment of ambient air takes place due to the motion of the plume relative to its environment as well as due to atmospheric turbulence. The external velocity and temperature fields may vary with height.

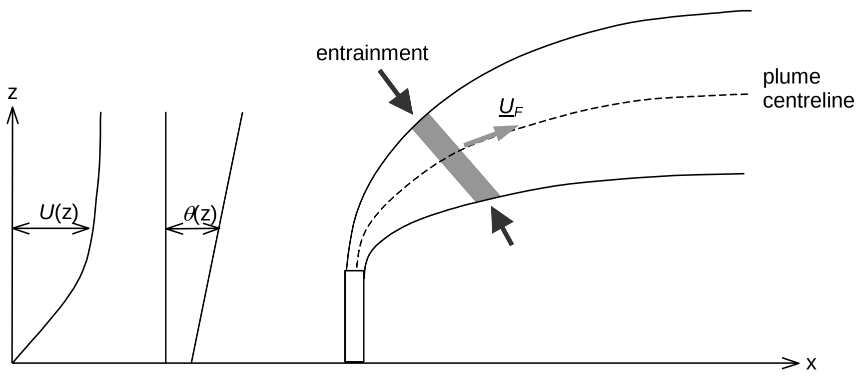


Figure 9.3 – Plume rise model.

The plume rise module is initialised from the following source conditions:

- exit diameter,
- emission velocity or volume flow rate,
- temperature or density.

The equations are then solved numerically by a Runge-Kutta numerical scheme with a variable internal time step.

9.4.1 Stack-induced downwash

The flow and pressure fields around a stack can influence the plume rise of the emission by reducing the mean height of the plume just downwind of emission. Only releases of relatively small upward momentum are affected, since all other emissions rise rapidly away from the zone of influence.

The algorithm used to predict stack downwash in ADMS-Urban is that used in many other models (Hanna *et al*., 1982; Snyder and Lawson, 1991) and applies to point sources only.

If the emission velocity ratio, w_S/U_H, is less than 1.5, the source height is corrected (reduced) by Δz_s, where

$$\Delta z_s = \begin{cases} 2\left(\dfrac{w_S}{U_H} - 1.5\right)D_S & \text{for} \quad \dfrac{w_S}{U_H} < 1.5 \\ 0 & \text{for} \quad \dfrac{w_S}{U_H} > 1.5 \end{cases} \tag{9.10}$$

with D_s the diameter of the stack, U_H the approach flow speed at the height of the stack top, w_s the emission speed, and Δz_s the correction to stack height.

If the **Buildings** option is used, the stack downwash correction is limited to ensure that the final source location is not inside one of the user-defined buildings.

ADMS-Urban includes an option to disable stack downwash for selected sources; please refer to Section 4.9 for more information.

Limitations

The stack downwash algorithm is quite widely used in dispersion models although it has never been adequately validated.

Strictly speaking, the stack downwash algorithm should only be applied to stacks of circular cross section, though again it could be applied, at the user's discretion, to other cases, such as stacks of square cross section.

9.5 Industrial sources

9.5.1 Point sources

Circular point sources are specified by the location of their centre, their diameter and their height. The circle that defines the point source is assumed to lie in a horizontal plane.

Pollutant emissions are specified in g/s.

9.5.2 Area, volume and line sources

Line sources with finite width, area sources with convex polygonal shape and volume sources with convex polygonal base areas can be modelled by ADMS-Urban. All these source types are modelled by ADMS-Urban as convex polygons.

Line sources are specified by the location of the midpoints of their ends, the height of both end points and a width. Area and volume sources are defined by the location of the vertices of their base area, a height and, in the case of a volume source, by a depth. The ends of a line source and the base of an area or volume source are assumed to lie in a horizontal plane.

A volume source does not have any plume rise. For instance, fugitive emissions that are continuously emitted from a building with negligible velocity, or the dust generated by quarrying which again is continuously emitted and is mixed throughout a depth at the source location but has no significant emission velocity, might be modelled as volume sources.

Pollutant emissions are specified in mass units/m/s for line sources, mass units/m^2/s for area sources and mass units/m^3/s for volume sources.

Method of calculation

In flat terrain, each source is decomposed into a maximum of 10 source elements, the decomposition being a function of the receptor location. The difference in streamwise distance between source elements and each receptor is constrained not to vary too rapidly, subject to the maximum number of 10 source elements. The concentration is then calculated by summing the contributions from each element.

In the case of a line or area source, the contribution from each element is approximated by a crosswind line source of finite length. The expression for the concentration $\overline{C}(x,y,z)$ from a finite crosswind line source of length L_s is given by equation (9.11). The source strength $\overline{Q_s}$ is in mass units/m/s.

(9.11)
$$\overline{C}(x,y,z)=\frac{\overline{Q}_s}{2\sqrt{2\pi}\sigma_z(x)U}\exp\left(-\frac{(z-z_s)^2}{2\sigma_z^2}\right)\times \left[\mathrm{erf}\left(\frac{y+L_s/2}{\sqrt{2}\sigma_y}\right)-\mathrm{erf}\left(\frac{y-L_s/2}{\sqrt{2}\sigma_y}\right)\right]+\text{reflectionterms}$$

Similarly, for a volume source, the contribution from each element is approximated by a crosswind vertical slice of finite length and height. The expression for the concentration $\overline{\overline{C}}(x,y,z)$ from a crosswind vertical slice of length L_s and height L_1 is given by equation (9.12). The source strength $\overline{\overline{Q}}_s$ is in mass units/m^2/s.

(9.12)
$$\overline{\overline{C}}(x,y,z)=\frac{\overline{\overline{Q}}_s}{4U}\left[\mathrm{erf}\left(\frac{y+L_s/2}{\sqrt{2}\sigma_y}\right)-\mathrm{erf}\left(\frac{y-L_s/2}{\sqrt{2}\sigma_y}\right)\right]\times \left[\mathrm{erf}\left(\frac{z+L_1/2-z_s}{\sqrt{2}\sigma_z}\right)-\mathrm{erf}\left(\frac{z-L_1/2-z_s}{\sqrt{2}\sigma_z}\right)\right] + \text{reflectionterms}$$

The calculation is accelerated by discarding those parts of the source that are at sufficiently large crosswind distances from a receptor that they do not contribute significantly to the concentration at that receptor. Redundant parts of the source are considered to be those for which a receptor lies outside the lines of $y = \pm 0.4\, x\, \sigma_y(z = z_m, x = 10 \text{ m})$ (see **Figure 9.4**).

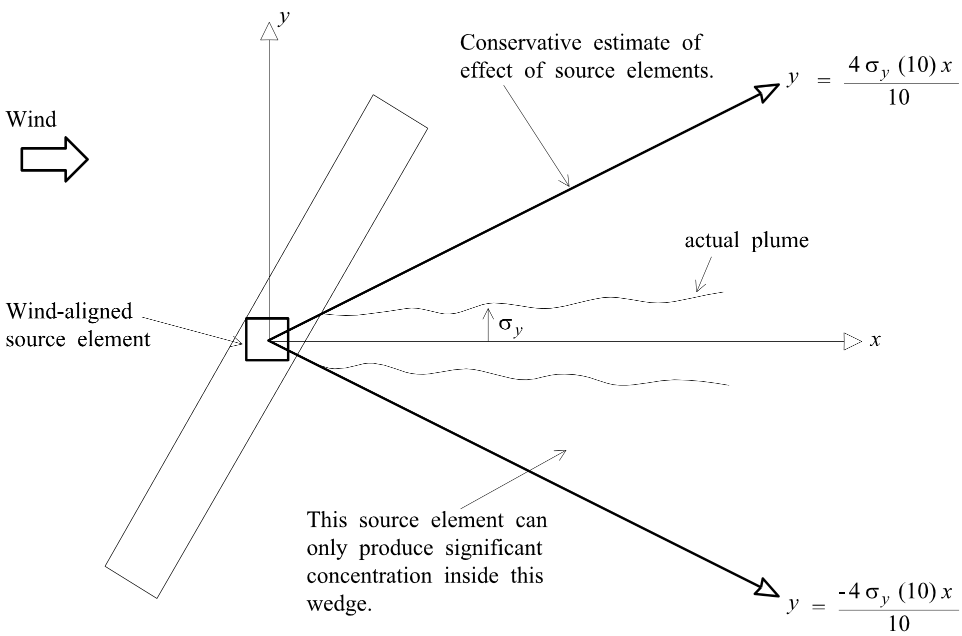


Figure 9.4 – The region of influence of a source element.

9.6 Road sources

Road sources are treated in the same way as one or more line sources (see Section 9.5.2), except that

- there is assumed to be no plume rise
- the initial vertical turbulence and emission height are adjusted to account for the range of heights of vehicle emissions
- the horizontal turbulence is adjusted to account for extra turbulence produced by the traffic flows (see Section 9.6.1)
- there is the option to model a road as a 'street canyon', if it is lined with buildings (see Section 9.8).

9.6.1 Traffic-produced turbulence

For flat, busy roads, where the wind direction is near to parallel to the road, extra lateral turbulence will be induced by the traffic. To model this, an extra component of σ_y is included when modelling road sources. (Note that this extra component is not included when modelling street canyons. The street canyon module includes a separate treatment of traffic produced turbulence.) The formulation of this extra component, $\sigma_{y_{road}}$, is as follows (formulation by D. J. Carruthers):

(9.13) $$\sigma_{y_{road}} = \sigma_{v_{road}} t \left\{ 1 + \left(\frac{t}{t_d} \right)^2 \right\}^{-1/2}$$

where

(9.14) $$\sigma_{v_{road}} = b \left(\frac{\sum_{i=1}^{n_v} N_i U_i A_i}{W} \right)^{1/2}$$

and the turbulence decay time, t_d, is given by

(9.15) $$t_d = \left(\frac{W}{\tau} \right) \Big/ \sigma_{v_{road}}$$

In the above definitions,

- t = time to travel from source to this point(s)
- b = constant (0.3) [from OSPM street canyon model]
- τ = constant (0.1) [chosen by CERC after testing]
- n_v = number of vehicle categories
- N_i = number of vehicles per second for that category

- U_i = speed of vehicles for that category (m/s)
- A_i = effective area covered by vehicles in that category (m^2)
- W = road width (m)

The formulation implies that $\sigma_{y_{road}} / \sigma_{v_{road}} \to 0$ as $t \to 0$ and $\sigma_{y_{road}} / \sigma_{v_{road}} \to t_d$ as $t \to \infty$.

Note that the effects of traffic-produced turbulence lead to a non-linear relationship between traffic flow and roadside concentration, as increased traffic flow increases both the emission rate and the rate at which these emissions disperse.

9.7 3D grid source

The ADMS-Urban approach to modelling 3D grid sources is based on the standard 2D grid source, as described in the ADMS Technical Specification Document P32/01 (CERC, 2025). The emissions of pollutants from the 3D grid source are treated as multiple 2D grid sources, one for each vertical layer, with varying elevation above ground and depth. Within each 2D grid source, the cells are modelled as volume sources, with some simplifications for computational efficiency.

The 3D grid source emission rate for each cell and each pollutant is read directly from the netCDF file for each hour, so no additional time-varying profile factors are required. When the file is created, different time-varying profiles may be applied to different component sources, allowing a more complex time variation than can be applied to a single 2D grid source.

9.7.1 Disaggregation options

As for 2D grid sources, the 3D gridded emissions file is assumed to include aggregated emissions from all sources within the modelling area, including any explicit sources (unless using the 'Disable grid disaggregation' additional input file option, as described in Section 3.2.3). This means that any explicit source emissions included in a run must be subtracted ('disaggregated') from the 3D gridded emissions by the model before the residual emissions are modelled, in order to avoid double-counting.

The horizontal approach of aggregating the explicit source emissions into horizontal grid cells is the same as for 2D grid sources, as described in the ADMS Technical Specification document P32/01 (CERC, 2025). The assumed aggregation approach for emissions from each explicit source type into the vertical layers of the 3D grid is given in **Table 9.5**. The vertical disaggregation approach used in ADMS-Urban corresponds to these aggregation assumptions, with options for altering the approach for point and road sources. Point, road, line or area sources with heights exactly on a cell boundary are allocated to the cell above the boundary, such that ground-level sources are included in the lowest layer of the grid.

Explicit source type	Aggregation approach
Point	Emissions distributed vertically according to source height incremented by an estimate of initial plume rise and spread (as described in Section 9.7.2). Optionally, emissions allocated only to the layer containing the point source height.
Road	Emissions allocated to the layer containing the road source height. Optionally, emissions allocated to the surface layer.
Line, Area	Emissions allocated to the layer containing the line or area source height
Volume	Emissions allocated to the layer(s) containing the volume source height and depth, according to the proportion of the source depth which is located in each layer.

Table 9.5 - Assumed aggregation approach for point, road, line, area and volume source

emissions in creating a 3D gridded emissions file.

9.7.2 Estimation of initial point source plume rise

The initial plume rise from a point source, ΔZ is estimated using expressions for buoyancy (ΔZ_s^{Buoy}) and momentum (ΔZ_s^{Mom}) effects, in the same way as it is estimated for assessing the position of a plume relative to building wakes. These expressions are given in the Source Conditions section of the ADMS Technical Specifications document P16/01 (CERC, 2025).

The initial plume spread for a point source is estimated as being equal to the initial plume rise, as illustrated in . A minimum plume spread of the greater of 0.1 m or the source diameter is imposed. In addition, in neutral and convective meteorological conditions, when there is a temperature inversion at the boundary layer height, the plume extent is restricted to be either entirely above or below the boundary layer following the initial plume rise.

The emissions from the point source are divided between layers of the 3D grid according to the proportion of the plume spread which is within the depth of each layer.

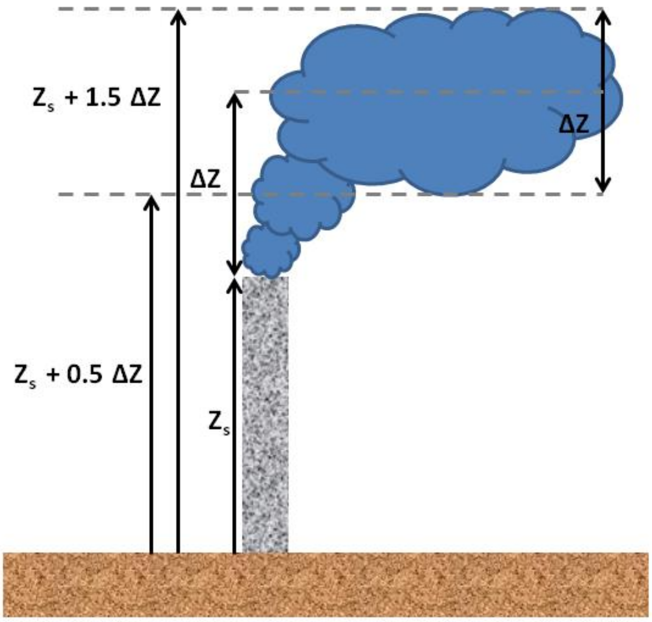


Figure 9.5 – Illustration of the initial plume rise and spread used for point source emissions disaggregation. Z_S is the physical source height and ΔZ is the plume rise.

9.8 Street canyons

The dispersion of pollutants from a road source within a street canyon may be altered by channelling of the flow by the canyon walls and a recirculating flow region driven by the component of the above-canopy flow perpendicular to the street. For street canyons with high aspect ratios, flow velocities may also reduce significantly near the ground.

ADMS-Urban includes two modelling options that simulate the effects of street canyons on local dispersion in different levels of detail; these are the ‘basic’ street canyon module and the ‘advanced’ street canyon module.

The basic street canyon module in ADMS-Urban is based on the Danish model OSPM. OSPM (Operational Street Pollution Model) was developed at the Danish National Environmental Research Institute (NERI), is described in a series of papers (Hertel and Berkowicz (1989a,b,c), Hertel *et al.* (1990)) and has been validated against Danish and Norwegian data. It uses a simplified flow and dispersion model and is able to represent the recirculation of flows within a street canyon. The basic street canyon module is described in Section 9.8.1.

The advanced street canyon module in ADMS-Urban, which has been developed in-house by CERC, addresses some of the limitations inherent in the basic street canyon module. In particular, the advanced street canyon module is able to model the channelling of flow along a street canyon, to represent asymmetric street canyons, to represent the effect of pavements within a canyon and to calculate the effect of a street canyon on the surrounding area. The module has been validated extensively by comparison with measurements from monitoring networks in Hong Kong and London. The advanced street canyon module is described in Section 9.8.2.

9.8.1 Basic street canyon module

The basic street canyon module is used for calculating the concentration at points which lie within roads lined with buildings with heights greater than 0.5 m. Concentrations within the road tend to the non-canyon results as the canyon height is reduced to zero. Concentrations at points outside the canyon are identical to those which would be obtained if the road were not a canyon.

The model ignores end effects such as junctions. It assumes a straight length of road which has a width L, and is lined on both sides by flat-roofed buildings of height H_B.

Figure 9.6 shows how a vortex is generated in the street canyon. The region occupied by the vortex is called the recirculation region. The velocity at street level in the recirculation region, u_b, is opposite in direction to the velocity at roof level u_t. A vortex will be generated if the roof level wind is not within 10^o of the street’s axis.

In some circumstances the recirculation region may extend all the way across the street; if not, it will be on the lee side of the street. Its width, L_R, is a function of roof level wind speed and canyon height.

(9.16) $$L_R = 2rH_B$$

where

$$r = \begin{cases} 1 & (u_t \geq 2m/s) \\ u_t - 1 & (1 < u_t < 2m/s) \\ 0 & (u_t \leq 1m/s) \end{cases} \quad (9.17)$$

Inside the recirculation region, the air is assumed to be well-mixed, resulting in a uniform pollutant concentration. The concentration is calculated by balancing the inflow from traffic exhausts against outflow at the top of the recirculation region.

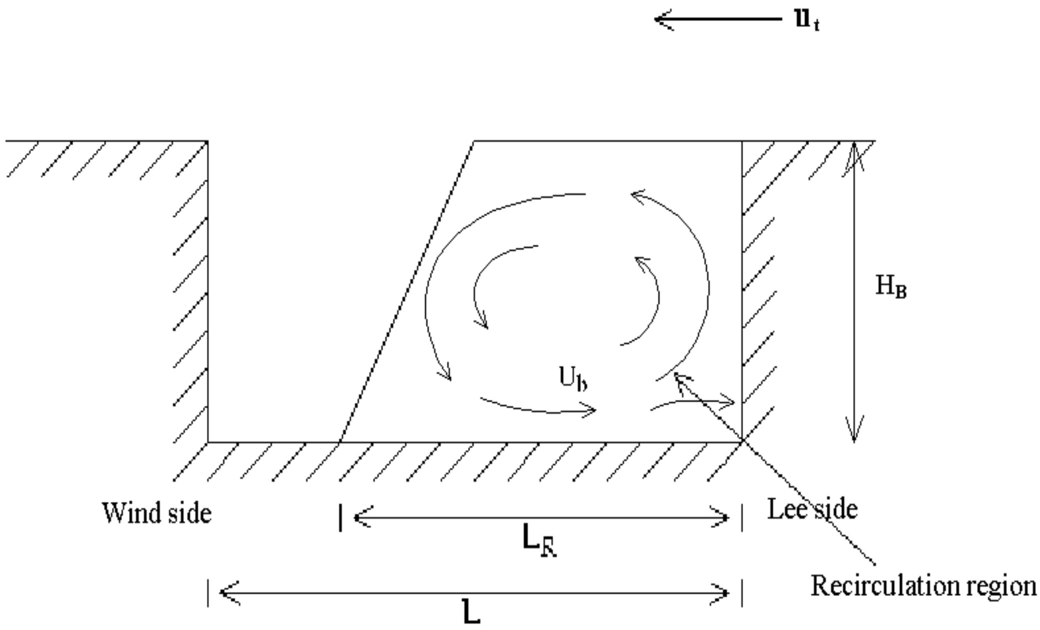


Figure 9.6 – Recirculation region in a street canyon

9.8.2 Advanced street canyon module

Input Data

In the advanced street canyon module, a street canyon is characterised by the following parameters, which must be specified for each side of the road: building height; distance from the road centreline to the canyon wall; and length (or porosity) of buildings adjacent to the road. The values of these parameters are processed by the model to obtain average canyon height H, total canyon width g and porosity α, which is defined as:

(9.18) $$\alpha = 1 - \frac{L_B}{L_R}$$

Where L_B is the length of road with adjacent buildings and L_R is the total length of road. A road source can be modelled using the advanced canyon module if at least one side has a canyon height and a length of road with adjacent buildings greater than user-definable minimum values. Canyon walls may be present on one or both sides of a road.

In contrast to the basic street canyon module, the road width and the canyon width are represented separately in the advanced street canyon module, which allows for a more accurate representation of traffic lanes, pavements etc.

Component Sources

The advanced street canyon module superimposes component sources to represent different aspects of street canyon dispersion. Seven component sources are used, which are subject to different wind directions and have various source directions and regions of influence. The seven component sources are shown diagrammatically in **Figure 9.7** and summarised in **Table 9.6**. More details of the formulations for each component source are given in the Technical Specification document (CERC, 2025) P28/02 (“Advanced street canyon specification”).

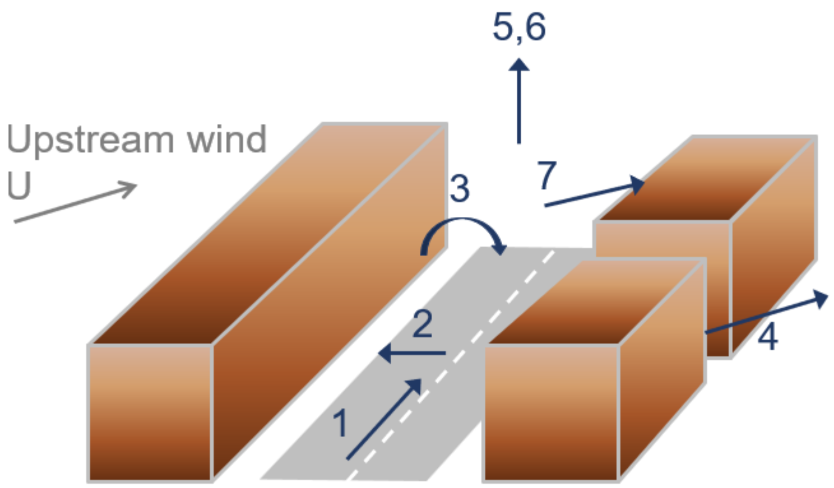


Figure 9.7 - Diagram of the seven component sources used in the advanced street canyon module

Component source		Canyon effect represented	Dispersion type	Wind direction	Region of influence
ID	**Name**				
1	Along-canyon	Channelling along canyon	Road source with wall reflections	Along canyon	Within canyon
2	Across-canyon	Direct dispersion across canyon by circulating flow	Simplified road source	Across canyon	Within recirculation region
3	Recirculation	Recirculation of pollution trapped within canyon	Well-mixed cells with vertical smoothing	n/a	Within recirculation region
4	Non-canyon	Dispersion through gaps between buildings	Road source	Upstream	Within and outside canyon
5	Canyon-top direct	Direct dispersion out of the top of the canyon	Volume source	Upstream	Outside canyon
6	Canyon-top recirculation	Escape of recirculating material out of the top of the canyon	Volume source	Upstream	Outside recirculation region
7	Canyon-end	Dispersion out of the end of the canyon	Volume source	Upstream	Downstream of canyon

Table 9.6 - Summary of component sources in the advanced street canyon module

Component source weightings

The total mass emission from a road source within a street canyon is distributed amongst the component sources according to a complex weighting that is dependent on the building height on each side of the road, the width of the street canyon, the calculated porosity of the street canyon and the local wind speed and wind direction above the buildings.

Full details are given in the Technical Specification document referred to above, but here are two example characteristics of the behaviour:

- If the upstream canyon wall is lower than the downstream canyon wall, the across-canyon source (source 2) is reduced and the non-canyon source (source 4) is increased.
- Similarly, if the downstream canyon wall is lower than the upstream canyon wall, the along-canyon source (source 1) is reduced and the non-canyon source (source 4) is increased.

Standard mode

If **Network mode** is not selected, the advanced canyon module assumes that each street canyon is part of a continuous network of street canyons. For each street canyon, it is assumed that all material leaving the end of the canyon is entering another canyon, and that the amount of material leaving the end of the canyon is balanced by an equal amount of material entering the canyon. The following assumptions are then made in the modelling:

- The along-canyon dispersion (source 1) is constant along the length of the canyon.
- No reduction in emissions is applied to the canyon-top direct source (source 5).
- There is no canyon-end source (source 7), as all material enters another canyon.

Network mode

In **Network mode**, the road sources are analysed for each meteorological condition to determine the road network, i.e. which street canyons have other street canyons upstream of them, note that only street canyons being modelled with the advanced canyon module are considered here. For each street canyon, the quantity of pollutant transported into that canyon, and the destination of the pollutant leaving that canyon are calculated. The following assumptions are then made in the modelling:

- The along-canyon dispersion (source 1) takes into account the material being transported into the canyon by modifying the source geometry to extend before the start of the road.
- If little, or no, material is being transported into the canyon then the emissions from the canyon-top direct source (source 5) are reduced to account for the time taken for material to disperse to the top of the canyon.
- The canyon-end source (source 7) is used to account for the material leaving through the end of the canyon which does not enter another canyon.

9.9 Noise barrier module

The algorithm implemented in the noise barrier module used by ADMS-Urban has been derived from investigations of explicitly modelled barriers using ADMS 3 (Version 3.3). It calculates the modified road source height, H_S. The results obtained from using this module are consistent with those from the model developed by TNO (Wesseling and Visser 2003).

Noise barriers have the effect of reducing downwind, near field ground-level concentrations. In ADMS-Urban, this decrease in ground-level concentration is represented by modelling an elevated road source. The height of the road source, H_S(m), is adjusted in the following way:

(9.19) $$H_S = \alpha_0 H_B + h_s$$

where H_B(m) is the noise barrier height, α_0=0.494 is a constant derived from investigations and h_s the height of the source in the absence of the barrier.

Figure 9.8 shows a road with two noise barriers, one on either side for part of the length of the road. This figure indicates that the region between the barriers is unaffected by the presence of the barriers i.e. the concentrations predicted by the model in this region are the same whether or not the barrier/barriers is/are present. Receptors placed to the North West of the road are affected by noise barrier A; similarly receptors placed to the South East of the road are affected by noise barrier B.

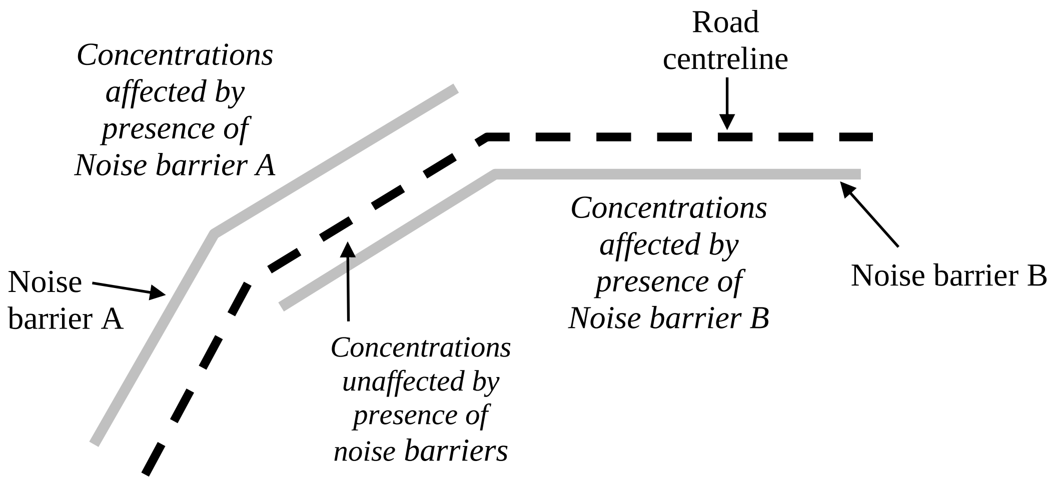


Figure 9.8 – Regions affected by noise barriers

9.10 Road tunnels module

The algorithms implemented in ADMS-Urban to represent road tunnel emissions are based on work by Ginzburg and Schattanek (1997) and have been validated against Austrian and UK road tunnels.

The approach is as follows:

- The ADMS-Urban road tunnels module models the effects of the tunnel on the surrounding area, *not* the air quality within the tunnel.
- Emissions of pollution from the end(s) of a tunnel (the tunnel ‘portals’) are assumed to be in the direction of traffic flow and follow traffic along an outflow road.
- Road emissions within a tunnel can be diverted from the tunnel portal(s) to tunnel vent(s).

Section 9.10.1 describes the modelling method for emissions from tunnel portals and Section 9.10.2 describes the modelling method for emissions from tunnel vents.

9.10.1 Dispersion from tunnel portals

The road tunnel module uses volume sources outside the tunnel portal to represent portal emissions. Three volume sources are modelled at each outflow end of the tunnel, with total length dependent on the wind speed, the outflow road traffic speed and the presence of an anti-recirculation wall. The fraction of the tunnel emissions assigned to each volume source decreases with distance from the portal, and depends on the traffic speed. Both the volume source lengths and source weightings are based on values reported by Ginzburg and Schattanek (1997), which were obtained from a wind tunnel modelling study.

The volume source height and depth are defined based on the portal base elevation and tunnel bore depth, as illustrated in **Figure 9.9**. The volume source width is set equal to the outflow road width unless the portal base elevation is negative, in which case the volume source width is defined as the average of the outflow road width and the ground level outflow width. The volume source footprint geometry depends on the outflow road geometry if an outflow road is defined, otherwise the volume sources are projected from the portal vertex along the direction of the final segment of the road tunnel. If the outflow road geometry is complex, the volume source footprint geometry may be limited by the model constraint that volume sources must be convex. More details of the method for determining volume source geometry with varying outflow road geometry are given in the Technical Specification document (CERC, 2025) P35/01 (“Road tunnel specification”).

The standard ADMS-Urban dispersion calculations for volume sources, described in Section 9.5.2, are used to model the road tunnel portal volume sources.

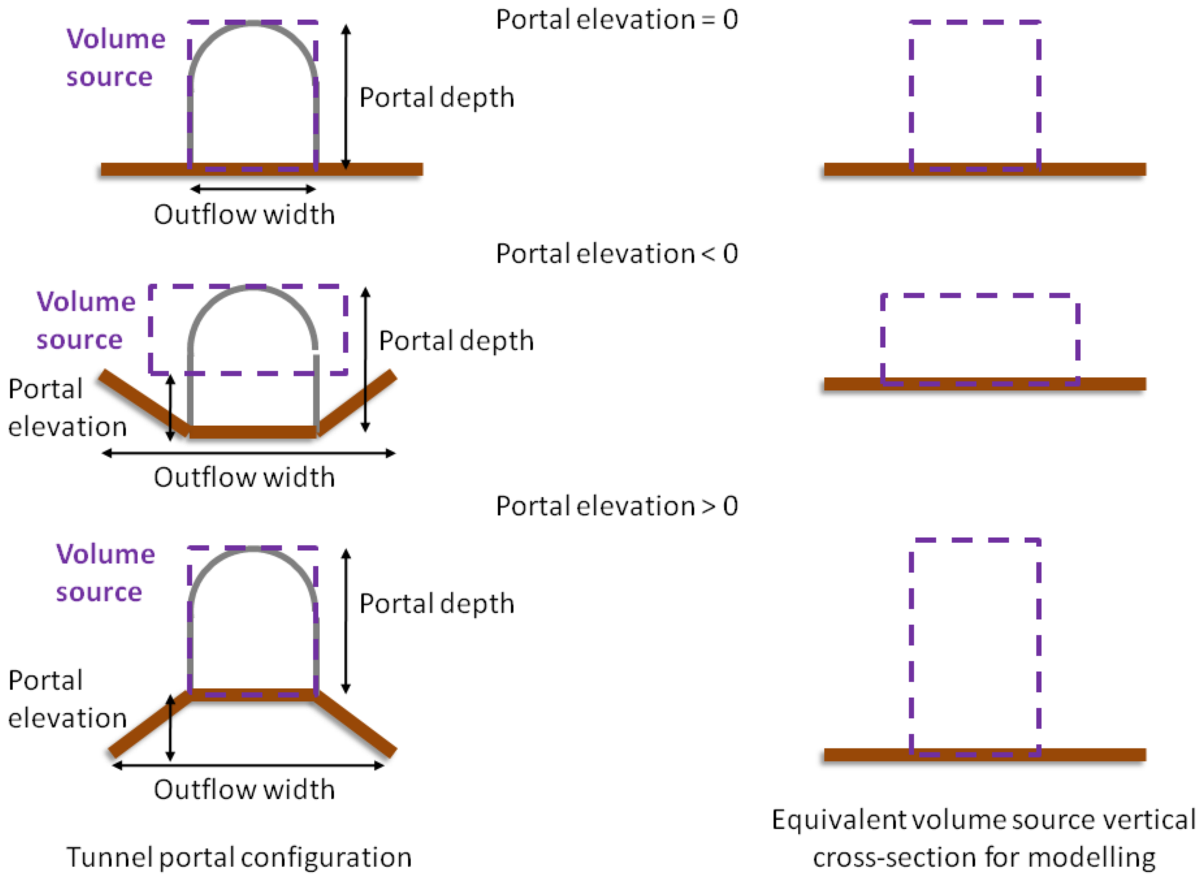


Figure 9.9 – Diagram of the vertical cross-section of the volume sources used for modelling road tunnel portal emissions for several portal geometry configurations.

9.10.2 Dispersion from tunnel vent sources

Tunnel vent sources are modelled using the standard ADMS-Urban calculations for point or area sources (described in Sections 9.5.1 and 9.5.2). Only the emission rates and the group membership of vent sources are altered by their definition as vent sources, the geometry and efflux parameters remain unchanged. The emissions from a vent source are calculated by multiplying together:

- the annual average road tunnel source emission;
- any time-varying emission factor applied to the road tunnel source;
- the specified vent fraction of emissions; and
- any time-varying emission factor applied to the vent source.

The tunnel portal emissions are reduced by the total fraction of emissions diverted to vent fractions in order to conserve total emission rates.

9.11 Flyovers

When an elevated road source is modelled as a flyover, the downward vertical plume spread, σ_z, is held constant at h_0, the initial road source mixing height (1 m), until the plume is advected past the downwind edge of the elevated road surface. It then proceeds to grow as it would have done from the source for a standard elevated road.

Recall that concentrations downwind of a source in ADMS-Urban are calculated as:

(9.20)
$$C = \frac{Q}{U} g(y) f(z),$$

where Q is the source strength, U is the wind speed (at the mean plume height), $g(y)$ and $f(z)$ are the Gaussian (in non-convective conditions) transverse and vertical concentration distribution functions, respectively, that both satisfy:

(9.21)
$$\int_{y=-\infty}^{y=+\infty} g(y)\,dy = \int_{z=-\infty}^{z=+\infty} f(z)\,dz = 1.$$

For flyover road sources, we split $f(z)$ into two piecewise continuous functions, $f_-(z)$ and $f_+(z)$, above and below the plume centreline height z_p:

(9.22)
$$f(z) = \begin{cases} f_-(z), & \text{for } z < z_p \\ f_+(z), & \text{for } z \geq z_p \end{cases},$$

subject to the following two constraints:

(9.23)
$$\int_{z=-\infty}^{z=z_p} f_-(z)\,dz + \int_{z=z_p}^{z=+\infty} f_+(z)\,dz = 1,$$

(9.24)
$$f_-(z_p) = f_+(z_p).$$

More details are provided in the Road Sources Technical Specification document P31/01 (CERC, 2025).

9.12 Multiple sources

The interaction of groups, sources, pollutants and particle sizes or gases is explained below.

9.12.1 Groups

Up to 20 user-defined groups can be entered in the user interface, and in addition a group containing all of the sources can be modelled. Each group may contain any of the sources, so that a source may be a member of more than one group, but a source can only appear once in any group.

9.12.2 Sources

Three types of sources can be modelled – road, industrial and grid. Industrial sources are modelled as point, line, area or volume sources. One grid source can be modelled consisting of a number of cells, each cell representing the total emission from all pollution sources within that cell.

Sources are defined by their position, dimensions (height, diameter/width/side length) and emission characteristics (vertical velocity/volume flow rate, temperature/density, molecular mass, specific heat capacity and pollutant emission rates). Emissions of up to 80 pollutants from each source can be specified.

All sources are assumed to emit a continuous plume (i.e. discrete ‘puff’ releases are not modelled). However, it is possible to vary the emission rate with the hour of the day, month and/or wind direction.

9.12.3 Pollutants

A pollutant is defined by a mass emission rate together with a units conversion factor. If the pollutant is particulate, up to 10 different particle sizes may be defined, but if gaseous, only one species may be defined.

9.12.4 Particle sizes and gases

For each particle size or gaseous pollutant the user is asked to define dry and wet deposition parameters, but these are only used if the **Dry deposition** and **Wet deposition** options on the **Setup** screen are selected (with the exception of ozone if using the Trajectory Model, in which case the dry deposition parameters will always be used).

9.13 Output grids and points

'Gridded' output points defined in a horizontal plane (see **Figure 9.10**) can be specified in two different ways:

1. Cartesian grid with regularly spaced grid lines,
2. Cartesian grid with variable spacing between the grid lines,

If 'gridded' output is being modelled then as well as the user-defined output grid there exists the option to allow further grid points to be automatically added with the locations of these output points dependent on the locations of the sources. Two types of source-oriented grid points exist:

- those aligned with road and line sources – 'intelligent' grid points
- those aligned with point, area and volume sources – 'nested' grid points

Details of the way the additional receptor points are added for each of these two options are outlined in Section 3.5.2 and Section 3.5.3.

With all grid types, specified points can also be defined. Up to 100 specified points can be defined in the interface, and additional points can be included in a model run by the use of an additional specified points (*.asp*) file.

The grid options apply to all types of model runs: short-term average, long-term average, complex terrain, etc.

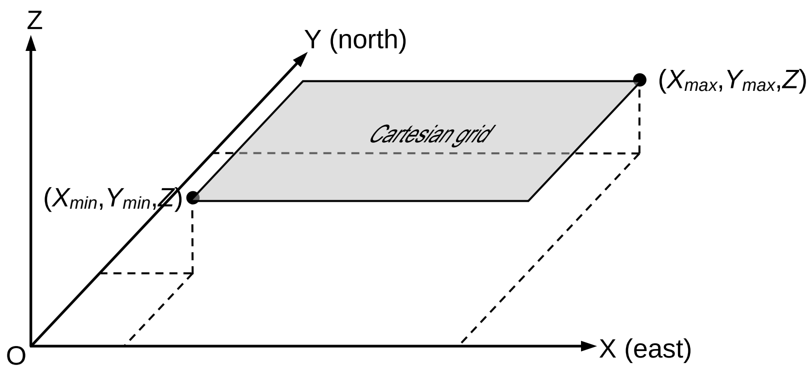


Figure 9.10 – Cartesian grids (cases 1 and 2) where (*X*,*Y*,*Z*) constitutes the coordinates of a point. OX is aligned due east and OY due north.

9.14 Averaging times and statistics

9.14.1 Averaging times of one hour or longer

The mean concentration averaging time or sampling time (t_s) is entered for each pollutant in the **Output** screen.

If an averaging time greater than one hour is specified, it must be a whole number of hours and hourly sequential meteorological data must be supplied. The program will sum successive hourly averages to calculate the mean and statistics of concentration for the required averaging time.

The program can calculate rolling or running averages. For example, if concentrations are calculated for 24 hours of data, $\{C_i, i = 1,2,\ldots,24\}$:

- A **short-term**, 8-hour (non-rolling) average would be calculated as

(9.25) at 8 hours of meteorological data: $C(t_s = 8\ h) = \frac{\sum_{i=1}^{8} C_i}{8}$

(9.26) at 16 hours of meteorological data: $C(t_s = 8\ h) = \frac{\sum_{i=9}^{16} C_i}{8}$

(9.27) at 24 hours of meteorological data: $C(t_s = 8\ h) = \frac{\sum_{i=17}^{24} C_i}{8}$

Note that the **Validity Threshold (%)** value for the pollutant, as specified in the **Output** screen, stipulates the percentage of met. lines in any given averaging period that must be valid in order for that averaging period to be considered valid.

- The **long-term**, 8-hour (non-rolling) average over this 24-h period would be

(9.28)
$$\frac{\frac{\sum_{i=1}^{8} C_i}{8} + \frac{\sum_{i=9}^{16} C_i}{8} + \frac{\sum_{i=17}^{24} C_i}{8}}{3}$$

- A **short-term** 8-hour **rolling** average would be (assuming a validity threshold of 75% and no invalid met. lines):

(9.29) before 6 hours of meteorological data: $C(t_s = 8\ h) = no\ value$

(9.30) at 6 hours of meteorological data: $C(t_s = 8\ h) = \frac{\sum_{i=1}^{6} C_i}{6}$

(9.31) at 7 hours of meteorological data: $C(t_s = 8\ h) = \frac{\sum_{i=1}^{7} C_i}{7}$

(9.32) at 8 hours of meteorological data: $C(t_s = 8\ h) = \frac{\sum_{i=1}^{8} C_i}{8}$

(9.33) at 9 hours of meteorological data: $C(t_s = 8\ h) = \frac{\sum_{i=2}^{9} C_i}{8}$

(9.34) at 10 hours of meteorological data: $C(t_s = 8\ h) = \frac{\sum_{i=3}^{10} c_i}{8}$

etc.

- The **long-term**, 8-hour **rolling** average would be the average of the 19 calculated short-term values, i.e. the short-term rolling averages from hour 6 to hour 24.

9.14.2 Averaging times shorter than one hour

The mean concentration averaging time or sampling time (t_s) is entered for each pollutant in the **Output** screen.

Averaging times up to and including one hour are used to calculate the component of lateral plume spread due to changes in the mean wind direction, unless σ_θ (standard deviation of the horizontal wind direction) is specified as a meteorological variable (see Section 3.3). It might be appropriate to use an averaging time of shorter than one hour as a first estimate to compare with an air quality standard that has an averaging time of shorter than one hour.

For averaging times greater than one hour, unless data are given by the user for σ_θ, the changes in mean wind direction within each hour of meteorological data are calculated using one hour.

9.14.3 Maximum daily output

Maximum daily output can be calculated for any averaging time. If maximum daily output is selected, then for each day, the maximum value of the averaging periods which end on that day is calculated (for those days for which the percentage of averaging periods that are valid is greater than the specified validity threshold).

9.14.4 Limitations

The longest period for which a rolling average can be calculated is 72 hours (3 days). The longest period for which a non-rolling average can be calculated is 168 hours (1 week).

9.14.5 Long-term statistics

Long-term averages of hourly means and percentiles can be calculated using meteorological data which have been statistically analysed and therefore have a frequency or weighting attached to each meteorological condition, or from raw data which have not been statistically analysed. The latter data may be sequential or a number of unrelated meteorological conditions (although then the long-term average is unlikely to be useful), in which case an equal frequency or weighting is attributed to each hour's data. The module outputs long-term average *mean concentrations*, long-term average *mean deposition fluxes*, and long-term average *percentiles of concentration*, and long-term average *exceedences per annum*.

For each set of meteorological data and associated frequency f (normalised by dividing the sum of the frequencies so that f lies between 0 and 1), the concentration at each point is calculated, say $C(x,y,z)$. Then the long-term *mean concentration* at that point, $C_{LT}(x,y,z)$, is given by the sum of Cf over all the data sets.

To calculate the concentration at each point corresponding to a specified *percentile p* at ground level, say $C_p(x,y,z)$, the values of $C(x,y,z)$ at one point for each combination of meteorological variables are considered along with their frequency of occurrence f. First of all the concentration values are arranged in descending order (i.e. highest at the beginning, lowest at the end) and the values of f rearranged accordingly. Then, starting at the highest concentration, the frequencies are summed until their cumulative value is $(100 - p)/100$. Percentile calculations may require a large amount of computing space and time, which is largely a function of the number of meteorological data sets.

Exceedences can also be calculated. In the same way as described above for percentiles, concentrations are ordered, along with their cumulative frequency. The number of concentrations above the exceedence value under consideration are calculated and output as an equivalent value of 'exceedences per annum'.

For example, consider a run of statistical data with 120 meteorological data lines. Take one output point where concentrations are greater than the exceedence value 12 times, say. The value output at this point for this exceedence is:

(9.35)
$$12 \times \frac{8760}{120} = 876$$

If maximum daily output is selected for a long-term output, the maximum daily values are used in any percentile or exceedence calculations but not in the calculation of the long-term average. The long-term average is always the average of all of the averaging periods, so as to approximate a period average.

9.15 Dry deposition

Dry deposition modifies the airborne concentration in two ways:

1. a reduction in plume strength (integrated flux of pollutant) Q with distance as material is removed from the plume at the surface, and
2. an adjustment of the vertical profile because removal of material occurs at the surface.

These effects are illustrated in **Figure 9.11**.

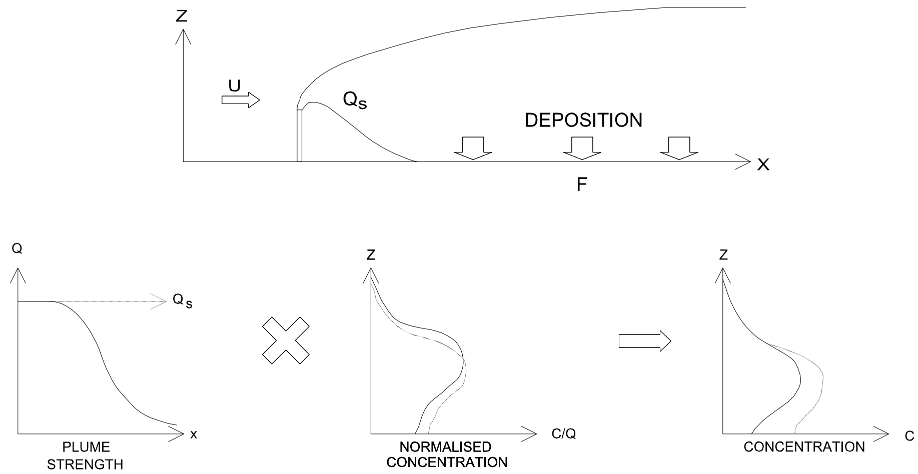


Figure 9.11 – Dry deposition modifies the vertical profile of concentration. Profiles of concentration and normalised concentration are shown.

The rate of dry deposition is assumed to be proportional to the near-surface concentration, i.e.:

(9.36)
$$F_{dry} = v_d C(x, y, 0)$$

where F_{dry} is the rate of dry deposition per unit area per unit time, $C(x,y,0)$ is the predicted airborne concentration at ground level and v_d is the deposition velocity. This velocity contains a diffusive part (v'_d) commonly referred to as the deposition velocity itself, and an element due to the gravitational settling (v_s), the terminal velocity of a particle. They are related to the overall deposition velocity v_d by the equation:

(9.37)
$$v_d = \frac{v_s}{1 - \exp(-v_s / v'_d)}$$

When v_s is zero, $v_d = v'_d$ and when v'_d is zero $v_d = v_s$. (These limits can be derived from equation (9.37), as well as making sense physically.)

One or both of v'_d and v_s may be known by the user and input directly to the model, or they can be estimated by the model on the basis of either gas type or particle size and density.

If v'_d is estimated by the model it is expressed as the reciprocal of the sum of three resistances:

(9.38)
$$\frac{1}{v_d'} = r_a + r_b + r_s$$

where r_a is the aerodynamic resistance, r_b is the sub-layer resistance and r_s the surface layer resistance, which can either be calculated by the model or input directly by the user.

These resistances depend on the pollutant species, nature of the surface and the wind speed. For particles $r_s = 0$, while for gases r_s is either input by the user or estimated by the model as 30 s/m for reactive gases (e.g. SO_2), 500 s/m for unreactive gases (e.g. O_2) and is taken to be infinite for inert gases.

The terminal velocity v_s is always zero for gaseous pollutants, while for particles it is estimated from the properties of the particle.

9.16 Wet deposition

The uptake of gases in clouds and rain, and their subsequent deposition at the ground in solution, is a complex kinetic process that must be simplified for application in practical models of wet deposition. The wet deposition module in ADMS-Urban allows users not only to model the amount of pollutant being deposited at the ground in solution, but also the corresponding depletion of the plume strength, as indicated in **Figure 9.12**.

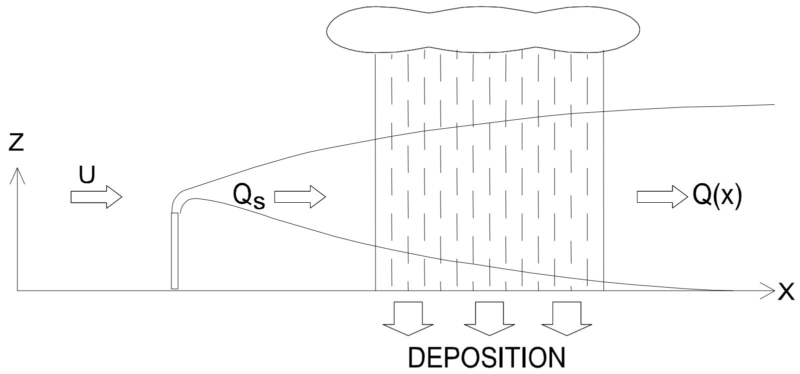


Figure 9.12 – Wet deposition model.

Wet deposition is modelled through use of a *washout coefficient* Λ, the amount of material incorporated into any falling rain or precipitation is ΛC per unit area per unit vertical distance per unit time, where C is the local airborne concentration.

Assuming irreversible uptake, the total wet deposition rate per unit horizontal area per unit time (F_{wet}) is found by integrating through a vertical column of air:

(9.39) $$F_{wet} = \int_0^{\infty} \Lambda C \, dz$$

The pollutant remaining in the plume, or plume strength, Q, therefore decreases with downwind distance.

The washout coefficient Λ is dependent on a large number of parameters, including the nature of the pollutant, rainfall rate, droplet size distribution and the pollutant concentrations in the air and in the raindrops. A value for Λ may be entered directly by the user or estimated by the system, in one of the following ways:

- by specifying a constant value Λ, which is then independent of the precipitation rate input to the meteorological input module, i.e. wet deposition will be calculated even when the precipitation rate is zero or if no precipitation is specified;
- by specifying constants A and B which give a washout coefficient dependent on precipitation rate of the form

(9.40) $$\Lambda = AP^B$$

The default values of A and B are $A = 10^{-4}$ and $B = 0.64$.

The following simplifications are made:

- uptake of pollutants is irreversible; uptake in rain does not lead to a *redistribution* of material in the plume. Also, there is no limit to the flux caused by chemical reactions. The model is therefore more appropriate for particulates than gases;
- all plume material lies in or below rain cloud; furthermore no distinction is made between in-cloud scavenging (*rainout*) or below-cloud scavenging (*washout*);
- the rainfall rate is constant and uniform over the area of calculation.

9.17 Chemistry

9.17.1 The Chemical Reaction Scheme

Within this scheme, both NOx chemistry and sulphate chemistry are modelled.

NOx chemistry: The Generic Reaction Set (GRS) scheme

Vehicles and industrial sources emit a complicated mixture of chemicals including many organic compounds (e.g. volatile organic compounds or VOCs) and oxides of nitrogen which are involved in reactions with ozone. It is beyond the scope of a fast, practical air dispersion model to include all the reactions that these chemicals undergo. Therefore, ADMS-Urban uses a scheme known as the Generic Reaction Set or GRS (Venkatram *et al.*, 1994), which models the important reactions involving nitrogen, VOCs and ozone. The GRS chemistry scheme is a semi-empirical photochemical model that reduces the complicated series of chemical reactions involving NO, NO_2, O_3 and many hydrocarbons to just the following seven reactions:

1. $ROC + h\nu \rightarrow RP + ROC$
2. $RP + NO \rightarrow NO_2$
3. $NO_2 + h\nu \rightarrow NO + O_3$
4. $NO + O_3 \rightarrow NO_2$
5. $RP + RP \rightarrow RP$
6. $RP + NO_2 \rightarrow SGN$
7. $RP + NO_2 \rightarrow SNGN$

where

ROC = Reactive Organic Compounds

RP = Radical Pool

SGN = Stable Gaseous Nitrogen products

SNGN = Stable Non-Gaseous Nitrogen products

Equations (3) and (4) represent exact chemical reactions that happen very quickly. The other equations are approximations.

An additional reaction

$$2NO + O_2 \rightarrow 2NO_2$$

is also included in the GRS chemistry scheme. This reaction will not have significant impact on NO and NO_2 concentrations unless the initial NO concentration is sufficiently high and the reaction takes place over a long period of time. For example, at 15°C, less than 3 $\mu g/m^3$ of NO_2 is produced from an NO concentration of

200 μg/m^3. The effect becomes more significant at NO concentrations of about 1000 μg/m^3 and is likely to be more significant if the Trajectory Model is used (see Section 9.17.2), where reactions can occur over several hours.

The model uses background concentrations of ozone, NO_X and NO_2. These values should be rural background values, i.e. values that do not take into account the effect of the sources being modelled, to avoid "double counting". During the day the reaction rate for equation (3) is modified by assuming the background values are in equilibrium or calculated based on solar radiation values, refer to Section 4.13.4.

A local night-time chemistry option is available, refer to Section 4.13.5. With this option, the chemistry scheme is applied to the NO and NO_2 values before the background values of these pollutants are added on. This allows the effects of chemistry to be taken into account, without incorrectly affecting the background concentrations.

Since the transport and chemistry schemes are uncoupled, the time period over which the chemistry scheme is applied must be estimated by the model. In order to do this the sources are, by default, split into two groups, 'near' and 'far' based on the source-receptor travel time. The number of said groups can be altered by the user using the option detailed in Section 4.13.7. For each of these groups the time period is taken to be the minimum of the 'weighted mean ages' of NO and NO_2, where the weighted mean age is calculated using the formula

(9.41)
$$Age = \frac{\sum_{i=1}^{N} C_i A_i}{\sum_{i=1}^{N} C_i}$$

where C_i is the concentration resulting from source i at the output point (specified point or output grid node) currently being considered, A_i is the age of the pollutant (which depends on the wind speed and the distance between source i and the output point) and N is the number of sources. Separate values of the age are calculated for the 'near' sources (Age_{near}) and the 'far' sources (Age_{far}). A maximum value of 1 hour is imposed on these values. The chemical reactions are firstly applied to the background and the contribution from the 'far' sources for time $Age_{far} - Age_{near}$. Then the contribution from the 'near' sources is added, and the chemical reactions are applied for a further time of Age_{near}.

The chemistry scheme uses an adaptive time stepping method where the time-step is dependent on the maximum rate of change of concentration occurring amongst all the pollutants. This allows the chemistry module to proceed quickly when concentration changes due to chemical reactions are small, and more slowly when detailed and rapid chemistry is occurring.

Sulphate chemistry

The reaction process used is taken from the EMEP model (Tsyro, 2001) developed by the MSC-west-Norwegian Meteorological Institute in Norway.

SO_2 is oxidised in the following reaction:

$$2SO_2 + O_2 \rightarrow 2SO_3.$$

The reaction rate depends only on the day of the year, and the maximum reaction time is 1 hour. Any SO_3 produced reacts immediately with water to form sulphuric acid:

$$SO_3 + H_2O \rightarrow H_2SO_4.$$

Sulphuric acid reacts with ammonia, which is assumed to be readily available, in the atmosphere to form ammonium sulphate,

$$H_2SO_4 + 2NH_3 \rightarrow (NH_4)_2SO_4$$

which is added to the concentration of particulate matter, PM_{10} and $PM_{2.5}$.

9.17.2 The Trajectory Model

A simple Lagrangian Trajectory Model is used to calculate background concentrations for the air approaching the main model domain. The main model domain can be thought of as containing all individually defined sources, receptor points and output grids. The Trajectory Model allows the main model domain to be nested within a larger domain, such as a large urban conurbation where the effects of emissions over the whole area need to be considered (**Figure 9.13**).

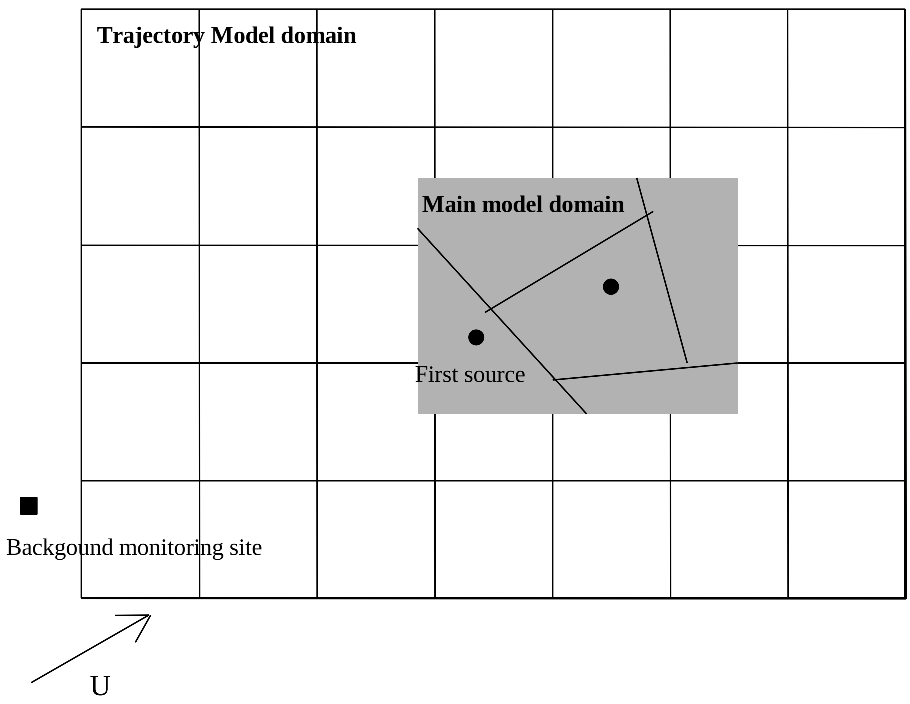


Figure 9.13 – Schematic showing the main model nested within large area-wide Trajectory Model

Computational time is a restricting factor on the complexity of the Trajectory Model. For this reason a single layer Trajectory Model with a regular grid is used. Meteorological parameters and emissions are assumed to be constant over each grid square within the Trajectory Model. Removal of pollutants from the atmosphere is modelled by way of dry deposition at the surface.

The governing equation for the Trajectory Model is

(9.42)
$$\frac{d}{dt}C(t) = \frac{F_e}{H_{mix}} - \frac{v_d}{H_{mix}}C(t)$$

where C(t) is the concentration of the chemical species at time t, H_{mix} is the depth of the atmospheric boundary layer, v_d is the deposition velocity and F_e is the emission flux of the species.

The solution of this equation for the travel time across a grid square, Δt (assuming steady state conditions), is given by

(9.43)
$$C(t_0 + \Delta t) = \begin{cases} C(t_0)\exp\left(-\frac{V_d \Delta t}{H}\right) + \frac{F_e}{V_d}\left(1 - \exp\left(-\frac{V_d \Delta t}{H}\right)\right), V_d > 0 \\ C(t_0) + \frac{F_e \Delta t}{H}, V_d = 0 \end{cases}$$

The Trajectory Model is employed in the following way. For each meteorological condition the most upwind point of the main model domain is calculated and this point is designated as the receptor point for the Trajectory Model. A back trajectory is calculated from the receptor point to the edge of the Trajectory Model domain. The Trajectory Model is initialised using the background concentrations specified by the user, which are typically measurements from automatic monitoring sites – for increased accuracy the user could supply data from different sites for different wind directions. The Trajectory Model is then run along the trajectory from the edge of the domain to the receptor point, grid square by grid square. The time period required to cross the grid square is estimated, and by assuming steady state conditions in each grid square, the change in concentration due to emission and dry deposition is computed easily using the above formulation. The algorithms used to determine values in the current grid square are the same as those used by the FRAME atmospheric transport model (Singles *et al*., 1997).

As the Trajectory Model moves from grid square to grid square the met conditions are updated which may result in increases in the boundary layer height along the Trajectory Model trajectory. Such boundary layer growth is accompanied by entrainment of air. This entrainment is modelled by the equation,

(9.44)
$$C_{NEW} = \alpha C_{OLD} + (1 - \alpha)C_{ENT}$$

where C is the concentration in the boundary layer, $\alpha = h_{OLD}/h_{NEW}$ where h is the height of the boundary layer, and subscripts 'OLD', 'NEW' and 'ENT' indicate values before boundary layer growth, values after boundary layer growth and entrained values, respectively. C_{ENT} is set to the maximum of 30ppb and the input background value for ozone, and to 0ppb for all other pollutants.

Once the new concentrations have been calculated for the current grid square, the GRS chemistry scheme for NO_X and the EMEP scheme for sulphate are applied to calculate the change in concentrations resulting from chemical processes. For the chemistry calculations, a time period is required. Here the travel time between grid squares is used, calculated by

(9.45) $$Time = \frac{\Delta X}{U(H/2)}$$

where ΔX is the distance travelled during the current step, and $U(H/2)$ is the wind speed at half the boundary layer height. Note that this expression is not pollutant-specific.

9.17.3 NOx-NO2 correlation

When the NO_X-NO_2 correlation option is selected, output concentrations of NO_2 are calculated from the NO_X values. The concentration of NO_2 is calculated using the following function:

(9.46) $$[NO_2] = 2.166 - \left(1.236 - 3.348A + 1.933A^2 - 0.326A^3\right)[NO_X],$$

where concentrations are hourly-averaged concentrations in ppb and $A = \log_{10}([NO_X])$.

The scheme is only valid for NO_X concentrations in the range 9 ppb to 1141.5 ppb and the correlation is independent of ozone concentrations. The correlation was originally derived from analysis of London ambient monitoring data although it has been tested against data collected in other cities in the United Kingdom. The user should be aware that correlation between NO_X and NO_2 concentrations can vary significantly with location. See Derwent and Middleton (1996) for further details of this scheme.

9.18 Buildings

The building effects module is used to calculate the dispersion of pollution from point sources near large structures. The ADMS-Urban model of building effects has the following features.

- Up to 25 cuboidal or cylindrical buildings may be defined by the user in terms of their height, length, width and orientation (the last two parameters are disregarded for cylindrical buildings). A main building is defined for each source (refer to Section 4.15.3 for advice on the choice of main building.) Then, for each wind direction the buildings are reduced to a single cuboidal effective wind-aligned building whose height is a function of the height of the main building (see Section 9.18.1).
- The disturbed flow field consists of a recirculating flow region or cavity in the lee of the building, with a diminishing turbulent wake downwind.
- Concentrations of the entrained part of the plume are uniform within the well-mixed recirculating flow region and based upon the fraction of the release that is entrained.
- The concentration at a point further downwind is the sum of two contributions: a ground-level plume from the recirculating flow region and an elevated plume from the non-entrained remainder. The turbulent wake reduces plume height and increases turbulent spread.
- If the source is upwind of the building and the plume impacts the façade of the building, the plume will be split into three plumes going around and over the building. These plumes are then used in the calculation of the fraction entrained into the cavity and represent the elevated plume for the non-entrained contribution in the main wake.
- The concentration and deposition are set to zero within the user-defined buildings.

The building effects module interacts with the rest of ADMS-Urban, using the standard concentration profiles, but with modified plume height and plume spread. The stages in the analysis of building effects are illustrated in **Figure 9.14**, while **Figure 9.15** shows how a complex of buildings is treated.

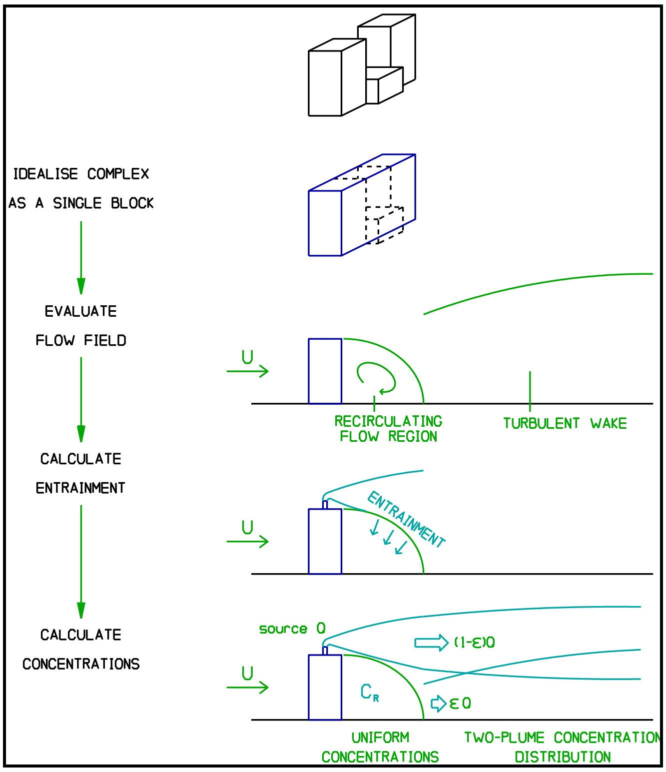


Figure 9.14 – Stages in the analysis of building effects.

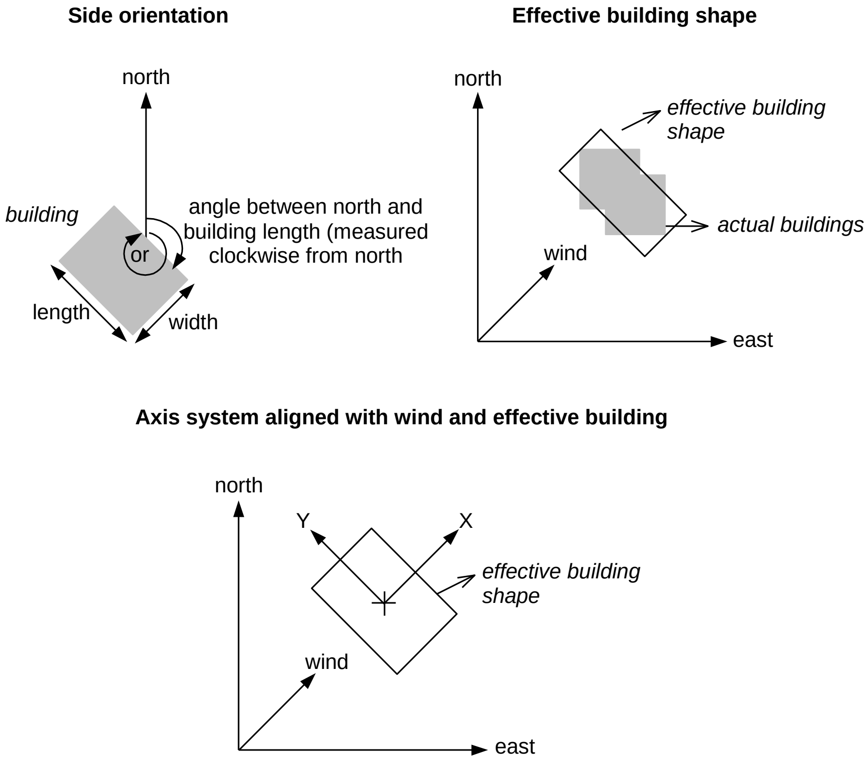


Figure 9.15 – Building effects module definitions.

9.18.1 Determination of the 'effective building'

The effective building is derived by the following algorithm:

1. Circular buildings are converted to 'equivalent' square blocks, with the same centre as the input circular building and side length $D_i/\sqrt{2}$, oriented such that the wind is normal to the building face.
2. Any buildings of height less than a fraction $1/\alpha$ of the source height are ignored, where

(9.47) $$\alpha = 1 + 2\ \min\left(1, \frac{W_i}{H_i}\right)$$

where W_i is the crosswind width of building i.

3. Any buildings that are greater than a certain distance from the plume centreline in the crosswind direction are ignored.

Specifically, a building will be ignored if all its vertices are greater than $\sigma_y(|x|)$ from the plume centreline in the crosswind direction, where x is distance from the source in the alongwind direction, and $\sigma_y(x)$ is the horizontal plume spread (not including building effects) at distance x downwind of the source, as illustrated in **Figure 9.16**.

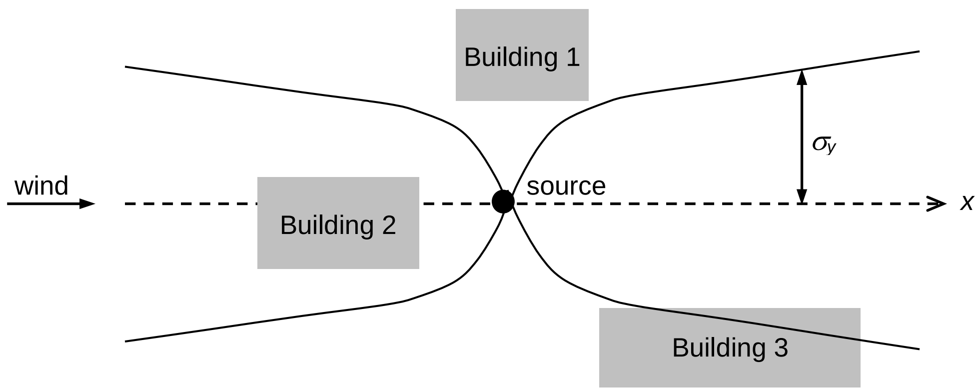


Figure 9.16 – Example source and building configuration. Buildings 2 and 3 will be included in the effective building, but building 1 will not.

4. The height, H_B, and 'orientation', θ_B, of the effective building are the height and orientation of the main building. Multiples of 90° are added or subtracted until -45° < $\theta_B \le$ 45°. Note that the effective building is always orthogonal to the flow; the 'orientation' θ_B is a parameter used to define some aspects of roof flow and near-wake behaviour.

A different main building may be selected for each source. If the main building is set to be calculated automatically it will be chosen as the closest building which satisfies criteria 2 and 3 above, this can differ on a per source and per met. line basis. A similar process is applied if the user-selected main building doesn't satisfy criteria 2 or 3 above.

5. A subset Σ is then defined by the main building plus all other buildings (a) that are at least 0.5 H_B high and (b) whose projected crosswind and along-wind separations from another subset member do not exceed half the projected crosswind width of the main building.

6. In the general case where Σ includes more than one building, W_E is the projected crosswind width and L_E is the along-wind projection from the furthest upwind mid-face to the furthest downwind mid-face, as illustrated in **Figure 9.17**.

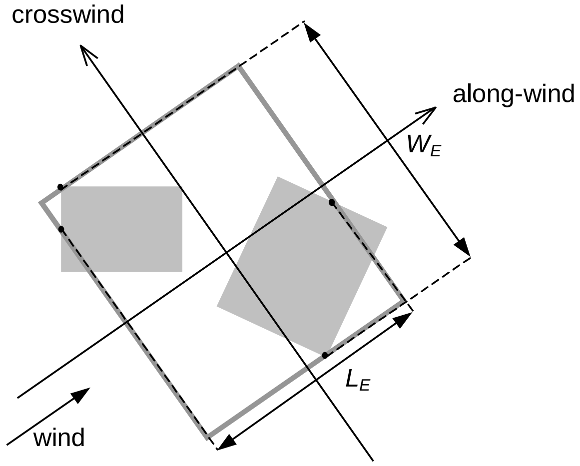


Figure 9.17 – Effective building for case where Σ includes two buildings. The grey shaded rectangles represent the user-input buildings and the grey open rectangle represents the effective building.

7. In the special case where Σ only includes one building, $L_E = min(L_F, L_D)$ where L_D is the along-wind length of the building, as seen when travelling along the wind direction, and L_F is the along-wind projection from the furthest upwind mid-face to the furthest downwind mid-face described in 6, as illustrated in **Figure 9.18**.

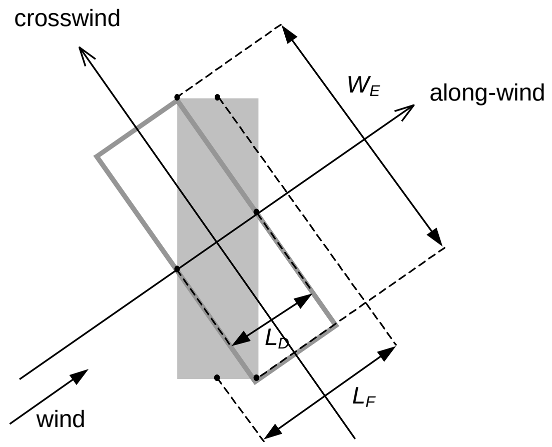


Figure 9.18 – Effective building for case where Σ only includes one building. The grey shaded rectangle represents the user-input building and the grey open rectangle the effective building.

9.18.2 Limitations

The buildings module is based on experiments in which there was one dominant site building and several smaller surrounding buildings less important for dispersion.

The buildings module only affects point sources. Other sources may be present in the same run but the effects of buildings on these sources will not be considered.

9.19 Complex terrain

The complex terrain module applies a three-dimensional flow and turbulence field to the dispersion modelling calculations. When variable terrain height is modelled there are two possible flow field solutions. The flow field solution used depends on the meteorological conditions. The critical parameter is the maximum height of the dividing surface, H_c, refer to Section 9.19.2 for more details. The critical dividing surface height is 20 m.

Table 9.7 summarises the dependence of the model behaviour on the maximum dividing surface height. Note that when modelling variable terrain height for very stable flows, the stable flow solution does not account for any spatial variation of surface roughness.

Complex terrain modelling option	$H_c > 20$ m	$H_c \leq 20$ m
Variable terrain height, with or without variable surface roughness	Very stable flow solution (assuming a constant roughness value)	FLOWSTAR-D solution
Variable surface roughness only	FLOWSTAR-D solution	FLOWSTAR-D solution

Table 9.7 – Summary of model behaviour when running the complex terrain option.

9.19.1 FLOWSTAR-D solution

In this case the flow field and turbulence values used in the dispersion modelling calculations are those output from the FLOWSTAR-D (F*) model. FLOWSTAR models air flow and dispersion over hills with roughness changes, including the effects of stratification. The model is described in some detail in Carruthers *et al*. (1988) while the basis of the theory is presented in Hunt *et al*. (1988b, 1988c) and Hunt (1985).

The airflow model uses linearised analytical solutions of the momentum and continuity equations, and includes the effects of stratification on the flow. The model is most accurate for hills of moderate slope and can typically be used for gradients up to about 1:2 but may not be reliable close to isolated slopes or escarpments with higher gradients or more generally if large parts of the modelling domain have slopes greater than 1:2. The influence of the terrain will vary with the source height and position and the local meteorology. The terrain height is specified at up to 770,000 points which are interpolated by the model onto a regular grid of up to 512×512 points. The best results are achieved if the points at which the elevation and/or surface roughness are specified are regularly spaced. The effects of roughness and terrain can be modelled separately. The airflow model has been extensively tested with laboratory and field data.

If spatially varying roughness is being modelled, values of the turbulence and flow fields below the local roughness height should be ignored as they will not be representative; the dispersion calculations never use values below 1.5 times the local roughness height.

9.19.2 Very stable flow conditions (variable terrain height, Hc>20m)

When very stable flows approach an isolated hill, the flow may divide, with the air above a certain height h_c (the *dividing surface*) flowing over the hill in a terrain-following manner and the air below the dividing surface flowing around the hill in a 2-dimensional flow. This is shown in **Figure 9.19**.

The maximum height of the dividing surface, H_c, is defined by an energy balance equation, which locates the lowest height at which the kinetic energy of an air parcel in the flow approaching the hill is equal to the potential energy attained by elevating an equivalent fluid parcel from this height to the top of the hill:

(9.48)
$$\frac{1}{2}U^2(H_c) = \int_{h_c}^{h_{max}} N^2(z) \times (h_{max} - z)\, dz .$$

The flow below the dividing surface is two-dimensional potential flow apart from upstream flow in a thin layer close to the terrain. Above the dividing surface, the FLOWSTAR-D solution is used except close to the ground where a weighted average of the FLOWSTAR-D and terrain following flow fields is used.

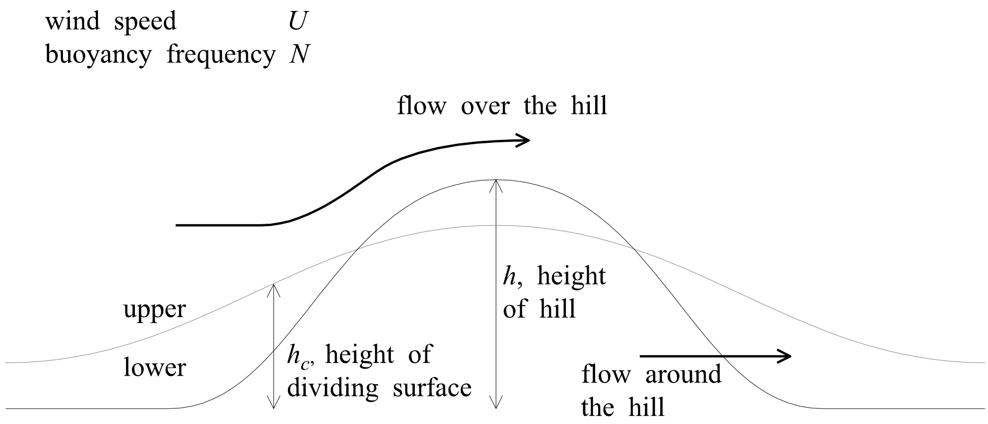


Figure 9.19 – Effect of a hill on the flow in very stable conditions. H_c is the maximum height of the dividing surface.

9.19.3 Regions of reverse flow

ADMS-Urban models plume dispersion for plumes starting in a reverse flow region. Regions of reverse flow occur when there is a component of the local velocity in the opposite direction to the upstream mean wind velocity. Regions of reverse flow sometimes occur in flows over complex terrain either upstream or downstream of hills, depending on the meteorological conditions and the terrain gradient. The flow field output from FLOWSTAR shows these regions of reverse flow.

If the source is in a region of reverse flow, then the pollutant becomes ‘well mixed’ within that region, i.e. it has a constant concentration, and the plume disperses out of the region from the downstream edge.

9.19.4 Running the complex terrain module

During a model run the following calculations take place:

- interpolation of terrain and roughness data (if variable) onto the local grid,
- calculation of mean air flow,
- calculation of shear stress and turbulence,
- calculation of mean streamlines through source,
- calculation of concentration distribution.

9.20 User-input 3D flow field

For this model option, the flow field and turbulence velocity values are used exactly as entered by the user. However, it was found that the turbulent kinetic energy (TKE) values in some mesoscale flow field datasets, which were used during validation of this module, were significantly lower than those usually used in ADMS-Urban. This may be because ADMS-Urban TKE includes some flow field effects such as convective eddies, which in the mesoscale data are shown in the vertical wind component rather than the turbulence.

For this reason, prior to use in the ADMS-Urban dispersion algorithms, it is possible to automatically adjust the user-input values of TKE ($TKE_{user}(x,y,z)$) to make them a similar order of magnitude to the ADMS-Urban TKE values ($TKE_{adjusted}(x,y,z)$). This is done in the following way:

(9.49)
$$TKE_{adjusted}(x,y,z) = \frac{\overline{U}_{user}(z)}{\overline{U}_{ADMS}(z)} \frac{\overline{TKE}_{ADMS}(z)}{\overline{TKE}_{user}(z)} TKE_{user}(x,y,z)$$

where $\overline{U}_{ADMS}(z)$ and $\overline{TKE}_{ADMS}(z)$ are the mean ADMS-Urban wind speed and TKE values at a particular height z, assuming neutral conditions, and $\overline{U}_{user}(z)$ and $\overline{TKE}_{user}(z)$ are the corresponding values calculated from the user-input data.

The ADMS-Urban turbulent velocities σ_u, σ_v and σ_w, and a local value of the frictional velocity u_{*local} are then derived from the adjusted TKE values. The way this is done depends on the stability of the upwind meteorological conditions. That is:

- for **stable/neutral conditions** ($h/L_{MO} \geq -0.3$):

  (9.50)
  $$(\sigma_u, \sigma_v, \sigma_w) = (2.5, 2.0, 1.3)\sqrt{\frac{2\,TKE_{adjusted}}{11.94}}$$

  and

  (9.51)
  $$u_{*local} = \begin{cases} \sqrt{\dfrac{2\,TKE_{adjusted}}{11.94 T_{wN}^2}} & for \quad -0.3 \leq \dfrac{H}{L_{MO}} < 1 \\ \sqrt{\dfrac{2\,TKE_{adjusted}}{11.94\left(1 - \alpha_s \dfrac{z_{min}}{h}\right)^{3/2}}} & for \quad \dfrac{H}{L_{MO}} \geq 1 \end{cases}$$

  where z_{min} is the lowest input height and the parameter α_s is used to represent the changing characteristics depending on whether conditions are ideal (i.e. no waves or instabilities) or disturbed (as defined in the Technical Specification document P09/01, CERC, 2025).

- for **convective conditions** ($h/L_{MO} < -0.3$), it has been assumed that the ratio (R) of the convective velocity scale w_* to the frictional velocity u_* for the user-input data is the same as that for the ADMS-Urban upwind meteorological data. Then, if

(9.52)
$$R_{upwind} = \frac{w_*}{u_*}$$

a local value of u_* can be defined in the following way:

(9.53)
$$u_{*local} = \sqrt{\frac{2\,TKE_{adjusted}}{R^2_{upwind}\left(0.6 + 0.4T^2_{wC}\right) + 11.94T^2_{wN}}}$$

where this expression is evaluated at the minimum user-input vertical height z_{min} and

(9.54)
$$T_{wC}(z_{\min}) = 2.1\left(\frac{z_{\min}}{h}\right)^{1/3}\left(1 - 0.8\frac{z_{\min}}{h}\right)$$

(9.55)
$$T_{wN}(z_{\min}) = 1 - 0.8\frac{z_{\min}}{h}$$

Then, the turbulent velocities σ_u, σ_v and σ_w can be defined in the following way:

(9.56)
$$\begin{aligned}\sigma_u^2 &= u^2_{*local}\left(0.3R^2_{upwind} + 6.25T^2_{wN}\right)\\ \sigma_v^2 &= u^2_{*local}\left(0.3R^2_{upwind} + 4.0T^2_{wN}\right)\\ \sigma_w^2 &= u^2_{*local}\left(0.4R^2_{upwind}T^2_{wC} + 1.69T^2_{wN}\right)\end{aligned}$$

Finally, note that the derived turbulent velocities σ_u, σ_v and σ_w are subject to a minimum value of turbulence that ranges between 0.01 and 0.2 m/s, depending on the user input minimum value of L_{MO}. This accounts for the fact that in urban areas (high minimum L_{MO}), where conditions never become very stable, there will always be some turbulence. Alternatively, the user may set a constant value of minimum turbulence, as described in Section 4.19.

9.21 Coastline

The coastline module models the case where the presence of a coastline or other thermal change has a large impact on surface concentrations over the land. This may occur during the day when there is advection of stably stratified air from the sea on to the heated land surface, with development of an internal convective layer (Venkatram, 1977) – see **Figure 9.20**. The module is invoked if the following conditions are all satisfied:

1. the sea is colder than the land,
2. meteorological conditions on land are specified as convective,
3. the wind is blowing onshore (i.e. from the sea to the land).

A release into the stable layer mixes only slowly but eventually it is advected into the convective layer and then is rapidly mixed down to the surface where large surface concentrations may occur.

The input required when modelling a coastline includes the usual meteorological data over the land and, in addition, the same parameters over the sea. These are estimated from the parameters over the land and the sea surface temperature (or sea-land temperature difference), which must be one of the variables in the data file. The rate of growth of the internal boundary layer is assumed to be independent of wind direction provided the air flows from sea to land.

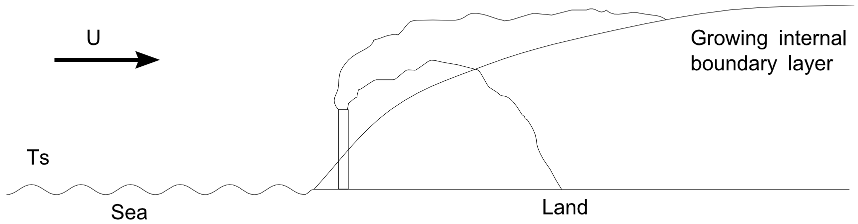


Figure 9.20 − Effect of coastline on plume.

9.21.1 Limitations

It is assumed that the coastline is straight near the location of interest.

The module has not been validated against field data.

9.22 Urban canopy flow

It is generally accepted that as wind approaches a built-up urban area, the vertical wind profile is displaced vertically by a height related to the mean height of the buildings, while the flow within the building canopy is slowed by the buildings (Belcher *et al.* 2013).

The ADMS-Urban urban canopy flow option incorporates above-canopy displaced flow and turbulence profiles linked to in-canopy profiles, allowing the flow field within urban areas to be characterised on a neighbourhood-by-neighbourhood basis. The urban canopy can be modelled as either spatially-varying or spatially homogeneous.

9.22.1 Input data

In the ADMS-Urban urban canopy module, the urban area is characterised by values of the average building height, the average street canyon width ($\hat{g}$), the fraction of the plan area occupied by buildings (λ_P) and, for a user-specified set of wind direction sectors, the ratio of the frontal area of buildings to plan area (λ_F).

For the spatially varying case, the urban canopy grid cell dimensions are set so that these parameters are able to represent the features of distinct urban neighbourhoods (typically 500 m to 2 km). Please refer to Section 4.18 for details of the input file format for urban canopy flow data.

The urban canopy module follows MacDonald et al. (1998) in calculating an effective roughness z_{0b} and a displacement height d from these input data.

9.22.2 Velocity profile

The full urban canopy vertical velocity profile consists of three sections as shown in **Figure 9.21**: above twice the displacement height (‘above-building’), below the displacement height (‘below-building’) and a transition region between these two regions.

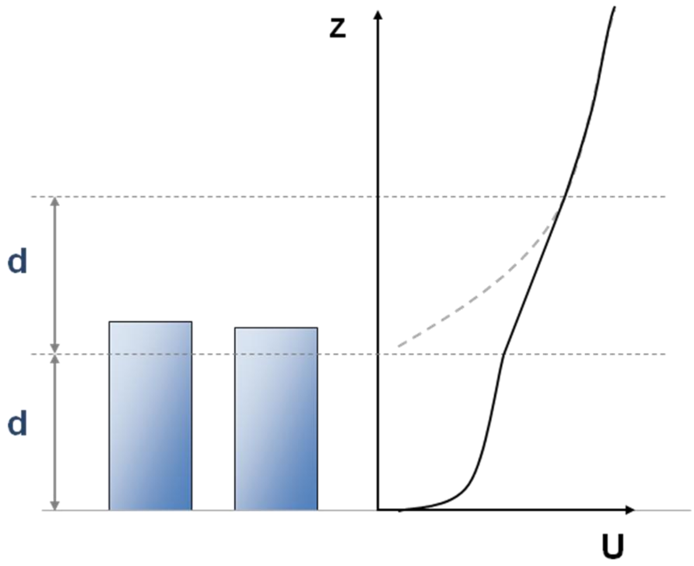


Figure 9.21 – Diagram of urban canopy vertical profile of velocity relative to displacement height *d*. The dashed line represents the standard velocity profile displaced upwards by *d*.

For the 'above-building' section, the standard ADMS velocity profile as described in the ADMS technical specification documents (CERC, 2025) is displaced upwards by *d* and a modified friction velocity is used, which is calculated using the local calculated roughness z_{0b}. Below the displacement height a logarithmic velocity profile is applied, with a 'below-building' roughness z_{0s} of 0.1 m, which represents the effect on the flow of small obstacles which may be present at street level. The effective friction velocity for the below-building flow is calculated by matching the below-building velocity at the displacement height to a fraction $(1 - \lambda_P)^2$ of the above-building velocity at twice the displacement height. A linear interpolation is performed in the transition region between the below-building velocity at the displacement height and the above-building velocity at twice the displacement height.

9.22.3 Turbulence profile

The full urban canopy vertical profile of turbulence consists of two sections: above and below the displacement height (see **Figure 9.22**). Above the displacement height, the standard ADMS stability-dependent turbulence profiles are displaced upwards by *d* and the modified friction velocity related to the local roughness z_{0b} is used. Below the displacement height, the turbulence velocities decay towards the ground according to $\exp(-(d - z)/2\hat{g})$, where *z* is the height above ground level.

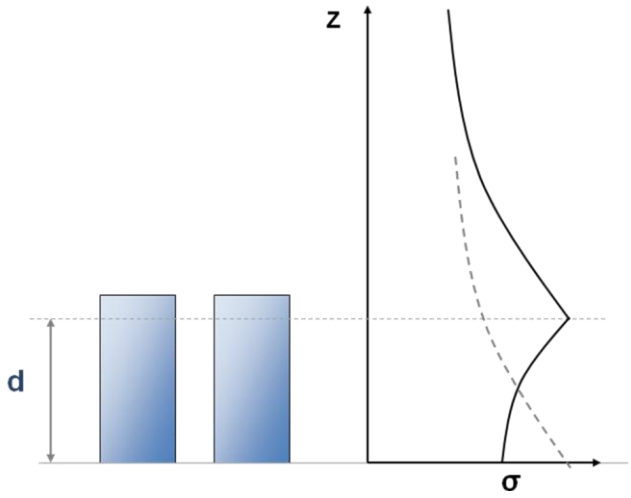


Figure 9.22 – Diagram of urban canopy vertical profile of turbulence relative to displacement height *d*. The dashed line represents the standard turbulence profile.

9.22.4 Flow regimes

The effects of the urban canopy module are divided into four regimes according to the values of z_{0b} and *d*. For the lowest values of *d*, below 1 mm, no urban canopy flow calculations are performed, while for the highest values of *d*, greater than the maximum of 2 m and half the average building height, full urban canopy flow calculations are performed as described below. Two intermediate regimes are defined: for low values of d, less than the maximum of 1 m and a tenth of the average building height, a 'standard' ADMS flow profile with the local roughness z_{0b} is used, known as the 'no displacement' solution; while for moderate values of *d* the flow is interpolated between the no displacement and full urban canopy profiles to give a 'low displacement' solution.

APPENDIX A Model Limits

The types of source available (Point, Line, Area, Volume, Road and Grid) are described in Section 3.2. The overall restrictions on the number of sources depend on the licence type and details for the standard licence types are given in **Table A.1**.

<table>
<tr><th colspan="2" rowspan="2">Source type</th><th colspan="4">Maximum number available in ADMS-Urban</th></tr>
<tr><th>Interface</th><th>Model
(Package A)</th><th>Model
(Package B)</th><th>Model
(Package C)</th></tr>
<tr><td colspan="2">Road</td><td>100000</td><td>150</td><td>600</td><td>3000</td></tr>
<tr><td>Point</td><td rowspan="4">Industrial</td><td rowspan="4">10000</td><td>3</td><td>25</td><td rowspan="4">1500</td></tr>
<tr><td>Line</td><td>3</td><td>5</td></tr>
<tr><td>Area</td><td>4</td><td>10</td></tr>
<tr><td>Volume</td><td>25</td><td>25</td></tr>
<tr><td colspan="2">Grid</td><td>1 (with up to 100000 cells)</td><td>1 (with up to 400 cells)</td><td>1 (with up to 3000 cells)</td><td>1 (with up to 3000 cells)</td></tr>
</table>

Table A.1 – Source limits in ADMS-Urban by licence type

The number of sources (of each type) permitted with your licence can be viewed by selecting **Licence details** from the **Help** menu, then clicking **Details**.

Other restrictions on the model output are as follows:

- The maximum number of user-defined groups that can be created and run in one model file is 20. Additionally a group containing all of the sources can be modelled.
- The maximum number of pollutants that can be entered into the pollutant palette, and emitted by any source, is 80. The maximum number of pollutants that can be included as output from the model is 25.

Table A.2 shows which model options can be used with each type of source, and with multiple sources.

Model option	Point source	Line, area or volume source	Road sources	Grid source
Deposition	✓	✓	✓	✓
Chemistry	✓	✓	✓	✓
Complex terrain	✓	✓	✓	[1]
Urban canopy	✓	✓	✓	[1]
Buildings	✓	[2]	[2]	[2]
Coastline	✓	[2]	[2]	[2]
Time-varying sources	✓	✓	✓	[3]
Odours	✓	✓	✓	✓

Table A.2 – Availability of model options with different source types.

1. A grid source and complex terrain or urban canopy flow (spatially varying) can be included in the same run but the effect of the complex terrain or urban canopy flow (spatially varying) will be ignored for the grid source. Spatially homogenous urban canopy flow will be taken into account when modelling a grid source.
2. Line, area, volume, road and grid sources and buildings or a coastline can be included in the same run but the effect of the buildings or the coastline will be ignored for these sources.
3. A single time-varying profile can be specified for a standard grid source. The use of hourly emission rates for a 3D grid source means that no time-varying factors are required for this source type.

Short-term output is available both with single and multiple source output. Line plotting output is only available with single source, short-term gridded output. Long-term output is not available with single source output; the **Groups** option must be selected on the **Output** screen if long-term output is required.

To calculate long-term output for a single source, create a group containing just this source and select the group on the ***Output*** *screen.*

All model options on the **Setup** screen can be used together, with following exceptions:

- **Odours** cannot be used with the **Chemistry** option. In addition, when the **Odours** option is selected, the **Calculate emissions using traffic flows** option for road sources cannot be used.

- **Coastline** cannot be used with the **Buildings** or **Complex terrain** option.

- **Spatial splitting** cannot be used with **Source exclusion by distance from receptors**.

Furthermore, the **Urban canopy flow** option cannot be used with the **Complex Terrain**, **Buildings** or **Coastline** options.

APPENDIX B NOx in ADMS-Urban

In ADMS-Urban, all NO_X data (emissions, background concentrations and output concentrations) in mass units are 'NO_X as NO_2'. This Appendix explains what this means and describes the implications for model users.

B.1 What is NOx?

NO_X is the name given to a mixture of two pollutants, nitrogen dioxide (NO_2) and nitrogen oxide (NO). Care must be exercised in specifying NO_X concentrations or emission rates in mass units (e.g. ng/m^3, µg/m^3, mg/m^3, g/s), since a specified volumetric concentration (for example in ppb) has a range of mass concentrations, depending on the relative proportions of NO and NO_2.

The conversion factor from ppb to µg/m^3 for any gaseous pollutant is M/24.06 (derived from the Ideal Gas Equation with an ambient temperature of 20°C and pressure of 1013 mb), where M is the molecular mass of the pollutant in grams. For example, the molecular mass of NO_2 is 46 g, so the conversion factor for NO_2 is 1.91. Similarly, the molecular mass of NO is 30 g, so the conversion factor for NO is 1.25. Therefore, for instance, 10 ppb NO_X could be 19.1 mg/m^3 if it is all NO_2, or 12.5 µg/m^3 if it is all NO, or somewhere in between for a mixture of NO and NO_2.

The following sections discuss how NO_X is treated in ADMS-Urban. This is especially important, as it is standard practice to express concentrations of NO_X in mass units as 'NO_X as NO_2'.

B.2 What does 'NOx as NO2' mean?

Pollutant levels can be measured in terms of volumetric concentration, and converted to a mass concentration using the Ideal Gas Equation, which is dependent on the molecular mass of the pollutant. Since NO_X is a composite pollutant, its effective molecular mass depends on the relative proportions of its constituents. This is often not known, so the assumption is often made in the conversion that NO_X has the molecular mass of NO_2; the resulting mass concentration is termed a '**NO_X as NO_2**' concentration.

More generally, the term 'NO_X as NO_2' refers to any mass emission rate or mass concentration of NO_X that has been calculated from a volumetric emission rate or concentration assuming that 100% of the NO_X is NO_2. Note that 'NO_X as NO_2' mass concentrations (and emission rates) generally overestimate the true situation, since the molecular mass of NO_2 (46 g/mol) is greater than the molecular mass of NO (30 g/mol).

When actual masses of each of NO and NO_2 are used to convert from volumetric to mass units, the NO_X concentration is referred to as '**True NO_X**' in this *User Guide*.

B.3 Treatment of NOx in ADMS-Urban

In ADMS-Urban, all NO_X data in mass units are ‘NO_X as NO_2’.

- NO_X emission rates must be input as ‘NO_X as NO_2’ values. If you have ‘true NO_X’ emission rate data, these must be converted to ‘NO_X as NO_2’ before being input to ADMS-Urban. Instructions for this conversion are given below (Section B.5).
- Background concentrations of NO_X entered into the model in mass units must be ‘NO_X as NO_2’. If background data have been obtained in the form of ‘True NO_X’, these must be converted to ‘NO_X as NO_2’, before being entered into the model. Instructions for these conversions are given below (Section B.5).
- NO_X concentrations output from ADMS-Urban in mass units are also ‘NO_X as NO_2’. When comparing ADMS-Urban results with, for example, measured concentrations, care should be taken that both sets of data are in the same form. ADMS-Urban output concentrations can be converted to ‘true NO_X’, if required, using the methods described in Section B.5 below.

B.4 NO2 emissions in ADMS-Urban

If the **Chemistry** option is used in ADMS-Urban, the relative proportions of NO and NO_2 in the emitted NO_X are required in order to calculate the effects of the chemical reactions. The user may input NO_2 emission rates in addition to NO_X emission rates. Alternatively, if NO_X emission rates are input to ADMS-Urban but no NO_2 emission rates are input, it is necessary for the model to make an assumption regarding the split between NO and NO_2. In this situation the percentage of emitted NO_X which is assumed to be NO_2 is dependent on the source type, as described in Section 4.13.2. For example, by default for industrial sources the proportion is 5%, i.e.

NO_2 emission rate = 0.05 * ‘NO_X as NO_2’ emission rate

B.5 Converting between ‘NOx as NO2’ and ‘true NOx’

This section describes how NO_X emission rates or concentrations in mass units can be converted from ‘true NO_X’ to ‘NO_X as NO_2’, and vice versa. In each case, it is just necessary to convert the NO part of the NO_X concentration. Therefore,

To convert from ‘true NO_X’ to ‘NO_X as NO_2’:

$$\text{‘}NO_X \text{ as } NO_2\text{’} = NO_2 + \frac{M_{NO_2}}{M_{NO}} (\text{‘true } NO_X\text{’} - NO_2)$$

where M_{NO_2} is the molecular mass of NO_2 (46 g/mol) and M_{NO} is the molecular mass of NO (30 g/mol). For example, an emission rate of 10 g/s ‘true NO_X’, of which 2 g/s is NO_2, is equivalent to 14.3 g/s ‘NO_X as NO_2’.

And to convert from ‘NO_X as NO_2’ to ‘true NO_X’:

$$\text{‘true } NO_X\text{’} = NO_2 + \frac{M_{NO}}{M_{NO_2}} (\text{‘}NO_X \text{ as } NO_2\text{’} - NO_2)$$

For example, a concentration of 10 μg/m^3 ‘NO_X as NO_2’, of which 5 μg/m^3 is NO_2, is equivalent to 8.26 μg/m^3 ‘true NO_X’.

Note that if you are converting an emission rate, and the emission rate of NO_2 is unknown, it will be necessary to make an assumption regarding the fraction of the NO_X emission rate that is NO_2. As noted above, if NO_2 emissions are not input and chemical reactions are modelled, ADMS-Urban assumes that a percentage of the ‘NO_X as NO_2’ emission rate is NO_2, with that percentage depending on the source type. Therefore, in order to be consistent with ADMS-Urban, the same assumption should be used for the conversion. For example for industrial source with a primary NO_2 of 5%, the conversion equation becomes

$$\text{‘}NO_X \text{ as } NO_2\text{’} = \frac{M_{NO_2}}{\left(0.95 M_{NO} + 0.05 M_{NO_2}\right)} \text{ ‘true } NO_X\text{’}$$

i.e.

$$\text{‘}NO_X \text{ as } NO_2\text{’} = 1.494 \text{ ‘true } NO_X\text{’}$$

Note also that to convert ‘NO_X as NO_2’ concentrations output from ADMS-Urban to ‘true NO_X’, the NO_2 concentration is required. To obtain this,

- If **Chemistry** is modelled, then select NO_2 on the **Output** screen in ADMS-Urban to calculate the NO_2 concentrations (selecting the same output units as for NO_X).
- If chemistry is not modelled, then it will be necessary to make an assumption regarding the split between NO and NO_2. This split will be dependent on the sources being modelled. In that case, the equation above can be used to calculate the ‘true NO_X’ concentration.

APPENDIX C   Emissions Inventory

Sections 3 and 4 of this User Guide have described how to set up ADMS-Urban model runs when there is a small amount of source data involved by typing the data into boxes in the interface. However, ADMS-Urban users will typically have a large volume of data, for instance, if they have compiled their own emissions inventory. Section 5 described one method of importing data into ADMS-Urban using comma-separated variable files, this section describes the legacy approach using the ADMS-Urban Emissions Inventory, which is a Microsoft Access database with a format such that the data can be directly imported into ADMS-Urban. The emissions inventory cannot be used when the **Odours** option is selected.

Selecting an ADMS-Urban Emissions Inventory file

You will probably want to work with several different inventories, for instance, where each inventory contains data for a different year or a different emissions-management scenario. You can create new, or change existing inventory files via the ADMS-Urban interface. In ADMS-Urban, select the **File**, **Emissions Inventory**, **Inventory Database** menu. This will open the screen shown in **Figure C.1** below. In order to:

- create a new inventory, click **New**.
- change to an existing inventory, click **Browse**.

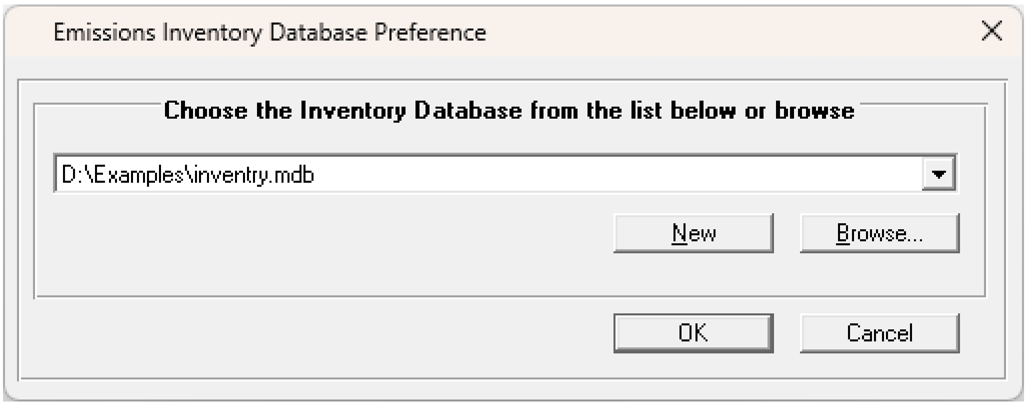


Figure C.1 – The **Emissions Inventory Database Preference** screen.

The screen in **Figure C.1** shows the path of the current Emissions Inventory *inventry.mdb*. To link ADMS-Urban to a different Emissions Inventory i.e. a different *.mdb* file, use the **Browse** button to locate the desired Emissions Inventory.

C.1 Transferring data between the Emissions Inventory and ADMS-Urban

Once data have been loaded into the Access Emissions Inventory they can be transferred into the ADMS-Urban interface. This process is two-way, i.e. it is also possible to enter data directly into the ADMS-Urban interface, and then to export the data from there into the ADMS-Urban Emissions Inventory. Both processes are described below.

C.1.1 Importing data from the Emissions Inventory into ADMS-Urban

To import data from the Emissions Inventory into the ADMS-Urban interface firstly select the correct Emissions Inventory, the one holding the data to be imported, by clicking **File**, **Emissions Inventory**, **Inventory Database** and browsing to locate the *.mdb* file of the Emissions Inventory.

Choose **File**, **Emissions Inventory, Import** from the ADMS-Urban menu. The **Import From Emissions Inventory (1: Pollutants)** dialogue box is displayed (**Figure C.2**).

Notice that this first import dialogue box lists the **Pollutants to Import** in the Emissions Inventory so that users can import the properties of the pollutants. If your Emissions Inventory does not contain any pollutant, no pollutants will be displayed and you should click **Next >** to proceed to the source data. If pollutants are displayed but you do not wish to use these pollutants properties, but instead wish to use those in the **Palette of Pollutants** in the ADMS-Urban interface, then click **Next >** to skip the import of pollutant properties.

If you *do* wish to import pollutant properties, you can choose which pollutants to import from the Emissions Inventory by clicking on the pollutant names in the **Emissions Inventory** list, and then clicking on the **< Add** button. The selected pollutants are added to the **Pollutants to import** list. To add all the listed pollutants, click on the **<< Add All** button.

To change your mind, click on a pollutant in the **Pollutants to import** list, and then click on the **Remove >** or **Remove All >>** button.

Click on the **Next >** button to display the second dialogue box (**Figure C.3**).

This second dialogue box lists the **Sources to import** in the Emissions Inventory. If there are no source data in the selected ADMS-Urban Emissions Inventory no sources will be listed in the right hand box.

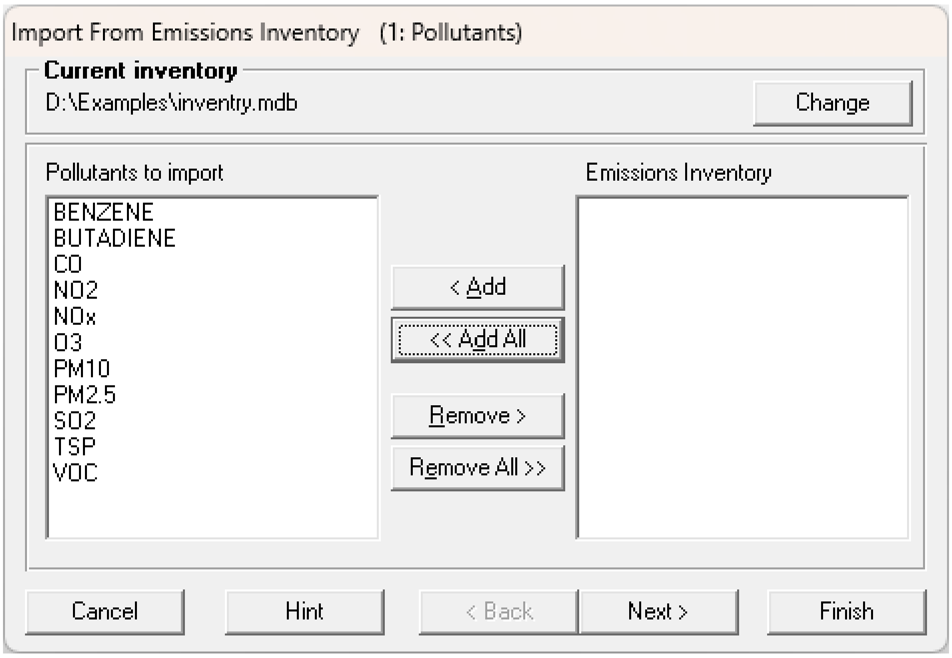


Figure C.2 – The Import From **Emissions Inventory (1: Pollutants)** screen.

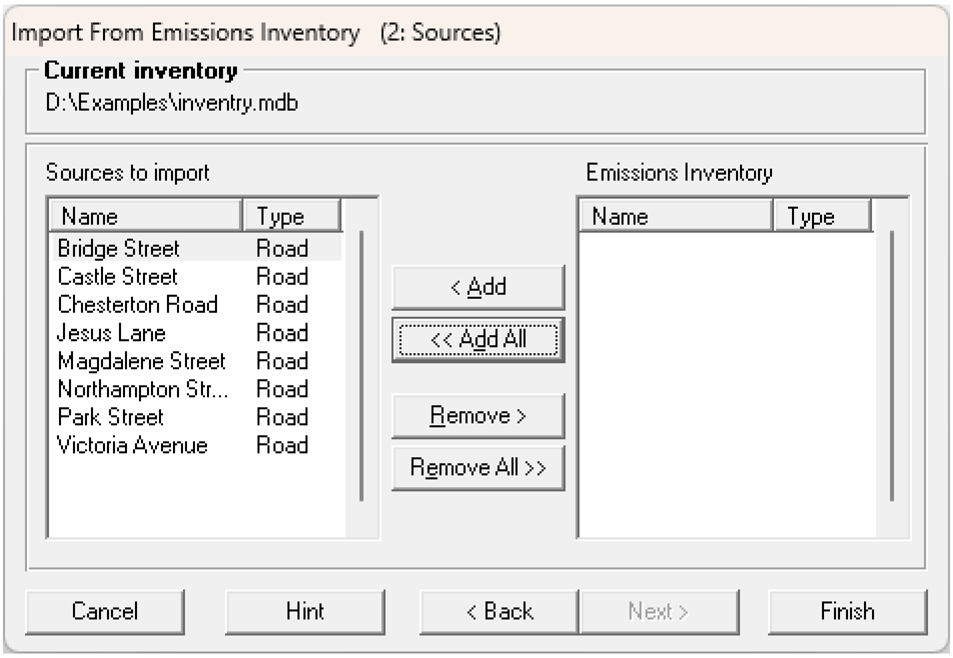


Figure C.3 – The Import From **Emissions Inventory (2: Sources)** screen.

To choose which sources to import from the Emissions Inventory, click on them in the **Emissions Inventory** list, and then click on the **< Add** button. They will then be added to the **Sources to import** list. To add all the listed sources, click on the **<< Add All** button. To select several sources that are listed successively, click on the first source name and then press down the **SHIFT** key and click on the last name. To select several sources that are not listed successively, click on the first source name and then press down the **CTRL** key and click on the other names. Then click **< Add** to add the selected sources to the **Source to import** list.

To change your mind, click on a source in the **Sources to import** list, and then click on the **Remove >** or **Remove All >>** button.

Clicking on the **Cancel** button at any time up until this point will stop the import operation and no data will be transferred. Click on the **Finish** button. The selected source and pollutant definitions are then imported from the Emissions Inventory into the ADMS-Urban interface.

The **Hint** button provides useful reminders at each step of the import process.

C.1.2 Exporting data from ADMS-Urban to the Emissions Inventory

It is also possible to transfer source data in the opposite direction to that described above, i.e. you can enter source data via the ADMS-Urban interface and then export it into the Access Emissions Inventory.

Any sources exported from the ADMS-Urban interface that are already defined in the Emissions Inventory will be updated. Any new sources will be added to the Inventory.

When you have entered the information into the ADMS-Urban interface that you wish to export, firstly select the correct Emissions Inventory. Click **File**, **Emissions Inventory**, **Inventory Database** and browse to locate the *.mdb* file of the Emissions Inventory.

Choose **File**, **Emissions Inventory**, **Export** from the ADMS-Urban menu. The **Export to Emissions Inventory** screen is displayed (**Figure C.4**). This screen indicates which source types can be exported.

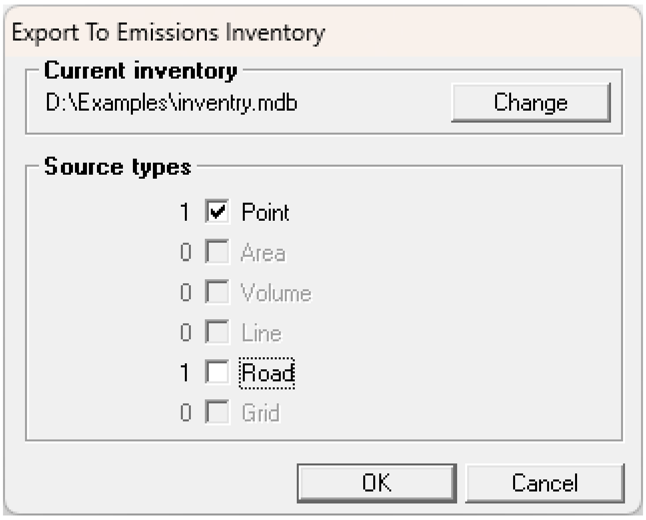


Figure C.4 – The **Export To Emissions Inventory** screen.

To choose which source types to add to the Emissions Inventory, check the box against that source type, all sources of that type will be exported.

Clicking on the **Cancel** button at any time up until this point will stop the export to database operation and no data will be transferred. Otherwise, click on the **OK** button. All sources of the selected types are now exported to the Emissions Inventory.

APPENDIX D Air Quality Limits and Guidelines

This section summarises air quality limits, guidelines and objectives. The limits are correct as of June 2025, but are subject to change. It is therefore important to ensure that you are referring to current limits.

The air quality limits, guidelines and objectives for the main pollutants are listed below as follows.

- Section D.1: UK Air Quality Strategy and Regulations.
- Section D.2: EU limit values.
- Section D.3: Lithuanian limit values
- Section D.4: French Limit Values
- Section D.5: US National Ambient Air Quality Standards.
- Section D.6: World Health Organisation guidelines.

Relationship between percentiles and exceedences

Many of the air quality limits are stated in terms of *"not more than N exceedences of a threshold value per year"*, where *N* is an integer. The limits can be restated in terms of percentiles by calculating the appropriate percentile from the maximum number of exceedences *N*, as described below. In ADMS-Urban, results can be calculated in terms of exceedences (in which case the threshold values are specified by the user on the **Output** screen) and/or percentiles (in which case the percentiles to be calculated are specified on the **Output** screen).

For example, the 2004 UK AQS particulates (PM_{10}) limit is *"not more than 35 exceedences per year of 50 mg/m³ as a 24-hour average"*. This means that 35 exceedences are acceptable, whereas 36 are not. The corresponding percentile to be calculated is therefore

$$\frac{365-35}{365}\times 100 = 90.41^{th} \text{ percentile}$$

where 365 is the number of 24 hour periods per year. If the limit is stated in terms of the hourly average, the percentile is calculated using

$$\frac{8760-N}{8760}\times 100$$

where 8760 is the number of 1-hour periods per year. Similarly for a 15-minute average, the number of 15-minute intervals per year is 35040, so the percentile to be calculated is

$$\frac{35040-N}{35040}\times 100.$$

D.1 UK Air Quality Strategy and Regulations

In the UK, concentrations of key pollutants in outdoor air are regulated by the Air Quality Standards Regulations[1,2,3,4,5,6].

Table D.1 lists UK air quality objectives for England, Wales, Scotland and Northern Ireland. The UK legislation website[7] is a useful source of information for the air quality regulations.

Pollutant	Region	Measured as	Limit, objective or target	Concentration[7]	Maximum no. of exceedences allowed	Exceedence expressed as percentile[10]
Benzene	UK	Running annual mean	Objective[8]	16.25 µg/m^3 (5 ppb)	-	n/a
	England and Wales	Annual mean	Limit[1-4]	5 µg/m^3 (1.54 ppb)	-	n/a
	Scotland and Northern Ireland	Running annual mean	Objective[8]	3.25 µg/m^3 (1 ppb)	-	n/a
1,3-Butadiene	UK	Running annual mean	Objective[8]	2.25 µg/m^3 (1 ppb)	-	n/a
Carbon monoxide (CO)	UK	Maximum daily running 8-hour mean	Limit[1-4]	10 mg/m^3 (8.6 ppm)	None	100th percentile
	Scotland	Running 8-hour mean	Objective[8]	10 mg/m^3 (8.6 ppm)	None	100th percentile
Lead (Pb)	UK	Annual mean	Limit[1-4]	0.5 µg/m^3	-	n/a
	UK	Annual mean	Objective[8]	0.25 µg/m^3	-	n/a
Nitrogen dioxide (NO_2)	UK	1-hour mean	Limit[1-4]	200 µg/m^3 (105 ppb)	18 times a year	99.79th percentile
	UK	Annual mean	Limit[1-4]	40 µg/m^3 (21 ppb)	-	n/a
Nitrogen oxides (NO_X)	UK	Annual mean	Critical level[1-4]	30 µg/m^3 (16 ppb) [11]	-	n/a

Table D.1 – UK air quality regulations.

Pollutant	Region	Measured as	Limit, objective or target	Concentration[7]	Maximum no. of exceedences allowed	Exceedence expressed as percentile[10]
Ozone (O_3)	UK	Maximum daily running 8-hour mean	Target[1-4]	120 μg/m³	25 days per year (averaged over 3 years)	-
	UK	Maximum daily running 8-hour mean	Long-term objective[1-4]	120 μg/m³	-	-
	UK	Maximum daily running 8-hour mean	Objective[8]	100 μg/m³	10 times a year	97.26th percentile
	UK	AOT40 calculated from 1 hour values from May to July, average over 5 years	Target[1-4]	18,000μg/m³ × h [11]	-	n/a
	UK	AOT40 calculated from 1 hour values from May to July	Long-term objective[1-4]	6,000μg/m³ × h [11]	-	n/a
Particulate matter (PM_{10})	UK	24-hour mean	Limit[1-4]	50 μg/m³	35 times a year	90.41th percentile
	UK	Annual mean	Limit[1-4]	40 μg/m³	-	n/a
	Scotland	24-hour mean	Objective[8]	50 μg/m³	7 times a year	98.08th percentile
	Scotland	Annual mean	Objective[8]	18 μg/m³	-	n/a
Particulate matter ($PM_{2.5}$)	UK	Annual mean	Limit[1-4]	25 μg/m³ [12]	-	n/a
	UK (except Scotland)	Annual mean	Objective[8]	20 μg/m³	-	n/a
	England	Annual mean	Target[5]	10 μg/m³	-	n/a
	Scotland	Annual mean	Objective[6]	10 μg/m³	-	n/a
Sulphur dioxide (SO_2)	UK	15-minute mean	Objective[8]	266 μg/m³ (100 ppb)	35 times a year	99.90th percentile
	UK	1-hour mean	Limit[1-4]	350 μg/m³ (132 ppb)	24 times a year	99.73th percentile
	UK	24-hour mean	Limit[1-4]	125 μg/m³ (47 ppb)	3 times a year	99.18th percentile
	UK	Calendar year and winter (1 October to 31 March)	Critical level[1-4]	20 μg/m³ (8 ppb) [11]	-	n/a

Table D.1 – UK air quality regulations. *(continued)*

Pollutant	Region	Measured as	Limit, objective or target	Concentration[7]	Maximum no. of exceedences allowed	Exceedence expressed as percentile[10]
PAHs (benzo[a]pyrene)	UK	Annual mean	Target[1-4]	1 ng/m^3	-	n/a
	UK	Annual mean	Objective[8]	0.25 ng/m^3	-	n/a
Arsenic	UK	Annual mean	Target[1-4]	6 ng/m^3	-	n/a
Cadmium	UK	Annual mean	Target[1-4]	5 ng/m^3	-	n/a
Nickel	UK	Annual mean	Target[1-4]	20 ng/m^3	-	n/a

Table D.1 – UK air quality regulations. *(continued)*

1. The Air Quality Standards Regulations 2010, UK Statutory Instruments 2010 no. 1001.
2. The Air Quality Standards (Wales) Regulations 2010, Wales Statutory Instruments, 2010 no. 1433 (W.126).
3. The Air Quality Standards (Scotland) Regulations 2010, Scottish Statutory Instruments, 2010 no. 204.
4. The Air Quality Standards Regulations (Northern Ireland) 2010, Northern Ireland Statutory Rules, 2010 no. 188.
5. The Environmental Targets (Fine Particulate Matter) (England) Regulations 2023, UK Statutory Instruments 2023, no. 96.
6. The Air Quality (Scotland) Amendment Regulations 2016, Scottish Statutory Instruments, 2016 no. 162. To be achieved by 31 Dec 2020.
7. https://www.legislation.gov.uk/
8. National air quality objectives available at https://uk-air.defra.gov.uk/air-pollution/uk-limits .
9. Conversions of ppb and ppm to µg/m^3 and mg/m^3 at 20°C and 1013 mb.
10. Percentile values are given for reference and are not defined explicitly in the air quality standards.
11. Objective for protection of vegetation and ecosystems.
12. 25 µg/m^3 is a cap to be seen in conjunction with 15% reduction.

D.2 EU limit values

The EU limit values[1] are listed in **Table D.2**.

Pollutant	Measured as	Limit concentration[2]	Maximum no. of exceedences allowed	Exceedence expressed as percentile[3]	To be achieved by
Benzene[1]	Annual mean	5 μg/m^3 [4]	-	n/a	11 Dec 2026
		3.4 μg/m^3 [4]	-	n/a	1 Jan 2030
Carbon monoxide[1] (CO)	Maximum daily running 8-hour mean	10 mg/m^3 [4]	None	100th percentile	11 Dec 2026
		10 mg/m^3 [4]	None	100th percentile	1 Jan 2030
	24-hour mean	4 mg/m^3 [4]	18 times a year	95.07th percentile	1 Jan 2030
Lead[1] (Pb)	Annual mean	0.5 μg/m^3 [4]	-	n/a	11 Dec 2026
		0.5 μg/m^3 [4]	-	n/a	1 Jan 2030
Nitrogen dioxide[1] (NO_2)	1-hour mean	200 μg/m^3 (104 ppb) [4]	18 times a year	99.79th percentile	11 Dec 2026
		200 μg/m^3 (104 ppb) [4]	3 times a year	99.97th percentile	1 Jan 2030
	24-hour mean	50 μg/m^3 (26 ppb) [4]	18 times a year	95.07th percentile	1 Jan 2030
	Annual mean	40 μg/m^3 (21 ppb) [4]	-	n/a	11 Dec 2026
		20 μg/m^3 (11 ppb) [4]	-	n/a	1 Jan 2030
Nitrogen oxides[1] (NO_X)	Annual mean	30 μg/m^3 [5]	-	n/a	

Table D.2 – EU limit values.

Pollutant	Measured as	Limit concentration[2]	Maximum no. of exceedences allowed	Exceedence expressed as percentile[3]	To be achieved by
Ozone[1] (O_3)	Maximum daily running 8-hour mean	120 μg/m^3 [4]	18 days per year (averaged over 3 years)	95.07th percentile	
	AOT40[6], calculated from 1-h values from May to July and averaged over 5 years	18000 μg/m^3 × h [5]	-	n/a	
	Maximum daily running 8-hour mean within one year	100 μg/m^3 [4]	3 days per year	99th percentile	
	AOT40[6], calculated from 1-h values from May to July	6000 μg/m^3 × h [5]	-	n/a	
Particulate matter[1] (PM_{10})	24-hour mean	50 μg/m^3 [4]	35 times a year	90.41th percentile	11 Dec 2026
		45 μg/m^3 [4]	18 times a year	95.07th percentile	1 Jan 2030
	Annual mean	40 μg/m^3 [4]	-	n/a	11 Dec 2026
		20 μg/m^3 [4]	-	n/a	1 Jan 2030
Particulate matter[1] ($PM_{2.5}$)	24-hour mean	25 μg/m^3 [4]	18 times a year	95.07th percentile	1 Jan 2030
	Annual mean	25 μg/m^3 [4]	-	n/a	11 Dec 2026
		10 μg/m^3 [4]	-	n/a	1 Jan 2030
Sulphur dioxide[1] (SO_2)	1-hour mean	350 μg/m^3 (130 ppb) [4]	24 times a year	99.73th percentile	11 Dec 2026
		350 μg/m^3 (130 ppb) [4]	3 times a year	99.97th percentile	1 Jan 2030
	24-hour mean	125 μg/m^3 (46 ppb) [4]	3 times a year	99.18th percentile	11 Dec 2026
		50 μg/m^3 (19 ppb) [4]	18 times a year	95.07th percentile	1 Jan 2030
	Annual mean	20 μg/m^3(7.4 ppb) [4]	-		1 Jan 2030
	Annual mean (calendar) and winter mean (1 October to 31 March)	20 μg/m^3 (7.4 ppb) [5]	-	n/a	

Table D.2 – EU limit values. *(continued)*

Pollutant	Measured as	Limit concentration[2]	Maximum no. of exceedences allowed	Exceedence expressed as percentile[3]	To be achieved by
Arsenic[1]	Annual mean	6 ng/m^3	-	n/a	
Cadmium[1]	Annual mean	5 ng/m^3	-	n/a	
Nickel[1]	Annual mean	20 ng/m^3	-	n/a	
PAHs (benzo[a]pyrene) [1]	Annual mean	1 ng/m^3	-	n/a	

Table D.2 – EU limit values. *(continued)*

1. Directive (EU) 2024/2881 of the European Parliament and of the Council of 23 October 2024 on ambient air quality and cleaner air for Europe.
2. Conversions of ppb and ppm to μg/m^3 and mg/m^3 at 20°C and 1013 mb.
3. Percentile values are given for reference and are not defined explicitly in the air quality standards.
4. Limit value for the protection of human health (see corresponding directive).
5. Critical level for the protection of vegetation and natural ecosystems (see corresponding directive).
6. AOT40 (see corresponding directive) means the sum of the difference between hourly concentrations greater than 40 ppb (80 μg/m^3) and 80 μg/m^3 over a given period (e.g. growing season) using only the hourly values measured between 8:00 and 20:00 Central European Time each day. It is expressed in μg/m^3 × hours.

D.3 Lithuanian limit values

The Lithuanian limits are identical to the EU limits (see Section D.2).

Below is a link to secondary Lithuanian national air quality standards for less-common regulated pollutants:

https://failai.gamta.lt/files/Nacionalines_oro_uzterstumo_normos.pdf

D.4 French Limit Values

The limit values for France[1] are listed in **Table D.3**.

Pollutant	Measured as	Limit concentration	Maximum no. of exceedences allowed	Exceedence expressed as percentile[2]
Benzene[3]	Annual mean	5 μg/m³	-	n/a
Carbon monoxide (CO)[3]	Maximum daily running 8-hour mean	10 mg/m³	None	100th percentile
Lead (Pb)	Annual mean	0.5 μg/m³	-	n/a
Nitrogen dioxide (NO_2)[3, 4]	1-hour mean	200 μg/m³ (104 ppb)	18 times a year	99.79th percentile
	Annual mean	40 μg/m³ (21 ppb)	-	n/a
Nitrogen oxides	Annual mean	30 μg/m³ (16 ppb)	-	n/a
Ozone[3,5] (O_3)	Maximum daily running 8-hour mean	120 μg/m³	25 days per year (averaged over 3 years)	-
	AOT40[6], calculated from 1-h values from May to July and averaged over 5 years	18000 μg/m³ × h	-	n/a
	Maximum daily running 8-hour mean within one year	120 μg/m³	None	100th percentile
	AOT40[6], calculated from 1-h values from May to July	6000 μg/m³ × h	-	n/a
Particulate matter[1] (PM_{10})	24-hour mean	50 μg/m³	35 times a year	90.41th percentile
	Annual mean	40 μg/m³	-	n/a

Table D.3 – French limit values.

Pollutant	**Measured as**	**Limit concentration**	**Maximum no. of exceedences allowed**	**Exceedence expressed as percentile**[2]
Particulate matter ($PM_{2.5}$)	Annual mean	25 $\mu g/m^3$	-	n/a
Sulphur dioxide (SO_2)[3, 7, 8]	1-hour mean	350 $\mu g/m^3$ (130 ppb)	24 times a year	99.73th percentile
	24-hour mean	125 $\mu g/m^3$ (46 ppb)	3 times a year	99.18th percentile
	Annual mean (calendar) and winter mean (1 October to 31 March)	20 $\mu g/m^3$ (7.4 ppb)	-	n/a
Arsenic	Annual mean	6 ng/m^3	-	n/a
Cadmium	Annual mean	5 ng/m^3	-	n/a
Nickel	Annual mean	20 ng/m^3	-	n/a
PAHs (benzo[a]pyrene)	Annual mean	1 ng/m^3	-	n/a

Table D.3 – French limit values. (*continued*)

1. Code de l'environnement Article R221-1, Modifié par Décret n°2010-1250 du 21 octobre 2010 - art. 1 accessible via www.legifrance.gouv.fr.
2. Percentile values are given for reference and are not defined explicitly in the air quality standards.
3. The volume must be reduced to conditions of temperature and pressure: 293 K and 101.3 kPa.
4. Alert concentration limit set to 200 or 400 $\mu g/m^3$ hourly mean (see corresponding directive).
5. Alert concentration threshold for progressive implementation of emergency measures:

   First level: 240 $\mu g/m^3$ average hourly exceeded for three consecutive hours;
   Second level: 300 $\mu g/m^3$ average hourly exceeded for three consecutive hours;

Third level: 360 μg/m^3 averaged hourly.

6. AOT40 (see corresponding directive) means the sum of the difference between hourly concentrations greater than 40 ppb (80 μg/m^3) and 80 μg/m^3 over a given period (e.g. growing season) using only the hourly values measured between 8:00 and 20:00 Central European Time each day. It is expressed in μg/m^3 × hours.
7. Objective of 50 μg/m^3 annual average.
8. Alert concentration limit set to 500 μg/m^3 hourly average exceeded for three consecutive hours (see corresponding directive).

D.5 US National Ambient Air Quality Standards

Table D.4 lists the US National Ambient Air Quality Standards (NAAQS) as given by the U.S. Environmental Protection Agency[1].

Pollutant	Primary or Secondary[2]	Measured as	Limit concentration[3]	Maximum no. of exceedences allowed	Exceedence expressed as percentile[4]
Carbon monoxide (CO)	Primary	1-hour mean	35 ppm (40 mg/m^3)	Once a year	99.99th percentile
	Primary	Running 8-hour mean	9 ppm (10 mg/m^3)	Once a year	99.99th percentile
Lead (Pb)	Both	Running 3-month mean	0.15 µg/m^3	-	n/a
Nitrogen dioxide (NO_2)	Both	Annual mean	53 ppb (28 µg/m^3)	-	n/a
	Primary	3-year average of the 98th percentile of the daily maximum 1-hour mean	100 ppb (52 µg/m^3)	-	n/a
Ozone	Both	3-year average of the fourth-highest daily maximum running 8-hour mean over each year	0.070 ppm (137 µg/m^3)	-	n/a
Particulate matter (PM_{10})	Both	24-hour mean	150 µg/m^3	Once a year (on average over 3 years)	-
Particulate matter ($PM_{2.5}$)	Both	3-year average of the 98th percentile of the 24-hour mean	35 µg/m^3	-	n/a
	Secondary	3-year average of the annual mean	15.0 µg/m^3	-	n/a
	Primary	3-year average of the annual mean	9.0 µg/m^3	-	n/a
Sulphur dioxide (SO_2)	Secondary	3-year average of the annual mean	10 ppb (4 µg/m^3)	-	n/a
	Primary	3-year average of the 99th percentile of the daily maximum 1-hour mean	75 ppb (195 µg/m^3)	-	n/a

Table D.4 – US National Ambient Air Quality Standards.

1. National Ambient Air Quality Standards (40 CFR part 50), U.S. Environmental Protection Agency. Available on https://www.epa.gov/criteria-air-pollutants/naaqs-table (see website for more detail).
2. Primary standards are defined for the protection of public health, whereas secondary standards are defined for the protection of public welfare, for example to minimise effects on visibility and vegetation.
3. mg/m^3 and μg/m^3 concentrations are approximately equivalent values.
4. Percentile values are given for reference and are not defined explicitly in the air quality standards.

D.6 World Health Organisation guidelines

The World Health Organisation (WHO) guidelines[1,2] are listed in **Table D.5**.

Pollutant	Measured as	Interim target				AQG level	Units	Maximum no. of exceedences allowed	Exceedence expressed as percentile
		1	2	3	4				
Carbon monoxide[1] (CO)	15-minute mean	-	-	-	-	100	mg/m^3	None	100th percentile
	1-hour mean	-	-	-	-	35	mg/m^3	None	100th percentile
	Running 8-hour mean	-	-	-	-	10	mg/m^3	None	100th percentile
	24-hour mean	7	-	-	-	4	mg/m^3	3 times a year	99th percentile
Lead[1] (Pb)	Annual mean	-	-	-	-	0.5	µg/m^3	-	n/a
Nitrogen dioxide[2] (NO_2)	1-hour mean	-	-	-	-	200	µg/m^3	None	100th percentile
	24-hour mean	120	50	-	-	25	µg/m^3	3 times a year	99th percentile
	Annual mean	40	30	20	-	10	µg/m^3	-	n/a
Particulate matter[2] (PM_{10})	24-hour mean	150	100	75	50	45	µg/m^3	3 times a year	99th percentile
	Annual mean	70	50	30	20	15	µg/m^3	-	n/a
Particulate matter[2] ($PM_{2.5}$)	24-hour mean	75	50	37.5	25	15	µg/m^3	3 times a year	99th percentile
	Annual mean	35	25	15	10	5	µg/m^3	-	n/a
Ozone[2] (O_3)	Daily maximum running 8-hour mean	160	120	-	-	100	µg/m^3	3 times a year	99th percentile
	Peak season[2]	100	70	-	-	60	µg/m^3	None	100th percentile
Sulphur dioxide[2] (SO_2)	10-minute mean	-	-	-	-	500	µg/m^3	None	100th percentile
	24-hour mean	125	50	-	-	40	µg/m^3	3 times a year	99th percentile
Cadmium[1] (Cd)	Annual mean	-	-	-	-	5	ng/m^3	-	n/a

Table D.5 – World Health Organisation guidelines.

1. WHO Regional Office for Europe, 2000: Air Quality Guidelines for Europe, Second Edition. WHO Regional Publications, European Series, No. 91. Available from www.euro.who.int.
2. WHO Regional Office for Europe, 2022: Air Quality Guidelines, Global Update 2021 − Particulate Matter, Ozone, Nitrogen Dioxide, Sulfur Dioxide and carbon monoxide. Available from www.euro.who.int/.
3. Average of daily maximum 8-hour mean O_3 concentration in the six consecutive months with the highest six-month running-average O_3 concentration.

APPENDIX E Surfer Tips

This appendix is intended only as a quick guide to some of the more frequently used features of Surfer, namely:

- how to create contour maps (2D),
- how to create surface maps (3D),
- how to overlay contours on digital map tiles or surface maps,
- how to customise a map,

This is not a comprehensive guide to all the features of the Surfer software package and it is recommended that users refer to the online help or to the Surfer user manual for further information. The information given in this section is based on Surfer 29; other versions of Surfer may differ.

E.1 Contour maps (2-dimensional)

Contour maps are useful to visualise gridded data such as concentrations, terrain data, etc.

Contour maps of concentrations should be produced using the **2-D Output Plotter (Plotting in Surfer)** utility (refer to Section 6.2 for details). Contour maps of other quantities can be produced as explained below, as long as the data format is XYZ ASCII, i.e. X, Y and Z coordinates in a text file.

There are two steps to creating a contour map:

1. create a grid (*.grd*) file from the original data,
2. make a contour map from the grid file.

Any contour map in Surfer is based on a grid file (.grd) that contains the data to be plotted, interpolated onto a regular grid.

Open Surfer and select **New Plot**, or select the **File**, **New**, **Plot document** menu option if Surfer is already open. This example shows how to create a contour map of terrain data.

Step 1 Click the **Grid Data** icon in the **Home** (or **Grids**) tab and open the terrain file.

You will need to choose **Files of type**: **All Files (*.*)** to be able to see *.ter* files and select the terrain file you want to use.

Step 2 On the **Data Import Options** screen, shown in **Figure E.1**, verify that the data will be read in as comma-delimited format and click on **OK**.

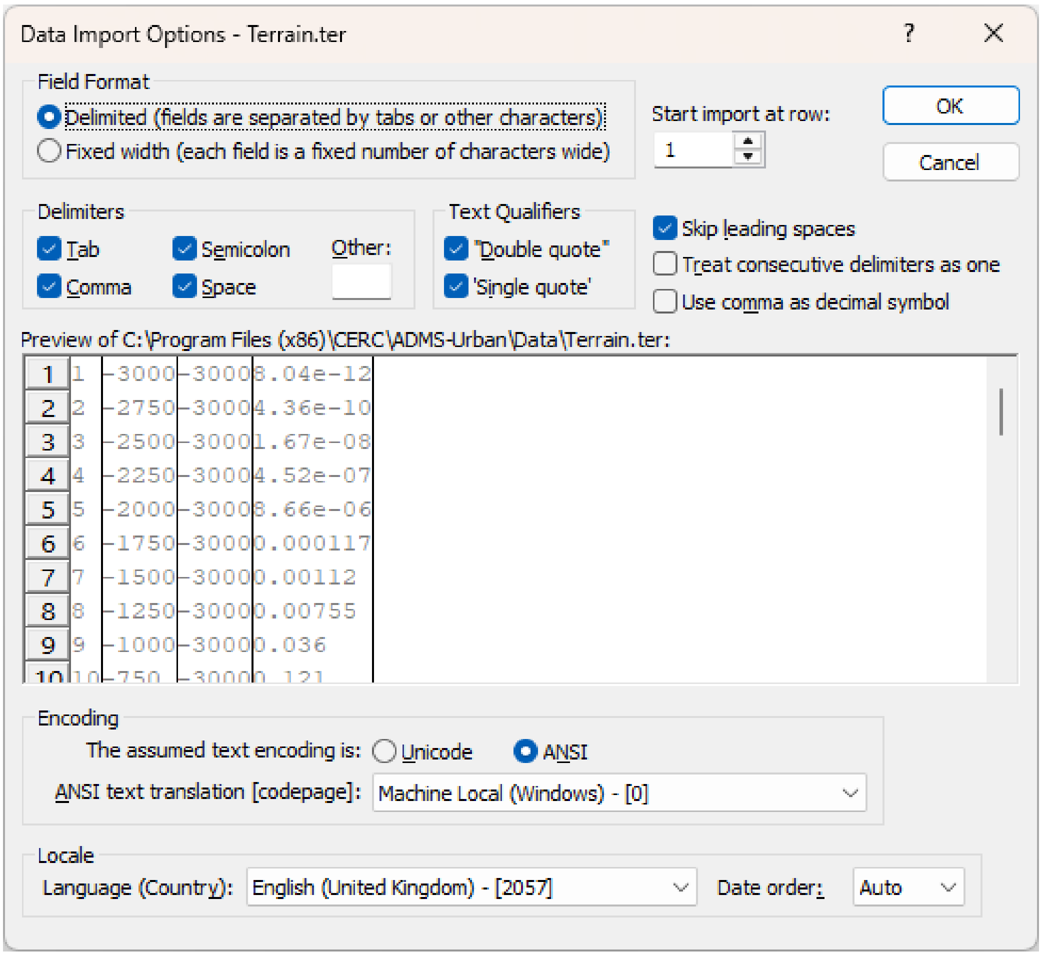


Figure E.1 – Data import options for the terrain file.

Step 3 On the **Grid Data** screen, shown in **Figure E.2**, select columns B, C and D corresponding to X, Y and Z respectively. The first column in the terrain file, column A, contains a counter that is not needed in the plotting.

In this screen you can change the grid file name or location.

Click on **Next>** to step through options relating to that gridding method in detail, or **Skip to End >>** to enter the resolution and output location, **Figure E.3**.

Click **Finish** to create the *.grd* file.

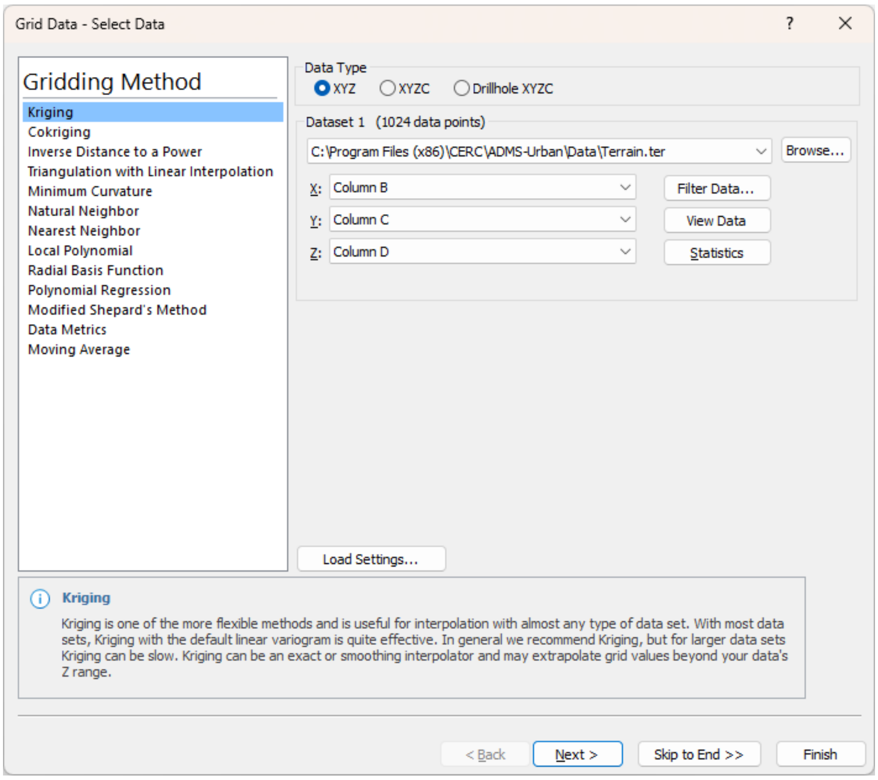


Figure E.2 – Select data settings for gridding the terrain file.

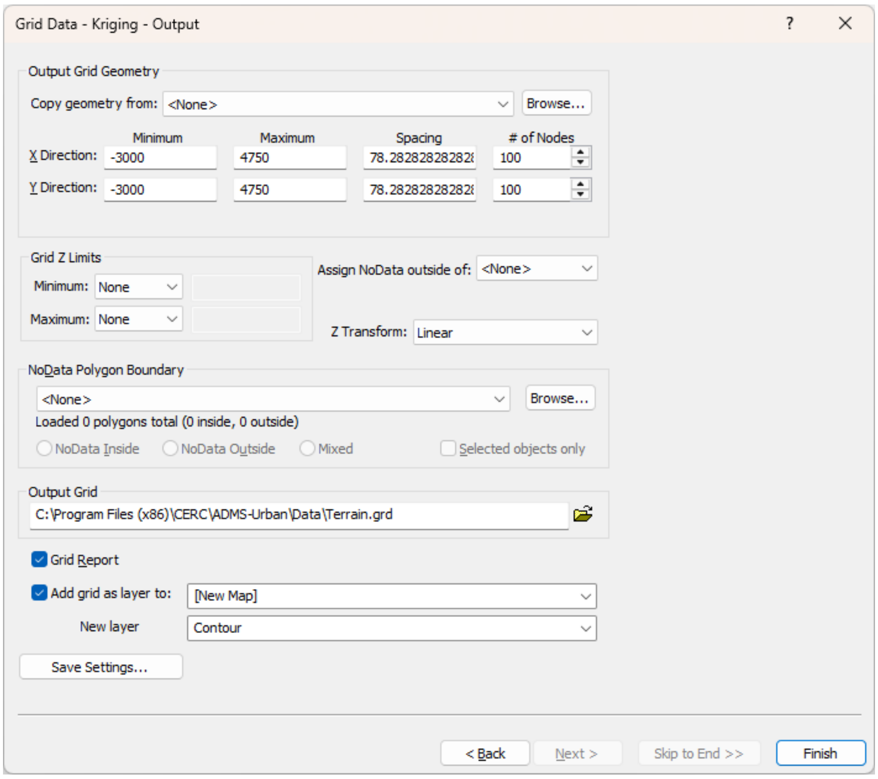


Figure E.3 – Output settings for gridding the terrain file.

Step 4 This should create a contour plot of the *.grd* file in the plot document, if not click the **Contour** icon, in the **New Map** section of the **Home** tab and then select the grid file just created.

The contour map is created with a default layout. Section E.4 describes how to customise the colours, labels and other features of a map.

An example of a final contour map is shown in **Figure E.4**.

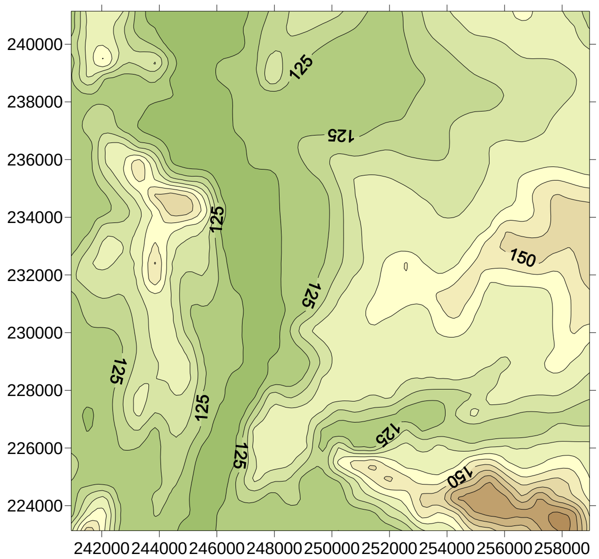


Figure E.4 – Example of a contour map of terrain data.

E.2 Surface maps (3-dimensional)

Creating a 3D plot, for instance from an ADMS-Urban terrain file (*.ter*), is an alternative way of visualising the terrain that can provide useful information as to which wind directions are likely to be most important in terms of dispersion modelling.

Open Surfer and select **New Plot**, or select the **File, New, Plot document** menu option if Surfer is already open.

As with the creation of contour maps, you first need to create a grid file (*.grd*) from your data. Create a *.grd* file as described in Section E.1. Click the **3D Surface** icon in the **Home** tab to create a 3-D surface map.

The remainder of this section describes how to create a contour map with contour lines (isopleths) from which a 3D map with isopleths is made.

Any surface map in Surfer is based on a grid file (.grd*) which contains the data to be plotted interpolated onto a regular grid.*

E.2.1 Create the grid file

In order to create both 2D and 3D maps, you will need a grid file of the data to be plotted. Refer to Section E.1 for instructions on how to create a grid file.

E.2.2 Create the 2D contour map

Once the grid file has been created, produce the contour map of the terrain data, as explained in Section E.1. Customise the map (see Section E.4).

The appearance of the final 3D surface map, in terms of contour levels, colours, etc. must first be set up at this stage in the 2D contour map.

For this example the final contour map of the terrain file is the same as that shown in **Figure E.4**.

E.2.3 Create the 3D surface map

Once the 2D contour map has been created and customised, open a new Surfer plot document (**File, New, Plot document**) and create the 3D surface map, from the same data, as follows.

Step 1 Open the terrain grid file after clicking the **3D Surface** icon.

A default 3D view of the terrain file is produced by Surfer as illustrated in **Figure E.5**.

This step could be achieved in the same Surfer plot as that of the 2D contour map. In such case, go directly to Step 3.

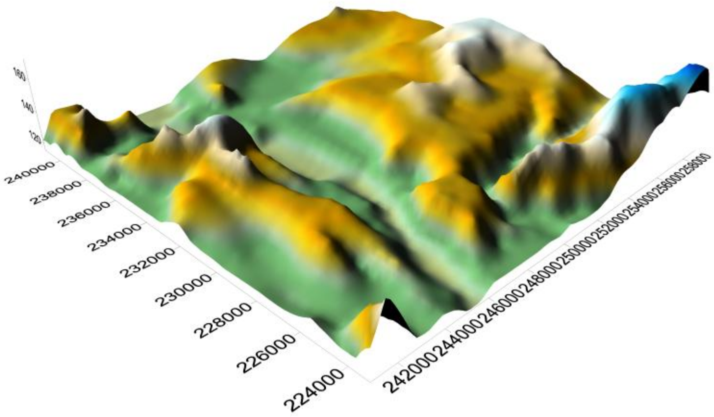


Figure E.5 – Default 3D view of the terrain file.

Step 2 To overlay the contour plot on the surface plot, go back to the contour plot, copy it (click the **Select All** and then **Copy** icons in the **Home** tab) and paste it onto the surface plot document (**Paste** icon in the **Home** tab).

Step 3 To set both plots on the same spatial scales, select and overlay them (**Select All** icon, followed by the **Overlay Maps** icon in the **Map Tools** tab).

Step 4 In order to use the colours of the contour plot only, click on **3D surface** in the **Contents** window. Then click on the **Overlays** tab in the **Properties** window. Click on the cell next to **Color modulation** and from the drop down menu select **Use Overlay color only**.

An example of a final 3D map of a terrain file is shown in **Figure E.6**.

Many elements of the plot can be modified. Please refer to Section E.4 for details of how to do this.

It is often useful to change the perspective of the map, add more details (level intervals), look at the coordinates to check that the required area is covered, etc. Click on **3D Surface** in the **Contents** window to access the **Properties** window for the 3D surface properties.

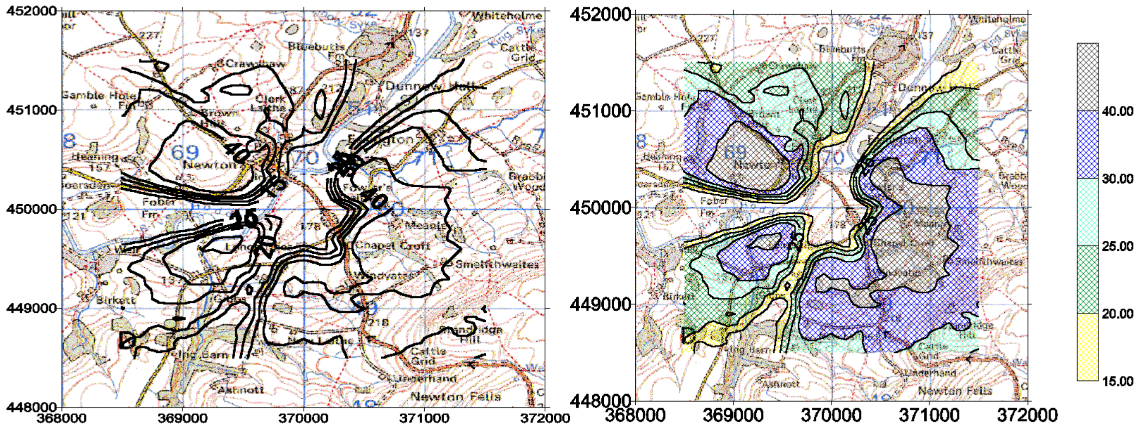
The colours of the surface map can be modified in the **General** tab by clicking on the colour ramp. X and Y grid lines can be overlaid (**Mesh** tab). Light and shadow effects are available in the **Lighting** tab. The **Overlays** tab controls how plots are to be overlaid. The 3D parameters of the plot can be changed by clicking on **Map** in the **Contents** window and then clicking on the **View** tab in the **Properties** window. The spatial scale and map extents are gathered in the **Scale** and **Limits** tabs.

Figure E.6 – Contour plot overlaid on the surface plot.

E.3 Overlay on a digital or surface map

Displaying standard contour maps of concentration, deposition, etc., on top of digital base maps or 3D surface maps allows attractive layouts for reports to be created without the need to purchase a Geographical Information System (GIS). **Figure E.7** shows examples of a digital map with either overlaid contour lines or colour-filled transparent contours. **Figure E.8** gives an example of site characteristics, such as the locations of the stacks and/or receptors, overlaid on a surface map.

The next section explains how to overlay an ADMS-Urban item (contour map of concentrations or site characteristics) on such maps. The same methodology can be used to overlay both types of data.



Reproduced from the Ordnance Survey 1:50,000 colour raster map
with the permission of The Controller of Her Majesty's Stationary Office © Crown copyright.

Figure E.7 – Digital map with contour lines overlaid (left) or colour-filled contours overlaid (right) produced with Surfer.

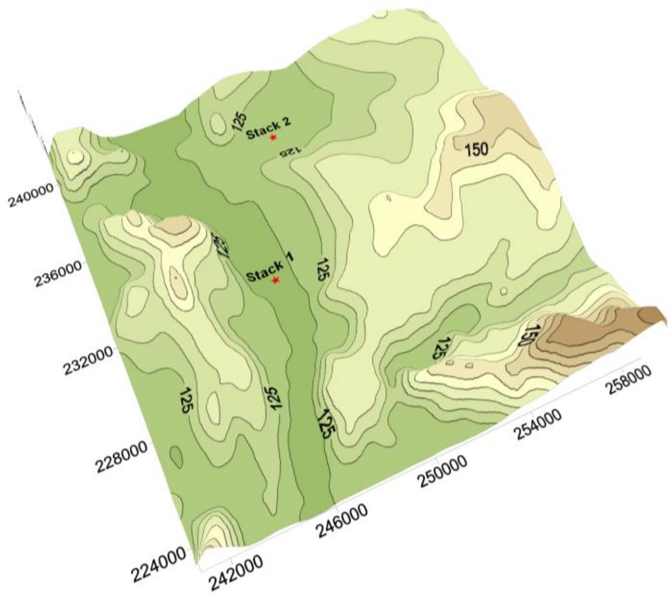


Figure E.8 – Surface map with overlaid site characteristics produced with Surfer.

To overlay an ADMS-Urban item on a map there are three steps:

1. prepare the map (digital base map or terrain surface map),
2. prepare the item to overlay (contour map of concentration, stack location, etc.),
3. merge both together to produce the final map.

The map and the item(s) to overlay should use the same coordinate system to ensure that the overlaid item is located at the correct geographical reference point on the map.

E.3.1 Prepare the map

The map can be any image, as long as it has appropriate geospatial information (e.g. uses a world file). Typically the base map will be a digital map (of the region of interest) or a surface map (showing the topography of the region).

Digital base map

Digital map data are available in a large number of graphical formats such as *.tif*, *.dxf*, etc. The tile position can be geographically referenced. For more information about appropriate formats of digital map data, please consult the Surfer user guide.

Step 1 Click the **Base** icon in the **Home** tab and locate the base map you want to use.

The map is loaded into Surfer as illustrated in **Figure E.9**. In this example, it covers approximately a 5 km × 5 km area. Depending on the extent and the resolution of your map, you might need to zoom into a small section to view the data in more detail and check that the tile is for the area you want.

Note that many different formats of base maps can be loaded, including AutoCAD drawings, bitmaps, Windows metafiles, etc.

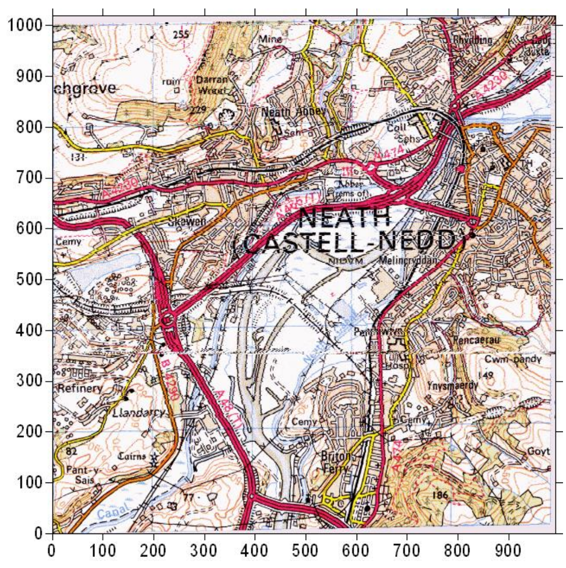


Figure E.9 – Base map loaded into Surfer.

Step 2 If the input base map does not contain geographical information, for instance through a world file, then the X and Y axes are automatically labelled relative to an origin (0,0) at the bottom left-hand corner. As previously explained, it may be necessary to reference the base map geographically to ensure the coordinate system matches that used by ADMS-Urban. To do this, select **Base** in the **Contents** window and click the **Georeference Image...** button in the **General** tab of the **Properties** window. Click the **Add Corner Points** button and change the **Target X** and **Target Y** values in the resulting table to the specify the actual minimum and maximum X and Y coordinates of the map. Close the window, choosing to save the changes and adjust the map limits, to apply the new coordinates to the map as shown in **Figure E.10**.

Step 3 Save this Surfer file.

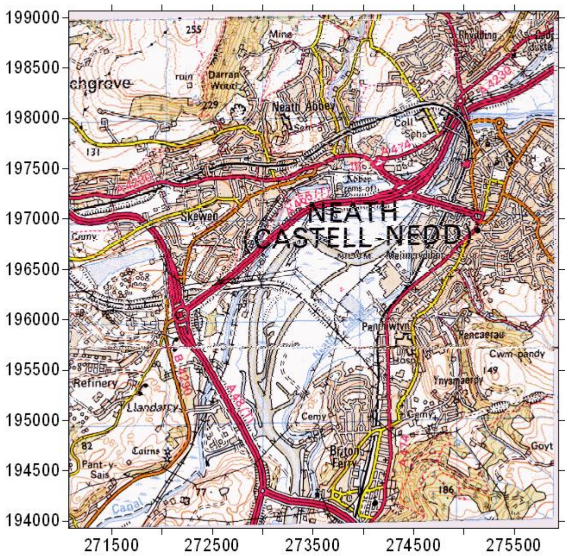


Figure E.10 – Synchronised map coordinates with ADMS-Urban coordinates.

Terrain surface map

A surface map of the terrain file can be created in the usual way (refer to Section E.2). When you are happy with its appearance, save it as a Surfer file.

It is always possible to modify the map appearance after an item has been overlaid on it. Just click on the surface map to access its appearance properties.

E.3.2 Prepare the item to overlay

Many different items can be overlaid on the map. The most commonly used are a contour map of concentrations and the locations of the stacks and receptors (if any). There is no restriction on the number of items that can be overlaid.

Each item to be overlaid should be prepared as explained below.

Contour map of concentration

Create the Surfer contour map in the usual way (refer to Section E.1), editing the levels, labels, fonts, etc., as detailed previously. Choose whether you require filled contours or just a series of contour lines, although this could be modified once the contour map has been overlaid on the map. When you are happy with the contour map save it as a Surfer file (*.srf*).

Site characteristics

In this example, we only consider two potential locations of the stack of an industrial site. The stack positions can be imported into Surfer by creating a *Post Map*, as follows.

Step 1 Go to **File, New** and select **Worksheet**, then enter the X and Y coordinates of the stack locations and, optionally, the stack names (see **Figure E.11**).

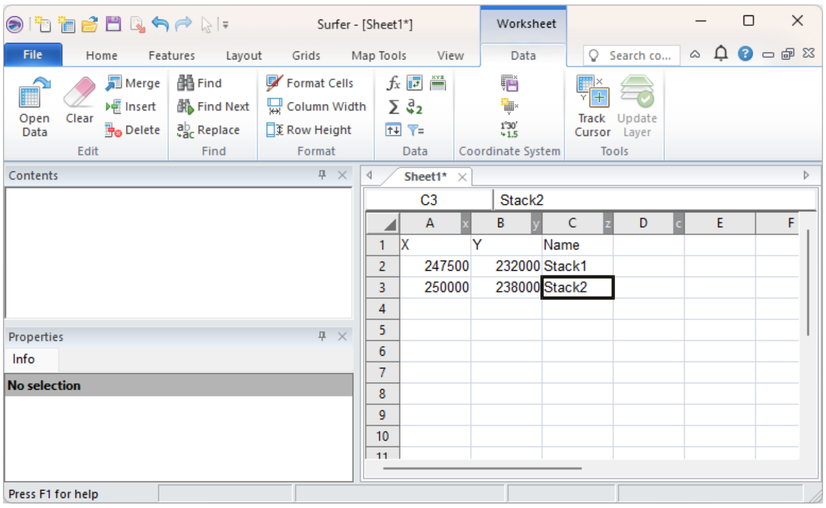


Figure E.11 – Stack locations.

Step 2 Save the file as **Text Data** (*.txt*). A **Data Export Options** dialogue window will appear. You can use this to select the **Delimiter** and the **Text Qualifier** before saving the file. Close the worksheet.

E.3.3 Overlay the item on the map

The items can be overlaid one at a time or with several items together.

Most of the appearance properties of the map and the item can be edited after the item has been overlaid on the map. The only exception to this is the coordinate system, which must be defined prior to the overlay step.

Overlay site characteristics

To overlay the stack locations on a map, proceed as follows.

Step 1 Open the digital base map or the terrain surface map previously saved.

Step 2 Click on the **Post** icon in the **Home** tab and select the text file previously created. This adds the stack locations to the map.

Step 3 To get the two images on the same spatial scale, click **Select All** followed by **Overlay Maps**. The stack locations are overlaid on the map.

Figure E.8 shows an example of the final 3D view of the terrain file with two possible locations of the stack of an industrial site.

You can edit the post map properties by clicking on **Post** in the **Contents** window. Several options are then available in the **Properties** window. The **Symbol** tab allows you to modify the stack symbol. In the **Labels** tab, you can choose to display the name of the stacks (if they are in the post map file) and set the font, colour and size to use to display the labels.

Overlay a contour map of concentration

To overlay the contour map of concentrations on a map, proceed as follows.

Step 1 Open the digital base map or the terrain surface map previously saved.

Step 2 Open the contour map you wish to overlay.

Step 3 Click on the contour map, click **Select All** and then **Copy**. Click on the digital base map, click **Paste**.

Step 4 To get the two images on the same spatial scale, click **Select All** followed by **Overlay Maps**.

If the contour map does not appear, it may lie below the map image. Select ***Base*** *in the* ***Contents*** *window and click the* ***Send to Back*** *icon in the* ***Layout*** *tab to set it as the bottom layer. The contour map will then be located on top of the base map.*

Figure E.12 shows an example of a contour map overlaid on a digital base map.

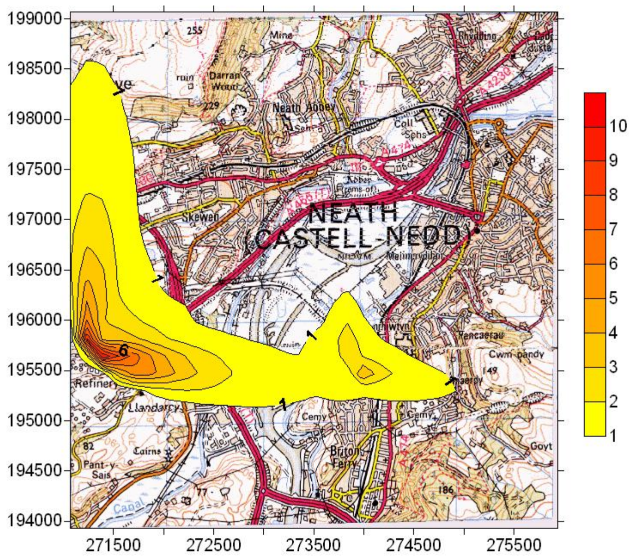


Figure E.12 – Contour plot of concentration overlaid on a base map.

Once the plot has been produced, you may wish to modify the map extent, change its scale, or modify the layout (colour-filled contours). If colour-filled contours are chosen, they should be made transparent so that the map underneath is visible. Refer to Section E.4 for an explanation of how to do this.

E.4 Customise a map

The easiest way to customise a feature present on a contour, surface or post map is to click on that feature in the **Contents** window (tick the **Contents** checkbox in the **View** tab if this window is not visible) to access the **Properties** window for it (similarly, tick the **Properties** checkbox if this window is not visible). Modifications can be applied to the map by editing the information in the various tabs of the **Properties** window.

This section contains explanations of how to edit and modify the most common features of the map (contour levels, colour, pattern and labels; line colour, thickness and pattern; colour scale; axes and grid lines; map scale and extent). The examples will use the contour map shown in **Figure E.13** and the contour map overlaid on the surface map shown in **Figure E.12**.

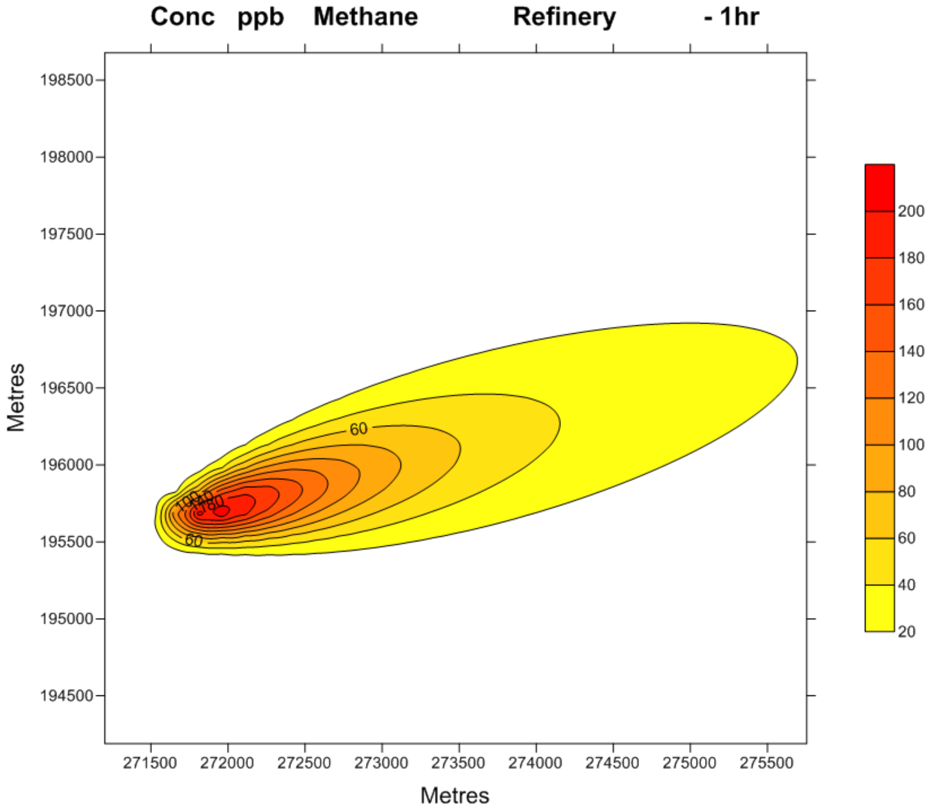


Figure E.13 – Example of contour plot of concentrations.

To access the properties of the contour map, click on **Contours** (or **ContourPlot** if created via the contour plotter) in the **Contents** window.

E.4.1 Levels

Levels can be edited by selecting the **Levels** tab in the **Properties** window for the contours. Simple modifications can be made by editing the values shown, but more advanced manipulation can be performed by clicking on the box next to the **Level method** item and selecting **Advanced** from the drop down menu. Clicking on **Edit Levels...** brings up a dialogue box for editing the levels properties, as shown in **Figure E.14**.

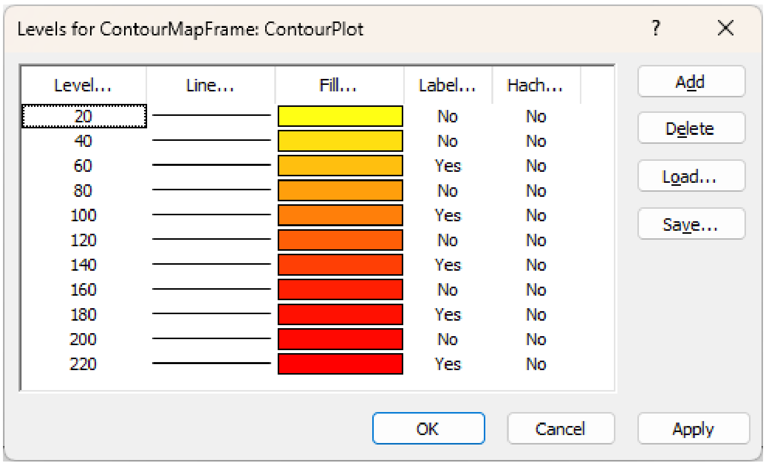


Figure E.14 – **Levels** editor screen.

The contour levels can be edited in three ways:

- *All existing levels together*: click on the **Level...** button on top of the level column to specify regular intervals. This may have changed the colours of the filled contours to a grey/black scheme but colour scheme and pattern can easily be changed.
- *One existing level at a time*: double-click on the value of the level to edit and change the value to the one you want.
- *Add a level between two existing ones*: to add a new contour level (e.g. 350 in **Figure E.14**), click on the next highest level (400 in this case) and then on the **Add** button. The new contour will be added, i.e. the halfway value between the one you clicked on (400) and the one below it (300).

Once you are happy with the contour levels that are shown in the contour plot, you can save them in a level file (*.lvl*) in order to re-use the configuration for other plots, for example plots you want to compare to each other. Click on **Save...** and save the level file with a unique name. This will not only save the contour levels but also the choice of colours, pattern of the filled contours, labels, line thickness and colour, label fonts, etc. (i.e. most of the parameters that can be edited in **Figure E.14**).

To apply an existing level file to a contour plot, select **Advanced** from **Level method** and click on **Edit Levels...** as described above to bring up the properties dialogue box. Click on the **Load...** button, and select the particular *.lvl* file required.

E.4.2 Contour colour and pattern

To edit the colour and pattern of contour maps, select **Edit Levels...** as described before to access the screen shown in **Figure E.14**. Click on the **Fill...** button at the top of the fill column to open the screen shown in **Figure E.15**.

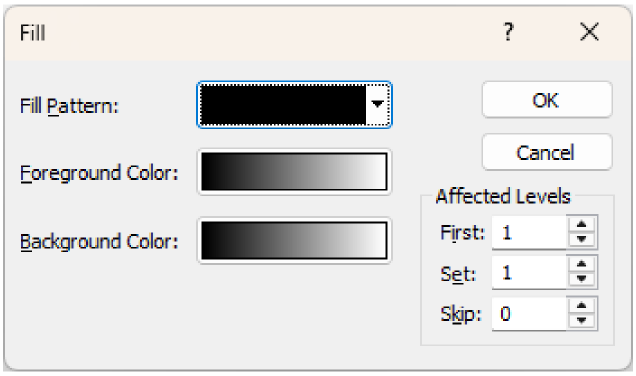


Figure E.15 – Setting the spectrum of colours and patterns for the contour plot.

By default, the pattern style is solid but other patterns such as lines, checks, etc., are available. For some patterns the choice will become available to make the pattern either **Opaque** or **Transparent**.

Click on the **Foreground Color** coloured box to access the screen shown in **Figure E.16**. In this **Colormap Editor** screen, you can choose the colours of the minimum and maximum contour levels as well as those for any intermediate level. The result will be a ramp of colours ranging from the minimum to the maximum colour (passing through intermediate level colours if specified).

---

*You must make sure that the **Fill Contours** box is checked in the **Levels** tab of the **Properties** window for the **Contours** layer in order for the colours to appear on the contour plot. If you just want a contour plot with labelled contour lines then leave this box unchecked.*

Alternatively, the user can specify a unique colour for every contour level. Similarly to the level values, each individual colour can be edited by double-clicking on the coloured box.

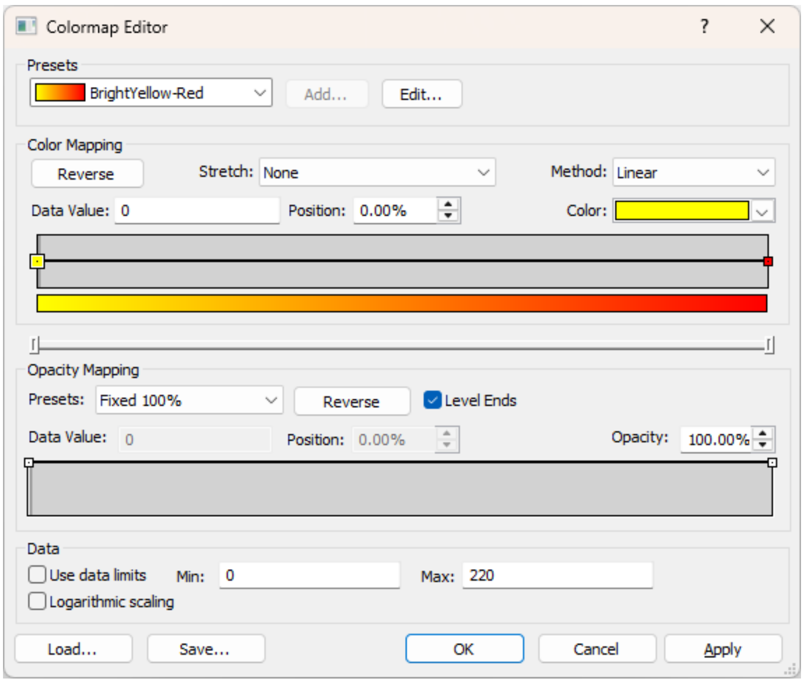


Figure E.16 – Choosing colours for filling contours.

Transparent colour-filled contours

Transparency is needed when you overlay a contour map of concentrations on a base map. Solid colour is used by default for the contours and the map underneath is not visible.

To set the transparency of colour-filled contours, select the contour layer in the **Contents** window, click on the **Layer** tab in the **Properties** window, and use the **Opacity** slide bar to set the transparency to the desired level.

Figure E.17 gives an example of a transparent filled-colour contour plot of concentration overlaid on a digital base map.

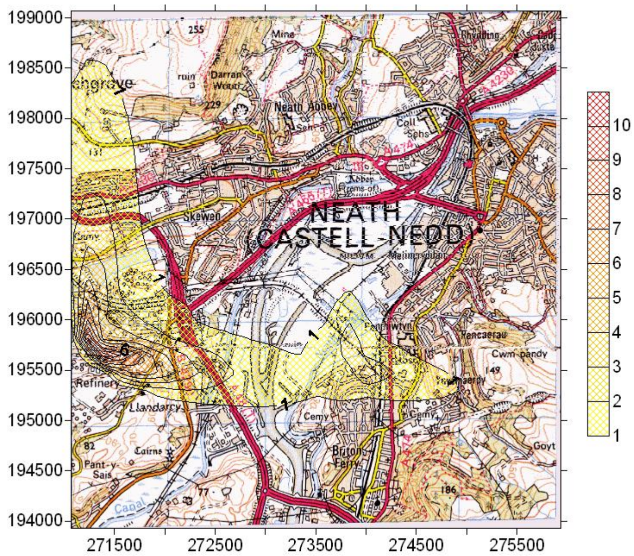


Figure E.17 – Example of transparent filled-colour contour plot of concentration overlaid on a digital base map.

E.4.3 Line colour, thickness and pattern

Colour, thickness and pattern of some or all the contour lines in the Surfer plot can be modified. To do so, in the levels editor screen shown in **Figure E.14**, click on the **Line...** button at the top of the line column to access the line properties. By default the lines are solid black lines but their colour, pattern and thickness can be modified. The lines can all be the same colour and thickness, they can ramp from minimum to maximum attributes or each have a unique colour and thickness.

E.4.4 Contour labels

To change the labelling format, in the levels editor screen shown in **Figure E.14** click on the **Label...** button at the top of the label column to access the properties of the contour labels.

The labelling frequency can be edited so as to label every line, every two lines, etc. The format and font of labels can also be adjusted: the size or colour changed, decimal digits added, exponential format selected (for example, for very small numbers) etc.

E.4.5 Colour scale

When a contour plot has filled-colour contours, it is useful to include a colour scale on the plot. This is included by default when contour plots are generated from the **2-D Output Plotter (Plotting in Surfer)** utility. For user-generated contour plots, the colour scale can be added to the plot by ticking the **Color Scale** checkbox in the **Levels** tab of the contour layer's **Properties** window. To change the format of these labels click on the colour scale bar on the contour plot to access its **Properties** window. The label format and font can be changed using the options in the various tabs.

E.4.6 Axes and grid lines

In order to access the properties of an axis, click on the axis to display the setting in the **Property Manager** window. A number of features can be edited here, such as the label format and angle, the ticks, the scale of the plot, and the grid lines.

It can be very useful to have grid lines at specific intervals on top of the contour map, and an example is shown in **Figure E.18**.

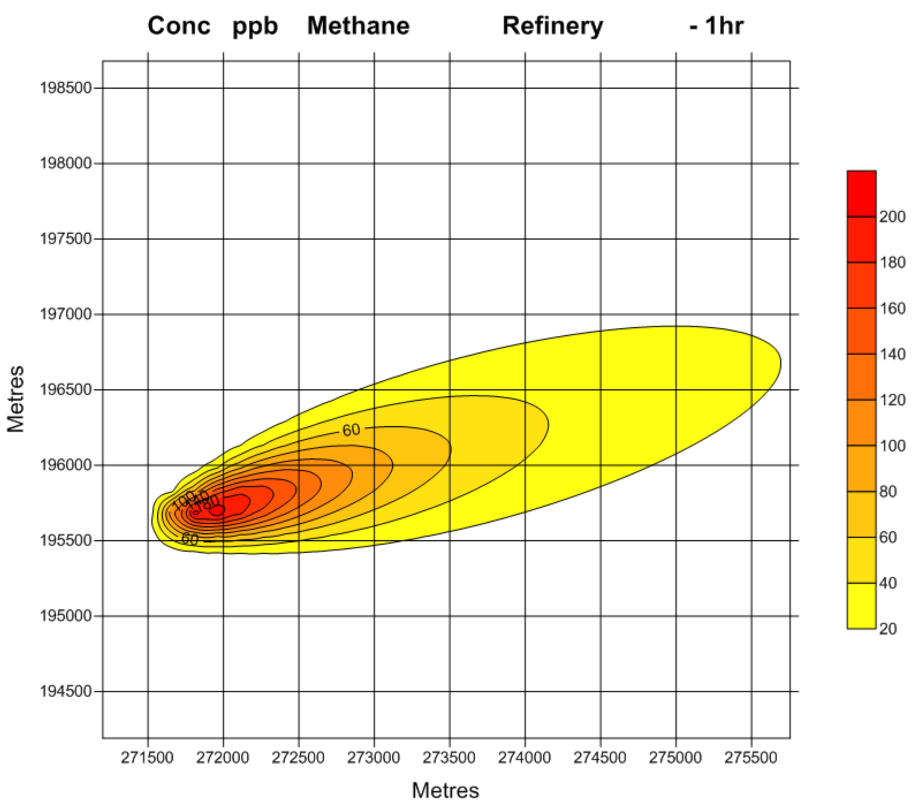


Figure E.18 – Use of grid lines superimposed on contour plot.

In the **Properties** window, click on the **Grid Lines** tab and check the **Show** check box of the **Major Grid Lines** (and **Minor Grid Lines** if you wish the minor grid lines to be shown too). The grid lines properties (style, colour, width) can be modified by clicking on the button showing the line. Major and minor grid line intervals are defined by the major interval of the axis label (**Scaling** tab) and the minor ticks options (**Ticks** tab).

E.4.7 Scale and extent

If you want the map to occupy more or less of the page then you can change the scale or stretch the map by dragging one of the corners. To modify the scale of a map, click on **Map** in the **Contents** window to view the **Properties** window for it and select the **Scale** tab. The **Proportional XY Scaling** option is useful to maintain the same scales in both X and Y directions.

When you open the ***Properties*** *window for the scale and extent, the units for the X and Y scales may be in map units per inch. To change this to centimetres, go to* ***File, Options*** *and change the* ***Page Units*** *under the* ***General*** *menu.*

Extent of a base map

It is often useful to make the base map extent a little larger than the extent of the contour map so that the surrounding land use can be viewed, or to zoom in on one particular area of the map.

To do this, click on **Map** in the **Contents** window to view the **Properties** window for it, select the **Limits** tab and indicate the new limits. Note that the scale is automatically at the scale of the original base map.

APPENDIX F Useful Contacts

This appendix contains information on:

1. ADMS-Urban: contact details, helpdesk, support contracts;
2. where to obtain input data for ADMS-Urban: meteorological, terrain and background data;
3. visualisation tools for ADMS-Urban output data: Surfer;
4. official organisations.

F.1 ADMS-Urban contact information

ADMS-Urban Helpdesk

Tel: +44 (0)1223 357 773 – ask for ADMS-Urban Helpdesk
Email: help@cerc.co.uk
Web site: https://cerc.co.uk

This service is available to those with valid support contracts (see below) between the hours of 9.30am and 5.00pm (GMT in winter, BST in summer), Monday to Friday (excluding UK public holiday periods and the period 25th December to 1st January inclusive).

Support contract

A valid support contract entitles the user to:

- use of the ADMS-Urban Helpdesk,
- model upgrades,
- access to further technical advice and downloads on the User Area of the CERC website https://www.cerc.co.uk/software-support/user-area.php,
- attendance at the annual ADMS-Urban User Group meeting,
- regular newsletters,
- participate in the user survey, which helps steer future model developments.

All period licence holders are entitled to support during the period of the licence. Permanent licence holders receive free support for one year, after which an annual support contract should be purchased.

Scope of the ADMS-Urban Helpdesk service

The scope of the ADMS-Urban Helpdesk Service is to provide answers to specific questions about using ADMS-Urban such as *"What do I select to model buildings effects?"*, to respond to any reported error messages which occur while running the model, and to record and report on any issues found. Where appropriate, CERC staff can also provide advice on setting up particular modelling scenarios and advice on interpreting the results.

Surfer problems should be directed to Golden Software.

F.2 Contact details for ADMS-Urban input data

Meteorological data

Meteorological data should be ordered in a format suitable for input into ADMS-Urban. Most ADMS-Urban users will require hourly sequential data for one or more years.

For the UK, meteorological data in ADMS format are available from a variety of suppliers, including the UK Met Office (https://www.metoffice.gov.uk). Hybrid datasets combining, for example, wind speed and direction from one location with cloud cover data from another may be appropriate; users are advised to discuss such options with their provider.

For France, meteorological data are available from Météo France (https://meteofrance.com). Supplied data include wind speed and direction measured at 10 m in height (for the last 10 minutes of each hour), temperature, cloud cover (note that it is usually measured only at the main station of each département, and that very often numerous night data are missing), and rainfall.

Terrain data and GIS mapping data

In order to model the effects of terrain on dispersion you will need to obtain terrain data. Many raster formats, e.g. OS terrain 50 or SRTM data, can be converted into the correct format for ADMS-Urban using the Mapper, refer to Section 6.1 of the Mapper User Guide for more details. For some older formats the ADMS Terrain Converter can be used, refer to Section 7.6 of this document for details. The ADMS Terrain Converter utility can convert the following formats:

- For sites in Great Britain, Landform PANORAMA or PROFILE data in NTF format.
- For sites in Northern Ireland, OS-NI (Ordnance Survey-Northern Ireland) data.
- For sites in France, BD ALTI 50, 75 or 100 (50, 75 or 100 m spatial resolution) data in XYZ ASCII format, provided by the Institut Géographique National (IGN).

Background data

Background concentration data for a wide range of sites in the UK are available from the UK National Air Quality Information Archive website

https://uk-air.defra.gov.uk/data/data_selector

Background data may also be available from local authorities.

F.3 Output visualisation tools

Surfer

Golden Software, LLC
PO Box 281
Golden
CO 80402-0281
U.S.A.

Tel: +1 (303) 279 1021 Email: info@goldensoftware.com or sales@goldensoftware.com

Web: https://www.goldensoftware.com

F.4 Official organisations

Department for Environment, Food and Rural Affairs (Defra)

Defra is responsible for the UK air quality strategy.

Web: https://www.gov.uk/government/organisations/department-for-environment-food-rural-affairs

Environment Agency (EA)

The Environment Agency regulates air quality in England.

Web: https://www.gov.uk/government/organisations/environment-agency

Scottish Environment Protection Agency (SEPA)

SEPA regulates air quality in Scotland.

Web: https://www.sepa.org.uk

Natural Resources Wales (NRW)

NRW regulates air quality in Wales.

Web: https://www.naturalresourceswales.gov.uk

Northern Ireland Environment Agency (NIEA)

The NIEA regulates air quality in Northern Ireland.

Web: https://www.daera-ni.gov.uk/articles/northern-ireland-environment-agency

European Environment Agency (EEA)

Web: https://www.eea.europa.eu

World Health Organization (WHO)

Web: https://www.who.int

APPENDIX G References

This is the main reference list for this User Guide, in particular Section 9.

Validation studies

The scientific modules contained in ADMS-Urban are almost identical to those used in the industrial air dispersion model ADMS 6, with the differences between the models being due to ADMS-Urban having been set up to specifically represent urban conditions. There are also differences in the number and type of sources available and the available model options. ADMS 6 has been widely validated and, in general, the validation papers available for ADMS 6 are also applicable to ADMS-Urban. The chemistry scheme used in ADMS-Urban is different to that used in ADMS 6 and has been validated separately.

ADMS 6 has been validated against a number of data sets in order to assess various configurations of the model such as flat or elevated terrain, line/area/volume sources and buildings. The model results have been compared to observational data or other model results if available. Documents and presentations containing results of ADMS 6 validation can be found on the CERC website https://www.cerc.co.uk.

The full ADMS 6/ADMS-Urban Technical Specification is available from the CERC website at the address https://cerc.co.uk/environmental-software/technical-specifications.html.

The section below lists published technical papers concerned with ADMS 6 and ADMS-Urban including validation studies.

Published technical papers

The following publications provide additional technical information on the ADMS-Urban model, its validation and application.

Belcher S.E, Coceal, O., Hunt, J.C.R., Carruthers, D.J., and Robins, A.G., 2013: A review of urban dispersion modelling. ADMLC report, available at https://webarchive.nationalarchives.gov.uk/ukgwa/20131102015843/http://www.admlc.org.uk/ADMLCReport7.htm (accessed April 2025)

Carruthers D.J., Edmunds, H.A., Lester, A.E., McHugh, C.A. and Singles, R.J., 2000: Use and validation of ADMS-Urban in contrasting urban and industrial locations. *Int. J. Environ. Pollut.* **14**, pp. 364-374.

Carruthers, D.J., Dixon, P., McHugh, C.A., Nixon, S.G. and Oates, W., 2001: Determination of Compliance with UK and EU Air Quality Objectives From High-Resolution Pollutant Concentration Maps Calculated Using ADMS-Urban. *Int. J. Environ. Pollut.* **16**, pp. 460-471.

Carruthers, D.J., Dyster, S. and McHugh, C.A., 2000: Contrasting methods for validating ADMS using the Indianapolis dataset. *Int. J. Environ. Pollut.*, **14**, pp. 115-121.

Carruthers, D.J., Dyster, S. and McHugh, C.A., 2003: Factors affecting interannual variability of NO_X and NO_2 concentrations from single point sources. *Clean Air and Environmental Protection*, **33**, pp. 15-20.

Carruthers, D.J., Edmunds, H.A., Bennett, M., Woods, P.T., Milton, M.J.T., Robinson, R., Underwood, B.Y. and Franklyn, C.J., 1995: Validation of the UK-ADMS Dispersion Model and Assessment of its Performance Relative to R91 and ISC using Archived LIDAR Data. Study commissioned by Her Majesty's Inspectorate of Pollution, published by DoE. DoE/HMP/RR/95/022.

Carruthers, D.J., Holroyd, R.J., Hunt, J.C.R., Weng, W-S., Robins, A.G., Apsley, D.D., Thomson, D.J. and Smith, F.B., 1994: UK-ADMS: a new approach to modelling dispersion in the Earth's atmospheric boundary layer. *J. Wind Eng. Ind. Aerod.*, **52**, pp. 139-153.

Carruthers, D.J., Holroyd, R.J., Hunt, J.C.R., Weng, W-S., Robins, A.G., Apsley, D.D., Smith, F.B., Thomson, D.J. and Hudson, B., 1991: UK Atmospheric Dispersion Modelling System. *Proceedings of the 19th NATO/CCMS International Technical Meeting on Air Pollution Modelling and its Application*, *September 1991*, *Crete*, *Greece*. Eds. Han van Dop and George Kallos, Plenum Publishing Corporation, New York.

Carruthers, D.J., McKeown, A.M. and McHugh, C.A., 1997: Air dispersion study of options for abatement of SO_2 emissions at Castle Cement, Clitheroe. *Environment Agency Report NW-9/97-25/B-BADQ.*

Carruthers, D.J., McKeown, A.M., Ellis, K.L. and McHugh, C.A., 1997: Air dispersion climate from the Castle Cement Works at Ribblesdale. *Environment Agency Report NW-7/97-20-BAZSQ.*

Carruthers, D.J., McKeown, A.M., Hall, D.J. and Porter, S., 1999: Validation of ADMS against wind tunnel data of dispersion from chemical warehouse fires. *Atmos. Environ.*, **33**, pp. 1937-1953.

CERC, 1998: Turbulence, Fluctuations and Averaging Times. CERC Technical Note, 22 December 1998.

CERC, 2025: ADMS Technical Specifications, available online at https://cerc.co.uk/environmental-software/technical-specifications.html

Dyster, S.J., Thomson, D.J., McHugh, C.A. and Carruthers, D.J., 2001: Turbulent Fluctuations and Their Use in Estimating Compliance Standards and in Model Evaluation. *Int. J. Environ. Pollut.*, **16**, pp. 57-68.

Eaton, B., Gregory, J., Drach, B., Taylor, K., Hankin, S., Caron, J., Signell, R., Bentley, P., Rappa, G., Höck, H., Pamment, A. and Juckes, M., 2011. NetCDF Climate and Forecast (CF) Metadata Conventions, Version 1.6. Available at https://cfconventions.org/cf-conventions/v1.6.0/cf-conventions.pdf

Fackerell, J.E. and Robins, A.G., 1982: Concentration fluctuations and fluxes in plumes from point sources in a turbulent boundary layer. *J. Fluid Mech.*, **117**, pp. 1-26.

Hood, C., Carruthers, D., Seaton, M., Stocker, J. and Johnson, K., 2014: Urban canopy flow field and advanced street canyon modelling in ADMS-Urban. Paper 67 in *Proceedings of the 16th International Conference on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Varna, 8-11 September 2014.*

Hunt, J.C.R., Holroyd, R.J., Carruthers, D.J., Robins, A.G., Apsley, D.D., Smith, F.B. and Thomson, D.J., 1990: Developments in Modelling Air Pollution for Regulatory Purposes. *Proceedings of the 18th NATO/CCMS International Technical Meeting on Air Pollution Modelling and its Application*, Vancouver, Canada.

Irwin, J.S., Carruthers, D.J., Stocker, J., and Paumier, J., 2003: Application of ASTM D6589 to evaluate dispersion model performance. *Int. J. Environ. Pollut.*, **20**, pp. 4-10.

McHugh, C.A., Carruthers, D.J., Higson, H. and Dyster, S.J., 1999: Comparison of Model Evaluation Methodologies with Application to ADMS 3 and US Models. Paper 58 in *Proceedings of the 6th International Conference on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Rouen, 11-14 October 1999*.

Pasquill, F. and Smith, F.B., 1983: Atmospheric Diffusion - Study of the dispersion of windborne material from industrial and other sources. Third edition. Ellis Horwood Ltd, Chichester.

Robins, A. and McHugh, C.A., 2001: Development and Evaluation of the ADMS Buildings Effects Module. *Int. J. Environ. Pollut.*, **16**, pp. 161-174.

Thomson, D.J., 1990: A stochastic model for the motion of particle pairs in isotropic high-Reynolds number turbulence and its application to the problem of concentration variance. *J. Fluid Mech.*, **210**, pp. 113-153.

Thomson, D.J., 1997: The 'inertial-meander' sub-range of a dispersing plume. *J. Appl. Meteorol.*, **36,** pp. 1046-1049.

Timmis, R., Wilkinson, S., Carruthers, D.J. and McHugh, C.A., 2000: Recent Studies to Validate and Compare Atmospheric Dispersion Models for Regulatory Purposes in the UK. *Int. J. Environ. Pollut.*, **14**, pp. 431-442.

Others

The following publications are references to papers and reports on dispersion relevant to the technical content of ADMS-Urban.

Briggs, G.A., 1969: *Plume Rise*. U.S. Atomic Energy Commission Div. Tech. Inf. TID-25075

Briggs, G.A., 1985: Analytical parameterizations of diffusion - the convective boundary layer. *J. Clim. Appl. Met.*, **14**, pp. 1167-1186.

Carruthers, D.J., Hunt, J.C.R. and Weng, W.-S., 1988: A computational model of stratified turbulent airflow over hills - FLOWSTAR I. *Computer Techniques in Environmental Studies*, pp. 481-492. Ed. P. Zanetti, Springer-Verlag.

Caughey, S.J. and Palmer, S.G., 1979: Some aspects of turbulence structure through the depth of the convective boundary layer. *Q. J. Roy. Meteor. Soc.*, **105**, pp. 811-827.

Cimorelli, A.J, Perry, S.G., Venkatram, A., Weil, J.C., Paine, R.J., Wilson, R.B., Lee, R.F., Peters, W.D., Brode, R.W. and Paumier, J.O., 2004: AERMOD Description of Model Formulation. United States Environmental Protection Agency, Office of Air Quality Planning and Standards, Emissions Monitoring and Analysis Division, Research Triangle Park, EPA-454/R-03-004.

CLAG, 1994: Critical Load of Acidity in the United Kingdom. Summary Report of the Critical Loads Advisory Group, Institute of Terrestrial Ecology, Penicuik, Midlothian.

Clarke, R.H., 1979: A model for short and medium range dispersion of radionuclides released into the atmosphere. NRPB Report NRPB-R91.

Davies, B.M., Jones, C.D., Manning, A.J. and Thomson, D.J., 2000: Some field experiments on the interaction of plumes from two sources. *Q. J. Roy. Meteor. Soc.*, **126**, pp. 1343-1366.

Derwent, R.G. and Middleton, D.R., 1996: An empirical function for the ratio NO_2:NO_X. *Clean Air*, **26**, 57-60

DfT 2009 Department for Transport road transport emission factors release (June 2009)

EN 13725:2003: Air Quality – Determination of odour concentration by dynamic olfactometry.

EFT v4.2, 2010, Defra and the Devolved Administrations emission factors release

EFT v5.1.3, 2012, Defra and the Devolved Administrations emission factors release

EFT v6.0.1, 2014, Defra and the Devolved Administrations emission factors release

EFT v7.0, 2016, Defra and the Devolved Administrations emission factors release

EFT v8.0, 2017, Defra and the Devolved Administrations emission factors release

EFT v9.0, 2019, Defra and the Devolved Administrations emission factors release

EFT v10.1, 2020, Defra and the Devolved Administrations emission factors release

EFT v11.0, 2021, Defra and the Devolved Administrations emission factors release

EFT v12.0, 2023, Defra and the Devolved Administrations emission factors release

EFT v12.1, 2024, Defra and the Devolved Administrations emission factors release

EFT v13.1, 2025, Defra and the Devolved Administrations emission factors release

Ginzburg, H. and Schattanek, G. (1997) Analytical Approach to Estimate Pollutant Concentrations from a Tunnel Portal Exit Plume. Presented at A&WMA 90th Annual Meeting, Toronto, Canada.

Hanna, S.R., Briggs, G.A. and Hosker, R.P., 1982: Handbook of Atmospheric Diffusion. US Dept of Energy Office of Scientific and Technical Information Publication DOE/TIC-22800.

Hanna, S.R. and Paine, R.J., 1989: Hybrid plume dispersion model (HPDM) development and evaluation. *J. Appl. Meteorol.*, **28**, pp. 206-224.

Hanna, S.R., Egan, B.A., Purdum, J. and Wagler, J., 1999: Evaluation of ISC3, AERMOD and ADMS dispersion models with observations from five field sites. Hanna Consultants Report P020. Fairfax, VA.

Hertel, O. and Berkowicz, R., 1989a: Modelling pollution from traffic in a street canyon. Evolution of data and model development. DMU Luft **A-129**. pp.77

Hertel, O. and Berkowicz, R., 1989b: Modelling NO2 concentrations in a street canyon. DMU Luft **A-131**. pp.31

Hertel, O. and Berkowicz, R., 1989c: Operational Street Pollution Model (OSPM). Evaluation of model on data from St Olavs Street in Oslo. DMU Luft **A-135**.

Hertel, O., Berkowicz, R. and Larssen, S. 1990: The Operational Street Pollution Model (OSPM). 18th International meeting of NATO/CCMS on Air Pollution Modelling and its Application, Vancouver, Canada, 741-749.

Highways Agency (1999) Design Manual for Roads and Bridges. Volume 11, Section 3, Part 1, Air Quality. The Stationery Office, ISBN 0 11 552129 1.

Highways Agency (2003) Design Manual for Roads and Bridges. Volume 11, Section 3, Part 1 - Air Quality. The Stationery Office.

Highways Agency (2015) Interim Advice Note 185/15, Updated traffic, air quality and noise advice on the assessment of link speeds and generation of vehicle data into 'speed-bands' for users of DMRB.

Holtslag, A.A.M. and van Ulden, A.P., 1983: A simple scheme for daytime estimates of the surface fluxes from routine weather data. *J. Clim. Appl. Meteorol.*, **22**, pp. 517-529.

Hunt, J.C.R., 1985: Turbulent diffusion from sources in complex flows. *Ann. Rev. Fluid Mech.*, **17**, pp. 447-458.

Hunt, J.C.R., Holroyd, R.J and Carruthers, D.J., 1988a: Preparatory studies for a complex dispersion model. CERC Report HB9/88.

Hunt, J.C.R., Leibovich, S. and Richards, K.J., 1988b: Turbulent shear flow over hills. *Quart. J. R. Met. Soc.*, **114**, pp. 1435-1470.

Hunt, J.C.R., Richards, K.J. and Brighton, P.W.M., 1988c: Stably stratified flow over low hills. *Q. J. Roy. Meteor. Soc.*, **114**, pp. 859-886.

Hunt, J.C.R., Stretch, D.D. and Britter, R.E., 1988d: Length scales in stably stratified turbulent flows and their use in turbulence models. *Proceedings IMA Conference on Stably Stratified Flow and Dense Gas Dispersion*, pp. 285-322. Ed. J.S. Puttock, Clarendon Press.

Lamb, R.G., 1982: Diffusion in the convective boundary layer. *Atmospheric Turbulence and Air Pollution Modelling*, pp. 159-230. Eds. F.T.M. Nieuwstadt and H. van Dop, D. Reidel, Dordrecht.

Lawson, R.E., Snyder, W.H. and Thompson, R.S., 1989: Estimation of maximum surface concentrations from sources near complex terrain in neutral flow. *Atmos. Environ.*, **23**, pp. 321-331.

Macdonald, R.W, Griffiths, R.F. and Hall, D.J., 1998: An improved method for estimation of surface roughness of obstacle arrays. *Atmos Environ.* **32**, pp. 1857–1864

Moore, D.J. and Lee, B.Y., 1982: An asymmetrical Gaussian plume model. Central Electricity Generating Board Report RD/L/2224N81.

Oke, T.R., 1987: Boundary layer climates. Second edition. Methuen.

Panofsky, H.A. and J.A. Dutton, 1984: Atmospheric Turbulence, Models and Methods for Engineers and Scientists. John Wiley and Sons, New York.

Perry, S.G., 1991: CTDMPLUS: A dispersion model for sources near complex topography. Part I: Technical formulations. *J. Appl. Meteorol.*, **31**, pp. 633-645.

Sehmel, G.A. 1980: Particle and gas dry deposition: a review. *Atmos. Environ.*, **14**, pp. 983-1011.

Singles, R.J., Sutton, M.A. and Weston, K.J., 1998: A multi-layer model to describe the atmospheric transport and deposition of ammonia in Great Britain. *Atmospheric Ammonia Special Issue* (Eds. Sutton, M.A., Lee D.S., Dollard G.J. and Folwer D.) *Atmos. Environ.* **32**, 393-399

Solera Garcia, M.A., Timmis, R.J., Van Dijk, N., Whyatt, J.D., Leith, I.D., Leeson, S.R., Braban, C.F., Sheppard, L.J., Sutton, M.A. and Tang, Y.S., 2017: Directional passive ambient air monitoring of ammonia for fugitive source attribution; a field trial with wind tunnel characteristics. In *Atmospheric Environment*, **167**, pp. 576-585. https://doi.org/10.1016/j.atmosenv.2017.07.043.

Snyder, W.H. and Lawson, R.E., Jr., 1991: Fluid modeling simulation of stack-tip downwash for neutrally buoyant plumes. *Atmos. Environ.*, **25A**, pp. 2837-2850.

Tsyro, S.G. 2001: Description of the Lagrangian Acid Deposition Model.

U.S. E.P.A., 1995a: User's Guide for the Industrial Source Complex (ISC3) Models. Volume I: User Instructions (Report EPA-454/B-95-003a) . U.S. E.P.A., Office of Air Quality Planning and Standards, Emissions Monitoring and Analysis Division, Research Triangle Park, NC 27711.

U.S. E.P.A., 1995b: User's Guide for the Industrial Source Complex (ISC3) Models. Volume II: Description of model algorithms (Report EPA-454/B-95-003b). U.S. E.P.A., Office of Air Quality Planning and Standards, Emissions Monitoring and Analysis Division, Research Triangle Park, NC 27711.

van Ulden, A.P. and Holtslag, A.A.M., 1985: Estimation of atmospheric boundary layer parameters for diffusion applications. *J. Clim. Appl. Meteorol.*, **24**, pp. 1194-1207.

Venkatram, A., 1977: Internal boundary-layer development and fumigation. In *Atmospheric Environment*, **11**, pp. 479-482.

Venkatram, A., Karamchandani, P., Pai, P. and Goldstein, R., 1994: The Development and Application of a Simplified Ozone Modelling System (SOMS). *Atmos. Environ.*, **28**, pp. 3665-3678.

Weil, J.C., 1985: Updating applied diffusion models. *J. Clim. Appl. Meteorol.*, **24**, pp. 1111-1130.

Wesseling, J. and Visser, G. 2003: Measuring and modelling of the A13 motorway near Overschie. *ArenA : een uitg. van de Vereniging van Milieukundigen*, **9,** pp. 159-162.

CERC

Cambridge Environmental Research Consultants Ltd
3 King's Parade, Cambridge, CB2 1SJ, UK
Tel: +44 (0)1223 357 773
Email: help@cerc.co.uk
Website: www.cerc.co.uk