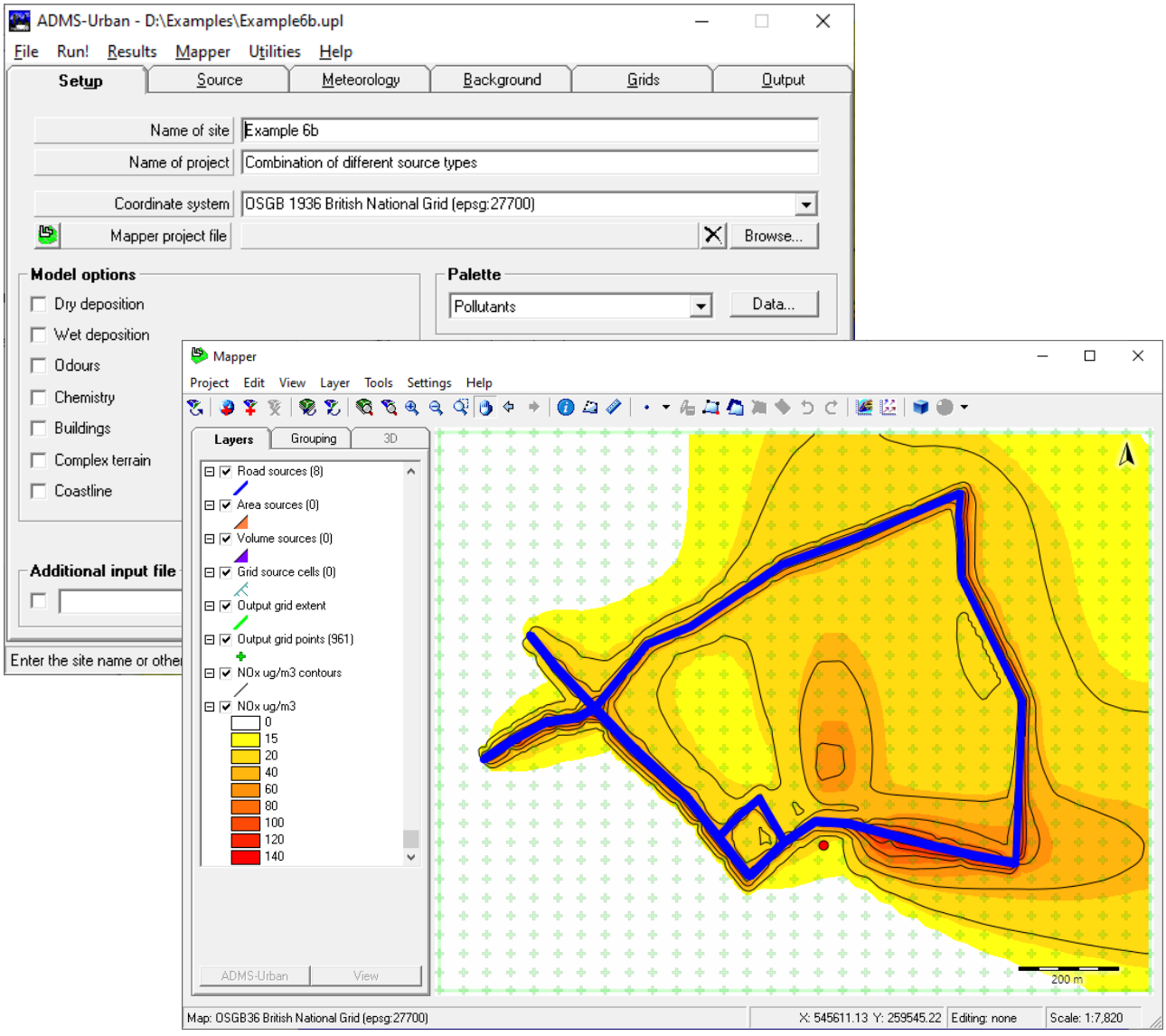
Mapper User Guide



User Guide CERC

Mapper

User Guide

Version 4.0

July 2025

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

Telephone: +44 1223 357773
Email: help@cerc.co.uk
Web site: http://www.cerc.co.uk

Contents

SECTION 1 Introduction to the Mapper

1.1 Overview of the Mapper

The Mapper is a visualisation and editing tool common to a number of CERC models, which allows model features to be created, edited and displayed graphically. The features you see in the Mapper vary depending on which model it is being used with; for example, some are available across all models and some are specific to ADMS 6, ADMS-Screen, ADMS-Urban, ADMS-Roads or ADMS-Airport.

The Mapper can be used to create, edit and display the following model features:

- Point, jet, line, area, volume and road sources,
- Buildings,
- Specified point receptors,
- Output grids.

Additionally,

- Grid sources and aircraft sources can be displayed,
- Specified points given in an **.asp* file can be displayed,
- Terrain and roughness files can be visualised,
- A background map can be added to the Mapper to assist in locating features,
- Model output can be displayed in the Mapper as contour or vector plots by using the 2D output plotter,
- A 3D visualisation of the features can be made,
- Data from raster layers in the Mapper can be extracted into a **.csv* file,
- Mapper layers can be exported into various file formats,
- Input files for use directly in the ADMS models' Advanced Street Canyon and Urban Canopy Flow modules can be created using the Canyon and canopy tool,
- Multiple **.shp* files can be combined into a single file using the merging tool.

Instructions on how to launch the Mapper are in Section 1.2.

Figure 1 shows the Mapper with the main features labelled. The menu options available from the menu are described in Section 1.3, the key interactions available are discussed in Section 1.4, right-click pop-up menus are described in Section 1.5 and the toolbar buttons in Section 1.6. The remaining parts of the Mapper, the layer panel and the map view window, are discussed in Sections 1.7 and 1.8 respectively. Instructions on how to set the coordinate system in the Mapper are given in Section 1.9.

Following this introduction, Section 2 describes how to set up and save a Mapper project file and edit features of the ADMS input using the Mapper; Section 3 explains how to view model results in the Mapper; Section 4 describes how to modify the

appearance of a layer; Section 5 deals with additional items such as importing background images, visualising input in 3D and exporting to Google Earth, together with other advanced features; finally SECTION 6 details the tools included in the Mapper for various applications such as extracting raster layer data and exporting layers into a file, among others.

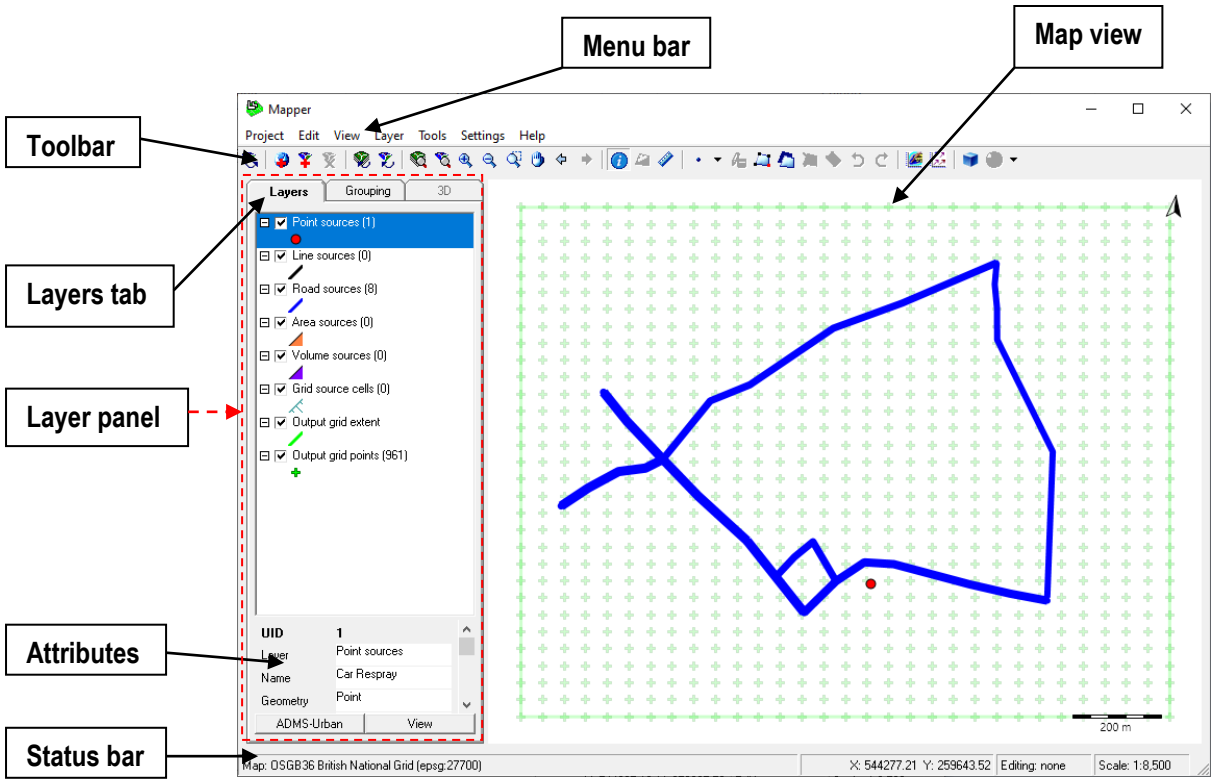


Figure 1 - The Mapper with the main features labelled.

1.2 Launching the Mapper

To launch the Mapper from the ADMS model interface, select the **Mapper** menu item or click on the  button on the **Setup** screen.

1.3 Menu options

There are six main menus: **Project**, **Edit**, **View**, **Layer**, **Tools** and **Settings**. The **Project** menu includes options to save, open and close a project; allows you to define whether the coordinates are part of a known projected or geographic system, e.g. Ordnance Survey OSGB coordinates, via the **Set map coordinate system** option – this is described further in Section 1.9; and to exit the Mapper. The **Edit** menu allows you to edit features in the **Sources**, **Buildings** and **Specified points** layers, which also applies any changes made to the model, and allows copying of images to the clipboard. The **View** menu provides options to change the map view. The **Layers** menu includes options to add, remove, export and

reorder the selected layer in the layer panel; to display the layer statistics; to clip the layer to polygons from another layer; and to use the layer coordinate system in the map view. The **Tools** menu launches utilities such as the Output Plotter, Extract data tool, Canyon and canopy tool and Shape file merging tool. The **Settings** menu includes options to edit preferences for viewing and gridding options; and to save the layer settings or restore the factory settings. Additionally, there is a **Help** menu. The menu options available are outlined in **Table 1**.

Menu	Option	Use
Project	**New**	Opens a new project.
	Open…	Open a previously saved project file.
	Save	Save the current project under the current file name.
	Save as…	Save the current project with a user-specified file name.
	Use the ADMS coordinate system	Sets the coordinate system used in the Mapper to the same as that defined in the ADMS interface.
	Set map coordinate system	This allows you to set the coordinate system to a projected or a geographic system, or to turn off the coordinate system.
	Close	Closes the Mapper.
Edit	**Add feature**	Add a feature to the selected layer, e.g. a point, a polygon etc.
	Edit feature	Edit the geometry of a feature in the selected layer.
	Shift feature	Move a feature in the selected layer.
	Delete feature	Delete the selected feature.
	Rotate a feature	Rotate the selected feature.
	Undo	Undo the current changes while editing.
	Redo	Redo the last Undo.
	Save edits	Saves the edits made to the model.
	Copy map to clipboard	Copies the current view in the map window to the clipboard.
	Copy legend to clipboard	Copies the top part of the layer panel to the clipboard.
	Copy scalebar to clipboard	Copies the scalebar to the clipboard.
View	**Zoom to layers**	Set the map view to show all the data in all the layers.
	Zoom to layer	Set the map view to the extent of the selected layer.
	Zoom in	Zoom in.
	Zoom out	Zoom out.
	Zoom	Zoom to display an area defined by clicking with the mouse on the map and dragging the cursor. A single left-click of the mouse on the map will make a fixed zoom in at the point clicked. A right-click will make a fixed zoom out.
	Previous extent	Return to the previously displayed extent in the map view.
	Next extent	Go to the next extent in the map view window.

Menu	Option	Use
	Pan	Move the map view without altering the scale.
	Refresh layers	Obtain the latest data from the model for all the layers and update the map view.
	Refresh layer	Obtain the latest data from the model for the selected layer and update the map view.
	Change view (2D/3D)	Toggle the map view between 2D and 3D mode.
Layer	**Add layer**	Allows a layer to be added, e.g. a background map.
	Remove layer	Removes the selected layer from the map view.
	Export layer…	Allows the selected layer to be exported to file in a choice of formats.
	Move to top	Moves the current layer to the top of the layer panel.
	Move to bottom	Moves the current layer to the bottom of the layer panel.
	Add WMS	Allows you to add a URL for a WMS layer.
	Add background map	Adds the background map to the map view window.
	Feature information	View information about a model feature in the map view window.
	Statistics	Displays statistics about the current layer. See Section 5.11
	Clip to polygon	Creates a new layer containing features from the current layer clipped to polygons from another layer. See Section 6.3
	Use CS for map	Sets the map coordinate system to that used by the current layer
Tools	**Measure**	Measure the distance between two or more points in the map view window.
	Contours	Launch the Output Plotter for concentration plots. See Section 3.1
	Flow field	Launch the Output Plotter for flow field plots. See Section 3.2
	Extract data from raster layers	Converts raster data to comma delimited file format. See Section 6.1
	Canyon and Canopy	Generates advanced street canyon and urban canopy flow parameter files for use in ADMS. See Section 6.4
	Merge SHP files	Combines multiple **.shp* files and allows editing of fields. See Section 6.5
Settings	**Snap to shape**	Option to snap to nearby shapes when adding and editing vertices.
	Display scalebar	Option to show/hide the scalebar.
	Display north arrow	Option to show/hide the north arrow.
	Gridding option	Allows selection of gridding method.

Menu	Option	Use
	Save all layer settings as default	Saves the current appearance of the layers (colour, symbol etc.) as the default appearance.
	Restore factory settings	Restores the default appearance of the ADMS layers.
Help	**User guide**	Opens the Mapper User Guide.
	About Mapper	Displays information about the Mapper, e.g. version number and CERC contact details.

Table 1 - Menu options in the Mapper.

1.4 Keyboard & mouse interactions

There are several keyboard interactions for functions in the Mapper, as listed in **Table 2**.

Action	Function
Layer panel	
Delete	Pressing the delete key while a user layer is selected in the layer panel will delete that layer.
Pan and zoom	
Shift – Mouse click	Holding down the shift key temporarily changes the mouse interaction with the map to pan, allowing you to move the map view.
Mouse wheel	Rolling the mouse wheel back and forth zooms the map view out and in
Ctrl – Mouse click	When in zoom mode, holding down the control key whilst using the mouse to zoom by click-and-drag will interactively scale the map image to indicate the amount of zoom (in or out) that will be applied when you release the mouse.
Editing	
Delete	Deletes the currently selected shape
Escape	Cancels/reverts current edits/changes
Enter	Completes current edits and deselects the shape, but doesn't save edits to the ADMS interface
Double click	
Information	
AltGr	Ensures that only features in the currently selected layer can be clicked

Table 2 - Key interactions in the Mapper.

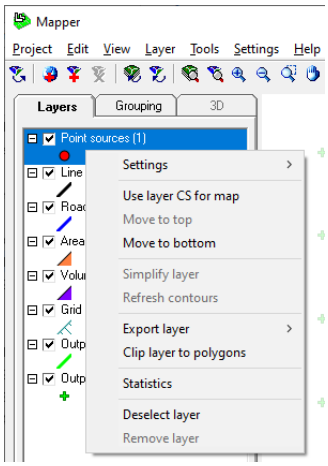
Note that the pan and zoom functions can be used whilst editing a feature. This is useful, for example, if you are drawing a source which covers a large distance such as a road.

1.5 Pop-up menus

You will find context-sensitive pop-up menus when you use the mouse to right-click in the Mapper. These give quick access to relevant actions while you are working with the Mapper.

1.5.1 The layer context menu

When you select a layer in the layer panel and right-click with the mouse, the following context menu will appear. The menu is context-sensitive and only the menu options that apply will be enabled.

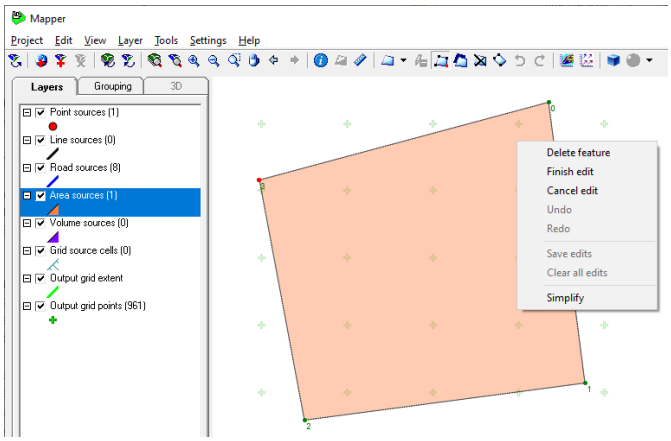


The details of these actions can be found in the relevant sections of this document.

Menu option		Use
Settings	**Layer settings**	Opens the layer properties dialog. See Section 4 and Section 5
	Import layer settings…	Changes the appearance of the layer. See Section 4.8
	Export layer settings…	Saves the layer settings to file for re-use later. See Section 4.8
	Reload default settings file	Resets the appearance of the layer. See Section 4.8
	Save default settings file	Saves the layer settings as the defaults so they will automatically be used the next time you open the Mapper. See Section 4.8
	View active settings file	Opens the current layer settings file in a text editor
	Share colour ramp	Shares the colour ramp of the current layer (if any) with every other layer of that type in that group
	Share renderer	Shares the renderer of the current layer (if any) with every other layer of that type in that group
Use layer CS for map		Sets the map coordinate system to that used by the current layer
Move to top		Moves the current layer to the top of the panel (only available in the Layers tab)
Move to bottom		Moves the current layer to the bottom of the panel (only available in the Layers tab)
Simplify layer		Simplifies the geometry of features belonging to the current layer if possible. See Section 2.3.3.
Refresh contours		Redraws contour lines to match up with their associated contour layer. See Section 4.5.
Export layer		Quick access to common file export options such as ADMS SPT, Google Earth and ESRI Shape. See Section 6.2
Clip layer to polygons		Creates a new layer containing features from the current layer clipped to polygons from another layer. See Section 6.3
Statistics		Displays statistics about the current layer. See Section 5.11
Deselect layer		Deselects the currently selected layer in the layer panel
Remove layer		Removes the selected layer from the layer panel and map view. This is only available for user layers. See Section 1.7.1.

1.5.2 The add/edit context menu

When adding or editing features in the map view you can display a pop-up menu specifically dealing with different edit options when you right-click on the map or an edited feature.



The details of these actions can be found in the relevant sections of this document.

Menu option	Use
Delete feature	Delete the selected feature from the current layer.
Finish edit	Finish editing the currently selected feature without saving all edits. See Section 2.3.2
Cancel edit	Cancel the edits to the currently selected and deselect it.
Undo	Undo the edits to the currently selected feature without deselecting it.
Redo	Redo the edits that have just been undone.
Save edits	Save the edits made during the current editing session to the model.
Clear all edits	Clear all edits made during the current editing session
Simplify	Simplify the geometry of the current feature if possible. See Section 2.3.3.
Simplify to [n] vertices	Simplify the geometry of the current feature such that there will be n (or fewer) vertices after the simplification. Option only available if the current feature has more than the maximum number of vertices (n) allowed by the ADMS model.

1.6 Toolbar buttons

The toolbar contains buttons that allow layers, features in the ADMS layers, and the map view to be edited, as well as providing access to the 2D output plotter. The purpose of each of the buttons is summarised in **Table 3** and **Table 4**. The remaining sections then illustrate how these are used to view, create and edit data.

When using a tool that requires the user to specify a location in the map view window, the appearance of the cursor will change.

Button	Name	Cursor	Function
	**Export layer**	N/A	Export the current layer to a file.
	**Add background map**	N/A	Adds a background map to the map view window.
	**Add layer**	N/A	Add a new layer from a file.
	**Remove layer**	N/A	Remove the selected layer from the layer panel and map view. This is only available for user created layers.
	**Refresh layers**	N/A	Obtain the latest data from the model for all the layers and update the map view.
	**Refresh layer**	N/A	Obtain the latest data from the model for the selected layer and update the map view.
	**Zoom to layers**	N/A	Set the map view to show all the data in all the layers.
	**Zoom to layer**	N/A	Set the map view to the extent of the selected layer.
	**Zoom in**	N/A	Zoom in.
	**Zoom out**	N/A	Zoom out.
	**Zoom**		Zoom to display an area defined by clicking with the mouse on the map and dragging the cursor. A single left-click of the mouse on the map will make a fixed zoom in at the point clicked. A right-click will make a fixed zoom out.
	**Pan**		Move the map view without altering the scale.
	**Previous extent**	N/A	Return to the previously displayed extent in the map view.
	**Next extent**	N/A	Go to the next extent in the map view window.
	**Information**		View information about a model feature in the map view window.
	**Extract data from raster layers**	N/A	Convert raster data to comma delimited file format.
	**Measure**		Measure the distance between two or more points in the map view window.
	**Add feature**		Add a feature to the selected layer, e.g. a point, a polygon etc.
	**Save edits**	N/A	Save the edits made during the current editing session to the model.
	**Edit feature**		Edit the geometry of a feature in the selected layer.
	**Shift feature**		Move a feature in the selected layer.
	**Delete feature**	N/A	Delete the selected feature.
	**Rotate**	N/A	Rotate the selected feature.
	**Undo**	N/A	Undo the current changes while editing.
	**Redo**	N/A	Redo the last Undo.
	**Contours**	N/A	Launch the Output Plotter for concentration plots.
	**Flow field**	N/A	Launch the Output Plotter for flow field plots.
	**Change view**	N/A	Toggle the map view between 2D and 3D mode. The icon changes depending on the map mode. When viewing in 3D mode additional buttons are available as shown in **Table 4**.

Table 3 - Mapper toolbar buttons

Button	Name	Cursor	Function
	**Camera position**		Changes the camera pan mode so that the camera position is changed by yaw and pitch rotations. Use the drop down arrow to change the mode.
	**Camera XYZ**		Changes the camera pan mode so that the camera position is changed by vertical translations and left-to-right horizontal translations.
	**Camera XY**		Changes the camera pan mode so that the camera position is changed by left-to-right and forward-and-backward translations in the horizontal plane.
	**Camera rotation**		Changes the camera pan mode so that the camera position is changed by point-of-view rotation.
	**Sun position**		Changes the light and shade of the display.
	**Zoom**		Zoom in and out.

Table 4 - Mapper toolbar buttons for 3D display

The **Add Feature** button will change appearance depending on the layer currently selected. For the **Buildings**, **Area sources** and **Volume sources** layers, the appearance of the **Add Feature** button depends on the shape of the building/source to be added; this can be changed using the drop down arrow to the right of the button. Some of the buttons are only available under certain conditions; for instance, the **Add Feature**, **Edit Feature** and **Shift Feature** buttons are only available if the current selected layer can be edited from within the Mapper. If a button is not currently available it will be greyed out.

1.7 The layer panel

There are three tabs in the layer panel: **Layers**, **Grouping** and **3D**. The first two tabs show the ADMS data layers and a feature attributes table. The **Layers** tab lists the layers in the order in which they appear in the map view. The **Grouping** tab groups the layers according to their type, e.g. all source layers will be listed under the **Sources** group. The **3D** tab shows the additional options for visualising in 3D. The 3D manipulation is described in Section 5.9 and the **Layers** and **Grouping** tabs are described next.

1.7.1 Layers and Grouping tabs

The **Layers** and **Grouping** tabs show the ADMS data layers along with any user defined layers, such as contour plots or background images. The layers that are present when the parent interface and Mapper are first launched are referred to as ADMS layers. These layers contain all the sources, buildings, and output locations for the **.apl* or **.upl* file currently open in the model interface. Unlike user layers, you will not be able to delete these layers directly from the Mapper layer panel. To update the display to reflect any changes that have been made in the model interface, e.g. through opening a new **.apl* or

*.*upl* file, or adding a new source, click on the **Refresh All Layers** button on the toolbar.

For each of the ADMS layers that can contain multiple features, the layer panel shows the name of the layer, the number of features in that layer (e.g. the number of point sources) and the symbol used to represent that layer. Details of how you can change the symbology and other layer options are given in Section 4. The **Output grid extent** layer only shows the name of the layer and the symbol used to represent the layer.

The number displayed in the layer panel provides a handy count of the number of sources of each type that have been entered into the interface.

User defined layers can be contours generated using the **Contours** button, vector or flow field plots generated using the **Flow field** button, or web map services (WMS), background images and data layers added using the **Add Layer** button (or drag-dropped onto the layer panel). Any user defined layers can be removed by first selecting the layer and then clicking on the **Remove Layer** button; alternatively they can be removed by selecting the layer in the layer panel, right-clicking on the layer and selecting **Remove layer** from the pop-up menu, or simply hitting the delete key on the keyboard. Details about creating contour layers can be found in Section 3 and instructions for adding a background map as a layer can be found in Section 5.2.

1.7.2 Attributes table

An **Attributes** table is displayed beneath the layer panel. The **Attributes** table is populated when you select the **Information** tool and click on a feature; refer to Section 5.6 for full details. The splitting and the sizes of the attributes table and layer panel can be adjusted by dragging the horizontal divider. Similarly, the vertical divider between the layer panel and the map view window can be adjusted.

1.7.3 Showing and hiding layers

If a layer is visible then the data contained in that layer are shown in the map view window. A layer can be hidden by unchecking the checkbox next to the layer name. To make the layer visible again recheck the checkbox.

1.7.4 Reordering layers

The ordering of the layers in the **Layers** tab determines the order in which they are shown in the map view window. Features from layers at the top of the **Layers** tab are shown on top of features from layers further down and thus may hide them from view. There are three different ways to reorder the layers within the **Layers** tab:

1. Click on the layer you wish to move, holding down the mouse button. Once a red bar appears, move the layer to its new position, and then release the mouse button.

2. Use the **Move to top** and **Move to bottom** right-click options on a particular layer.

3. Click on the layer you wish to move. Press and hold the **Ctrl** button and then press the ↑ or ↓ arrow keys to move the layer up or down.

The symbol used to represent a layer can be made partially transparent to allow features in layers below them to be seen. See Section 4.1 for more details on making layers transparent. Note that layers cannot be reordered from the **Grouping** tab.

1.8 The map view window

The map view window is where the data from the current visible layers are shown. Both the ADMS and user-defined layers are shown in this window. The scale of the map view window is given as a scalebar in the bottom right corner of the map view window and as a numerical scale on the right hand side of the status bar. The position of the cursor in the map view window is given in the status bar, as well as the value at that position if a raster layer containing data values is selected.

The scale and extent of the map view window is controlled by various toolbar buttons:

- To zoom and centre the window on the features of all the (unhidden) layers click on the **Zoom To Layers** button on the toolbar. 
- To zoom and centre the window on the features of the currently selected layer click on the **Zoom To Layer** button on the toolbar. 
- To zoom in to a central point in the map view window click on the **Zoom In** button on the toolbar. 
- To zoom out from a central point on the map view window click on the **Zoom Out** button on the toolbar. 
- To view a selected region in the map view window click on the **Zoom** button on the toolbar. 

  Click in the top left corner of the region you want to view, and while holding down the mouse button drag the cursor to the bottom right corner of the region you want to view, then release the mouse button to set the extent of the map view window.

- To move the view in the map view window without altering the scale click on the **Pan** button on the toolbar. 

  Then click and drag the mouse in the map view window to move the view.

- To return to a previous viewing extent in the map view window click on the **Previous Extent** button on the toolbar. 
- To go to the next viewing extent in the map view window click on the **Next Extent** button on the toolbar  . This option is only available after the **Previous Extent** button has been used.

1.9 Coordinate systems

There is an option to set the coordinate system within the Mapper. The choice of the coordinate system affects the way the ADMS layers are shown in the Mapper map view window. Note that the coordinate system must be selected before certain features of the Mapper can be used; for example, before exporting Mapper files to **.kml* files for use in Google Earth.

Coordinate systems can also be defined separately for each layer. This is necessary to ensure that additional features, such as background maps, use the correct coordinate system for the data they are displaying. Typically you would set the map coordinate system to match that used by the parent interface. However, if you have a lot of data in a user layer with a different coordinate system, you might choose that for the map to avoid CPU-intensive re-projection of the data as you move about in the map view. The coordinate system for the ADMS layers should not be changed in the Mapper – this is set for all ADMS layers in the model interface; see the relevant model User Guide for more details.

Full details of how to set the coordinate system are given in Section 5.1.

SECTION 2 Using the Mapper

This section outlines how to use the Mapper to view, create and modify the geometry of the model data. The model data that can be manipulated with the Mapper varies depending on which model it is being used with, but include sources, buildings, specified output points, output grids and terrain and roughness files.

2.1 Mapper projects

Mapper projects are a convenient way to manage files and data you wish to view in the Mapper that are associated with a given task. Any data added to the Mapper project, such as a background map or contour plot, will remain in the project after saving. The project will also remember any custom settings for the ADMS layers that you may have configured. The file name extension of a Mapper project file is **.mpj*.

To save the current Mapper project, select the **Save project** command from the **Project** menu, or, to save the current project under a new name, select **Save project as**... from the **Project** menu. Once the project has been saved, the filename will appear in the form title at the top of the Mapper. When using the Mapper with a compatible model, for example ADMS, the project will also be entered into the **Mapper project file** panel on the **Setup** screen of the ADMS interface. In this way you can associate **.apl* or **.upl* files with a project so that all your added data is loaded automatically when you start the Mapper.

Note that the project file has a required associated file with the file extension *.ttkgproject*. If the Mapper project is moved to a different directory after it has been created, the associated* *.ttkgproject *file must be moved with it.*

You can choose to open a Mapper project at any time, regardless of any **.apl* or **.upl* file you may have loaded in the parent interface. This will display all the user layers referenced by the project, and also apply the project configuration to the ADMS layers. It also reassigns the Mapper project file in the parent (e.g. ADMS) interface.

To close the Mapper project, select the **New** command from the **Project** menu, the current project will then close, clearing all the user layers and project configuration settings, and leaving only the ADMS data displayed in the Mapper using the default layer configurations.

To view existing data from an **.apl* or **.upl* file refer to Section 2.2; or to add, modify the geometry of, move, or delete a feature refer to the instructions in Sections 2.3 to 2.10.

2.2 Viewing existing data

To view existing sources, buildings, specified output points, the output grid and output points, first open the **.apl* or **.upl* file in the model interface. Then return to the Mapper and click on the **Refresh Layers** button  on the toolbar. Some layers will only appear in the Mapper once they have been selected for modelling in the parent interface.

This can include buildings, specified output points, the output grid, complex terrain and surface roughness options. For details of how to set these options see the model user guide.

2.3 General source editing

This section covers how to add, move or delete sources using the Mapper. To follow these instructions, make sure you already have the **.apl* or **.upl* file which is to be edited open in the model interface, i.e. the file to which you wish to add, move or delete sources. Also make sure that the Mapper is open.

2.3.1 Adding a source

Step 1 Select the appropriate layer in the layer panel e.g. the **Point sources** layer, the **Area sources** layer etc.

Step 2 Click on the **Add Feature** button on the toolbar to select it. Note that the **Add feature** button changes appearance depending upon which type of source is being edited. For example the **Add feature** button for point sources uses a dot symbol, . For area and volume sources, click on the drop down arrow and select either **Polygon** or **Circle**.

Step 3 Click in the map view window at the location where you want the source to be placed. If you are adding a line source in ADMS 6 you will need to click at the location of either end of the line; remember a line source must have precisely two vertices in ADMS 6. If you are adding a polygonal area or volume source, or a road or line source in ADMS-Urban, ADMS-Roads or ADMS-Airport, you will need to click in the map view window at each vertex location, and double click when placing the last vertex to finish defining the source. If you don't double click at the end of the drawing, you can still complete the edit by pressing enter on the keyboard or right-clicking with the mouse and choosing **Finish edit** from the pop-up menu. If you are adding a circular area or volume source, you will need to click in the map window on the location of the centre of the source, and while holding down the mouse button, move the cursor until the circle is of the required diameter before releasing the mouse button. This will create a polygon representation of the circle.

*When drawing road sources, for example, enable the **Snap to shape** option from the **Settings** menu in order to easily connect road ends of adjoining sources, or disable it for fine edits.*

Step 4 You will need to click on the **Save edits** button on the toolbar, , to complete all editing in the layer and commit the new features (alternatively, right click the mouse to bring up a pop-up menu and select the **Save edits** option).

Step 5 A new source of the appropriate type is created and displayed in the **Source** screen of the model interface. The source coordinates have been filled in automatically but you will need to fill in the other parameters for the

source. Refer to the model user guide for further details.

2.3.2 Editing a source

Follow these steps to edit an existing source from within the Mapper.

Step 1 Select the appropriate layer in the layer panel e.g. the **Area sources** layer.

Step 2 Click on the **Edit Feature** button on the toolbar to select it. 

Step 3 Select the source you want to edit by clicking on it.

Step 4 If you are editing a source with multiple vertices (e.g. area source) and wish to add a new vertex, do as follows. Click on the starting vertex of the edge you wish to split into two, e.g. click on vertex 3 if you wish to add a new vertex between the current vertices 3 and 4. The selected vertex should turn red, with the rest of the vertices coloured green. Alternatively, use either the full stop (.) or comma (,) keys on the keyboard to select a different vertex. Then click on the location of the new vertex.

For multiple vertex sources, the modified shape of the selected source will appear as a dashed line during editing.

Step 5 To move a vertex, click and drag that vertex to the new location before releasing the mouse button.

Step 6 To delete a vertex, click on the vertex you wish to delete in order to select it, and then click on that vertex again.

Step 7 If you want to edit other sources in the same layer without saving all edits, you can complete the current edit by pressing enter on the keyboard or right-clicking with the mouse and choosing **Finish edit** from the pop-up menu. You can now select another source to edit as in Step 3.

Step 8 Click on the **Save edits** button on the toolbar to save the changes to the model.  . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

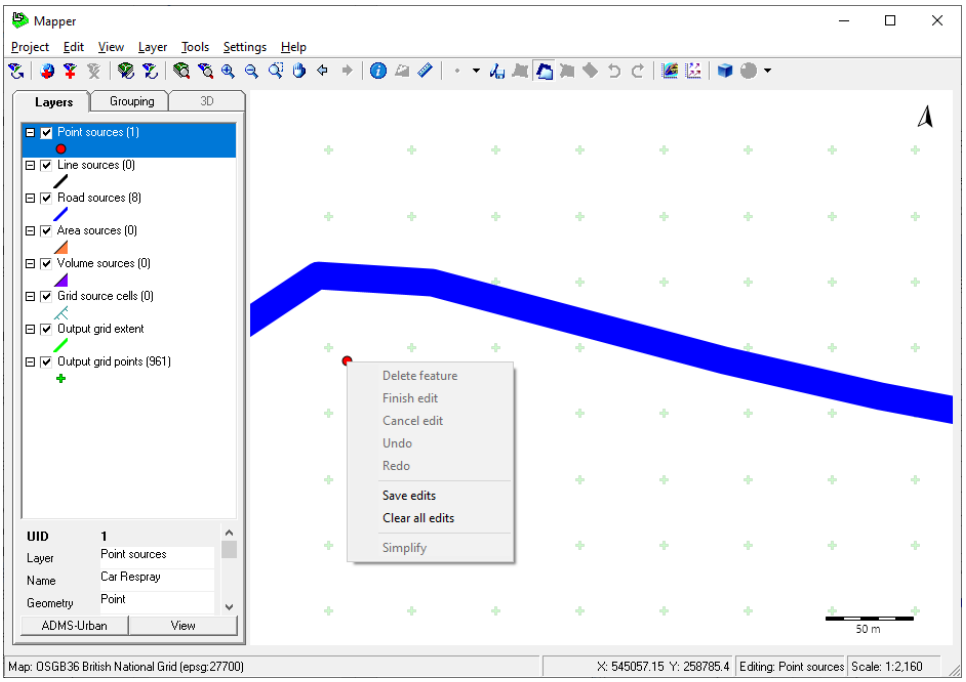


Figure 2 - Pop-up menu showing editing options.

2.3.3 Simplifying source geometry

When you are editing a source with multiple vertices (e.g. a road source) you can use the **Simplify** option from the right-click pop-up menu in **Figure 2** to reduce the number of vertices. This will remove as many vertices as possible while preserving the approximate shape of the original source. **Simplify** uses the Douglas-Peucker vertex removal algorithm with a tolerance of 1 m to systematically remove vertices and simplify the polyline or polygon.

If the source being edited has more than the maximum number of vertices allowed by the ADMS model, another option, **Simplify to [n] vertices**, will become available from the right-click pop-up menu. This will force the simplification to result in n (or fewer) vertices, increasing the default tolerance of the algorithm if necessary.

There is also an option to simplify the geometry of all sources of a particular type simultaneously. To do this, right-click on the appropriate layer in the layer panel, e.g. the **Road sources** layer, and click **Simplify layer**. This generates a **Simplify report** that displays the results of the layer simplification before they have been applied; an example is shown in **Figure 3**. You should then choose whether to apply the layer simplification by clicking either **Save changes** or **Discard changes** as appropriate. There is also a **Copy report** button that allows you to copy the report text to the clipboard.

---

*Running **Simplify layer** will ensure that no sources in the layer have more than the maximum number of vertices allowed by the ADMS model.*

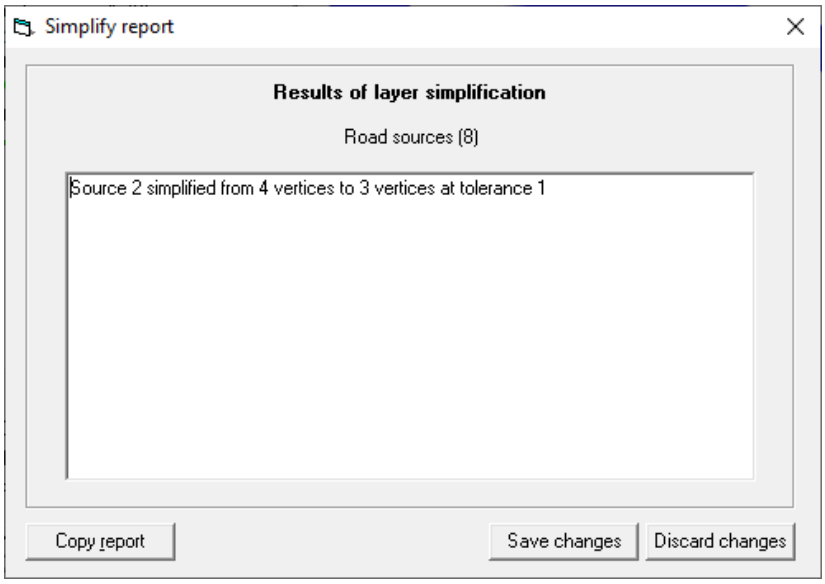


Figure 3 - Example Simplify report

2.3.4 Moving a source

Follow these steps to change the location of an existing source from within the Mapper.

Step 1 Select the appropriate layer in the layer panel e.g. the **Point sources** layer.

Step 2 Click on the **Shift Feature** button on the toolbar to select it.

Step 3 Click on the appropriate source, and while holding down the mouse button move the cursor to the new location for the source before releasing the mouse button.

Step 4 Click on the **Save edits** button on the toolbar to save the changes to the model. Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

2.3.5 Rotating a source

Line, road, area and volume sources can all be rotated in the horizontal plane within the Mapper. To do this, follow these steps:

Step 1 Select the appropriate layer in the layer panel e.g. the **Area sources** layer.

Step 2 Click on the **Rotate** button on the toolbar to select it.

Step 3 Click on the appropriate source, away from its centre, and while holding down the mouse button, move the cursor around the centre of the source until you are happy with its new location (indicated by a red outline), before releasing the mouse button.

Step 4 Click on the **Save edits** button on the toolbar to save the changes to the model. Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option

2.3.6 Deleting a source

Follow these steps to delete an existing source from within the Mapper.

Step 1 Select the appropriate layer in the layer panel e.g. the **Point Source** layer.

Step 2 Click on the **Edit Feature** button on the toolbar to select it.

Step 3 Select the source you want to delete by clicking on it in the map view window.

Step 4 Click on the **Delete Feature** button on the toolbar. . Or press the `Delete` key on your keyboard.

Step 5 Click on the **Save edits** button on the toolbar to save the changes to the model. . Alternatively, right click the mouse to bring up a pop-up menu and select the **Save edits** option.

Step 6 Repeat this process to delete further sources.

2.4 Grid sources and aircraft sources

Grid sources and aircraft sources can be displayed in the Mapper, but they cannot be added, edited or deleted.

To view a grid or aircraft source add the source in the model interface following the instructions in the ADMS-Urban or ADMS-Airport User Guide. In the Mapper click on the **Refresh Layers** button. The sources will now be displayed.

2.5 Specified output points

In this section we look at how to add, move or delete specified output points using the Mapper. To follow these instructions, make sure you already have open in the model interface the **.apl* or **.upl* file to which specified points are to be added, edited or deleted. Also make sure that the Mapper is open. Also ensure that **Specified points** is selected as an output location type in the **Grids** screen of the model interface.

2.5.1 Adding an output point

Step 1 Select the **Specified Points** layer from the layer panel.

Step 2 Click on the **Add Feature** button on the toolbar to select it.

Step 3 Click on the location in the map view window where you wish to place the specified output point.

Step 4 Click on the **Save edits** button on the toolbar to save the changes to the model. . Alternatively, right click the mouse to bring up a pop-up menu and select the **Save edits** option.

Step 5 A new specified output point has been created and is displayed in the **Grids** screen of the model interface. The **X** and **Y** co-ordinates have been automatically entered but the rest of the parameters need to be filled in. See the model user guide for more details.

Step 6 Repeat this process to add further specified output points.

2.5.2 Moving an output point

Step 1 Select the **Specified Points** layer from the layer panel.

Step 2 Click on the **Shift Feature** button on the toolbar to select it. 

Step 3 Click on the specified output point you wish to move, and while holding down the mouse button, move the cursor to the new location for the specified output point before releasing the mouse button.

Step 4 Click on the **Save Edits** button on the toolbar to save the changes to the model.  . Alternatively, right click the mouse to bring up a pop-up menu and select the **Save edits** option.

Step 5 Repeat this process for all the specified output points you wish to move.

2.5.3 Deleting an output point

Step 1 Select the **Specified points** layer from the layer panel.

Step 2 Click on the **Edit Feature** button on the toolbar to select it. 

Step 3 Select the specified output point to delete by clicking on it in the map view window. The selected specified output point will have a red dot on it.

Step 4 Click on the **Delete Feature** button on the toolbar,  , or press the Delete button on your keyboard.

Step 5 Click on the **Save Edits** button on the toolbar to save the changes to the model.  . Alternatively, right click the mouse to bring up a pop-up menu and select the **Save edits** option.

Step 6 Repeat this process to delete further specified output points.

2.6 Specified output points file

If you wish to add a large number of specified points you can do this quickly by including an **.asp* file in the ADMS model interface to define their locations. Refer to your model user guide for instructions on how to do this. The locations of the specified points defined in the **.asp* file can be viewed in the Mapper by following these steps.

Step 1 Add the file path for the **.asp* file to the model interface, refer to the model user guide for instructions on how to do this.

Step 2 In the Mapper click on the **Refresh Layers** button  . The specified points in the **.asp* file will now be displayed.

*The specified points in the *.asp file cannot be edited or deleted using the Mapper; this can be done only by editing the *.asp file directly.*

*For very large *.asp files, only the first 500,000 points will be displayed.*[1]

2.7 Output grid

In this section we look at how to define a new output grid using the Mapper. To follow these instructions, make sure you already have open in the model interface the **.apl* or **.upl* file with the output grid you wish to redefine, and ensure that the Mapper is also open.

Regular and variable grids can be displayed in the Mapper, but only regular grids can be defined using the Mapper. Variable grids must be defined in the ADMS interface. Therefore, if you wish to define the output grid with the tool in the Mapper, ensure that **Gridded output points** is selected as an output location type and the gridded output spacing set to **Regular** in the **Grids** screen of the model interface.

You can only redefine or move an output grid; you cannot edit or delete an existing output grid.

2.7.1 Defining an output grid

Step 1 In the ADMS interface, ensure that **Gridded output points** is selected as an output location type and the gridded output spacing set to **Regular** in the **Grids** screen.

Step 2 Select the **Output grid extent** layer in the layer panel.

Step 3 Click on the **Add Feature** button on the toolbar to select it. 

Step 4 Click on the location of one corner of the output grid in the map view window, and while holding down the mouse button, move the cursor to draw the output grid required before releasing the mouse button.

Step 5 A new output grid has been defined and is displayed in **Grids** screen of the model interface. The new extent of the grid will have been filled in automatically but if any other parameters need to be changed these must be altered from the interface. Refer to the model user guide for more details.

*Note that the grid point locations are shown in the **Output grid points** layer but they cannot be edited in the Mapper. The number of points in the grid is displayed in brackets.*

Any type of grid point can be displayed in the Mapper.

[1] The maximum number of data points shown in any layer can be edited manually in the Mapper settings file.

2.7.2 Editing the output grid

Step 1 Select the **Output grid extent** layer from the layer panel.

Step 2 Click on the **Edit Feature** button on the toolbar to select it. 

Step 3 Click anywhere within the current output grid extent. Three of the grid vertices will appear as green dots and the fourth vertex as a red dot.

Step 4 The vertex with the red dot indicates which corner of the output grid will be movable. To select a different vertex, use either the full stop (.) or comma (,) keys on the keyboard.

Step 5 Click in the map window, and while holding down the mouse button, move the cursor until the red grid vertex is in the desired location before releasing the mouse button.

Step 6 Click on the **Save edits** button on the toolbar to save the changes to the model.  . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

2.7.3 Moving the output grid

Step 1 Select the **Output grid extent** layer from the layer panel.

Step 2 Click on the **Shift Feature** button on the toolbar to select it. 

Step 3 Click anywhere within the current output grid extent, and while holding down the mouse button, move the cursor to the new location for the output grid before releasing the mouse button.

Step 4 Click on the **Save edits** button on the toolbar to save the changes to the model.  . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

2.8 Buildings

This section outlines how to add, move, edit or delete buildings using the Mapper. To follow these instructions, make sure you already have open in the model interface the **.apl* or **.upl* file to which buildings are to be added, edited or deleted. Make sure that the Mapper is also open. Also make sure that the **Buildings** option is selected in the **Setup** screen of the model interface.

Rectangular buildings must consist of 4 vertices and be rectangular in shape. If the shape entered is not rectangular but is convex and has 4 vertices then the Mapper will convert the shape to be rectangular.

The main building will be shown with a red outline. See the relevant model User Guide for more details about setting the main building.

2.8.1 Adding a rectangular building

Step 1 Select the **Buildings** layer from the layer panel.

Step 2 Use the drop down arrow on the **Add Feature** button on the toolbar to select **Polygon** and then click on this button to select it.

Step 3 Click in the map view window on the locations of the four vertices of the building in order, either clockwise or anticlockwise. Double click when placing the last vertex to finish defining that building. Alternatively, press enter on the keyboard, or right-click with the mouse and choose **Finish edit** from the pop-up menu, after placing the last vertex.

Step 4 Repeat this process to add further rectangular buildings.

Step 5 Click on the **Save Edits** button on the toolbar to finish editing and save the changes to the model. . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option

Step 6 The new buildings will have been created and are displayed in the **Buildings** screen in the model interface. Enter the rest of the parameters for these buildings and then exit the **Buildings** screen. See the model user guide for more details.

2.8.2 Adding a circular building

Step 1 Select the **Buildings** layer from the layer panel.

Step 2 Use the drop down arrow on the **Add Feature** button on the toolbar to select **Circle** then click on the button to select it.

Step 3 Click in the map view window on the location of the centre of the building, and while holding down the mouse button, move the cursor until the circle is of the appropriate diameter, and then release the mouse button.

Step 4 Repeat this process to add further circular buildings.

Step 5 Click on the **Save Edits** button on the toolbar to finish editing and save the changes to the model. . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

Step 6 The new buildings will have been created and are displayed in the **Buildings** screen in the model interface. Enter the rest of the parameters for these buildings and then exit the **Buildings** screen. See the model user guide for more details.

2.8.3 Editing a rectangular building

Step 1 Select the **Buildings** layer from the layer panel.

Step 2 Click on the **Edit Feature** button on the toolbar to select it.

Step 3 Select the rectangular building whose geometry you wish to alter in the map view window. The vertices of the selected building will appear as red or green dots.

Step 4 For each vertex you want to move, click on that vertex, and while holding down the mouse button, move the cursor to the new location of that vertex before releasing the mouse button.

Step 5 Click on the **Save Edits** button on the toolbar to finish editing this building and save the changes to the model. . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

Step 6 Repeat this process to edit the geometry of further rectangular buildings.

2.8.4 Editing a circular building

Step 1 Select the **Buildings** layer from the layer panel.

Step 2 Click on the **Edit Feature** button on the toolbar to select it.

Step 3 Select the circular building whose geometry you wish to alter in the map view window. The outline of the selected building will appear as a series of green or red dots.

Step 4 Click in the map window, and while holding down the mouse button, move the cursor until the circle is of the required diameter before releasing the mouse button.

Step 5 Click on the **Save Edits** button on the toolbar to save the changes to the model. . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

Step 6 Repeat this process to edit the geometry of further circular buildings.

2.8.5 Moving a building

Follow these instructions to use the Mapper to change the location of an existing building, either rectangular or circular, without altering its shape.

Step 1 Select the **Buildings** layer in the layer panel.

Step 2 Click on the **Shift Feature** button on the toolbar to select it.

Step 3 Click on the building you wish to move, and while holding down the mouse button, move the cursor to the new location of this building before releasing the mouse button.

Step 4 Click on the **Save Edits** button on the toolbar to save the changes to the model. . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

Step 5 Repeat this process to move other buildings.

2.8.6 Rotating a building

Buildings can also be rotated in the horizontal plane within the Mapper. To do this, follow these steps:

Step 1 Select the **Buildings** layer in the layer panel.

Step 2 Click on the **Rotate a feature** button on the toolbar to select it.

Step 3 Click on the appropriate building, away from its centre, and while holding

down the mouse button, move the cursor around the centre of the building until you are happy with its new location (indicated by a red outline), before releasing the mouse button.

Step 4 Click on the **Save Edits** button on the toolbar to save the changes to the model. Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

2.8.7 Deleting a building

Step 1 Select the **Buildings** layer in the layer panel.

Step 2 Click on the **Edit Feature** button on the toolbar to select it.

Step 3 Select the building you wish to delete by clicking on it in the map view window. The vertices of the selected building will appear as red or green dots.

Step 4 Click on the **Delete Feature** button on the toolbar, . Or press the **Delete** button on your keyboard.

Step 5 Click on the **Save Edits** button on the toolbar to save the changes to the model. . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

Step 6 Repeat this process to delete further buildings.

2.9 Coastline

This section outlines how to edit the coastline properties using the Mapper. To follow these instructions, make sure you already have open in the model interface the **.apl* or **.upl* file to which the coastline properties are to be edited. Make sure that the Mapper is also open. Also make sure that the **Coastline** option is selected in the **Setup** screen of the model interface.

2.9.1 Coastline

The coastline is represented by a line between the two coastline vertices. The **Coastline layer** can be manipulated in the following ways:

- The **Add feature** button can be used to redraw the coastline. Press the mouse button down at the location of the first vertex, hold the mouse button down and drag the cursor to the location of the second and then release the mouse button.
- The **Edit feature** button can be used to move one of the vertices of the coastline. First click on the coastline then click and drag the vertex to move it to its new location. Then click the **Save edits** button .
- The **Shift feature** button can be used to move the whole coastline. Click and drag on the coastline to move it to the new location and then click the **Save edits** button .

- The **Rotate a feature** button  can be used to rotate the coastline about its centrepoint. Click and drag on the coastline to put it in the new orientation and then click the **Save edits** button  .

2.9.2 Coastline point on land

The coastline point on land is represented by a single point. This point can be moved in the Mapper by first selecting the **Coastline land layer** in the layer panel, then selecting the **Add feature** button and clicking in the new location. This point must be at least 0.1 m away from the line representing the coastline.

2.10 Terrain and roughness files

Any complex terrain or variable surface roughness that has been included in the model can be visualised in the Mapper. To view this data in the Mapper, first make sure that the complex terrain option has been selected in the model interface and that either the terrain file path (**.ter*) or the roughness file path (**.ruf*) has been selected and specified. Refer to the model user guide for further details on modelling and including complex terrain and variable roughness.

After the complex terrain file(s) have been included click on the **Refresh Layers** button  in the Mapper. A representation of the variable terrain or surface roughness, or both, will then be displayed in the map view window. Initially, the variable terrain or roughness layer is shown with a default colour scale and default number of height or roughness levels. These can be edited; see Section 4.6.

It is possible to view a 3D visualisation of the terrain and other input. Further details on this are given in Section 5.9.

Extremely large terrain and roughness files can be displayed in the Mapper, but only the first 500,000 points will be displayed. This limit can be changed in the Mapper settings file, or alternatively the terrain file can be converted to a grid instead; see Section 5.6 for more details.

SECTION 3 Viewing model results in the Mapper

Contour and vector plots of model output results can be created and viewed in the Mapper. The 2D output plotter can produce contour plots using the in-built interpolator, or using Golden Software's Surfer, if the user has this program installed. Section 3.1 describes how to plot model output. Section 3.2 describes how to plot flow field output. For details on changing the appearance of a contour layer once it has been produced refer to Section 4.5.

3.1 Contour plotting

Step 3 Use the **Settings > Gridding option** menu to select the **Interpolator** or the **Surfer** option as desired and then click on the **Contours** button.

Step 4 The **2-D Output Plotter** screen will appear. The **Interpolator** mode is shown in **Figure 4**.

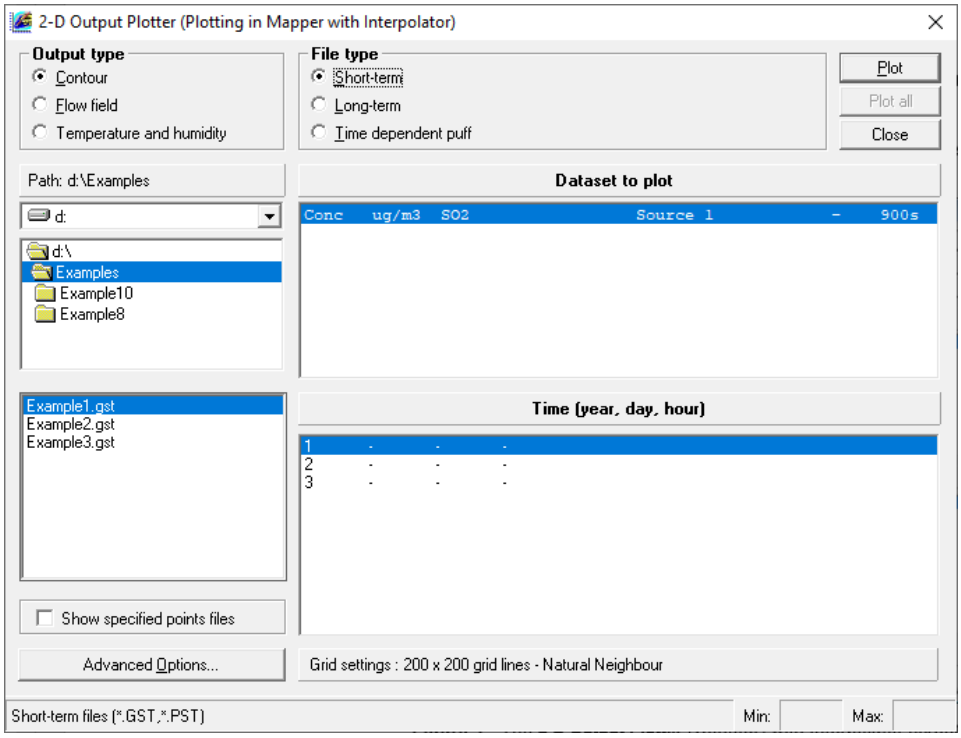


Figure 4 - The **2-D Output Plotter** (Contour) with Interpolator screen.

Step 5 Select the file type (short *or* long term), the file containing the results, the **Dataset** and the **Time** (if required) to be plotted.

Step 6 If required, use the **Advanced Options...** button to change the number of grid lines or to specify user defined contour levels, as shown in **Figure 5**. The **Gridding method** can also be changed. The options available with the **Interpolator** option are **Inverse Distance**, **Kriging**, **Kriging (all points)**, **Natural Neighbour** and **Triangulation** (additional options are available with Surfer). The

appearance of the output contours will depend on which of these has been chosen. **Natural Neighbour** is a suitable choice in most instances. Once advanced options have been selected, click **Close** to return to the main output plotter screen.

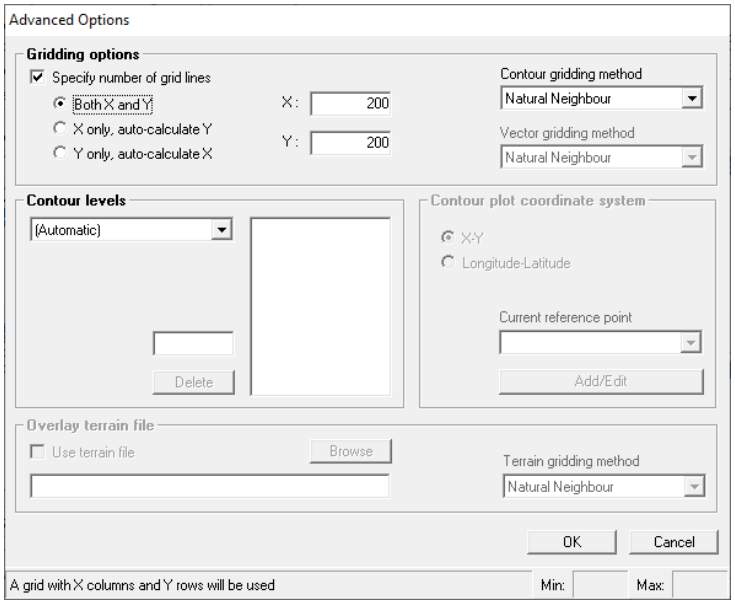


Figure 5 - The **Advanced Options** screen for the ADMS Contour Plotter with Interpolator.

Step 7 Click on the **Plot** button.

Step 8 You are asked to enter a name for the grid file. After entering a file name click on the **Save** button.

Step 9 The contour layer will appear in the Mapper map view window, along with a layer for the contour lines.

Figure 6 shows an example of a contour plot in the Mapper.

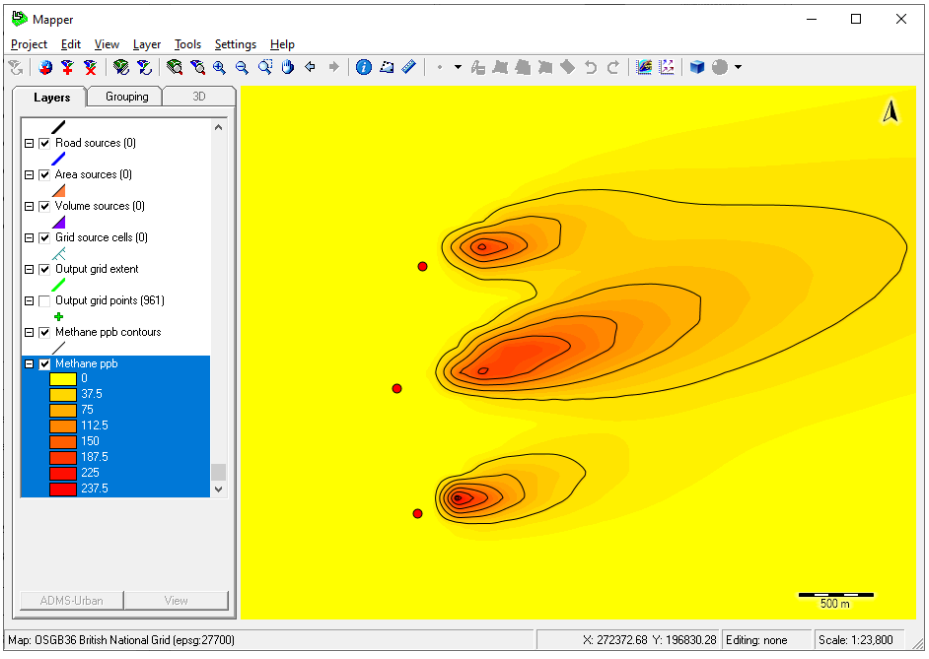


Figure 6 - Example contour plot.

3.2 Flow field plotting

Step 1 Use the **Settings > Gridding option** menu to choose either the **Interpolator** or **Surfer** option. Click on the **Flow field** button. The rest of this example assumes that the in-built interpolator has been selected.

Step 2 The **2D Output Plotter** screen will appear, as shown in **Figure 4**.

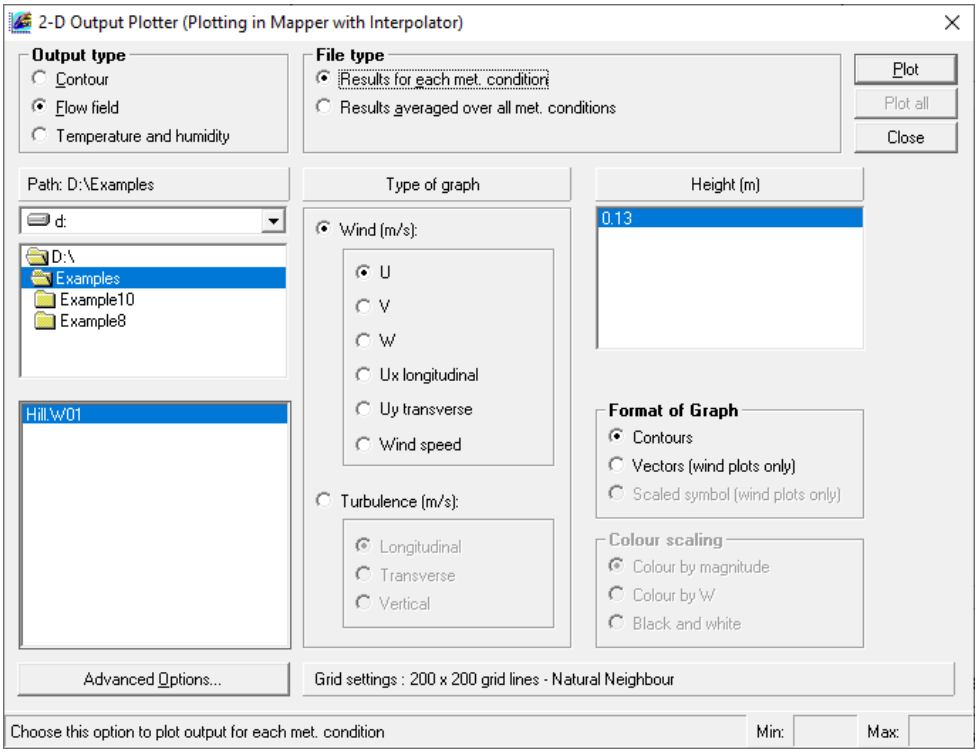


Figure 7 - The **2-D Output Plotter** (Flow field) with Interpolator screen.

Step 3 Select the file type (**Results for each met. condition** *or* **Results averaged over all met. conditions**), the file containing the results, the type of graph (**Wind** or **Turbulence**), the height for which the flow field is to be plotted and the format of the graph (**Contours** or **Vectors**). If contours are being plotted then you must select which component of the flow field or turbulence that you wish to plot.

Step 4 If required, use the **Advanced Options...** button to change the number of grid lines or to specify user defined contour levels, as shown in **Figure 5**. The **Gridding method** can also be changed. The options available with the **Interpolator** option are **Inverse Distance**, **Kriging**, **Kriging (all points)**, **Natural Neighbour** and **Triangulation** (additional options are available with Surfer). The appearance of the output contours will depend on which of these has been chosen. **Natural Neighbour** is a suitable choice in most instances. Once advanced options have been selected, click **OK** to return to the main output plotter screen.

Step 5 Click on the **Plot** button.

Step 6 You are asked to enter a name for an intermediate file. After entering a file name click on the **Save** button.

Step 7 The plot will appear in the Mapper map view window.

Figure 8 shows an example of a flow field vector plot in the Mapper.

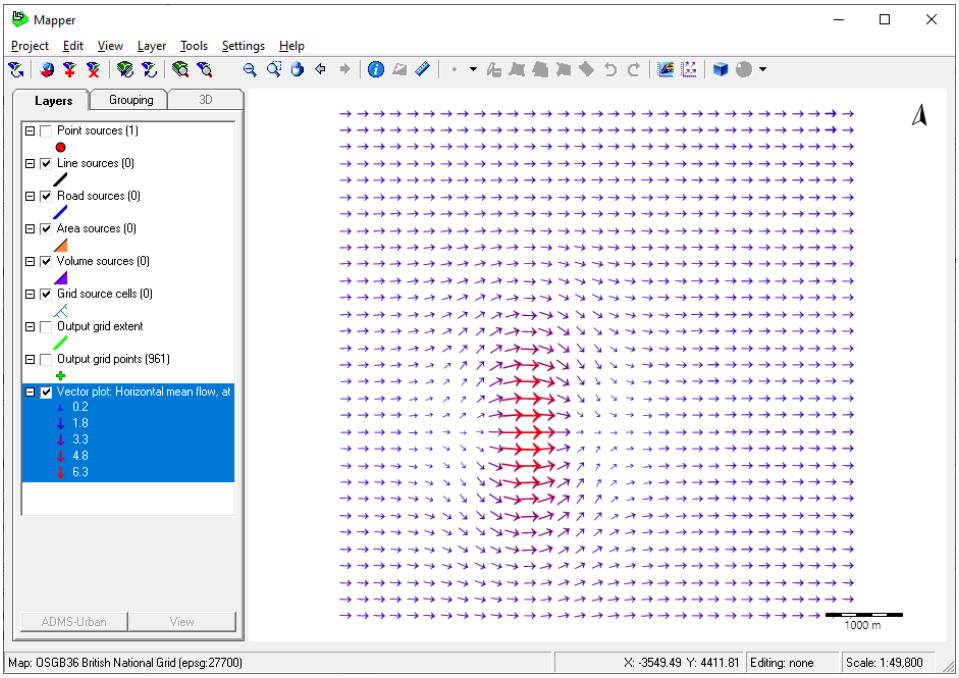


Figure 8 - Example vector flow field plot.

3.3 Displaying footprints

After performing an ADMS model run in which the **Output per source** option was selected, it is possible to visualise the data in the extra output files created by this option (*.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 receptor point. The data to be viewed in the Mapper is controlled via a form that is launched from the model interface; refer to the model user guide for full details.

An example visualisation in the Mapper is shown in **Figure 9**. Each included source is filled with a colour that indicates 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; refer to Section 4.

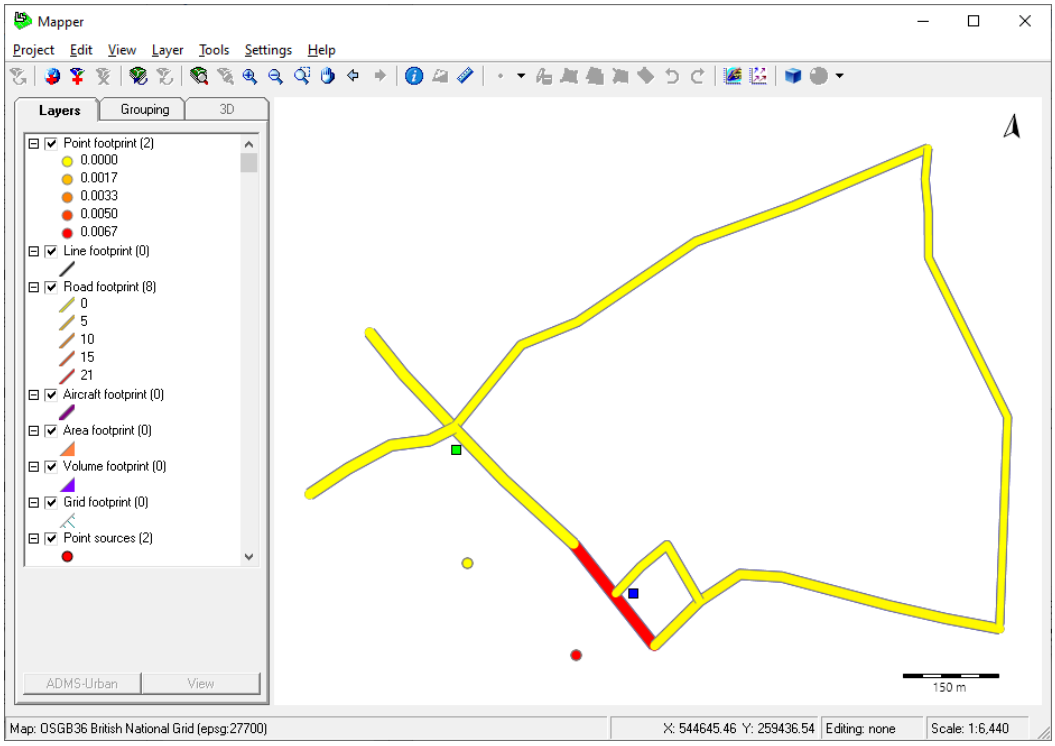


Figure 9 - Example footprint plot.

3.4 Viewing numerical results

After an ADMS model run numerical results are written to **.gst*, **.glt*, **.pst*, or **.plt* files (see the relevant model user guide for further details). The data in these files can be displayed as a new layer in the Mapper. The example below is for a **.gst* file, but the method is similar for the other file types.

Any data file that is in comma-separated format can be displayed as a layer in the Mapper provided it has some spatial information to place the data on the map. See Section 5.5 for more details about this feature.

Step 1 Click on the Add Layer tool. A new window will appear as shown in **Figure 10**.

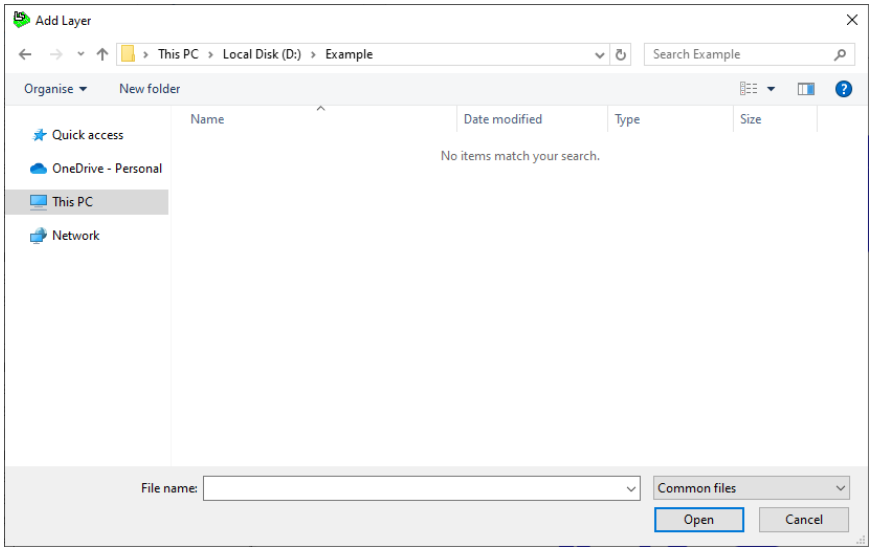


Figure 10 - Add Layer window.

Step 2 Change the file type from **Common files** to **All Files (*.*)**.

Step 3 Navigate to the **.gst* file you wish to view and click **Open**. A new window will appear as shown in **Figure 11**.

As an alternative to Steps 1-3, the file can be drag-dropped from Explorer onto the layer panel.

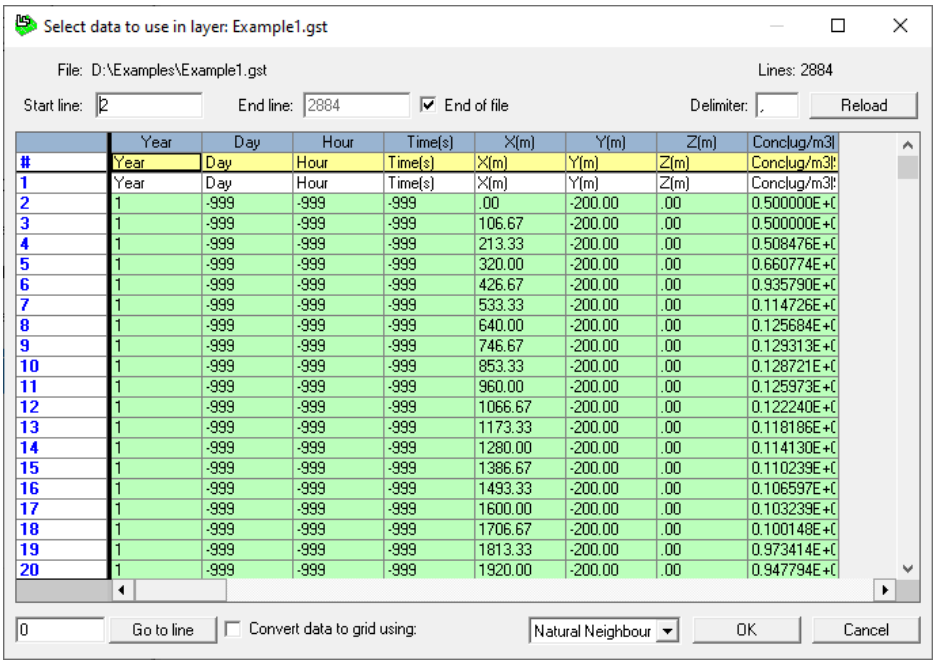


Figure 11 - Data from **.gst* file

Step 4 This window can be used to select which data you wish to display in the Mapper. By default all of the data is selected. To change the selection first scroll to the desired start line of data and right click on the index (the number in blue) for that line. When the menu appears select **Set as first data line**. An example is shown in **Figure 12** for which the data for hour 2 is being selected for display in the Mapper.

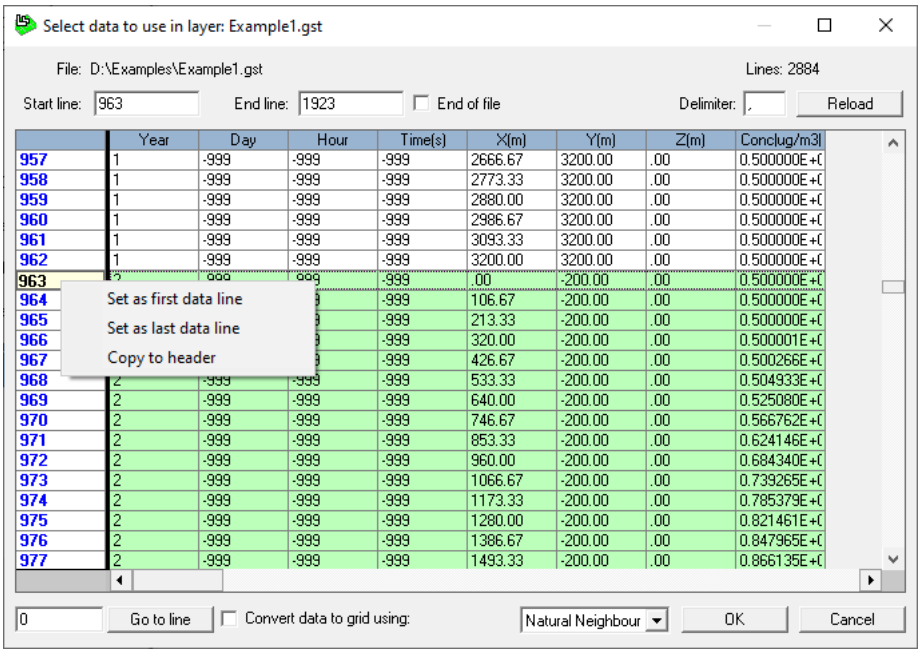


Figure 12 - Setting the first data line

Step 5 Similarly, to set the last data line scroll to the desired line, right click on the index, and click in **Set as last data line**. The selected lines will be highlighted in green.

The information at the top of the window shows you how much data is in the file and which lines of data have been selected.

Step 6 At this point you have the option to grid the selected data or add it as a set of discrete points. To grid the data select the tick box beside the **Convert data to grid** option; see Section 5.5 for more details about displaying data. Once you have selected the data that you wish to display click **OK**. If you have not gridded the data, it will then be displayed in the Mapper using the default red square symbol as shown in **Figure 13**.

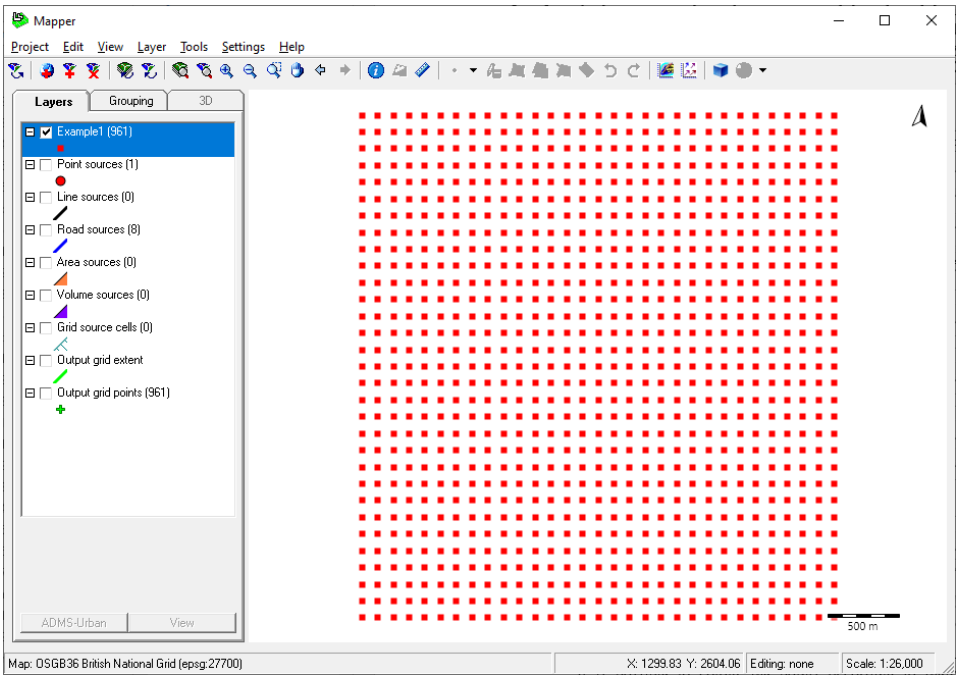


Figure 13 - Data from the **.gst* file displayed in the Mapper.

Step 7 Information about the data can then be viewed. Ensure that the data layer is selected in the layer panel. Select the information tool and click on one of the data points. Full details on the information tool can be found in Section 5.6. Information is then displayed at the bottom of the layer panel in the **Attributes table**. The information includes the name of the point, its elevation and location, the hour or time for which the data comes from, the pollutant concentrations at that point, and the spatial extent of the whole data in the data file.

*It is possible to colour the points according to their concentration, for example, as shown in **Figure 14**. By importing a configuration file you can quickly apply the same colour settings to multiple layers. See Sections 4.6 and 4.8 for further details.*

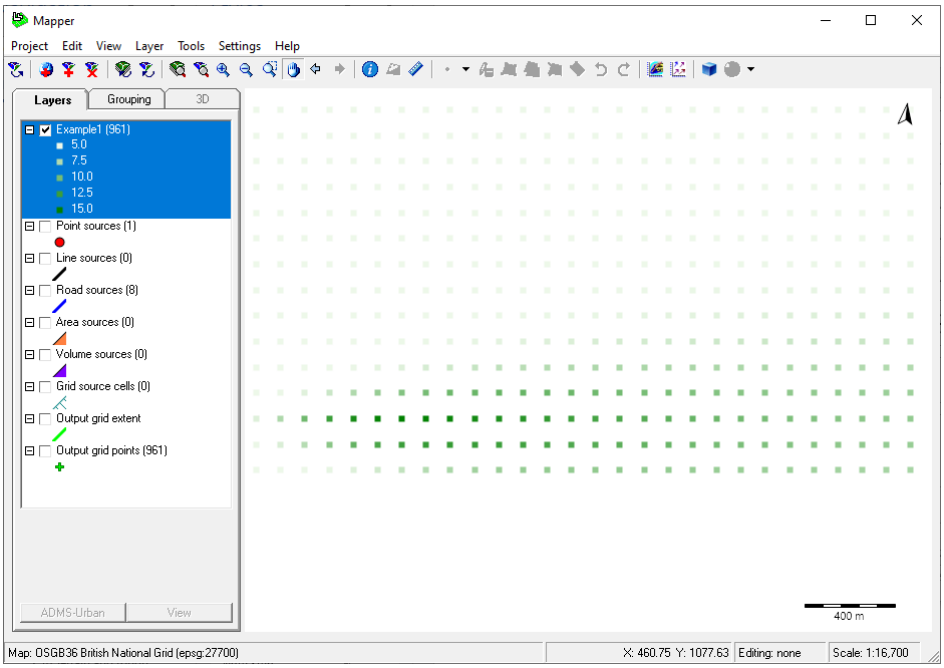


Figure 14 - Numerical data displayed in the Mapper coloured by concentration.

SECTION 4 Modifying the appearance of layers

This section outlines how to modify the appearance of layers. The types of modification that can be made depend on the type of layer you wish to modify. Sections 4.1 to 4.7 give details of various different types of modification that can be made and the types of layer to which these modifications can be applied. After modifying the appearance of the ADMS layers the new appearance can be saved as the default appearance for ADMS layers by selecting **Save all layer settings** from the **Settings** menu. The default appearance for ADMS layers can be restored at any time by selecting **Restore factory settings** from the **Settings** menu. Note that this will save the current appearance of all of the ADMS layers. The settings for individual layers can be saved or reloaded by right-clicking on the layer and selecting **Settings > Save default settings file** or **Settings > Reload default settings file** respectively. There is also the option to **View active settings file**; selecting this option brings up a text file containing the current codes for the selected layer. The settings for an individual layer can also be exported and then imported to other layers, or imported in new maps later. This can be achieved by right-clicking on the layer and selecting **Settings > Import layer settings...** or **Settings > Export layer settings...** as appropriate. These options are discussed in Section 4.8.

4.1 Modifying the transparency of a layer

The transparency of any layer can be changed. The transparency determines how "see-through" the layer is and thus whether any features in a lower layer can be seen through features in this layer. To change the transparency of a layer, follow these steps:

Step 1 Double click on the layer in the layer panel or select **Settings > Layer settings** from the **Layer context menu** to bring up the layer properties window.

Step 2 Select **General** from the left panel and scroll to the **Painting** section, as shown in **Figure 15**.

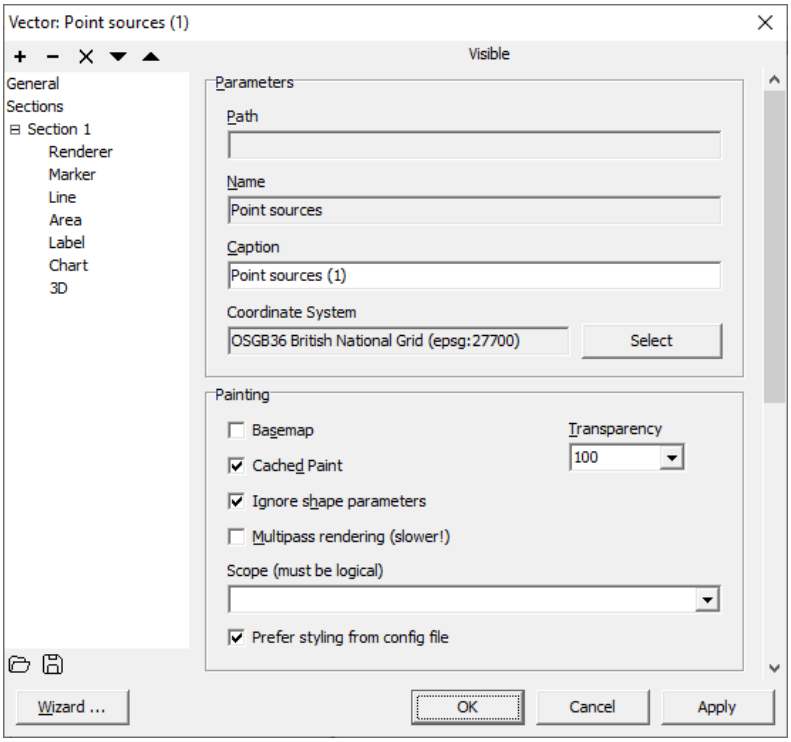


Figure 15 - The **General** section of the layer properties screen.

Step 3 Alter the **Transparency** value as required. 0 is completely transparent and 100 is completely opaque.

Step 4 Click on the **OK** button to make the changes and return to the Mapper window, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the Mapper window.

4.2 Modifying the appearance of a marker layer

Markers are used to represent any layer of data defined as single points. For instance, the ADMS layers for point sources are marker layers. To modify the appearance of one of these layers follow these steps:

Step 1 Double click on the layer of interest in the layer panel to bring up the layer properties window, e.g. the **Point sources** layer.

Step 2 Select **Marker** from the left panel, as shown in **Figure 16**.

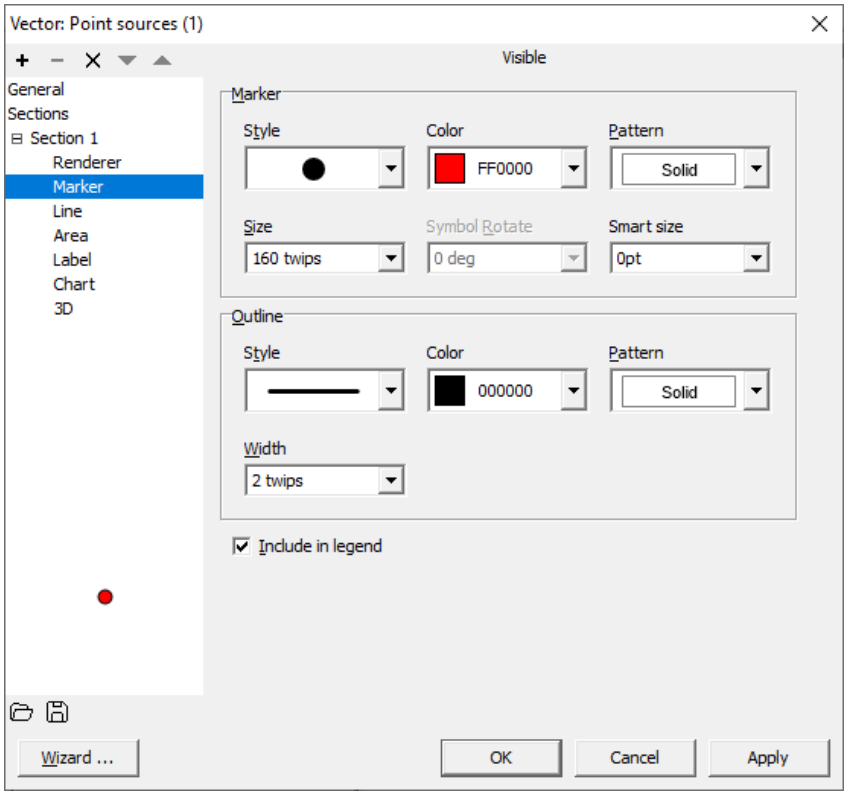


Figure 16 - The **Marker** section of the layer properties screen.

Step 3 The **Marker** and **Outline** sections can be used to alter the look of the markers used to represent the features in this layer.

Step 4 Click on the **OK** button to make the changes and return to the Mapper window, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the Mapper window.

4.3 Modifying the appearance of a line layer

Coloured lines are used to represent any layer of data defined as lines such as line sources. To modify the appearance of this layer follow these steps:

Step 1 Double click on the layer of interest in the layer panel to bring up the layer properties window, e.g. the **Line sources** layer.

Step 2 Select **Line** from the left panel, as shown in **Figure 17**.

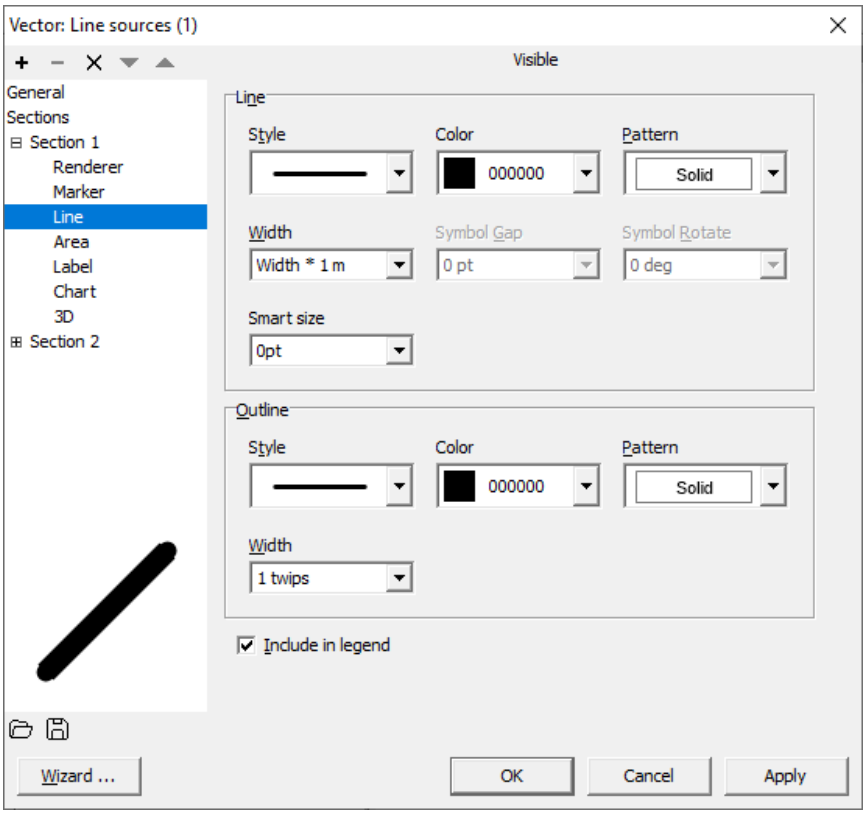


Figure 17 - The **Line** section of the layer properties screen.

Step 3 The **Line** and **Outline** sections can be used to alter the appearance of the lines used to represent the features in this layer.

Step 4 Click on the **OK** button to make the changes and return to the Mapper window, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the Mapper window.

4.4 Modifying the appearance of an area layer

Coloured areas are used to represent any layer of data represented by two-dimensional regions in the Mapper, such as area or volume sources. To modify the appearance of one of these layers follow these steps:

Step 1 Double click on the layer of interest in the layer panel to bring up the layer properties window, e.g. the **Area sources** layer.

Step 2 Select **Area** from the left panel, as shown in **Figure 18**.

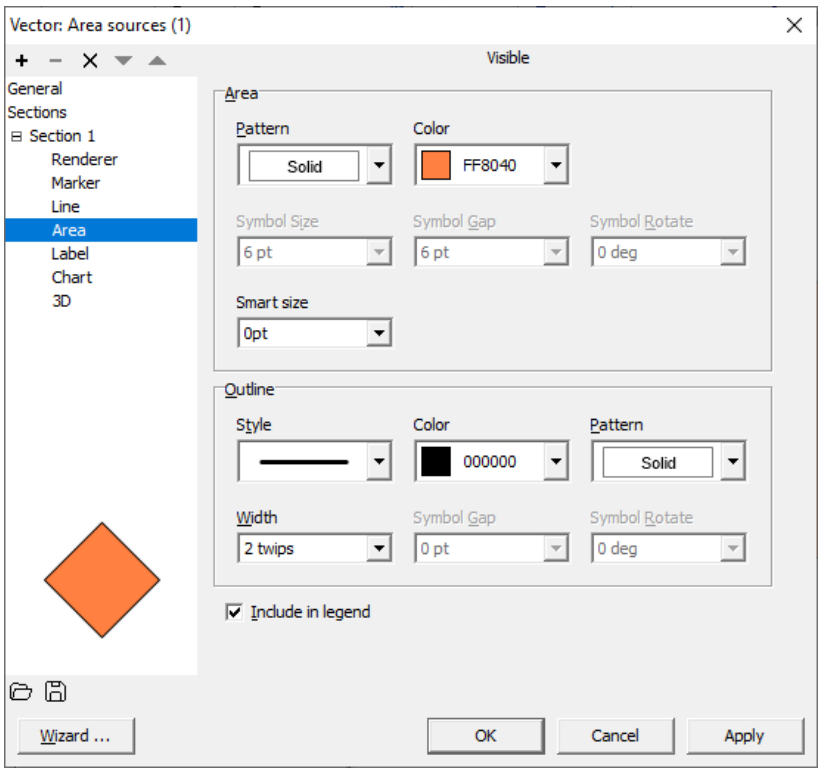


Figure 18 - The **Area** section of the layer properties screen.

Step 3 The **Area** and **Outline** sections can be used to alter the appearance of the shapes used in this layer.

Step 4 Click on the **OK** button to make the changes and return to the Mapper window, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the Mapper window.

4.5 Modifying the appearance of a contour layer

The colour scheme and number of contour levels used for a contour plot can be changed either manually, by altering each of the individual levels, or by applying a new colour scheme.

The contour levels can be manually altered by double clicking on the appropriate contour layer in the layer panel to open up the layer properties window and then selecting **Grid** from the left panel, as shown in **Figure 19**.

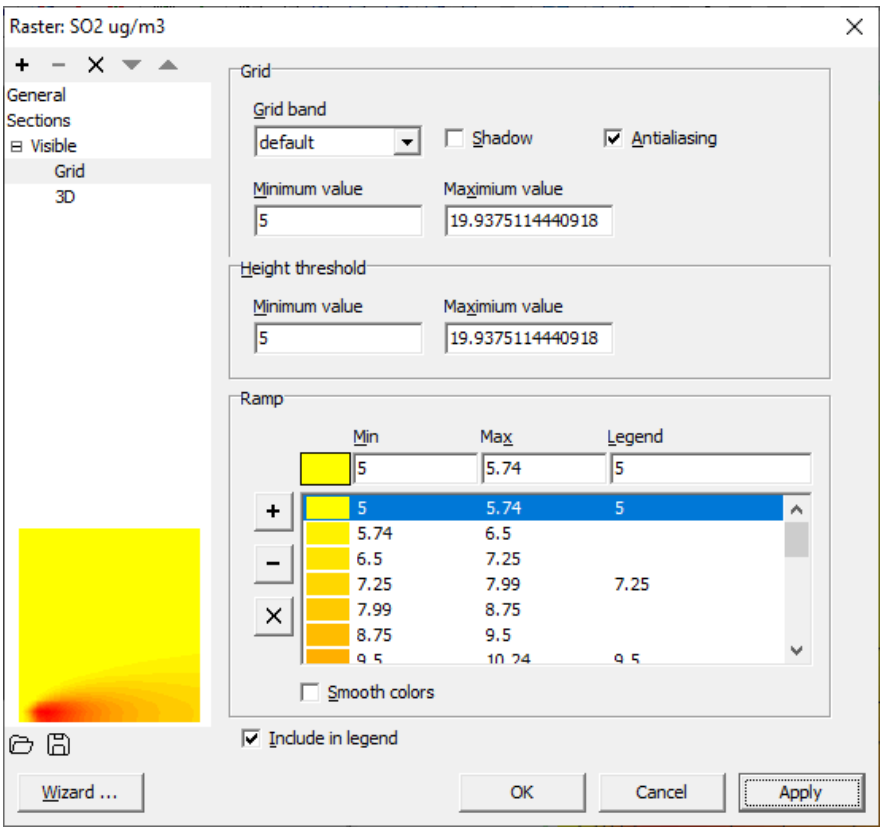


Figure 19 - The **Grid** section for the layer properties screen for a contour layer.

To select a contour level, click on it. The following options are then available:

- Add a new contour level with the same properties as the current selected contour level just above the current contour level by clicking on the **Plus** button  next to the contour levels.
- Delete the current selected contour level by clicking on the **Minus** button next  to the contour levels.
- Delete all of the contour levels by clicking on the **Cross** button  next to the contour levels.
- Alter the **Min**, **Max** or **Legend** values of the contour level by editing the contents of the text boxes.

- Alter the colour of a particular contour level by selecting it and then clicking on the coloured rectangle to the left of the **Min** text box  . This brings up the **Color** screen, as shown in **Figure 20**. The colour can be changed by either clicking on the colour wheel, using the sliders or editing the contents of the text boxes. There are also some pre-set colours which can be viewed by clicking the '**>**' button. The lower slider (or **Alpha** text box) alters the colour transparency, which can be useful for 'seeing through' certain regions of the contour plot. Click **OK** or **Cancel** to return to the layer properties screen.

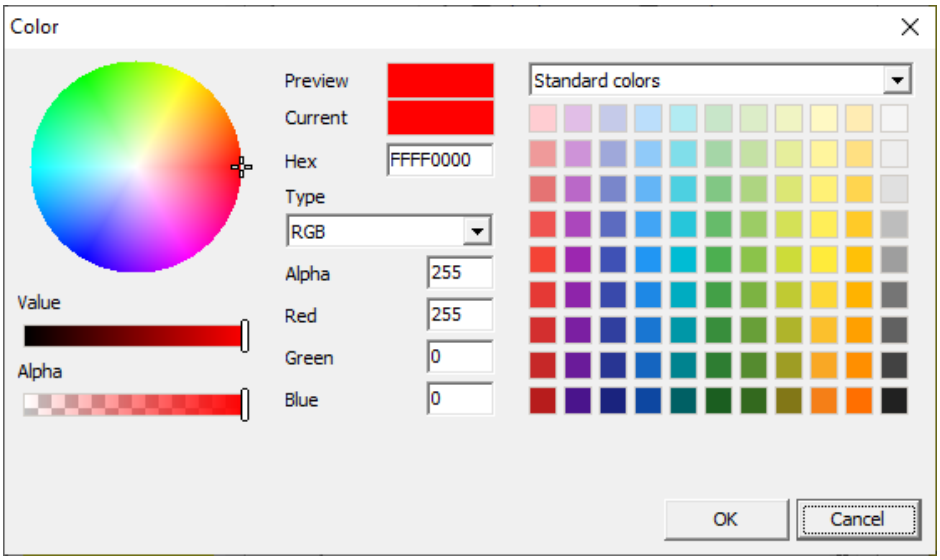


Figure 20 - The **Color** screen

When the appropriate changes have been made, click on the **OK** button to make the changes and go back to the Mapper window, click the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the Mapper window.

A facility exists to automatically create a new graduated colour scheme for a contour layer. To alter the colour scheme in this way, follow these instructions.

Step 1 Double click on the contour layer you wish to alter in the layer panel to bring up the layer properties window.

Step 2 Click on the **Wizard**... button to start the **Grid Ramp Wizard**.

Step 3 Choose **Simple classification** to define a colour ramp with equally spaced intervals or **Advanced classification** to get more interval spacing options, then click **Next >>**. Choosing **Simple classification** brings up the window shown in **Figure 21**.

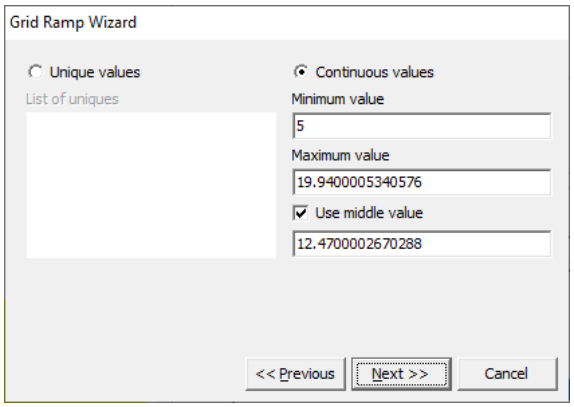


Figure 21 - The **Grid Ramp Wizard** screen for setting the range of values.

Step 4 This screen allows a choice between colouring by **Unique values** or by **Continuous values**. For the contour layer, it is appropriate to choose **Continuous values** so there is a smooth gradation in colours from low to high values. From here set the **Minimal** and **Maximal** values.

Step 5 If required, set a middle value. If this is not required then uncheck the **Use middle value** check box, then click **Next >>**. Setting the values brings up the window shown in **Figure 22**.

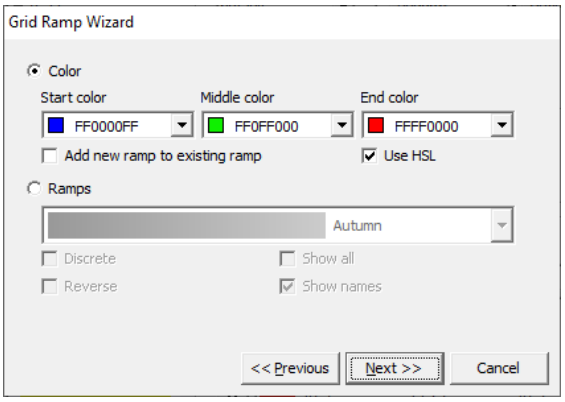


Figure 22 - The **Grid Ramp Wizard** screen for creating a new contour colour scheme.

Step 6 Choose **Color** to select your own **Start color**, **Middle color** and **End color**, or **Ramps** to choose from a pre-defined set of colour ramps, then click **Next >>**. Choosing **Color** brings up the window shown in **Figure 23**.

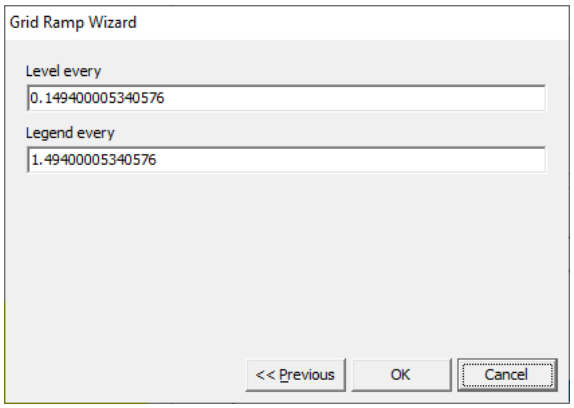


Figure 23 - The **Grid Ramp Wizard** screen for defining the intervals.

Step 7 Set the spacing required between colour levels, **Level Every**, and the spacing between legend entries, **Legend Every**.

Step 8 Click on the **OK** button.

Step 9 Uncheck the **Shadow** checkbox in the **Grid** section.

Step 10 If required, manually edit the contour levels.

Step 11 Click on the **OK** button to make the changes and return to the Mapper window, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the Mapper window.

Step 12 To update the contour lines so that they match the new contour levels, right-click on the contour lines layer associated with the modified contour layer in the layer panel, and click **Refresh contours**.

4.6 Colouring a layer according to its properties

You may wish to colour a layer according to its properties. For example, you may wish to colour a terrain file according to elevation; or a spatially varying roughness file according to the roughness value; or you may wish to colour sources according to their emission rates. This section describes how to make these changes.

Step 1 Double click on the layer which you wish to modify, for example the terrain layer, to bring up the properties window, see **Figure 24**.

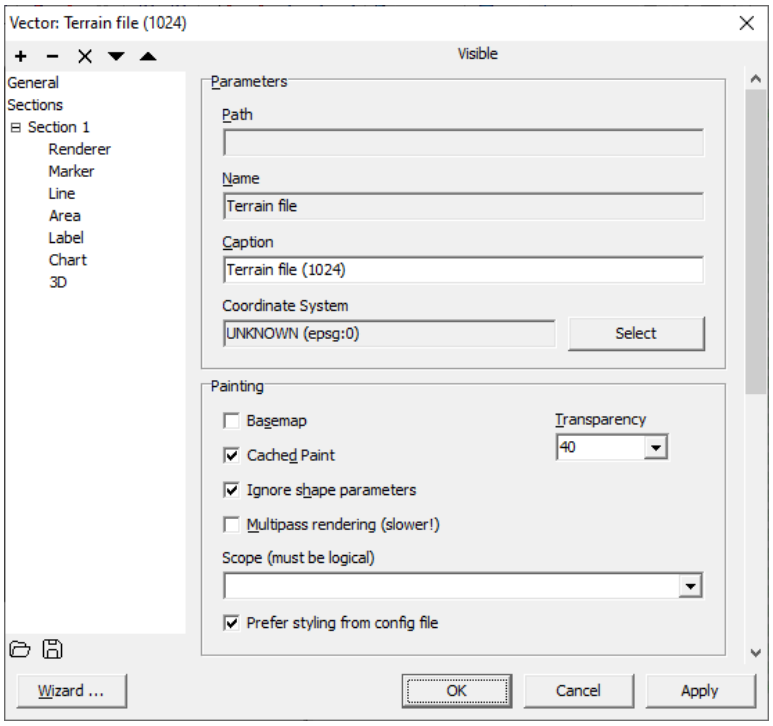


Figure 24 - The layer properties screen for a terrain file

Step 2 Click on the **Wizard...** button to start the **Rendering Wizard**.

Step 3 Choose whether to use a **Simple** or **Advanced classification**, then click **Next >>**. Choosing **Simple classification** brings up the window shown in **Figure 25**.

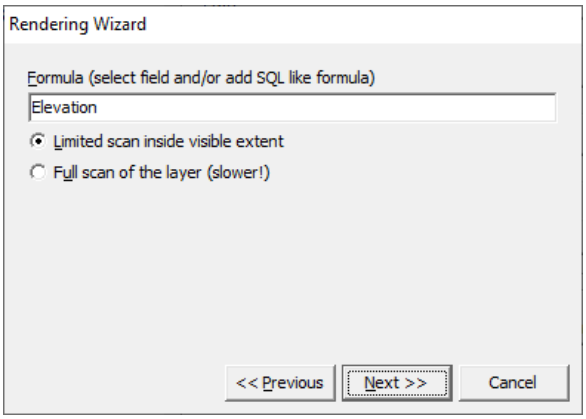


Figure 25 - The **Rendering Wizard** dialogue box

Step 4 Using the drop down menu select the property according to which you wish to colour the layer. For example, for a terrain layer select **Elevation**. Click on **Next >>**.

Step 5 The next screen allows a choice between colouring by **Unique values** or by **Continuous values**. The choice depends on the property you are using to colour the layer. For the terrain layer, for example, it is appropriate to choose **Continuous values** so there is a smooth gradation in colours from low elevations to high elevations. On the other hand, when plotting surface roughness, for example, the changes in roughness may be discrete, and it is likely that there will be only a small number of different roughness values across the region, so colouring by **Unique values** is more appropriate. The choice can be made by selecting the appropriate radio button, and in the continuous case you can also specify the range of values to use when colouring the layer, see **Figure 26**. Click on **Next >>**.

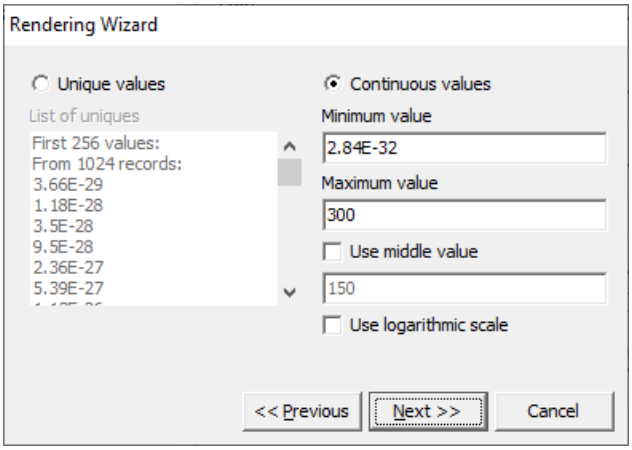


Figure 26 - Selecting unique or continuous values

Step 6 On the next screen make sure that **Render by Color** is selected and then click on **Next >>**.

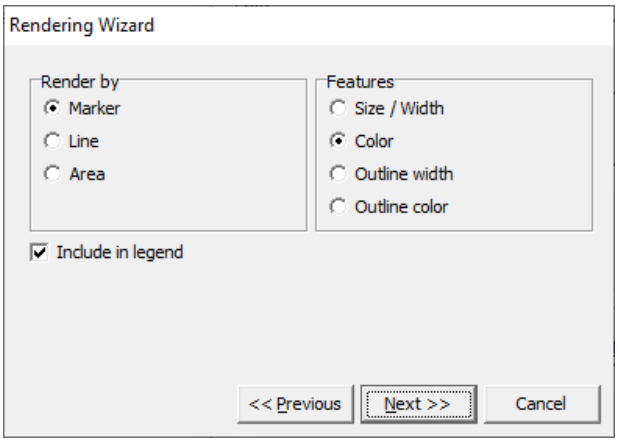


Figure 27 - **Render by Color**

Step 7 The next step is to choose the colour scheme for the layer. The method for doing this depends on whether the layer is being coloured by continuous or unique values.

For continuous values select **Renderer** from the left panel, see **Figure 28**. You

can then change the minimum and maximum values at which the start and end colours will be applied, as well as the start and end colours themselves using the drop-down menus.

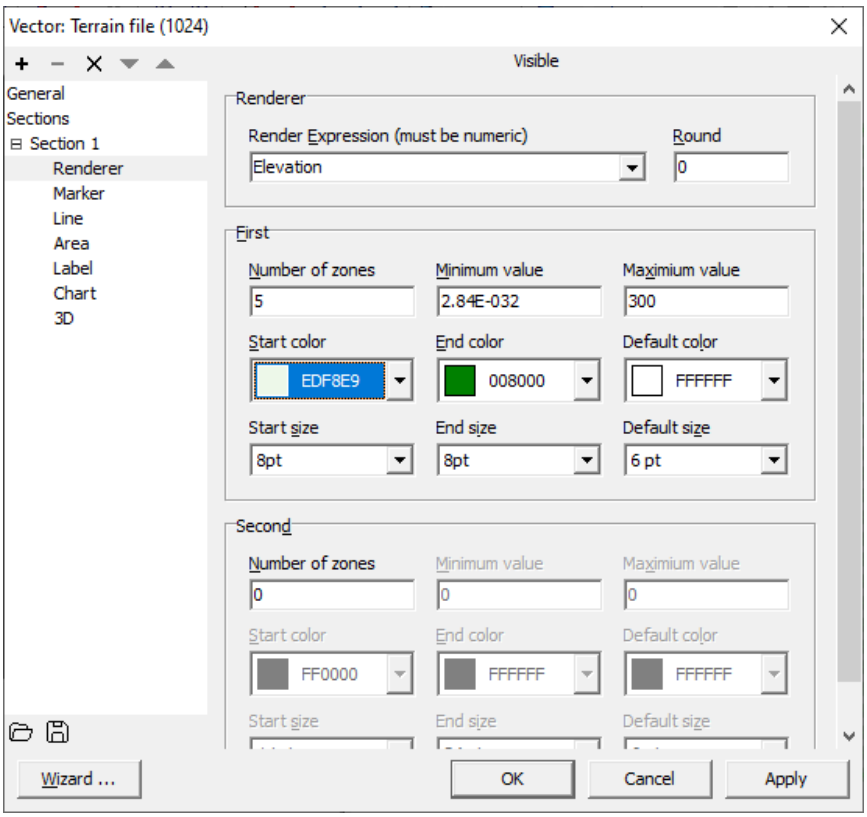


Figure 28 - The **Renderer** section of the layer properties screen

For unique values, a separate colour can be selected for each individual level. To do this, select the appropriate field, e.g. **Marker**, from each section listed in the left panel in turn, as shown in **Figure 29**. Then choose the colour for that level using the **Color** drop-down menu.

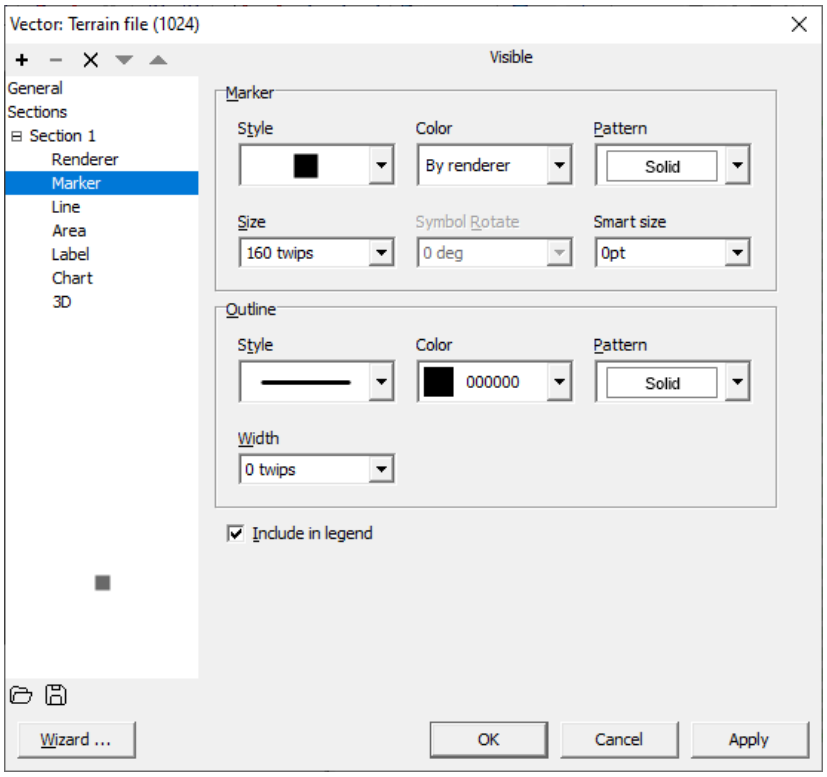


Figure 29 - The **Marker** section of the layer properties screen

Step 8 Finally, click **OK** to apply your changes.

If you wish to colour point sources by their emission rates, for example, then the steps above can be followed, but at the stage when you select the formula from the drop-down window in the **Rendering Wizard** you should choose the pollutant emission rate with which you want to colour the layer, e.g. [NOx (g/s)].

---

*It is possible to define new formulas in the Rendering Wizard, e.g. if you wished to colour area sources by their total NO_X emission rate you could specify the formula: [NO_X (g/m2/s)]*GIS_AREA by typing in the formula box.*

4.7 Displaying feature names

Any of the default ADMS layers can display the names of the features within that layer. To show these names follow these instructions.

Step 1 Double click on the layer of interest in the layer panel to bring up the layer properties window.

Step 2 Select **Label** from the left panel, as shown in **Figure 30**.

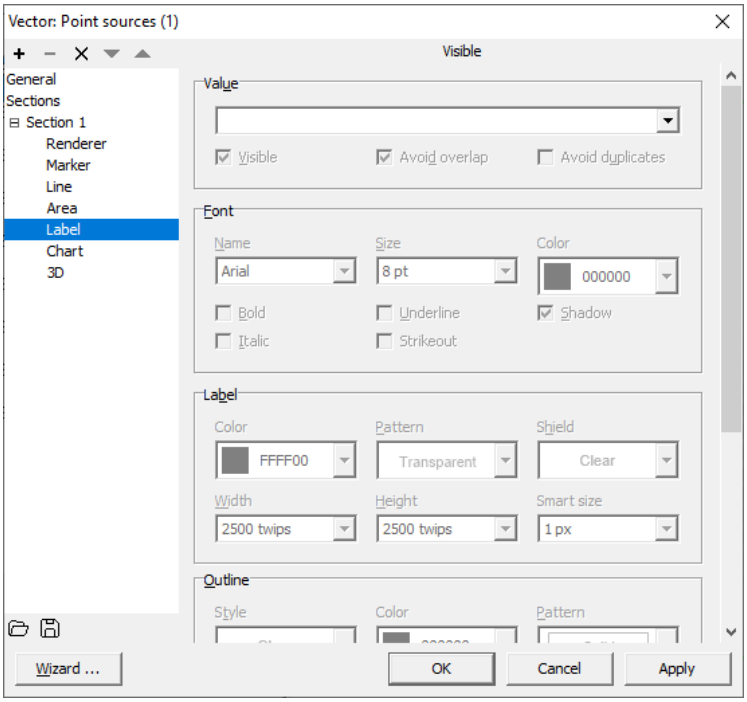


Figure 30 - The **Label** section of the layer properties screen

Step 3 In the **Value** section, use the drop-down menu to set the **Field** to **Name**.

Step 4 The **Font**, **Label**, **Outline** and **Position** sections can then be used to alter the properties of the labels.

Step 5 The labels can be removed by unchecking the **Visible** checkbox in the **Value** section.

Step 6 Click on the **OK** button to make the changes and return to the Mapper window, or click on the **Apply** button to make the changes but remain in the layer properties window, or click on the **Cancel** button to discard the changes and return to the Mapper window.

4.8 Exporting, importing, saving and reloading layer settings

Once the layers have been modified as desired the layer settings can be exported so that they can be used in other layers or in other maps. Similarly, existing layer settings can be imported. The layer settings can also be saved and reloaded. The steps are highlighted in the examples below.

Step 1 Right click on the layer you wish to export. This brings up the menu shown in **Figure 31**.

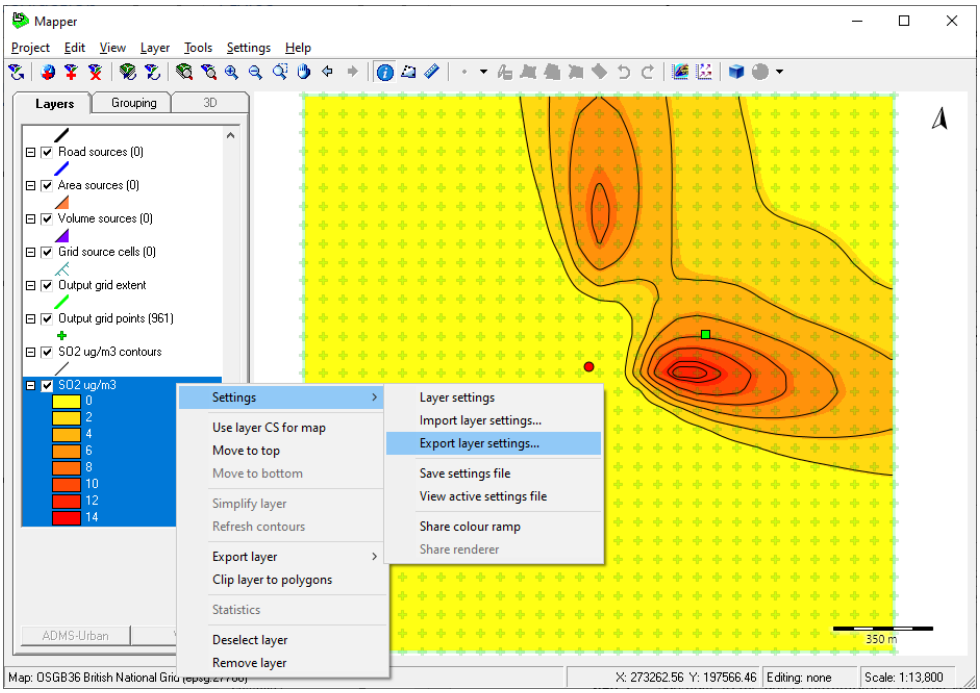


Figure 31 - The menu for exporting/importing layer settings.

Step 2 Click on **Settings > Export layer settings...** .

Step 3 A window opens prompting you to save the configuration file for the layer. Choose a location and save the file with extension **.ttkstyle*.

Step 4 Right click on the layer to which you wish to import the settings. From the menu select **Settings > Import layer settings...** .

Step 5 Navigate to the layer configuration file that you have just created and click on **Open**.

Step 6 The layers are now displayed using the same layer settings.

The settings for an individual layer can be saved by right-clicking on the layer and selecting **Settings > Save settings file**. These settings are then stored in a file so if you wish to return to the saved settings, after making modifications, you can right-click on the layer and select **Settings > Reload settings file**. This undoes any changes you have made.

SECTION 5 Additional features

This section describes some of the more advanced features available in the Mapper.

5.1 Setting the map coordinate system

There are two aspects to consider when setting coordinate systems within the Mapper: the map and the individual layers. The map coordinate system determines the way the layers are displayed in the map view window, and you can choose one that suits your needs provided it is compatible with the data. It can be either a projected or geographic system. On the other hand, the layers each have a coordinate system, but these must correspond to the data values in the layer. For example, the ADMS layers' coordinate system is set in the parent interface and acts on all the ADMS layers. This should be set to whatever coordinate system you are using for your work in the model. For user layers you often find the data have associated projection or world files that will define the coordinate system for the layer automatically. This allows layers with data in different coordinate systems to be projected correctly onto the map coordinate system for easy comparison.

The map coordinate system is used as the default coordinate system for user layers with no associated world file or other georeferencing information.

The coordinates of a point in the map view window (as given in the status bar) are in the units associated with the map coordinate system. For example, if the map coordinate system has been set to **WGS 84** then the coordinates are degrees longitude and latitude.

The map coordinate system must be selected before certain features of the Mapper can be used; for example, before exporting Mapper layers to **.kml* files for use in Google Earth.

The coordinate systems for the ADMS layers should not be changed in the Mapper – these are set in the parent interface. See the relevant model User Guide for more details.

To set the coordinate system that is used in the Mapper select either **Use the ADMS coordinate system** or **Set map coordinate system** from the **Project** menu, as shown in **Figure 32**. The first option ensures that the system currently defined in the ADMS interface is used in the Mapper; the second option allows you to choose the coordinate system from an extensive list of options.

You should choose a map coordinate system that is compatible with the ADMS coordinate system.

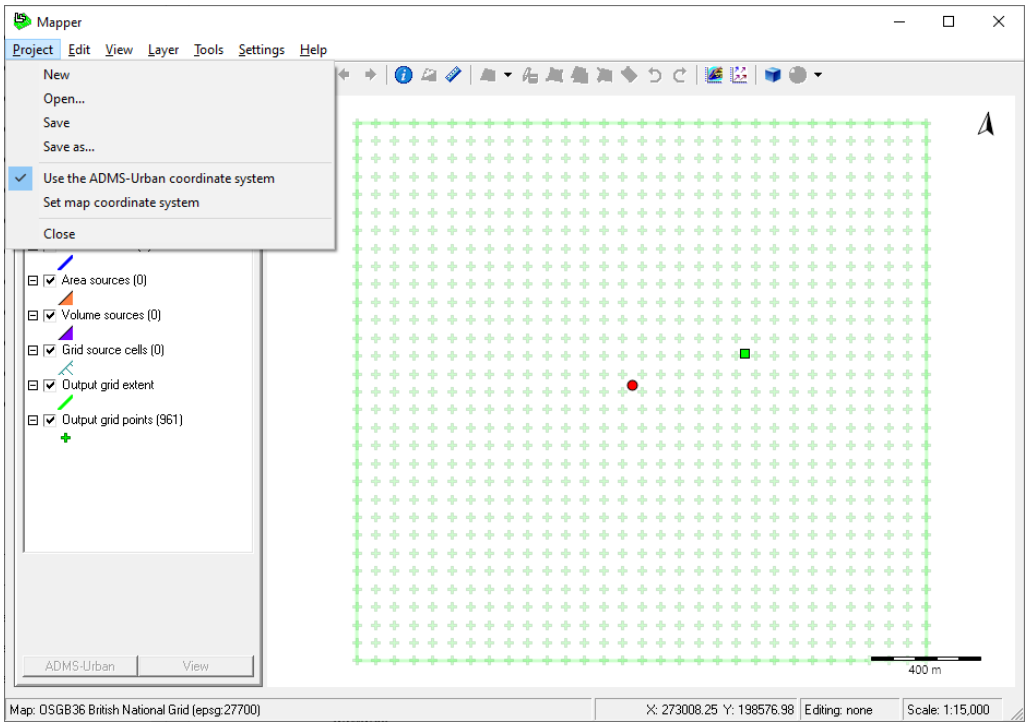


Figure 32 - The coordinate systems options in the **Edit** menu.

If the **Set map coordinate system** option is selected then a new window appears, as shown in **Figure 33**, where the coordinate system can be chosen from an extensive list of options. To select a **Projected system**, click on the radio button and then choose the system from the drop-down list.

---

*Rather than scrolling through the entire list of options to find your desired choice you can click in the box and type the name of the coordinate system you wish to use. For example, to quickly find the Ordnance Survey grid for Great Britain type **OSGB**. You can also type part of the name and then click on the drop-down arrow to see a shorter list of options.*

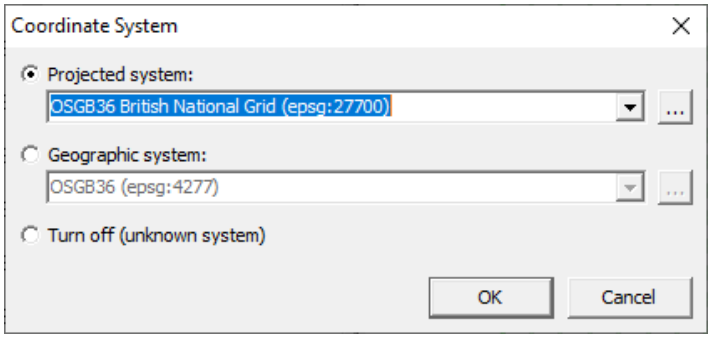


Figure 33 - Choice of the coordinate system

A **Geographic system** can be chosen in a similar way to a **Projected system**.

If you wish to use the **Unspecified regular Cartesian** grid from the model interface, i.e. if the coordinates are site specific, for example distances relative to a source centre, then the **Turn off (unknown system)** option must be chosen.

5.2 Adding a background map

Background maps can easily be added to the map view window in the Mapper. This is done using the **Add background map** button on the toolbar. This uses a WMS link to add a dynamic layer to the map view. See Section 5.4 for more details.

Step 1 Check that a valid coordinate system has been set for the Mapper. See Section 5.1.

Step 2 Click on the **Add background map** button on the toolbar. Alternatively, you can select the **Add background map** option from the **Layer** menu.

Step 3 A map layer will be added to the Mapper layer panel.

The map service used for the background map will depend upon the BackgroundMap setting in the [MapperInterface] section of the parent model's configuration file. An example service is supplied with the model install via a Protocol Layer Connector file (.ttkwp*), but you can change this if you have an alternative WMS you would prefer to use.*

5.3 Displaying background images

Background images can be added to the map view window in the Mapper. This is done using the **Add Layer** button on the toolbar. The add layer feature allows files with a wide variety of formats to be loaded into the Mapper.

To load an image to use as a background map, the image must be correctly georeferenced, which can be done using world files or MapInfo **.tab* files. For British OS Map data, world files are available from the Ordnance Survey website. These instructions show how to load an OS map tile as a background map using the corresponding world file to georeference it.

Step 1 Ensure the map tile image file (**.tif*) and world file (**.tfw*) have the same name and are in the same directory.

Step 2 Check that a valid coordinate system has been set for the background layer. See Section 5.3.1.

Step 3 In the Mapper click on the **Add Layer** button on the toolbar. Alternatively, you can select the **Add layer** option from the **Layer** menu.

Step 4 The **Add Layer** screen is then displayed, as shown in **Figure 34**. Browse to find the **.tif* file for the map tile, then click on the **Open** button.

As an alternative to Steps 3-4, the file can be drag-dropped from Explorer onto the layer panel.

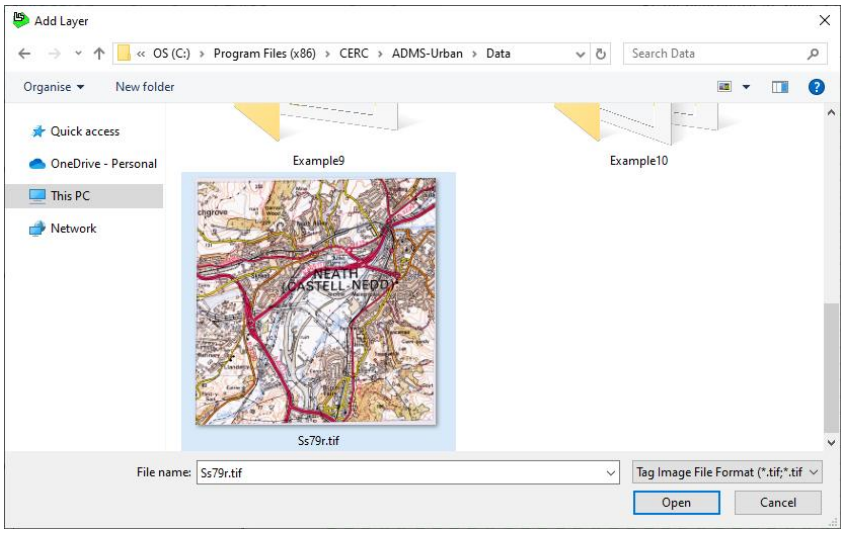


Figure 34 - The **Add Layer** screen.

Step 5 The map tile will be added to the map view window. This layer will be at the top of the **Layers** tab and will need to be moved down to allow the ADMS layers to be seen.

5.3.1 Setting the coordinate system for a background layer

To set a coordinate system for a background map you will need to create a **.prj* file as follows. Suppose that the background map you wish to add has the filename *map.tif*. Then follow these steps:

Step 1 Double click on one of the layers in the layer panel. A new window will appear.

Step 2 Select **General** from the left panel to view the layer information, as shown in **Figure 35**.

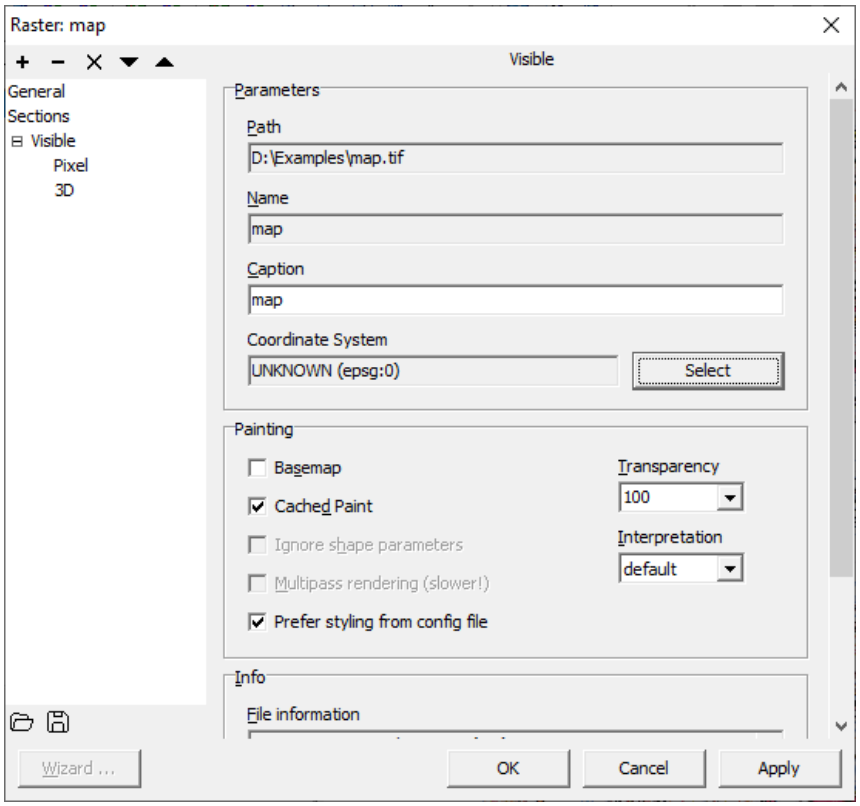


Figure 35 - The **General** section of the layer properties screen

Step 3 Click on the **Select** button next to the **Coordinate System** cell to bring up the dialogue box shown in **Figure 33**.

Step 4 Choose the coordinate system you wish to use in the background layer from the available list and then click on the ellipsis (...). This will bring up the **Coordinate System Setup** screen shown in **Figure 36**.

You must choose a coordinate system that is the same as, or consistent with, the coordinate systems used for the other layers.

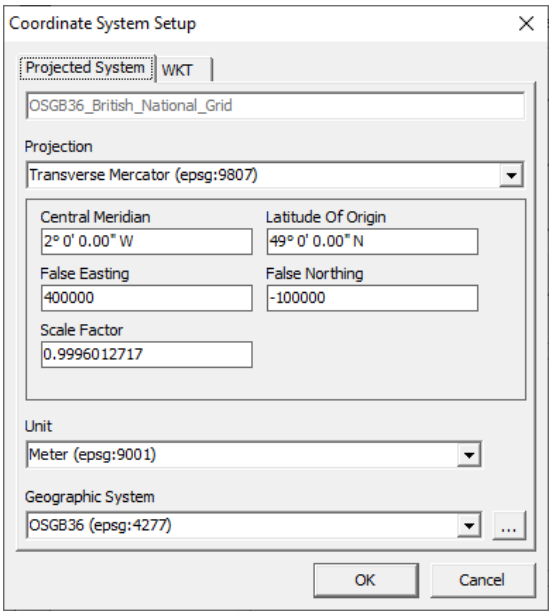


Figure 36 - The **Coordinate System Setup** screen.

Step 5 Click on the **WKT** tab. The screen will be similar to that shown in **Figure 37**.

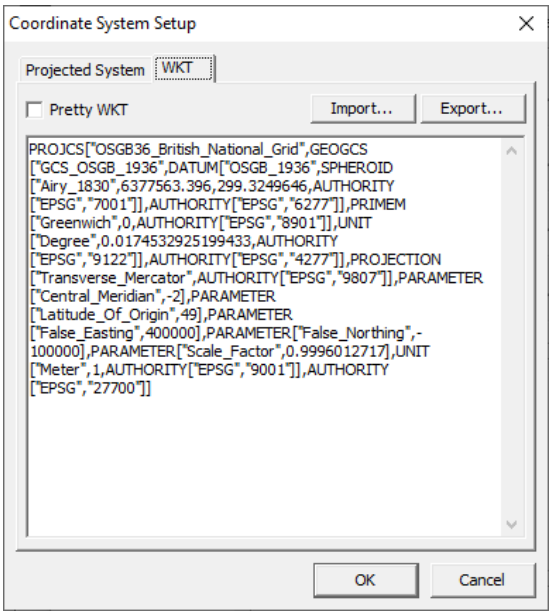


Figure 37 - The **WKT** tab.

Step 6 Click on **Export...** and save the file in the same location as the *map.tif* file with the file stem *map.tif* and extension *.prj*, i.e. save the file with name *map.tif.prj*.

The same procedure applies to other file types, not just *.tif *files. In general for a file with name* <filename>.ext, *where* .ext *is any extension, the name of the projected file must be* <filename>.ext.prj

---

Step 7 After you have saved the **.prj* file click **Cancel** to close the three sub-screens that appeared in the process of creating the **.prj* file.

5.4 Adding background map imagery from a Web Map Service (WMS)

It is possible to add a background map layer in the Mapper from a Web Map Service (WMS). The background map imagery can help display and locate your data and results. There are two methods for adding background map imagery: one is to use the **Add WMS** command from the **Layer** menu to specify a URL to a web map service; the other is to use a special Protocol Layer Connector file (*.ttkwp*).

5.4.1 Adding background map imagery with the Add WMS layer command

Step 1 First ensure that the coordinate systems in the ADMS interface and the Mapper are set appropriately for the map that you wish to use. For example, in the UK the coordinate system for the ADMS interface is often set to OSGB 1936 British National Grid. (Refer to your relevant model user guide for full details on how to do this.) In the Mapper the coordinate system can either be set to use the ADMS coordinate system (i.e. the coordinate system specified in the ADMS interface), or a particular coordinate system can be defined (e.g. WGS 84). This is done under the **Project** menu. See Section 5.1 for full details on how to set the coordinate system in the Mapper. Also ensure that all the coordinates in the interface are given using the specified coordinate system.

Step 2 Select the **Add WMS** command from the **Layer** menu.

Step 3 A window appears as shown in **Figure 38**. Specify the URL for the web map service and click **OK**.

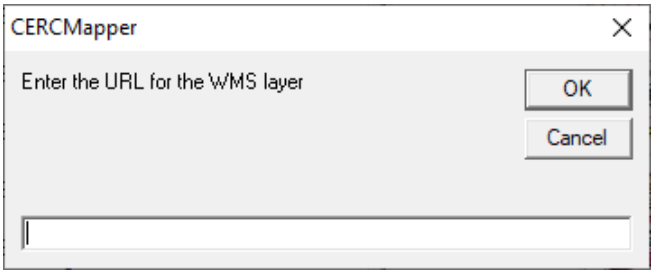


Figure 38 - The **Add WMS Layer** window.

Step 4 The map will be displayed. The layers may need to be reordered to view the map and other layers properly (see Section 1.7.4). An example is shown in **Figure 39**.

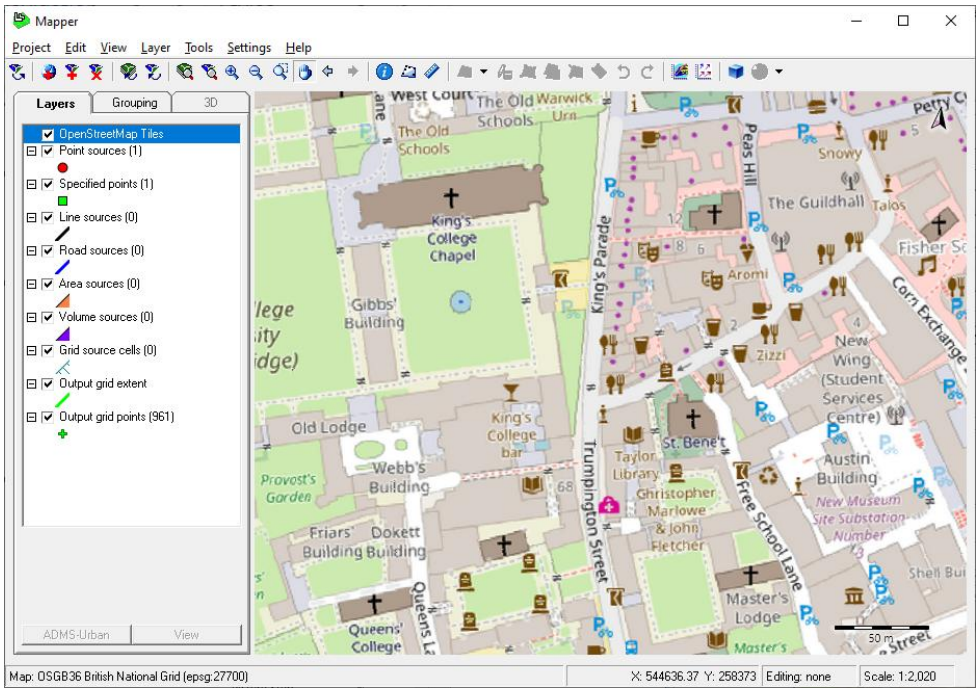


Figure 39 - Background map imagery in the Mapper; © OpenStreetMap contributors www.openstreetmap.org/copyright.

For best results, Map layers and data layers should use the same coordinate system. Some coordinate systems may not be mutually compatible.

5.4.2 Adding background map imagery using a Protocol Layer Connector file

Alternatively a background map may be added by using a Protocol Layer Connector file as described here. An example file is included in your ADMS install directory: this links to the OpenStreetMap WMS.

Step 1 Ensure that the coordinate systems in the ADMS interface and the Mapper are set appropriately for the map that you wish to use. (Refer to the relevant model user guide and Section 5.1 of this user guide).

Step 2 Click on the **Add Layer** tool.

Step 3 Navigate to the directory containing your **.ttkwp* file. Select **Files of type: Protocol Layer Connector (*.ttkwp)** to display the files:

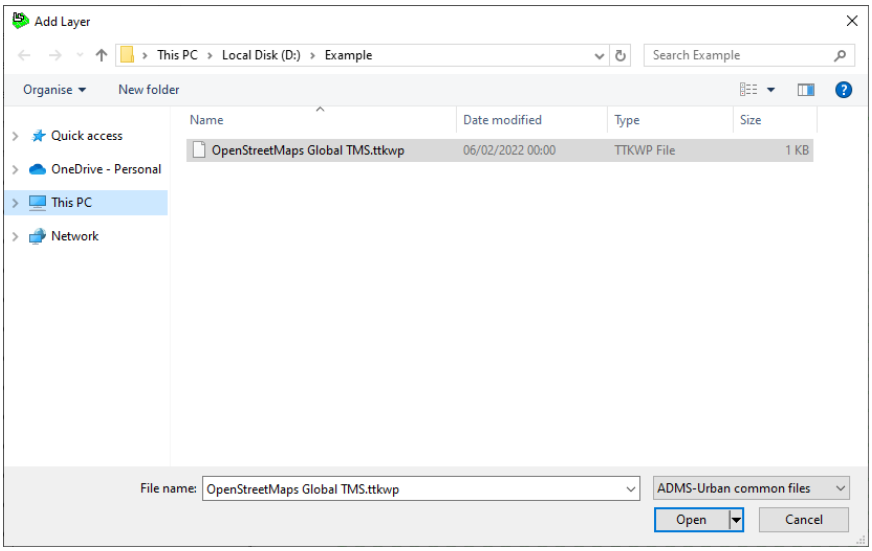


Figure 40 - Selecting Protocol Layer Connector files.

Step 4 Select a (**.ttkwp*) file that covers the geographic location of your sources and click **Open**.

As an alternative to Steps 2-4, the file can be drag-dropped from Explorer onto the layer panel.

Step 5 The new layer will be displayed in the Mapper. By default, the view is zoomed to the full extent of the new layer (**Figure 41**). The layers may need to be reordered so that the background image does not obscure the view of the other layers. Information on reordering layers can be found in Section 1.7.4.

Contact the CERC helpdesk for help setting up *.ttkwp *files for other maps.*

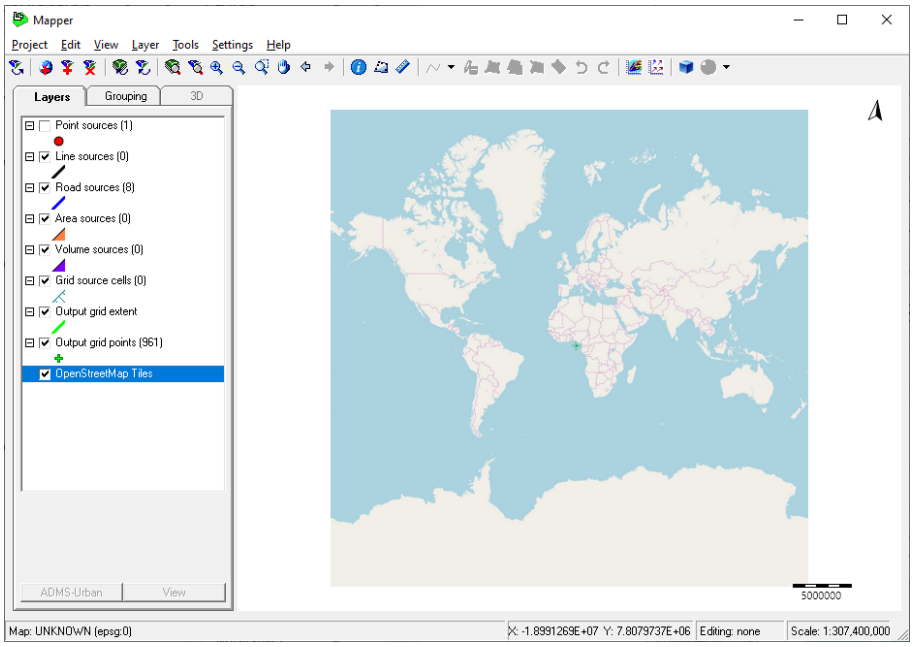


Figure 41 - New layer in the Mapper; © OpenStreetMap contributors www.openstreetmap.org/copyright.

Step 6 Select the output grid extent or largest source layer and select **Zoom to layer**.

Step 7 This will zoom to the correct location on the map (**Figure 42**).

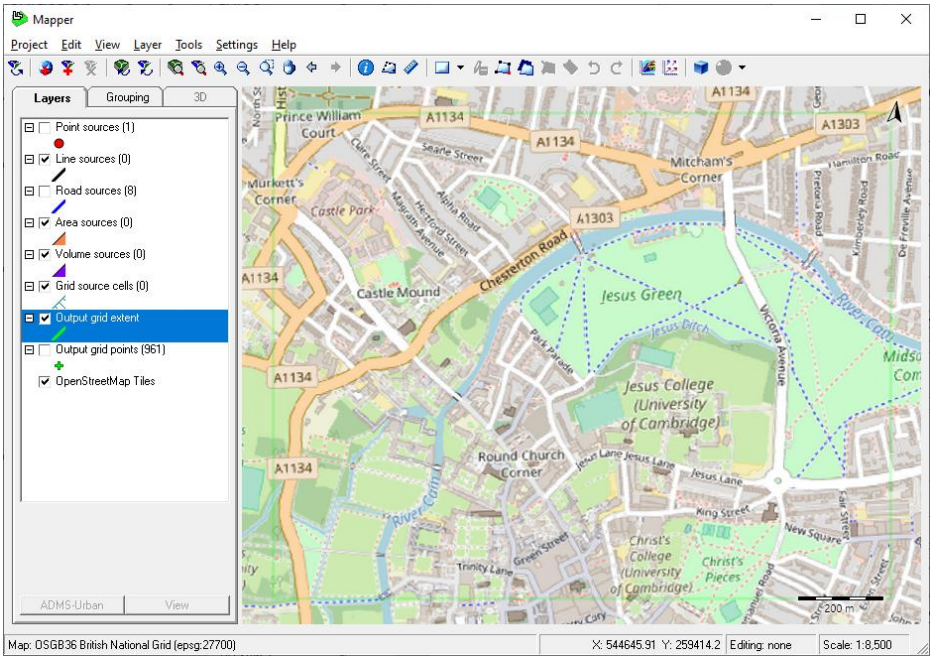


Figure 42 - Background image displayed in the Mapper; © OpenStreetMap contributors www.openstreetmap.org/copyright.

For best results, Map layers and data layers should use the same coordinate system. Some coordinate systems may not be mutually compatible.

5.5 Displaying data layers

Data from external files can be displayed in the Mapper using the **Add layer** toolbar button. Recognised formats such as ESRI shape files (*.shp*) and MapInfo Interchange Format files (*.mif*) will be displayed automatically, but generic delimited data files such as comma separated variables (*.csv*) will first prompt you to define data fields as well as selecting the data that you wish to display.

Data layers created from files are not editable, but their appearance can be modified as outlined in Section 4.

5.5.1 Using drag and drop

When you are selecting data files to add to the Mapper you can click on the **Add layer** tool, and browse to the file.

Alternatively you can locate the file that you wish to add to the Mapper in Windows Explorer, click and drag it to the layer panel of Mapper. If the file is a recognised format it will then be added immediately, otherwise further steps need to be followed as detailed in Section 5.5.3.

5.5.2 Data from recognised file formats

If you choose **Add layer** and select a file in a recognised format, such as an ESRI shape file (*.shp*), the Mapper will automatically display it in the map as a new layer.

This applies to all supported vector files; see Appendix A.3 for a list of recognised file types. For pixel (image) or grid files, see Section 5.2.

Step 1 Click on the **Add Layer** tool. A new window will appear as shown in **Figure 43**.

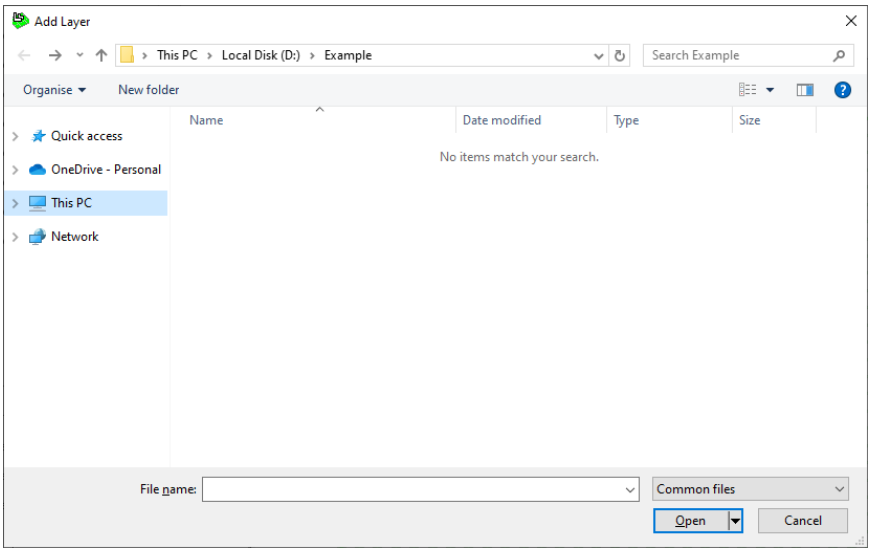


Figure 43 - Add Layer window.

Step 2 Change the file type from **Common files** to **ArcView Shape Files (*.shp)**.

Step 3 Navigate to the file you wish to view and click Open. The data will then be displayed in the Mapper as shown in **Figure 44**.

As an alternative to Steps 1-3 you can use drag and drop.

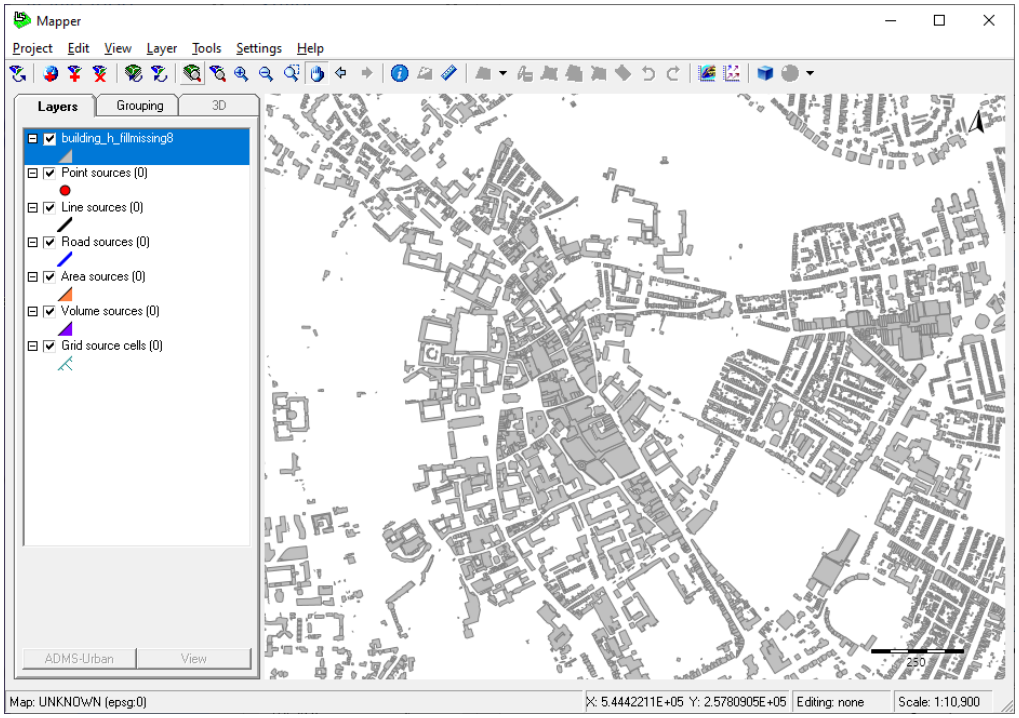


Figure 44 - Data from the shape file displayed in the Mapper.

Step 4 Information about the data can then be viewed. Select the information tool and click on one of the shapes. (Full details on the information tool can be found in Section 5.6). Information is then displayed in the **Attributes** table, as shown in **Figure 45**. The information available will depend on the file type and the file data.

It is possible to colour and label shapes according to their properties. See Sections 4.6 and 4.7 for further details.

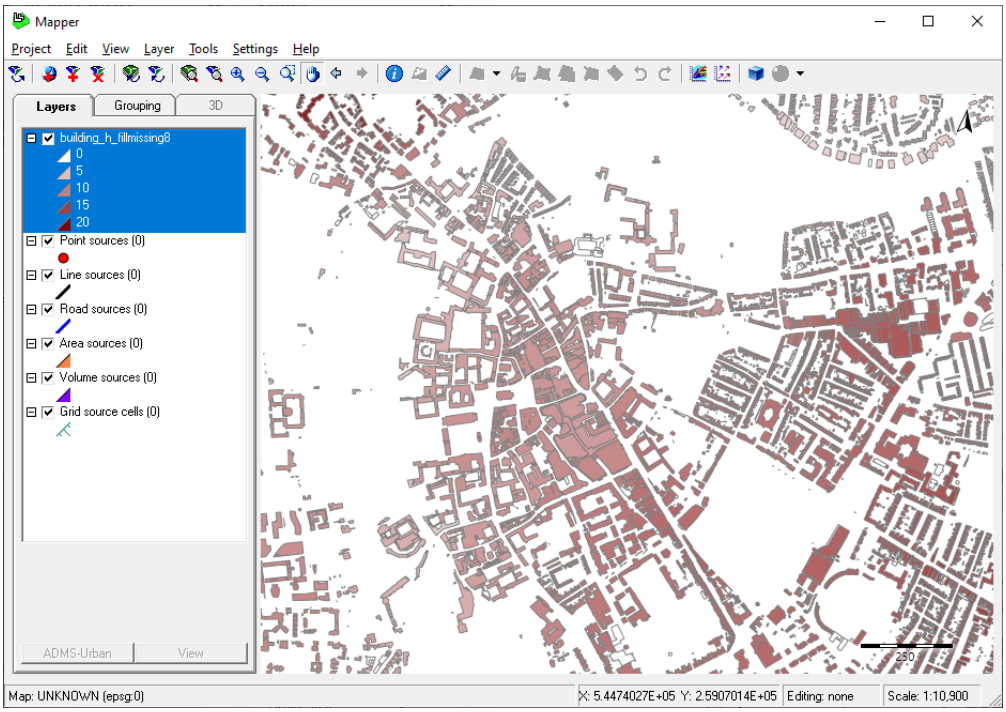


Figure 45 - Information from the numerical data displayed in the Mapper.

5.5.3 Data from CSV and other delimited text files

If you display data from a CSV file or unrecognised text file, the Mapper will attempt to load it into a layer, providing an intermediary window for selecting data from the file. If the file contains a header line, the Mapper will attempt to find it and identify the start of the data lines; otherwise you need to manually enter the column headers.

Data can be displayed as points, polylines or polygons but must be identified as such in the column headers by following a specific naming convention, while all other columns are treated as properties. In each case this can be done in the file or in the Mapper data window:

* For point layers, there must be an easting and a northing named appropriately in order to display the data.
    * The easting must be named as one of X, X(m), or X (m)
    * The northing must be named as one of Y, Y(m), or Y (m)
    * The elevation will be recognised if it is one of Z, Z(m), Z (m), or ELEVATION. Elevation can also be set to any column by right clicking the column header and selecting **Set as elevation field.** To clear, right click on the elevation field again and select **Clear elevation field**.

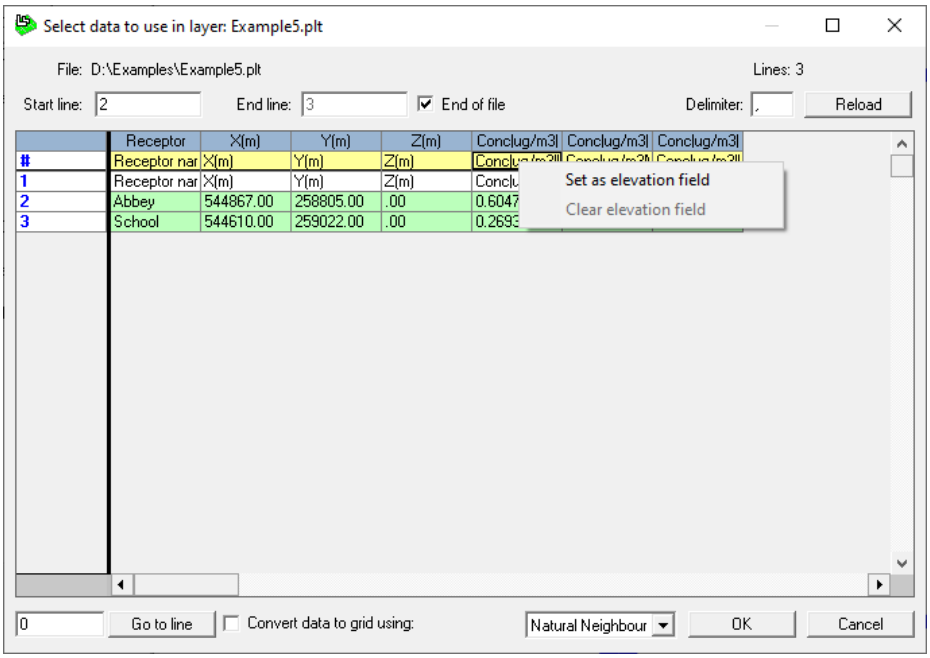


Figure 46 - Setting the Elevation field in the Mapper.

* For polyline layers, at least two appropriately named vertices must be supplied in order to display the data.
    * For the start point, the easting must be named as X1 and the northing must be named as Y1
    * For the next point, they must be named as X2 and Y2 respectively
    * Subsequent points can be provided, continuing the naming sequence. For each point both X and Y must be provided and they must form a continuous sequence starting from 1.
    * The elevation must be named as one of Z, Z(m), Z (m), or ELEVATION
* For polygon layers, at least three appropriately named vertices must be supplied in order to display the data.
    * For the start point, the easting must be named as pX1 and the northing must be named as pY1
    * For the next point, they must be named as pX2 and pY2 respectively
    * For the next point, they must be named as pX3 and pY3 respectively
    * Subsequent points can be provided, continuing the naming sequence. For each point both pX and pY must be provided and they must form a continuous sequence starting from 1.

    The polygon will be closed automatically, so the first and last vertices must be in different locations.

    ---

    * The elevation must be named as one of Z, Z(m), Z (m), or ELEVATION

The data window – shown in **Figure 47** – has the following options for data selection:

* The option to enter the **Start line** and **End line** (or select **End of file**). The delimiter can also be entered.
* An editable header line (yellow background) lets you choose which columns are included as properties in the layer. Columns that have '-' or whitespace for their header not included as layer properties.
* A right-click menu on the **#** column that has the following options:
  * **Set as first data line** – selects a new data start line.
  * **Set as last data line** – choose a new data end line.
  * **Copy to header** – replaces the contents of the editable header line with the selected line.
* **Go to line** button – Brings the line specified in the box to its left into view in the table.
* **Reload** button – Refreshes the table
* **Convert data to grid using:** – check this option at the bottom of the data window if you require a **.grd* file of your data. Once the selection box has been checked, set the grid field by right clicking the mouse over the relevant column of data. Then press the **OK** button.

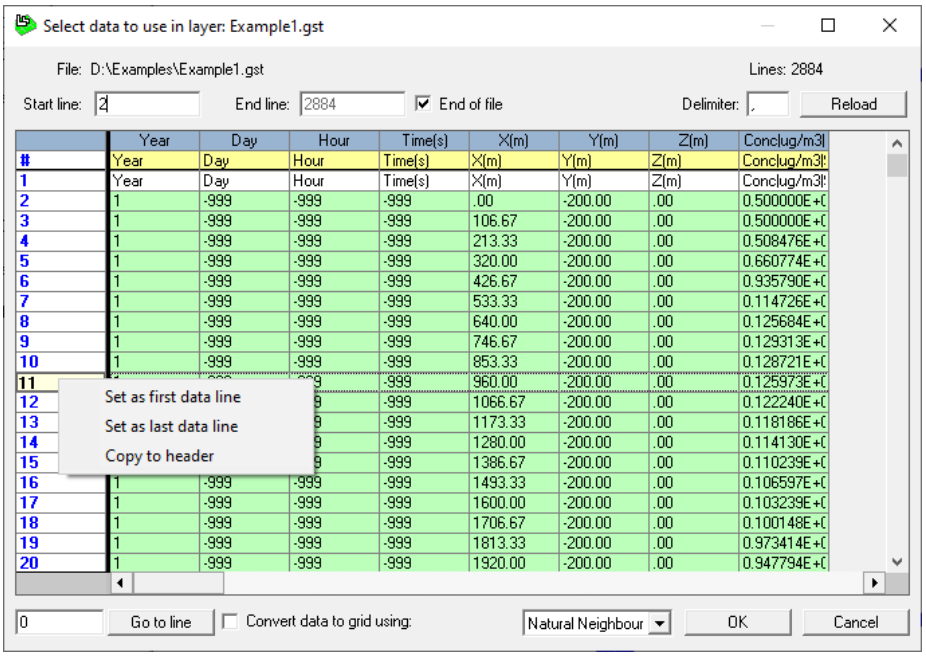


Figure 47 - Data window

Suppose that you wish to display data from a tunnel file *Tunnels.csv* alongside data in *Example6b.upl* (these examples files are not supplied with ADMS 6). Ensure that the UPL file is loaded and follow these steps:

Step 1 Click on the Add Layer tool. A new window will appear as shown in **Figure 48**.

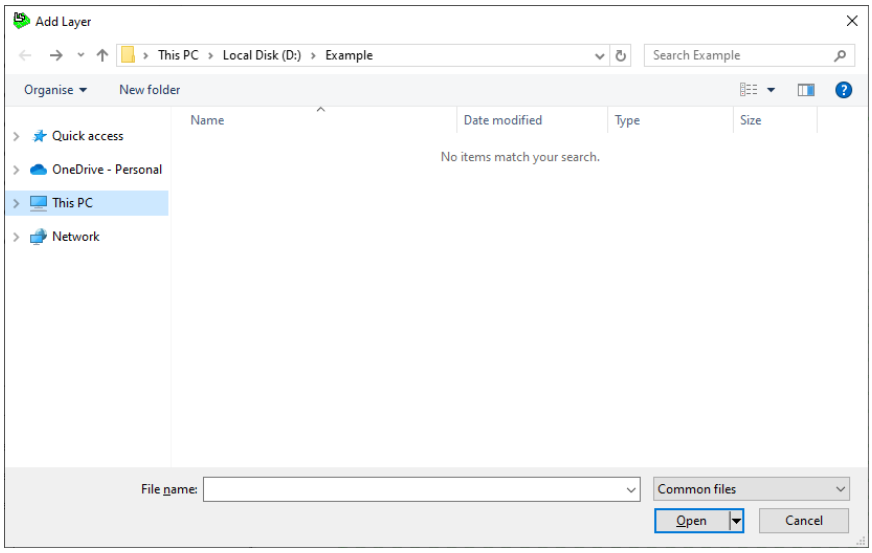


Figure 48 - Add Layer window.

Step 2 Change the file type from **Common files** to **All Vector Files**.

Step 3 Navigate to the **.csv* file you wish to view and click Open. A new window will appear as shown in **Figure 49**.

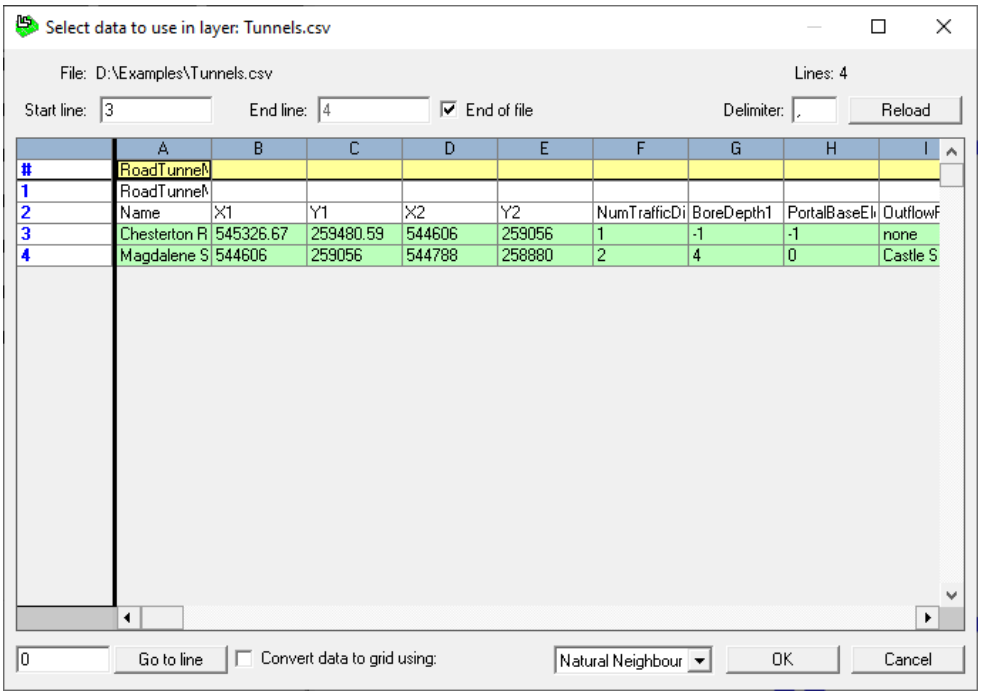


Figure 49 - Data from *Tunnels.csv*

Step 4 This window can be used to select which data you wish to display in the Mapper. See Section 3.4 for an example that requires data selection.

Step 5 Once you have selected the data that you wish to display click **OK**. The data will then be displayed in the Mapper as shown in **Figure 50**. The default line appearance is a thin black line.

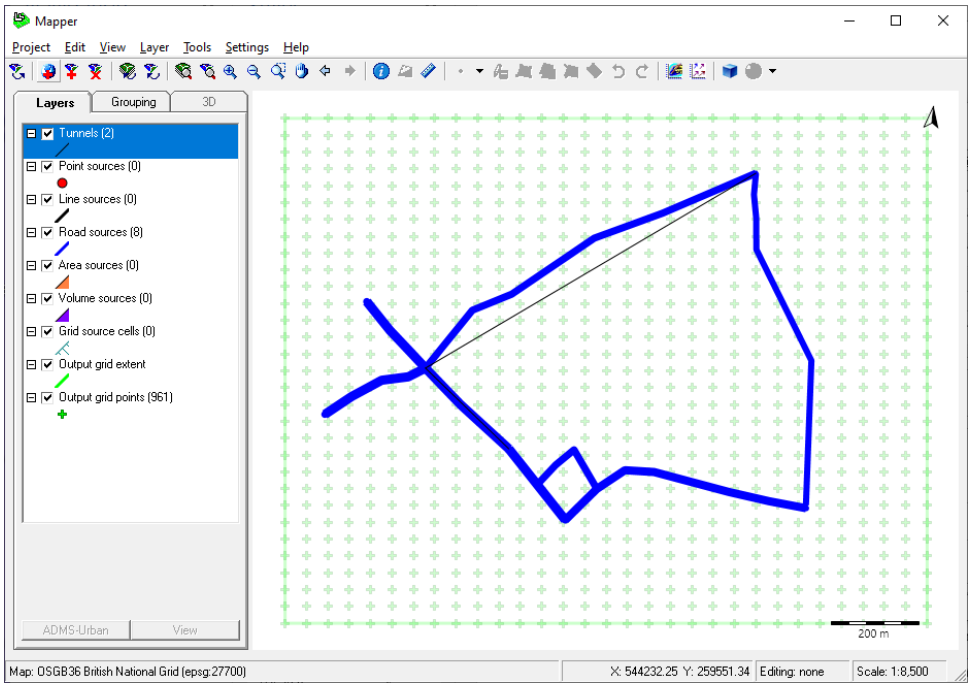


Figure 50 - Data from the *Tunnels.csv* file displayed in the Mapper.

Step 6 Information about the data can then be viewed. Ensure that the data layer is selected in the layer panel. Select the information tool and click on one of the data points. (Full details on the information tool can be found in Section 5.6). Information is then displayed in the **Attributes** table, as shown in **Figure 51**. The information includes all of the columns shown in the data window.

It is possible to colour the data according to its properties. See Section 4.6 for further details.

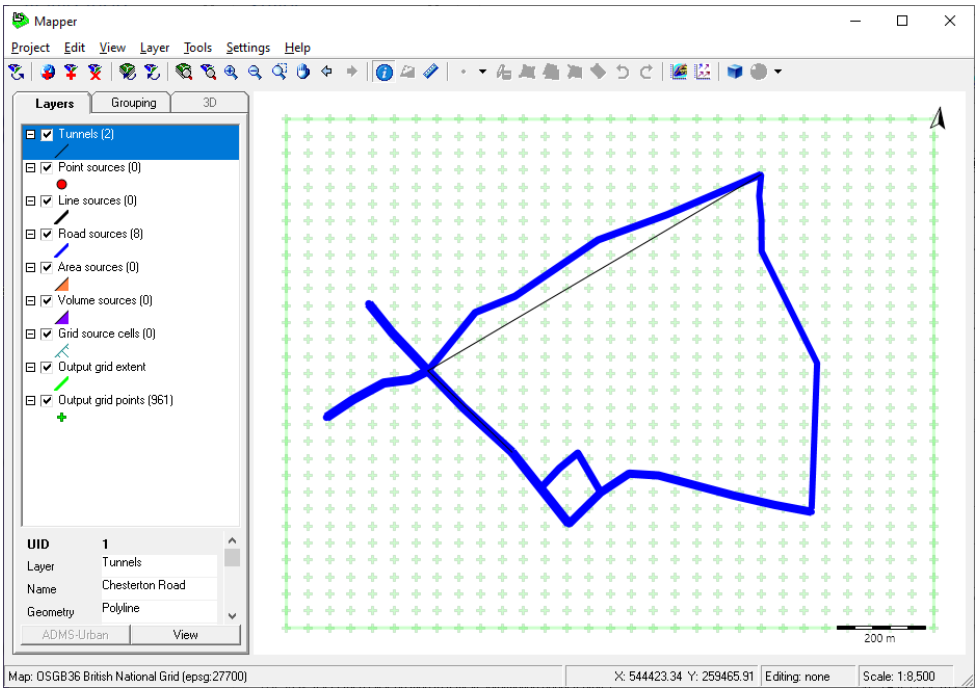


Figure 51 - Information from the *Tunnels.csv* file displayed in the Mapper.

5.6 Information about a feature

The Mapper allows you to view information about a model feature or a raster layer from within the Mapper. To display the information about a model feature, follow these instructions.

Step 1 Click on the **Information** button on the toolbar.

Step 2 Then click on the feature in the map view window.

Step 3 Information will be displayed about the feature in the attributes table, as shown in **Figure 52**.

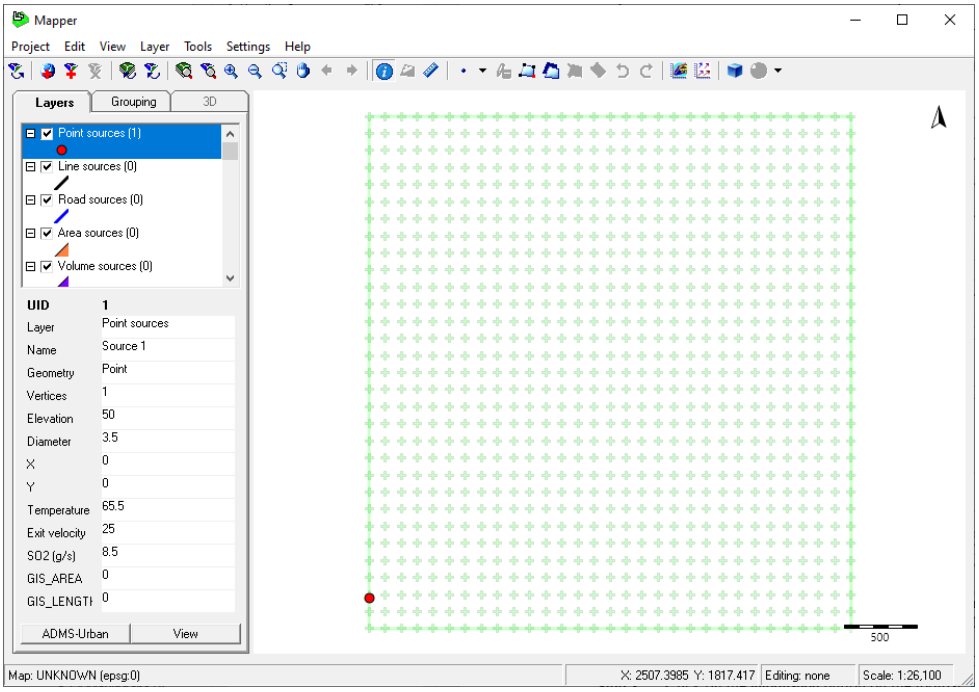


Figure 52 - The feature information screen for a point source.

Step 4 Click on the model name button (e.g. **ADMS**) to be directed to the feature in the model.

Step 5 Click on the **View** button for the feature to be highlighted in the Mapper.

---

*Holding down **AltGr** ensures that only features in the currently selected layer can be clicked.*

To display information about a raster layer containing data values, including any contour plots created with the 2D output plotter, follow these instructions:

Step 1 Ensure that the raster layer is selected in the layer panel

Step 2 Click on the **Information** button on the toolbar.

Step 3 As you move the cursor across the raster layer, the value at that point will be displayed on the right hand side of the status bar, as shown in **Figure 53**.

Step 4 Hold down **AltGr** and click on the raster layer to display the value and X, Y coordinates at that particular location in the attributes table, also shown in **Figure 53**.

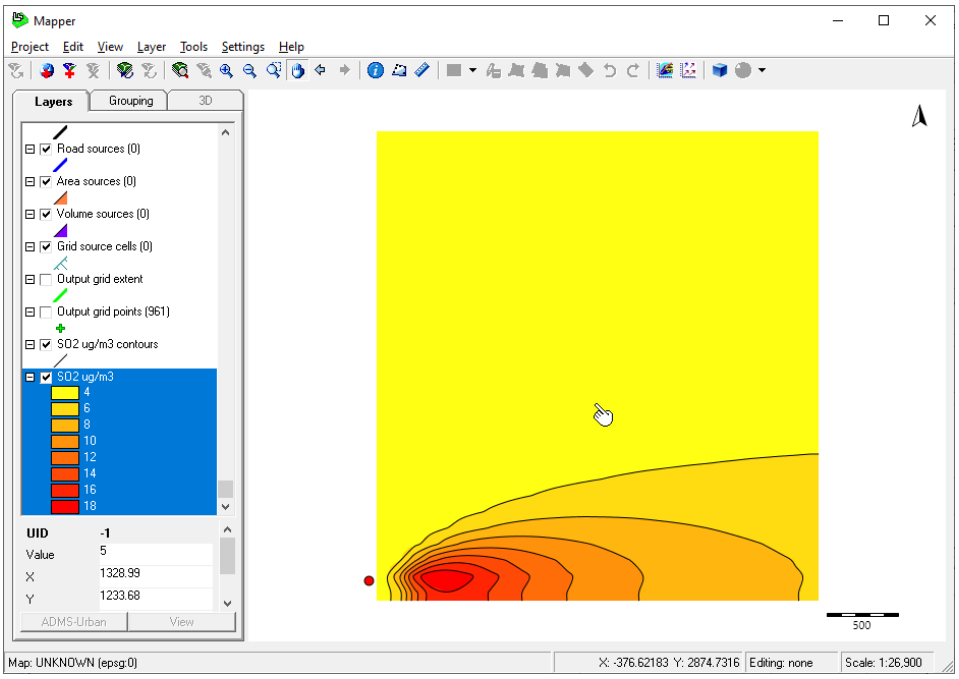


Figure 53 - The feature information screen for a raster layer.

5.7 Measuring distances

The Mapper comes with a tool that allows you to measure the distance between two or more points in the map view window. Follow these instructions to measure distances in the map view window.

Step 1 Click on the **Measure** button on the toolbar.

Step 2 Click on the map view window at the first point of interest, and then click on a second point. A dotted line is produced between the two points of interest and the distance between the two points is shown on the status bar in the bottom left hand corner of the Mapper window, as shown in **Figure 54**.

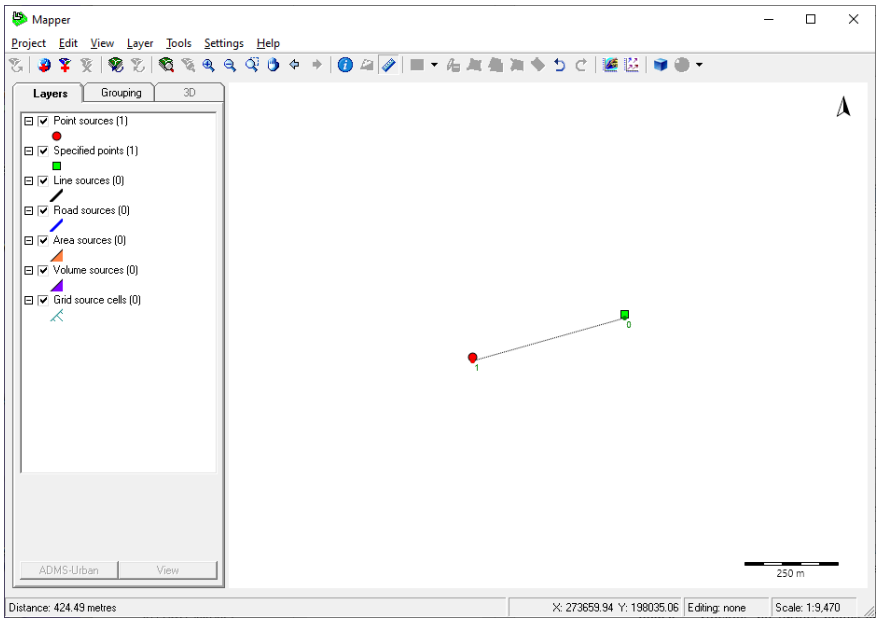


Figure 54 - Measuring distances in the Mapper. The distance is given in the bottom left corner.

Step 3 Clicking on further points adds further lines and the cumulative distance is displayed in the status bar.

Step 4 The position of the points can be modified as follows:

- To add a point, click on the existing point that lies before the desired location of the new point and then click where you want the new point to be.
- To move a point, click on the point, and while holding down the mouse button, move the cursor to the new location for the point before releasing the mouse button.
- To delete a point, first click on the point to select it, and then click on it again.

The points that the distance is being measured between appear as coloured squares. The current selected point is red and the other points are green.

Step 5 To remove all the defined points whilst using the measuring tool, double click anywhere in the map view window.

5.8 Displaying the north arrow and scalebar

It is possible to toggle the display of an arrow in the Mapper that indicates the direction of north and the scalebar, see **Figure 55** . These indicators are only displayed when the plot is in 2D mode.

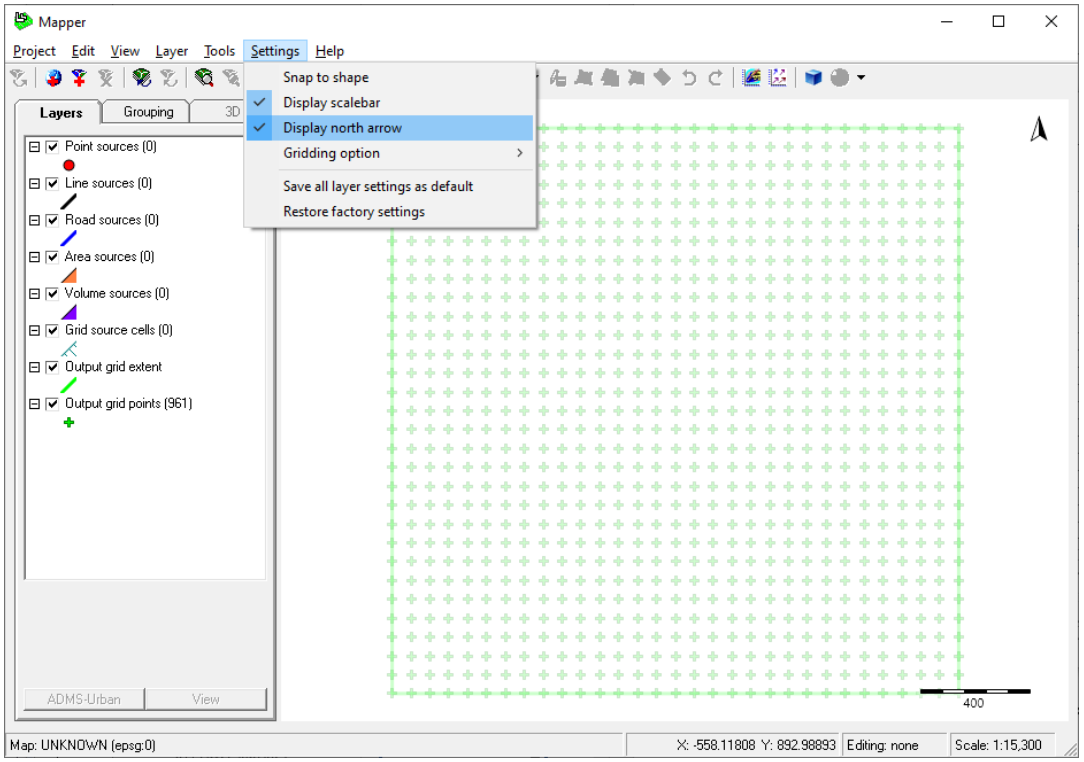


Figure 55 - The north arrow and scalebar are displayed in the top and bottom right-hand corners, respectively, of the map window.

To toggle the indicators, select/deselect the **Display north arrow** or **Display scalebar** option from the **Settings** menu as desired.

5.9 Viewing features in 3D

Features in the Mapper can be viewed in 3D. This can be especially helpful when checking building or source heights, or when examining complex terrain, variable roughness or concentration fields.

To see a 3D visualisation of the features in the Mapper first click on the Change View icon. After doing this, the map view will change and the Change View icon will also change. To return to the 2D planar view, click again on the new Change View icon.

When in 3D mode, three axes are shown: vertical (blue), east-west (red) and north-south (green). By clicking and holding the mouse you can rotate the map to examine the 3D visualisation from various points of view. A new button appears which enables you to change the method of panning the camera – the options available are summarised in **Table 4**.

See for example **Figure 56** which shows a view of buildings in the Mapper when in standard mode and when in 3D mode.

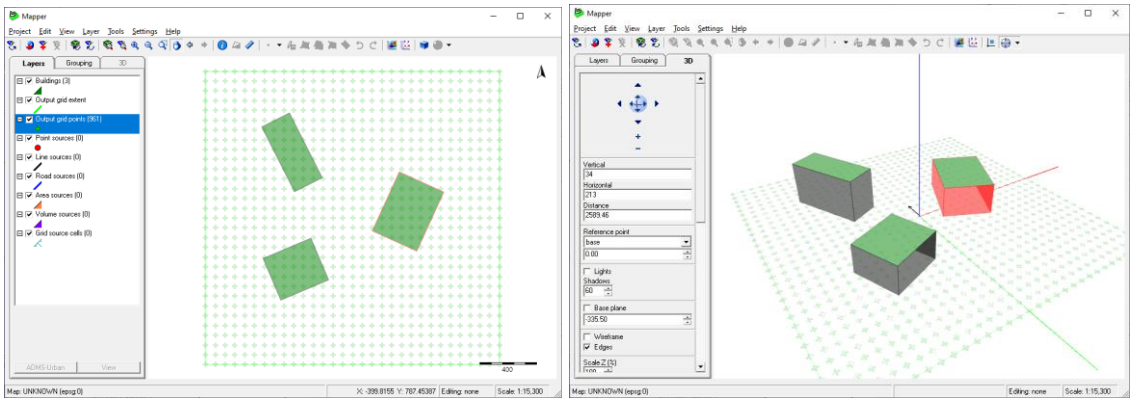


Figure 56 - Buildings viewed in 2D and 3D modes.

Some layers will have both a height (Z) and a depth (M), for example a building can be at a terrain height Z, with a depth of M.

*The vertical scale can be exaggerated using the **Scale Z (%)** and **Scale M (%)** options in the 3D tab. This may be required for clarity if the horizontal extent of the modelling region is much larger than the vertical extent of the buildings, sources, terrain, etc.*

5.9.1 Options in the 3D panel

When in 3D mode the view can also be changed a variety of ways by using the options under the 3D tab in the layer panel.

- **Vertical**, **Horizontal** and **Distance** – allows manual setting of the position of the camera in spherical coordinates. The **+** and **–** buttons change the distance of the camera from the origin of the X,Y,Z axis.
- **Reference point** – positions the origin along the Z axis (altitude) relative to the

layers shown:

- **Base** – if there is at least one layer which is a digital elevation model (DEM), then the base is set to the lowest Z value found in the DEM
- **Zero** – the value is set to zero regardless of the Z coordinates found in the open layers
- **Lowest** – lowest value of all Z coordinates in all the layers
- **Highest** – highest value of all Z coordinates in all the layers
- **On DEM** – the value of the Z coordinate read from the DEM at the longitude and latitude of the origin pointer; this does not change if the origin pointer changes
- **Fly on DEM** – as **On DEM**, but the value updates if the position of the origin pointer changes

- **Lights** – toggles the directional light on/off
- **Shadows** – sets the intensity of shadows when the directional light is turned on; varies from 0 (no shadows) to 100 (darkest shadows)
- **Base plane** – toggles a base plane on/off
- **Wireframe** – toggles the wireframe display mode on/off
- **Edges** – toggles edges on/off on the 3D objects
- **Scale Z (%)** – scales the height (Z) coordinate of all open layers; 100 is the actual size
- **Scale M (%)** – scales the depth (M) coordinate of all open layers; 100 is the actual size
- **Flood** – toggles on/off a floodplains simulation; the numerical value is the flood level (in metres)
- **Wall** – if there is at least one digital elevation model (DEM) among the open layers, this option sets the appearance of the DEM boundaries:
  - **Off** – no DEM walls
  - **Colour** – grey walls with contour lines
  - **Texture** – textured walls with contour lines

5.10 Copying the map view window to the clipboard

To copy the current view from the map view window select **Copy map to clipboard** from the **Edit** menu. This image can then be pasted into a document or picture editing software and saved.

The **Copy legend to clipboard** option from the **Edit** menu can be used to copy an image of the layer panel to the clipboard. If only a part of the layer panel is required, for instance the legend for a contour plot, then the image should be pasted into image processing software and then cut down as appropriate.

The Attributes section of the layer panel is not copied to the clipboard.

The **Copy scalebar to clipboard** option from the **Edit** menu can be used to copy an image of the scalebar to the clipboard.

5.11 Layer statistics

Statistics about a particular layer can be viewed using the **Layer statistics** tool. Namely, the minimum, maximum and X, Y location of each are reported for each numerical field in the layer. Up to five filters can also be applied to the layer to constrain the number of features/data points on which to calculate the statistics.

To view the statistics for a particular layer, right click on that layer in the layer panel of the Mapper and select **Statistics**. This brings up the **Display statistics** screen, as shown in **Figure 57**.

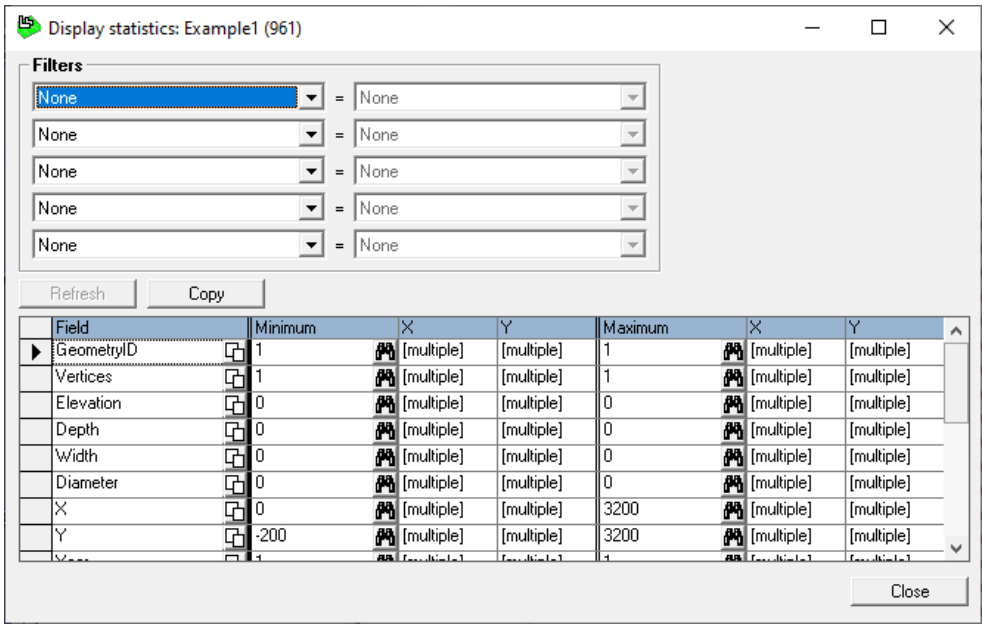


Figure 57 - Display statistics screen

You can use the **Filters** section to apply up to five filters to the data in this layer, for

example you may want to view the statistics for only those points in a *.pst* file that correspond to the first line of meteorological data and/or have a receptor height of 2 m. Whenever the filters are modified, it is necessary to click the **Refresh** button for the changes to take effect.

Other useful features include:

- The **Copy** button can be used to copy the entire table to the clipboard, which can then be pasted into a text editor (e.g. Notepad) or spreadsheet package (e.g. Microsoft Excel).

- Similarly, the buttons can be used to copy the statistics for a particular field to the clipboard. If there are multiple X, Y locations where the minimum or maximum occur for that field, the data for all locations will be copied, thus expanding on the information shown in the table.

- The X, Y location(s) of the minimum or maximum can also be highlighted in the map view window using the relevant binoculars icon . For example, this can be used to identify the location(s) of the lowest and highest concentrations in a *.glt* file, as demonstrated in **Figure 58**.

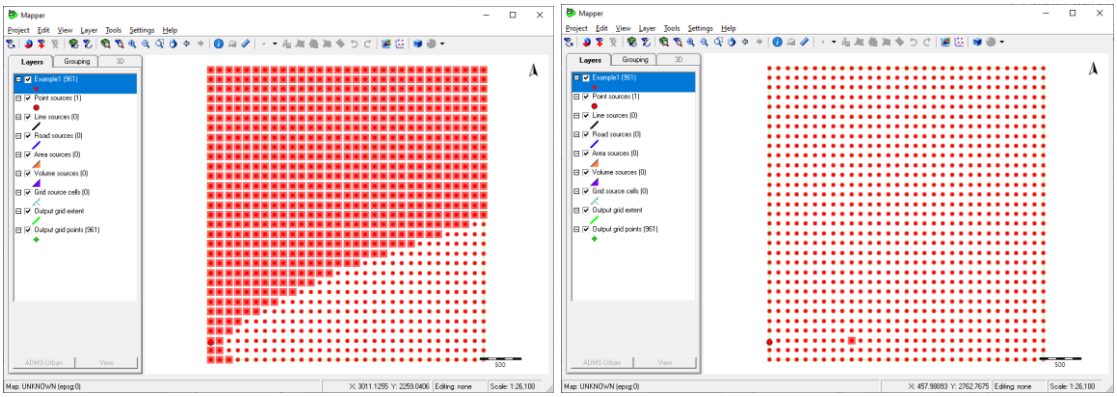


Figure 58 - Displaying the minima (left) and maxima (right) of a *.glt* field

SECTION 6 Tools

6.1 Extract data tool

If a raster layer has been added to the Mapper, it is possible to extract the data from this layer into **.csv* format. This can be achieved through a simple process, although there are more advanced options to satisfy precise requirements for the output data.

6.1.1 Using the Extract data tool

This can be done by following these instructions.

Step 6 Add the raster data file to the Mapper. In this example, an OS terrain 50 file will be added, and a section of the terrain data extracted for use with ADMS.

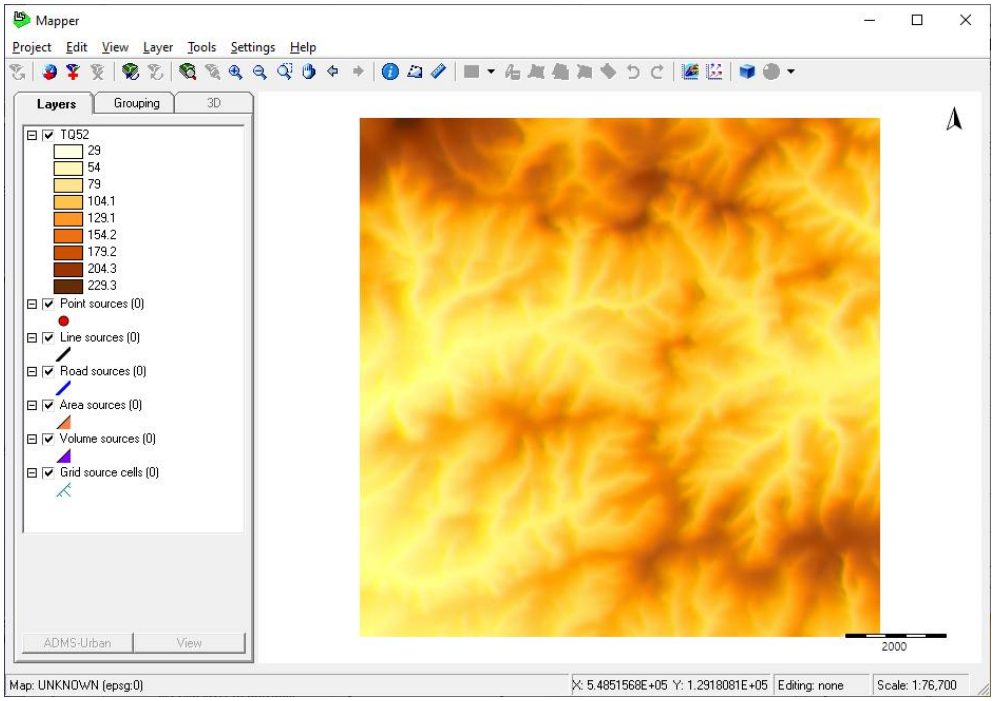


Figure 59 - Mapper showing an OS terrain 50 file, which has been coloured by terrain height.

Step 7 Press the button **Extract data from raster layers,**, to bring up the **Extract data** screen. Alternatively, you can select the **Extract data from raster layers** option from the **Tools** menu.

Step 8 The Extract Data Extent is now displayed in the map as a new layer. By default, it covers the whole of the current map view window. If you wish to change the extent, you can redefine, move or edit it using the standard Mapper editing tools (see below). Alternatively, you can edit the extent manually in the **Extract data** screen.

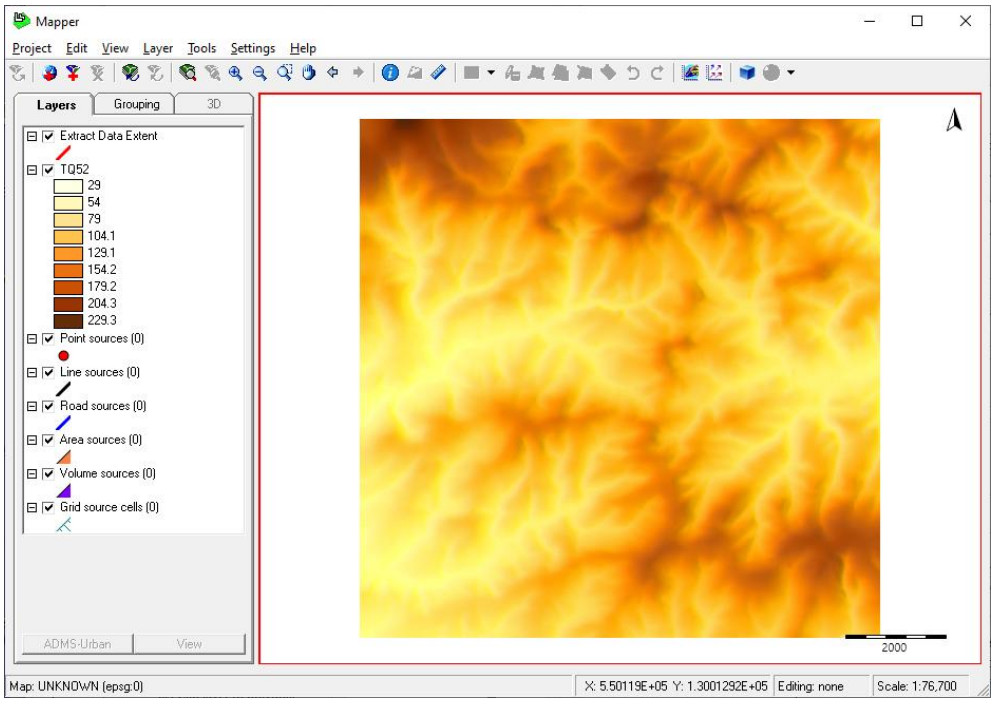


Figure 60 - Mapper showing the Extract Data Extent as a red rectangle.

Step 9 Check the selected data layers and customise output data options if necessary; see Section 6.1.5 for a description of the available options.

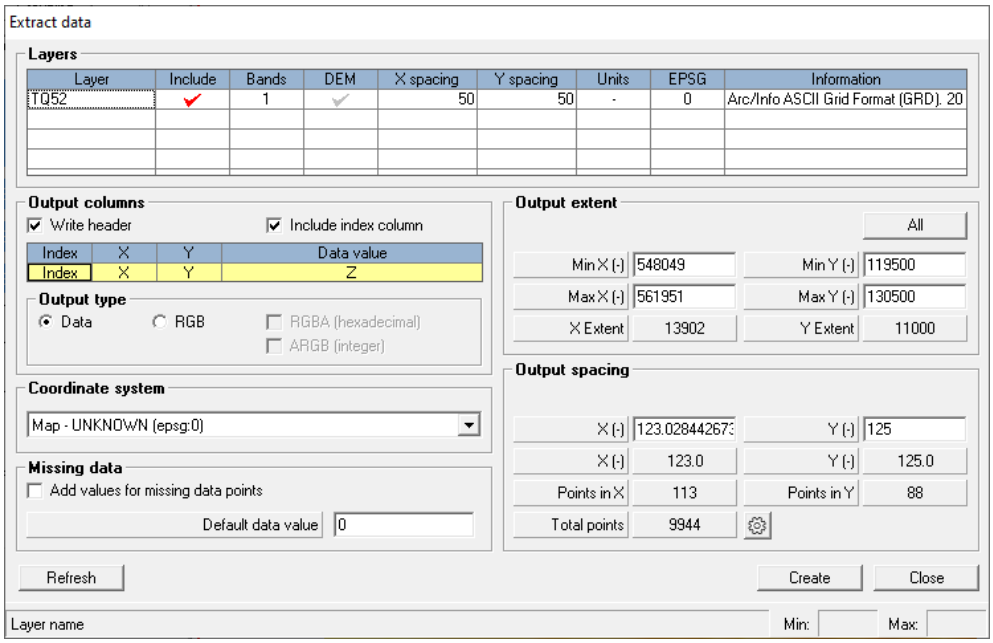


Figure 61 - Extract data screen

Step 10 When you have confirmed all the settings are as you desire, click **Create**. You will be prompted to enter the name of the output file. Once the file has been successfully created, a notification dialog will inform you of the number of points written to the file.

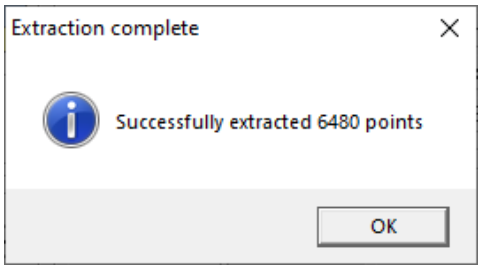


Figure 62 - Notification of successful data extraction

*If you have been making changes in both the Mapper and the **Extract data** screen and you want to ensure they are synchronised before continuing, you can click the **Refresh** button.*

Step 11 The resulting output **.csv* file can be added to the Mapper or viewed in a text editor (e.g. Notepad) or spreadsheet package (e.g. Microsoft Excel).

6.1.2 Redefining the Extract Data Extent

Step 1 Select the **Extract Data Extent** layer from the layer panel.

Step 2 Click on the **Add Feature** button on the toolbar to select it.

Step 3 Click on the location of one corner of the extent in the map view window, and while holding down the mouse button, move the cursor to draw the extent required before releasing the mouse button.

Step 4 A new extent has been defined and the **Output extent** section of the **Extract data** screen will have been updated automatically to show its new coordinates. These coordinates can now be manually adjusted further if desired.

6.1.3 Editing the Extract Data Extent

Step 1 Select the **Extract Data Extent** layer from the layer panel.

Step 2 Click on the **Edit Feature** button on the toolbar to select it.

Step 3 Click anywhere within the current extent. Three of the extent vertices will appear as green dots and the fourth vertex as a red dot.

Step 4 The vertex with the red dot indicates which corner of the extent will be movable. To select a different vertex, use either the full stop (.) or comma (,) keys on the keyboard.

Step 5 Click in the map window, and while holding down the mouse button, move the cursor until the red grid vertex is in the desired location before releasing the mouse button.

Step 6 Click on the **Save Edits** button on the toolbar to save the changes to the **Extract data** screen. . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

6.1.4 Moving the Extract Data Extent

Step 1 Select the **Extract Data Extent** layer from the layer panel.

Step 2 Click on the **Shift Feature** button on the toolbar to select it. 

Step 3 Click anywhere within the current extent, and while holding down the mouse button, move the cursor to the new location for the extent before releasing the mouse button.

Step 4 Click on the **Save Edits** button on the toolbar to save the changes to the **Extract data** screen.  . Alternatively right click the mouse to bring up a pop-up menu and select the **Save edits** option.

6.1.5 Extract data options

There are a range of options that can be used to customise the extraction of data including altering the location, spacing and coordinate system.

* Basic options:
  * Choose which **Layers** to extract data from. Layers that overlap the selected area are included by default. In general, layers with a single data band (e.g. OS terrain data) and layers with multiple data bands (i.e. Red-Green-Blue raster data) should not be included at the same time.
  * Choose whether or not to write a header row to the data file by selecting or deselecting the relevant checkbox. If you choose to write a header row, the column headings can be edited.
  * Select whether or not to include an index column.

When creating an ADMS format terrain file, it should not have a header row, but an index column should be included.

---

  * Select whether to extract the data with a single band (e.g. elevations) or with multiple bands (e.g. RGB values). If RGB values are selected then additionally chose whether to also output **RGBA (hexadecimal)** or **ARGB (integer)** values.
  * The **Output spacing** should be set to the required resolution in the X and Y directions. You can specify a different spacing in each direction if required. If the output spacing entered would result in more than this number of output points it is automatically reset to the nearest value that reduces the number to within tolerance. You can also use **Total point target** option,  , to specify an overall number of points to use. This option will set the spacing to give a total number of points as close to the target as possible while maintaining the current ratio between the X and Y spacings.
* Additional options:
  * The **Coordinate system** for the output data can be selected. By default this is set to the ADMS coordinate system, but it can be changed to match the raster layer or map coordinate systems.

When creating an ADMS format terrain file, the coordinate system should match the ADMS model data that it will be used with.

* It is also possible to manually edit the **Output extent** in the **Extract data** screen if you wish to change the selected range of X and Y coordinates. To do this, type in the **Min X**, **Max X**, **Min Y** and **Max Y** values in the boxes provided. This will automatically update the **Extract Data Extent** layer in the map view window.
* Some layers can have regions with no data; you can specify the value to be entered in these regions, e.g. some formats of terrain data files do not report values over areas of sea, and in this case the terrain height should be set to zero.

6.2 Export features

The Mapper layers can be exported to a variety of formats. Some popular formats available are

- ArcView Shape Files (*.*shp*)
- Autocad (*.*dxf*)
- Digital Line Graphs (*.*opt*) – line sources only
- Geographic Markup Language (*.*gml*)
- GPS Exchange Format (*.*gpx*)
- Keyhole Markup Language (*.*kml*)
- MapInfo Interchange (*.*mif*)
- Portable Network Graphic (*.*png*)
- JPEG File Interchange Format (*.*jpg*)
- Tag Image File Format (*.*tif*)
- Windows Bitmap (*.*bmp*)

To export a layer into any of these formats first click on the layer in the layer panel to select it. Then click on the **Export Layer** button on the toolbar. This brings up the export layer screen, as shown in **Figure 63**. Select the file type you wish to export to, give the file a name then click on the **Save** button.

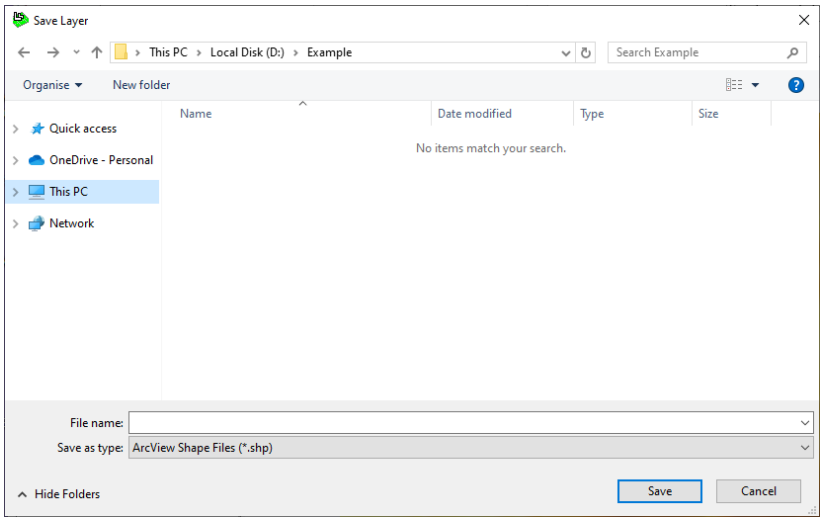


Figure 63 - The **Export Layer** window.

Alternatively a layer can be exported by selecting it and then right-clicking to display the menu. Select **Export layer** and then select the file type from the sub-menu which offers four formats, **to SHP...**, **to KML...**, **to CSV...** and **for SPT...**, for data layers containing points, polylines or polygons including ADMS layers.

With the first three of these options, a new window will appear which you can use to

save the file in your desired location. The file type will be selected automatically and a default name for the file is suggested, based on the name of the layer.

The **for SPT...** option opens a series of forms that automate the generation of comma-separated *.spt* (and related) files that can be imported directly into the ADMS models. See Section 6.2.2 below for more details.

For raster layers, the options are **to PNG...**, **to JPG...**, **to TIF...** and **to BMP...**. Similar to non-raster layers, a new window will appear which you can use to save the file in your desired location. The file type will be selected automatically and a default name for the file is suggested, based on the name of the layer.

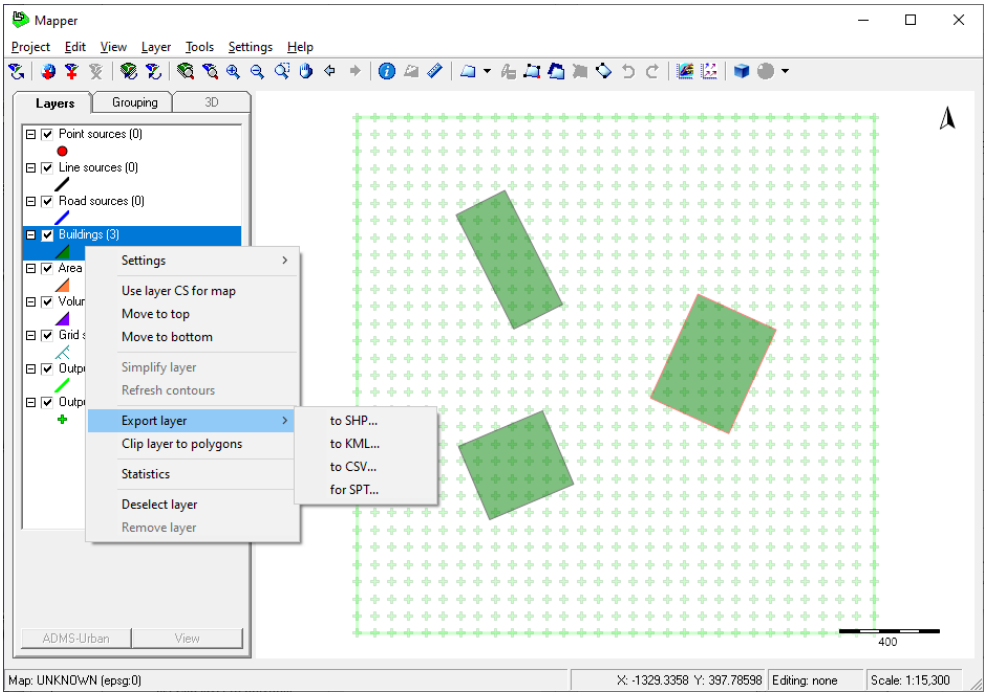


Figure 64 - The **Export Layer** right-click option.

6.2.1 Exporting to Google Earth

The layers can be exported so that they can be viewed in Google Earth. This may be useful for checking the locations of sources and buildings etc. against a satellite image of the area being modelled. To do this, follow these steps:

Step 1 Set the coordinate system for the map, see Section 5.1.

Step 2 Ensure that a valid projected or geographic coordinate system has been set for the layers that are being exported. Refer to Section 5.2 for details on how to do this.

*The coordinate systems for the ADMS layers do not need to be changed in the Mapper. These are set automatically from the option defined in the ADMS interface. The **Unspecified regular Cartesian** grid cannot be used if you wish to export an ADMS layer to Google Earth.*

Step 3 Export the layer to Keyhole Markup Language (**.kml*) by following the

instructions given above.

Step 4 Open the **.kml* file in Google Earth.

6.2.2 Exporting for SPT

Non-ADMS layers containing source information, e.g. a shape file of point or road source data, can be exported for *.spt* format so that they can be easily imported into the relevant ADMS model for inclusion in the dispersion calculations. Once imported, the sources will appear in the relevant ADMS source layer in the Mapper; the original non-ADMS layer can then be removed/unticked to avoid duplicating the visualisation of these sources.

To do this, follow these steps:

Step 1 Right-click on the relevant non-ADMS layer in the layer panel and select **Export layer > for SPT**.... This brings up the **Export files for SPT** form, as shown in **Figure 65**.

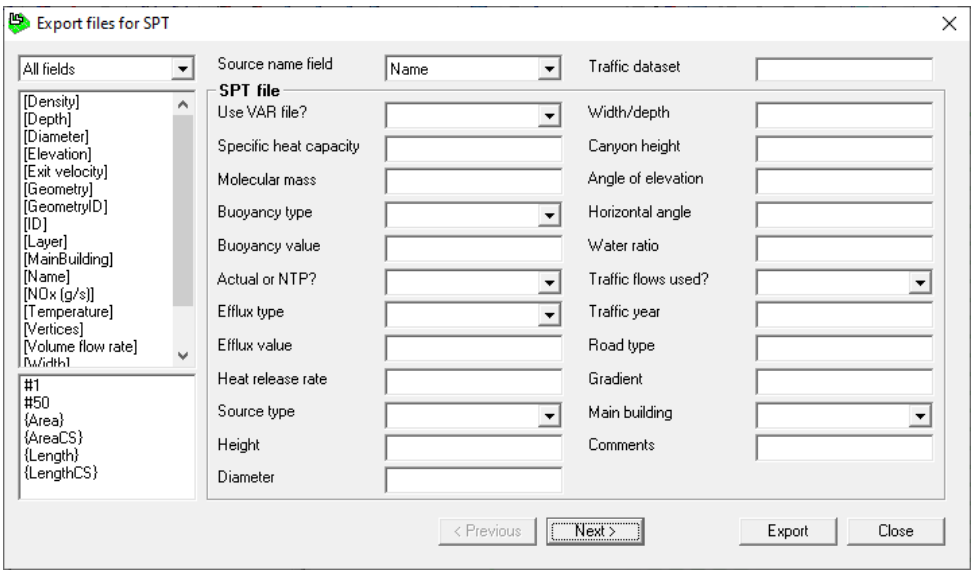


Figure 65 - Export files for SPT, SPT file section

Step 2 Using the drop-down menu next to **Source name field**, select which of the fields in the non-ADMS layer contains the source names.

Step 3 If exporting road source data with emissions calculated using traffic flows, specify name of the **Traffic dataset**, e.g. "UK EFT v13.1 (2 VC)" – this should exactly match one of the traffic datasets used in ADMS-Urban/ADMS-Roads. If exporting data for a different source type, or road traffic emissions will be user-defined, this box can be left blank.

Step 4 Fill in as much of the **SPT file** section (source properties) as possible. Boxes can be filled either by using the drop-down menu (where applicable), typing directly into the box or drag-dropping the name of a field from the current layer from the list of fields on the left-hand side of the form (enclosed in square brackets, e.g. '`[Name]`') into the relevant box. Field names will automatically be converted to the relevant values for each source in the layer upon export. Entering a constant value in a box will

result in every source taking that value for that source property in the *.spt* file. Fields that are not relevant/required can be left blank (or filled with 'na' or 'n/a'). Fields that are required in the *.spt* file for successful import into the ADMS model but that don't have an equivalent field in the current layer can also be left blank, however this information will need to be manually added to the *.spt* file after creation before it can be imported into the ADMS model.

Step 5 Press **Next >** to move onto the **EIT file** section (pollutant emissions), as shown in **Figure 66**. If the current layer does not contain direct emissions data, skip to Step 7.

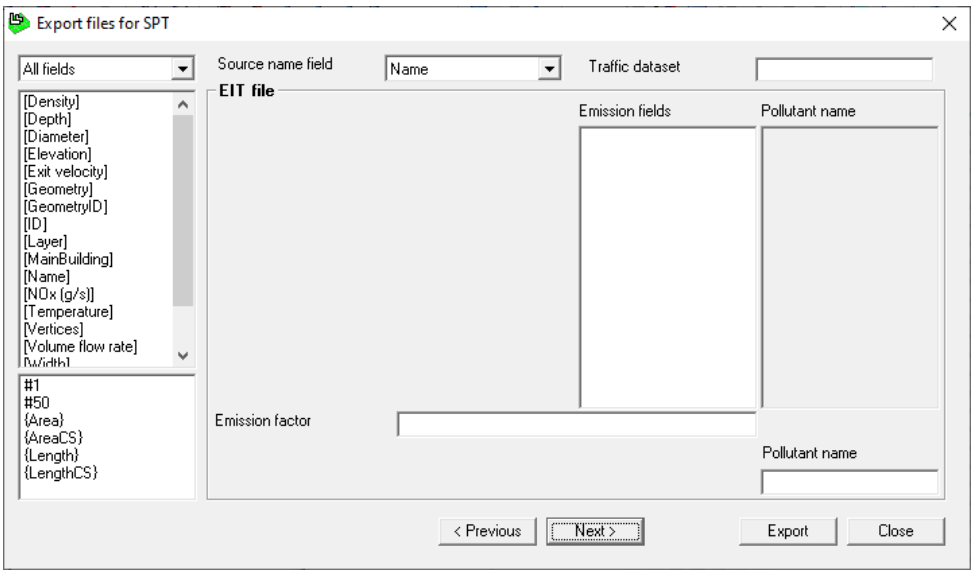


Figure 66 - Export files for SPT, EIT file section

Step 6 Fill in as much of the **EIT file** section as possible in a similar manner. Note that the **Emission fields** box can only be filled by drag-dropping field names, the **Pollutant name** box can only be filled by typing and the **Emission factor** box can also be filled by drag-dropping the name of a parameter associated with the current layer from the list of parameters (enclosed in curly brackets, e.g. '`{Length}`') on the left-hand side of the form. The **Pollutant name** box should be used to ensure that the pollutant name associated with a particular emission field matches a pollutant name as defined in the ADMS model's pollutant palette The **Emission factor** box can be used if emission data in the current layer are not in the appropriate units. For example, area source emissions should be given in $g/m^2/s$ in the ADMS models, so if the current layer contains area source emission rates in g/s, it would be necessary to multiply the emission field by the area of each source, i.e. specify '/`{Area}`' in the **Emission factor** box.

Step 7 Press **Next >** to move onto the **TFT file** section (traffic flows), as shown in. If the current layer does not contain traffic flow data, skip to Step 9.

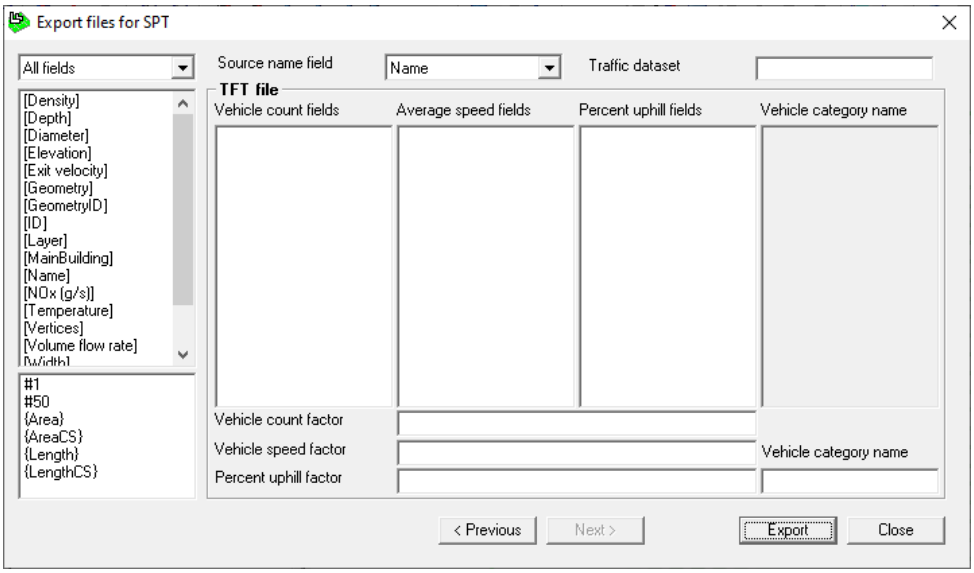


Figure 67 - Export files for SPT, TFT file section

Step 8 Fill in as much of the **TFT file** section as possible in a similar manner. The **Vehicle category name** box should be used to ensure that the vehicle category name associated with a particular vehicle count field matches a vehicle category name as defined in the ADMS-Urban/ADMS-Roads interface. The '#1' parameter can be used with the **Average speed fields** and/or **Percent uphill fields** box to indicate a constant value. The constant value itself should be entered into the **Vehicle speed factor**/**Percent uphill factor** box. The '#50' parameter is there as a convenience for use with the **Percent uphill fields** box (50% implies flat-road emission factors will be used throughout). The **Vehicle count factor**, **Vehicle speed factor** and **Percent uphill factor** boxes can be used to apply any appropriate conversions to the data in the current layer. For example, vehicle speeds should be given in km/hr in the ADMS-Urban/ADMS-Roads interface, so if the vehicle speed data in the current layer are in mph, a value of 1.61 should be specified in the **Vehicle speed factor** box.

Step 9 Press **Export** to bring up a **Save As** window. Browse to where you want the files to be saved, give them a name (common stem), then click on the **Save** button. This will generate up to four files: an *.spt* file (containing source properties), a *.vgt* file (vertex information), an *.eit* file (pollutant emissions) and a *.tft* file (traffic flows).

Step 10 (optional) These files can then be modified further in a text editor (e.g. Notepad) or spreadsheet package (e.g. Microsoft Excel), if required, before being imported into the ADMS model via the **File > Import** menu item from the model interface; see the relevant model user guide for full details on importing sources from *.spt* files.

Step 11 (optional) Once the sources have been imported into the model interface and the Mapper has been refreshed, it may be desirable to remove/untick the original non-ADMS layer to avoid duplicating the visualisation of these sources.

6.3 Clip layer to polygons

The **Clip layer to polygons** tool allows individual features from one layer to be 'clipped' to the features from a separate layer, i.e. retained/removed depending on whether they fall inside or outside the spatial boundaries of the second set of features. The resulting set of clipped features are saved to a new shape file rather than replacing the original layer. If clipping zero-dimensional features, e.g. output points, there is also an option to save the clipped data to an *.asp* file, which can then be used in an ADMS model run. It is possible to clip onto any layer that contains polygons or polylines.

6.3.1 Clipping point, polyline or polygon layers

Step 1 Right-click on the layer that you wish to clip, e.g. the **Line sources** layer, and select **Clip layer to polygons**. This brings up the **Clip from:** form, as shown in **Figure 68**.

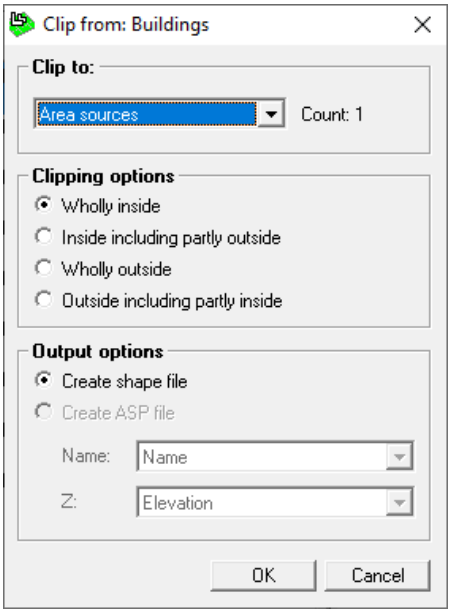


Figure 68 - Clip layer to polygons

Step 2 Select which layer to **clip to** using the drop-down menu.

Step 3 Select from one of the following **Clipping options**:

- **Wholly inside**: Any feature that falls completely inside one of the features being clipped to will be retained.
- **Inside including partly outside**: Any feature that falls at least partly inside one of the features being clipped to will be retained. This option is not available if clipping zero-dimensional features, e.g. point or jet sources.
- **Wholly outside**: Any feature that falls completely outside all of the features being clipped to will be retained.
- **Outside including partly inside**: Any feature that falls at least partly outside one of the features (and not wholly inside any of the features) being

clipped to will be retained. This option is not available if clipping zero-dimensional features, e.g. point or jet sources.

The **Count** in the **Clip to:** section shows how many features will be retained with the current clipping option. The map view window will also interactively show which features will be retained as different clipping options are selected.

Step 4 In the **Output options** section, select how the clipped features should be saved by choosing either **Create shape file** or (if clipping zero-dimensional features) **Create ASP file**. For the latter option, you should also use the **Name** and **Z** drop-down menus to specify which of the fields in the layer being clipped contain the specified point names and heights, respectively.

Step 5 Press **OK** to create the output file. If **Create shape file** was selected, the resulting shape file will automatically be added as a new layer in the Mapper, and the original layer being clipped will be unticked.

6.3.2 Clipping raster layers

Step 1 Right-click on the raster layer, i.e. the contour plot, that you wish to clip and select **Clip layer to polygons**. This brings up the **Clip from** form, as shown in **Figure 69**.

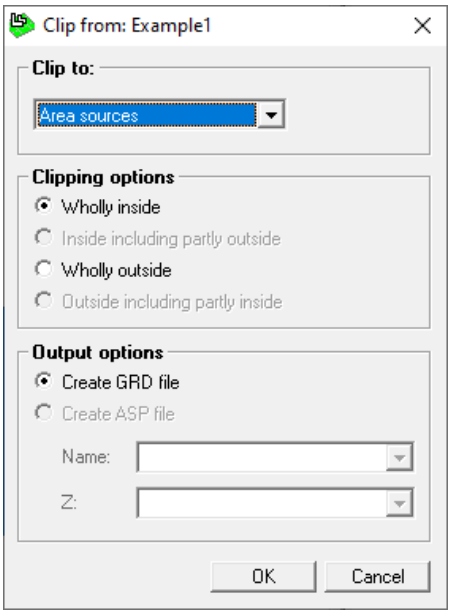


Figure 69 - Clip layer to polygons

Step 2 Select which layer to **clip to** using the drop-down menu.

Step 3 Select from one of the following clipping options:

Wholly inside: Data inside all of the features being clipped to will be retained.

Wholly outside: Data that falls completely outside all of the features being clipped to will be retained.

The map view window will interactively show the resulting layer as different clipping options are selected.

Step 4 **Create GRD file** is selected in the **Output options** section as clipped raster layers can only be saved as **.grd* files. Press **OK** to create the output file. The resulting file will automatically be added as a new layer in the Mapper, and the original layer being clipped will be unticked.

6.4 Create Street Canyon and Urban Canopy files

The **Canyon and Canopy** tool creates **.csv* format parameter files that can be used directly with the Advanced Street Canyon and Urban Canopy Flow modules in ADMS-Urban. The tool outputs all of the required headers together with the following Street Canyon parameters for each feature in the roads layer:

- Canyon heights (maximum, mean and minimum)
- Canyon widths
- Canyon lengths

And the following Urban Canopy parameters which characterise an urban area as defined on a rectangular grid:

- Average building heights
- Average canyon widths
- LambdaP (λ_P) values, a measure of building coverage at ground level
- LambdaF (λ_F) values, a measure of building fascias for multiple directions

6.4.1 Using the Canyon and canopy tool

The **Canyon and Canopy** tool generates the parameter files using a buildings layer and a roads layer that have been added to the Mapper. To use the tool:

Step 1 Select the **Canyon and Canopy** option from the **Tools** menu to bring up the **Canyon and canopy tool** screen. Ensure that ADMS is set to a valid coordinate system as the tool uses the coordinate system in the ADMS interface. Otherwise, the menu option will not be available. If the map coordinate system is different from the ADMS coordinate system, a notification dialog will inform you that the map will be set to the system currently used in the ADMS interface.

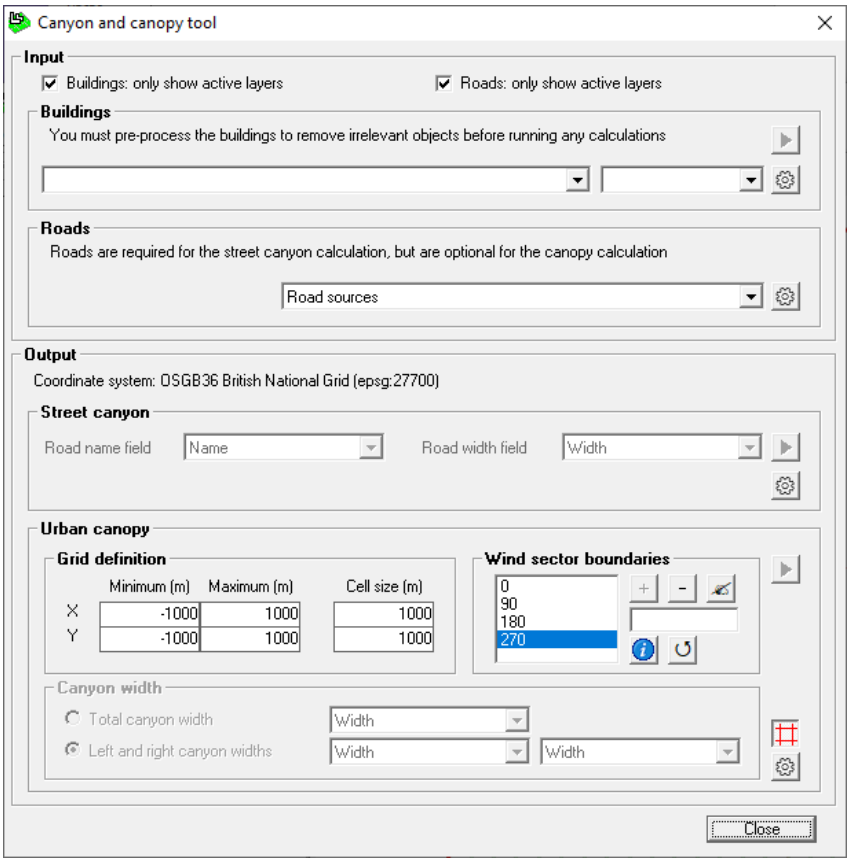


Figure 70 - Canyon and canopy tool screen

Step 2 The Canopy Output Cells and Extent are now displayed in the map as new layers. Initially, the extent covers the whole of the current map view window. The generated cells will be used in the Urban Canopy calculations, see 6.4.4.

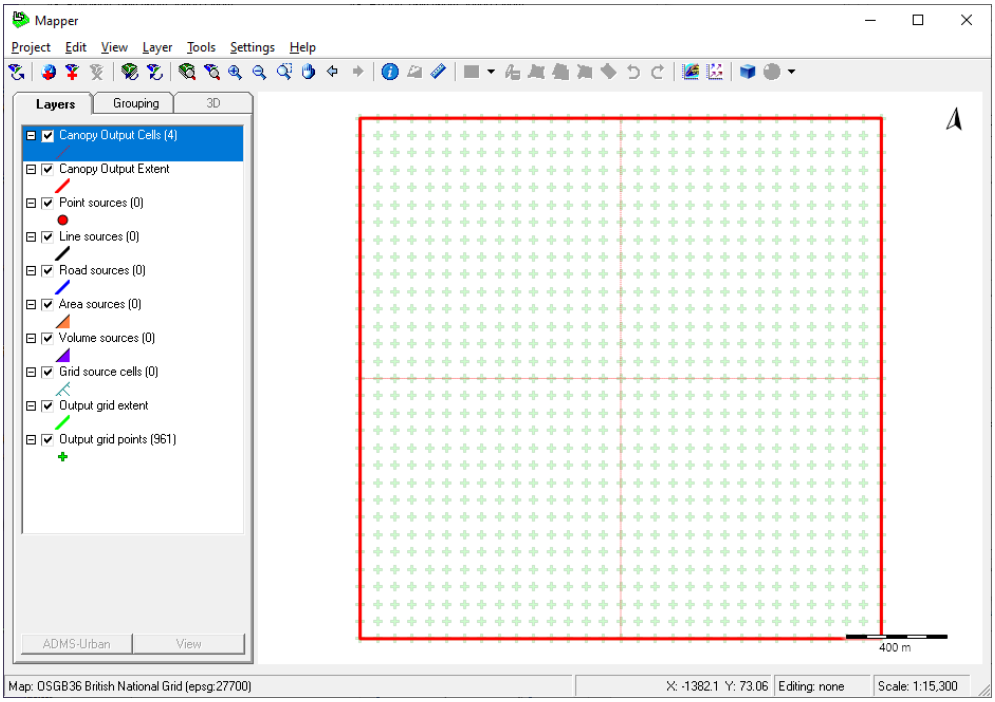


Figure 71 - Canopy Output Cells and Extent layers in the Mapper as red rectangles

Step 3 You must pre-process the buildings data before any calculations can be carried out. Select the layer you wish to process and the name of the building height field from the **Buildings** drop-down menus. Configure the processing options if necessary, see Section 6.4.5 for a description of the available options. If the buildings data have already been pre-processed, skip to Step 5.

Some data sources define buildings data across multiple sets of *.shp *files. The* ***Shape file merging tool*** *can be used to combine such data into a single file. See Section 6.5 for details about the tool.*

Only polyline or polygon layers displayed in the Mapper are included in the drop-down menu. Uncheck the ***Buildings: only show active layers*** *checkbox to also include hidden layers.*

Step 4 When you have confirmed all the settings are as you desire, click the ▶ button to start the pre-processing. You will be prompted to enter the name of the output file. Once the file has been successfully created, it will be automatically added to the map as a new layer and selected in the **Buildings** layer drop-down menu.

Step 5 Select the roads data layer that you wish to use in the calculations from the **Roads** drop-down menu.

It is possible to use the roads data defined in the ADMS model by selecting the ***Road sources*** *layer. On the other hand, using roads data is not required in Urban Canopy calculations. If so, select* ***[No road layers]****.*

Only polyline or polygon layers displayed in the Mapper are included in the drop-down menu. Uncheck the ***Roads: only show active layers*** *checkbox to also include hidden layers.*

Step 6 If generating Street Canyon files, continue to Step 7. Otherwise if generating Urban Canopy files, skip to 0.

If generating Urban Canopy files and the selected roads data layer is not defined with canyon widths, you need to generate the Street Canyon files first. Select the advanced setting ***Create canyon shape file*** *and then use the additional output* *.shp *file as roads data for Urban Canopy.*

Step 7 Select the **Road name field** from the drop-down menu to be written to the output **.csv* file. Select the **Road width field** to use in the canyon width checks if it is defined in the roads data layer. Otherwise, select **[No width defined]**. Configure the settings for the street canyon calculations if necessary, see Section 6.4.5 for a description of the available options.

Step 8 When you have confirmed all the settings are as you desire, click the ▶ button in the **Street canyon** section to create the output file(s). You will be prompted to enter the name of the output file. If the advanced setting **Create canyon shape file** is selected, the additional output **.shp* files will be created in the same directory with the same file stem. Once the files have been successfully created, the **Canyon and canopy tool** screen will close.

Step 9 Define the output grid and cell size, shown as the Canopy Output layers in the map and is set to the current map view window by default. If you wish to change the extent, you can redefine, move or edit it using the standard Mapper editing tools (see **Figure 71**). You can hide the layers in the map view by clicking on the button. Alternatively, you can edit the extent manually in the **Grid definition** section.

Step 10 Specify the **Wind sector boundaries** for the LambdaF calculations. The pre-set boundaries cover the sectors 0 to 90 degrees, 90 to 180 degrees, 180 to 270 degrees and 270 to 0 degrees. The following actions are available if you wish to change the list of wind sector boundaries:

- Add a sector boundary by typing the angle in degrees (from 0 to 359) in the text box and then clicking on the **Plus** button.
- Remove the selected item on the list by clicking on the **Minus** button.
- Generate the set of boundaries for uniformly sized sectors by clicking on the **Edit** button. In the **Create sectors** sub-screen, select whether specifying a starting angle for **Wind sector boundaries** or **Wind directions** and then type the **Start angle** and **Number** of sectors. The panels show the resulting wind sector boundaries and corresponding wind directions. Update the list in the **Canyon and canopy tool** screen by clicking **OK**.

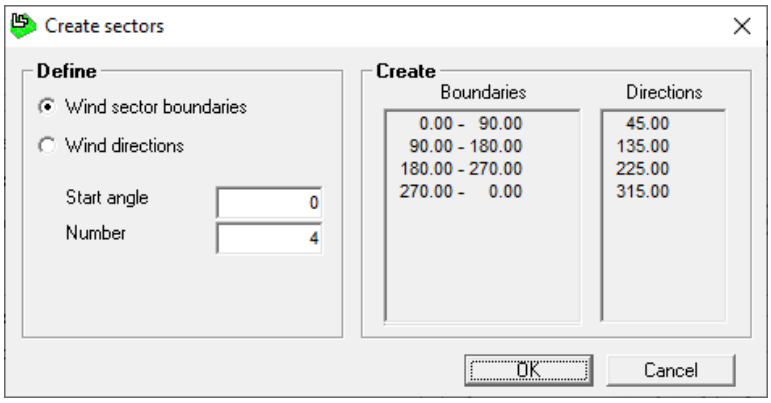


Figure 72 - Create wind sectors

- Check the wind directions for which LambdaF will be calculated by clicking on the **Information** button.

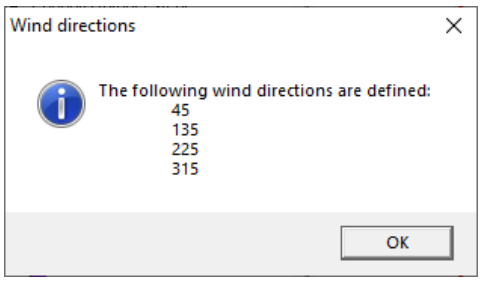


Figure 73 - Wind directions for the pre-set sectors

- Set the wind sector boundaries to default by clicking on the **Reset** button.

Step 11 Select the canyon width fields from the drop-down box(es), which could either be the **Total canyon width** or **Left and right canyon widths**.

Step 12 When you have confirmed all the settings are as you desire, click the ▶ button in the **Urban canopy** section to create the output file(s). You will be prompted to enter the name of the output file. If the advanced setting **Create canopy shape file** is selected, the additional output **.shp* files will be in the same file stem. Once the files have been successfully created, the **Canyon and canopy tool** screen will close.

6.4.2 Pre-processing buildings

The buildings layer needs to be prepared before any calculations can be carried out by the tool. The pre-processing cleans up the buildings layer by removing all features that are smaller than the minimum height set in the **Advanced settings**, takes applicable actions to overlapping features and exports the resulting layer to new ESRI shape files (**.shp*) with the following new fields added:

- **Tower height field**: Height (m) of the part of an enclosed building that is above a shorter enclosing building (see example below).
- **Intersecting building field**: True if a building intersects with another building.

The removed building features are stored in additional set of shape files with the same file stem as the processed files affixed with '_X' e.g. Output_X.shp.

The buildings layer is analysed and searched for intersecting features as these may affect the main calculations carried out by the tool. Two or more buildings found in the same location generally represent a single building rendered into multiple parts. **Figure 74** shows a building feature enclosing another building of larger height. In this example, the **Tower height field** for the taller building will be assigned as the difference between height a and height b while for the shorter building, zero.

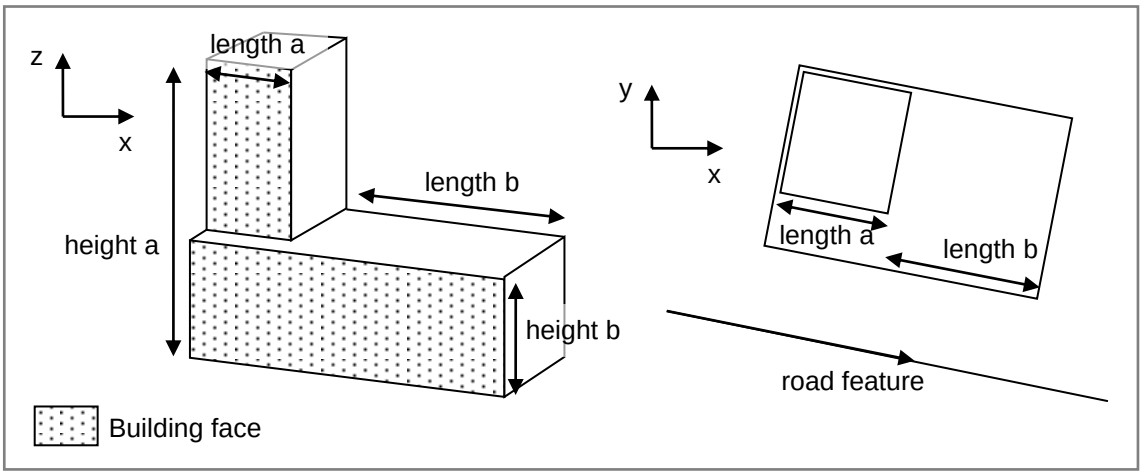


Figure 74 - Illustration of the approach taken with enclosed buildings

The **Tower height field** is only used in the calculations for the urban canopy parameter LambdaF, which requires the cross-wind vertical area of buildings shown in **Figure 74** as the building face with respect to the perpendicular wind.

If the enclosed building is smaller in height, it is therefore fully contained inside the larger building. Fully contained features are removed from the buildings layer during

pre-processing as they have no impact to the calculations.

Alternately, although less common, a building feature may not be enclosed by the intersecting building(s). In such cases, the tool assumes that every feature is separate. Each building is assessed and the common feature is assigned to the larger building and subtracted from the smaller building(s).

Figure 75 shows a variation of the previous example where building a is not enclosed by building b. In this example, building a will retain its geometry as it is the taller building while the intersecting part will be clipped from building b.

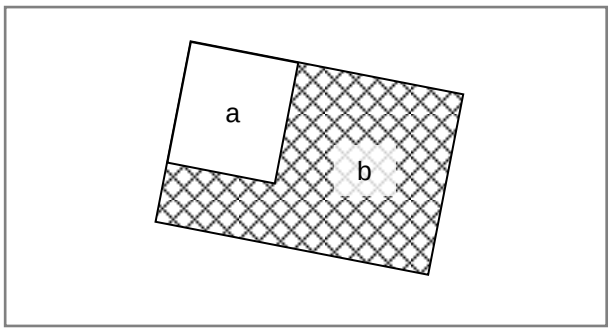


Figure 75 - Intersecting buildings without enclosing

6.4.3 Creating Street Canyon files

The **Canyon and canopy tool** generates perpendicular lines from the road centreline extending outward at equal intervals along the road to "search" or identify all building features that may affect the road. The length of the lines is the maximum distance from the road centreline that the tool will search buildings. The length of perpendicular lines and other settings for carrying out searches can be altered by the user in the **Advanced settings** (see Section 6.4.5).

The canyons are identified by the name of the corresponding road source. The left and right sides of the canyon are assigned according to the orientation of the first and second vertices, where the first vertex represents the start and the second vertex represents the end of the first straight-line segment of the road. The left and right side calculations are carried out independently.

For each road and each side of the road, the tool:

- Creates lines from the road centreline perpendicular to the road with a length equal to the **Maximum canyon width (m)** used to search for buildings that interact with the line. The first perpendicular line will be located at half the **Search spacing (m)** distance from the first road vertex. The succeeding lines are created every **Search spacing (m)** until the entire road is covered as in **Figure 76** for one side of the road. Each line will be assigned with a canyon width and height value to represent that portion of the road.

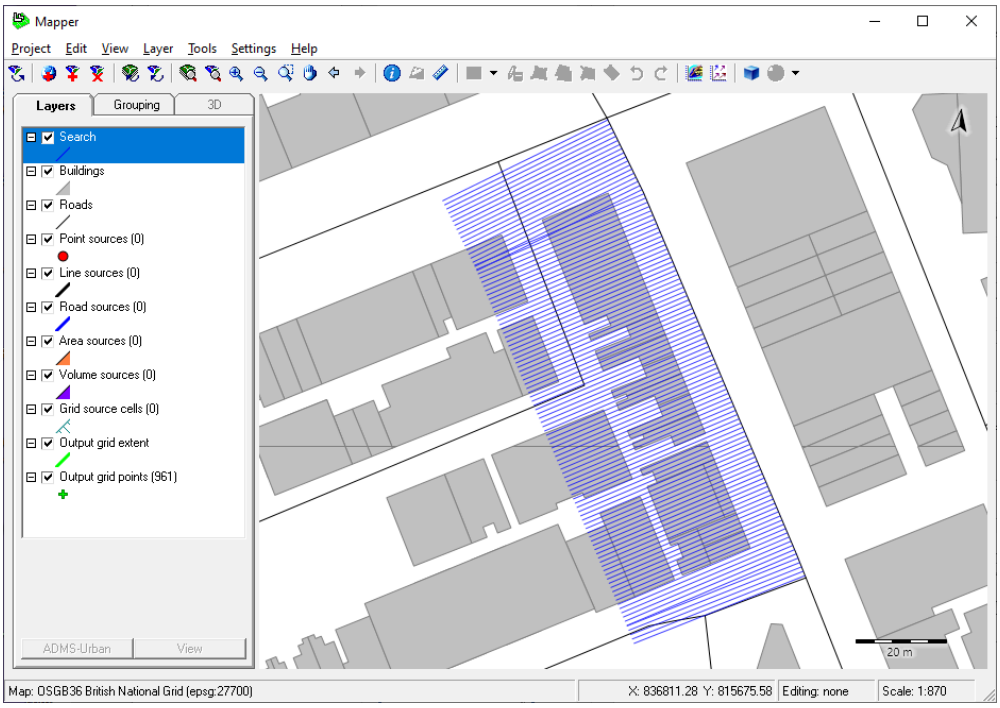


Figure 76 - Perpendicular lines generated on one side of the road

- Evaluates all interacting buildings to determine the values to assign each line. The canyon width is the distance from road centreline to the nearest interacting building. Only buildings above the **Gradient (height-to-width)** setting are evaluated to avoid including small structures that do not affect the road in a meaningful way. In the example in **Figure 77**, building a falls below the gradient and thus is not considered an interacting building.

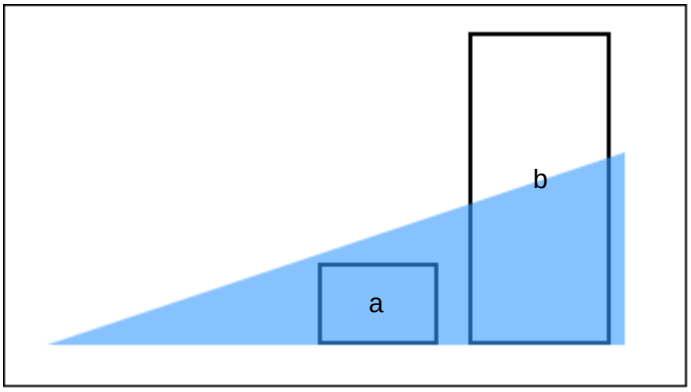


Figure 77 - The gradient setting used in evaluating which buildings to include

- Selects the building to take the canyon height from based on a dynamic, adaptive gradient. This gradient is adjusted according to the buildings included with the **Gradient (height-to-width)** setting. Buildings are included in the selection if their height is at or above the adaptive gradient, shown as red dashed lines in **Figure 78**. As a result, a building can still be excluded in the selection if it falls below the adaptive gradient. The canyon height is the height of the most significant building or the building with the largest gradient. Included buildings are numbered according to how far they are from the road, 1 being the nearest and so on. The selection limit is determined by the **Building search limit** setting, with a value of 0 being no limit. In the example, the height of building 2 is used.

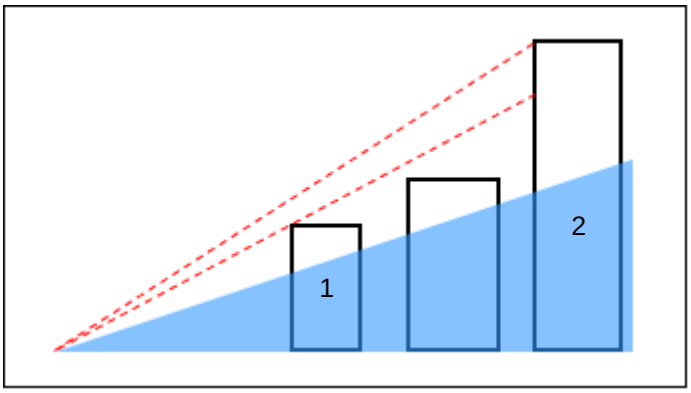


Figure 78 - The gradient setting used in evaluating which buildings to include

- Calculates the median and average of all the canyon width values assigned to each perpendicular line. Each canyon width is then compared against both the median and average. A tolerance for the comparison is set by multiplying the median and average by the **Quality tolerance factor**. The quality is determined by the number of individual width values, i.e. number of perpendicular lines the widths are assigned to, that fall within this tolerance. The tool selects the resulting width for the road between the median and average based on which has the higher quality.

In addition, the tool checks if the resulting canyon overlaps with the road carriageway by checking the canyon width against the road half-width.

- Calculates the average, minimum and maximum values of canyon heights assigned to each perpendicular line. Similarly, the quality of the average height is determined by the number of individual height values that fall within the tolerance calculated by multiplying the average by the **Quality tolerance factor**. However, the median value is not calculated and the quality is only used to indicate the reliability of the average height.
- Calculates the length of the road with adjacent buildings as the number of perpendicular lines with interacting buildings multiplied by the spacing.

The following limitations apply:

- A road with no adjacent buildings will be assigned values of 0 for all of the Advanced Canyon parameters.
- Buildings that touch or cross the road centreline are ignored for that road segment.
- Buildings must be above the **Minimum building height (m)** and **Minimum building footprint (m²)** requirement to be included in the calculation for any road segment.
- The parameter fraction of canyon area covered by overhanging features is not calculated by the tool and will be set to 0.

The parameters output by the **Canyon and canopy tool** are described in **Table 5** below. Parameters unique to the output shape file are described in **Table 6**.

Variable name in **.csv* file	Variable name in shape file	Units	Description
Name	-	-	Source (road) name
X1, Y1	-	m	x and y coordinates of first vertex (start)
X2, Y2	-	m	x and y coordinates of second vertex (end of first segment)
width_N[1]	c_width_n[2]	m	Distance from road centreline to canyon wall
avgHeight_N	c_height_n	m	Average canyon height
minHeight_N	c_h_min_n	m	Minimum canyon height
maxHeight_N	c_h_max_n	m	Maximum canyon height
buildLength_N	c_b_len_n	m	Length of buildings in canyon
fracCovered	-	-	Fraction of canyon area covered by overhanging features (always zero)

Table 5 - Advanced Canyon parameters. [1]Refers to variables for the left (L) and right (R) sides of the canyon. [2]Likewise but in lower case.

Variable name	Units	Description
c_widthtot	m	Total distance between canyon walls on left and right sides; sum of c_width_l and c_width_r
c_length_n	m	Length of canyon including gaps between buildings
c_start_n	m	Distance from start of road to first building in canyon
c_w_ave_n	m	Average canyon width
c_w_aveq_n	-	Quality of calculated canyon widths against the average value expressed as percentage of perpendicular lines per road source
c_w_med_n	m	Median canyon width
c_w_medq_n	-	Quality of calculated canyon widths against the median value expressed as percentage of perpendicular lines per road source
c_w_aorm_n	-	Indicates if the more accurate value of canyon width is the average ("A"), median ("M") or they are equally accurate ("E"); comparison of c_w_aveq_n and c_w_medq_n
c_w_ok_n	-	Blank if the canyon wall is outside or does not overlap with the road carriageway; otherwise, assigned with "x"
c_w_stdv_n	m	Standard deviation of canyon width values
c_h_qlty_n	-	Quality of calculated canyon heights against the average value expressed as percentage of perpendicular lines per road source
c_b_por_n	-	Amount of perpendicular lines not representing a canyon expressed as percentage per road source
c_b_pnmx_n	-	Amount of successive perpendicular lines not representing a canyon expressed as percentage per road source
c_b_plmx_n	m	Longest length of the road that is not a canyon
c_b_srch_n	-	Degree of how close to the road the buildings from which the canyon height was taken are, expressed as the average of the building number for each perpendicular line

Table 6 - Advanced Canyon parameters in the output shape file

6.4.4 Creating Urban Canopy files

The **Canyon and canopy tool** first defines the grid layout of the urban canopy data. The limiting coordinates of the user-specified **Grid definition** are rounded to start at a suitable coordinate divisible by the **Cell size (m)**. The rectangular grid is defined starting from the south-west corner extending eastward and northward by the cell size covering the entire buildings layer and, if applicable, roads layer. Each of the urban canopy parameters is then calculated for each cell.

- For the coordinates of a point in the cell, the cell centre is used.
- The average height of buildings in the cell is weighted by plan area. Buildings with a larger footprint will have a larger influence on the calculated average.
- If there are buildings present in the cell, the average canyon width is calculated for all canyon widths within the cell weighted by the length of each road. If there are no roads present in the cell or the user has selected **[No road layer]** from the **Roads** drop-down menu, the **Default canyon width (m)** is used.

  The urban canopy canyon widths provide the width from the road centreline to the buildings on each side of the road, not the width of the carriageway itself.

- LambdaP is calculated as the ratio of the horizontal area taken up by buildings or the building footprint within the cell to the total area of the cell.
- LambdaF is calculated as the ratio of the cross-wind vertical area of buildings to the total area of the cell, for each of the **Wind sector boundaries** provided by the user. The **Wind directions** used are the centre angles of each wind sector. For each direction, the vertical area of the building is calculated from the product of the cross-wind extent of the building and its height.

  When two or more buildings of varying height occur in the same location, the height and length of each part of the buildings is apportioned into multiple segments. This is illustrated in **Figure 74**. The tower height (see Section 6.4.2) is used to calculate the area of the building face for building a.

  Adjoining (touching) buildings are counted separately, not merged. The tool does not treat internal building faces, it only takes into account the widest points of a building perpendicular to the wind direction.

The parameters output by the **Canyon and canopy tool** are described in **Table 7** below.

Variable name in **.csv* file	**Variable name in shape file**	**Units**	**Description**
X, Y	X, Y	m	x and y coordinates of cell centre
H	H	m	Average building height
G	CanWidth	m	Average canyon width
LambdaP	LambdaP	-	LambdaP value
LambdaF <angle range>	LF_<angle range>	-	LambdaF values for binned wind sectors

Table 7 - Urban Canopy parameters

6.4.5 Advanced settings

Table 8 shows the available settings that the user can configure.

Setting			Description
Processing buildings	**Canyon**	**Minimum building height (m)**	Minimum height of buildings to include in the street canyon calculations.[1]
		Minimum building footprint (m²)	Minimum plan area of buildings to include in the street canyon calculations.
	Canopy	**Minimum building height (m)**	Minimum height of buildings to include in the urban canopy calculations.[1]
	Processed file field names	**Tower height field**	Field name to use in the processed **.shp* file for the tower height.
		Intersecting building field	Field name to use in the processed **.shp* file for the intersecting building indicator.
Canyons	**Search**	**Maximum canyon width (m)**	Distance from the road centreline the tool will search to find interacting buildings.
		Gradient (height-to-width)	Ratio of the height of building features on the side of the road and its distance to the road centreline, above which the building will be considered a canyon.
		Search spacing (m)	Distance between the generated perpendicular lines.
		Building search limit	Number of buildings to include in the selection for the canyon height based on an adaptive gradient.
	Quality tolerance factor	**Canyon width**	Multiplier for the calculated median and average canyon width. The resultant is used as tolerance for checking the accuracy of individual canyon widths.
		Canyon height	Multiplier for the calculated average canyon height. The resultant is used as tolerance for checking the accuracy of individual canyon heights.
Canopy	**Roads**	**Default canyon width (m)**	Value to use as average canyon width if no road features occur in a cell.
Output	**Additional output**	**Create canyon shape file**	Whether to create additional **.shp* file with street canyon parameters.
		Create canopy shape file	Whether to create additional **.shp* file with urban canopy parameters.
	CSV output	**Data delimiter**	Delimiter to use in the output **.csv* file.

Table 8 - Advanced settings

1. Building features that are below the smaller height value are removed from the layer during pre-processing.

6.5 Merge shapefiles

The **Shape file merging tool** allows the user to merge multiple **.shp* files and select which fields from those files to include in the merged output. The tool can also be used for renaming fields in the output file.

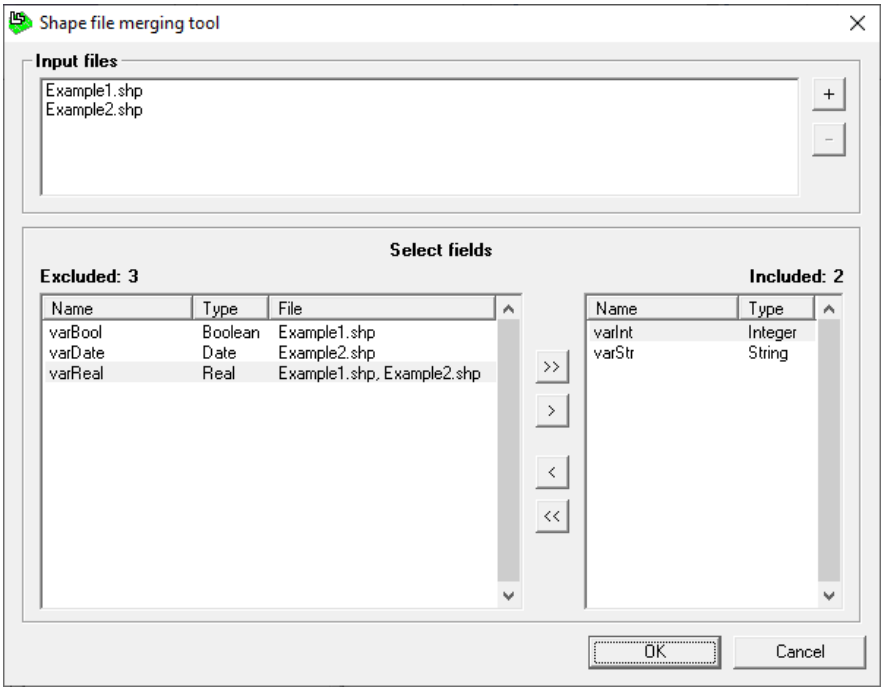


Figure 79 - Shape file merging tool

Files for merging are generally expected to have similar field structures. In cases where the field structure is not consistent across input files, mismatched fields are dealt as follows:

- If a field selected for the merged output is not defined in an input file, the tool sets it to a default value configurable by the user (see Section 6.5.2).
- If fields of the same name are defined as different data types in the input files, the tool allows the user to select which data type from a corresponding file to take (see Section 6.5.3).

The following limitations apply:

- Files cannot be empty i.e. must contain at least one feature.
- Files must be of the same coordinate system, shape type and dimension type.
- The shape file format only allows a maximum of 255 fields.
- Field names in the shape file format cannot be longer than 10 characters. The tool uses an aliasing system to allow for longer field names. However, the field names in aliased files may not be displayed correctly if viewed using other software.
- The maximum length of string type fields in the shape file format is 254. Also, float type fields are stored with 15 (positive values) and 14 (negative values) significant figures.

6.5.1 Using the Shape file merging tool

Step 1 Select the **Merge SHP files** option from the **Tools** menu to bring up the **Shape file merging tool** screen.

Step 2 Add the input shape files by clicking on the **Plus** button. The **Open Shape File** screen is then displayed.

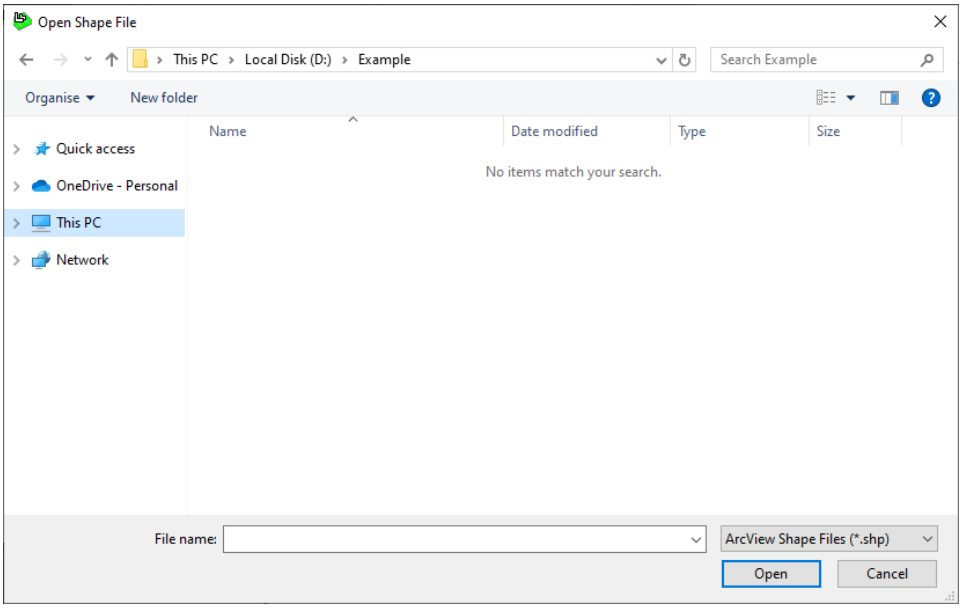


Figure 80 - Open Shape File

Step 3 Browse to find the **.shp* files to be merged then click on the **Open** button. The selected files will be added to the **Input files** list. The fields found in the files are automatically added to the **Excluded** list, showing the **Name** of the field, its **Type** and the **File(s)** it is found in. Remove a file from the list by selecting it and clicking the **Minus** button.

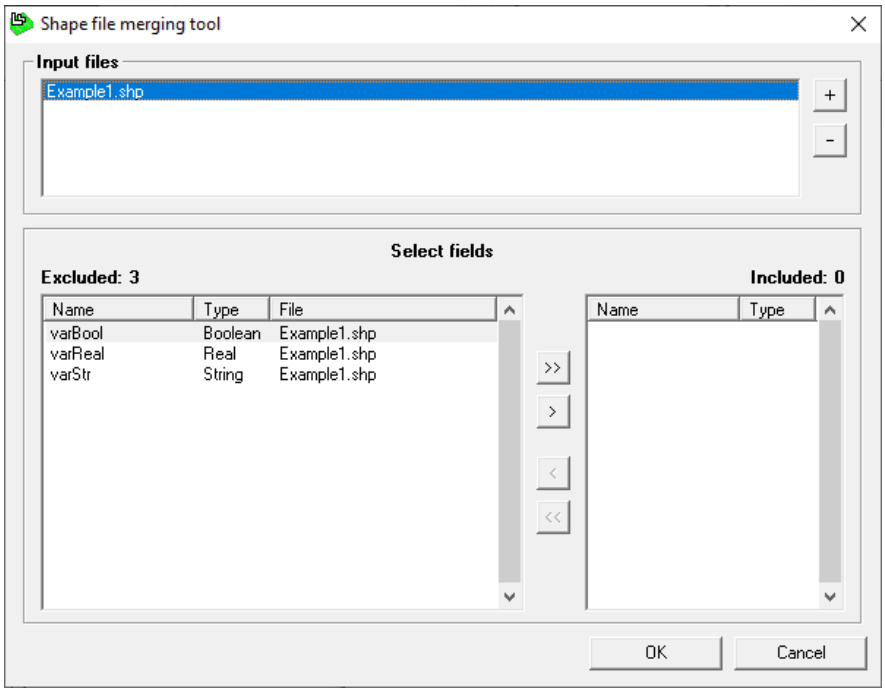


Figure 81 - Fields are added to the **Excluded** list upon loading input files

*As an alternative to Steps 2-3, file(s) can be drag-dropped from Explorer onto the **Input files** list.*

Step 4 Select the fields to be included in the output file by using the arrow buttons between the **Excluded** and **Included** lists.

- Move all fields by clicking on the **>>** or **<<** button.
- Move one or more selected fields by clicking on the **>** or **<** button.

Step 5 When you have confirmed that all the necessary fields to include in the output file have been selected, click **OK** to proceed with the merge. You will be prompted to enter the name of the output file. Once the file has been created, the **Shape file merging tool** screen will close.

6.5.2 Editing field names and default values

Step 1 Open the **Shape file merging tool** screen and add the files to be merged. The fields found in the files will be added to the **Excluded** list.

Step 2 Select the fields to be included in the output file by using the arrow buttons between the **Excluded** and **Included** lists.

Step 3 Edit a field in the **Included** list by double-clicking on it. The **Edit field** screen is then displayed where the following actions are available:

- Specify the new field name on the first textbox to rename it.
- Specify the default value on the second textbox to set the field to if it is not found in one or more files. This is necessary as shape files cannot contain null values.

For specifying the default value of Boolean-type fields, a checkbox is used rather than a textbox; for date-type fields, a date selector is used.

- Navigate to other items, if any, in the **Included** list by clicking on the arrow buttons.

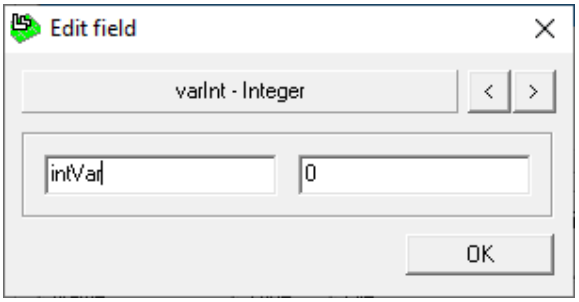


Figure 82 - Edit field screen

Step 4 Once you are satisfied with your edits, click the **OK** button. The **Included** list will be updated to use the new names.

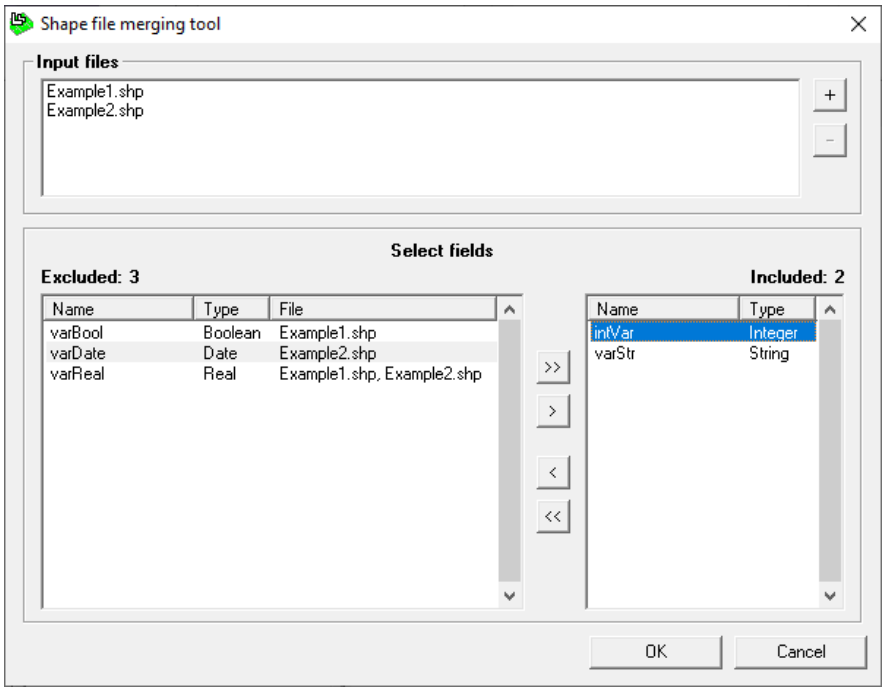


Figure 83 - Updated field name after editing

6.5.3 Resolving field type conflicts

Fields defined in multiple files as different data types are marked as **Conflict** under the **Type** column of the **Excluded** list. Fields marked as conflict must be resolved before they can be added to the **Included** list.

Step 1 Open the **Shape file merging tool** screen and add the files to be merged. The fields found in the files will be added to the **Excluded** list.

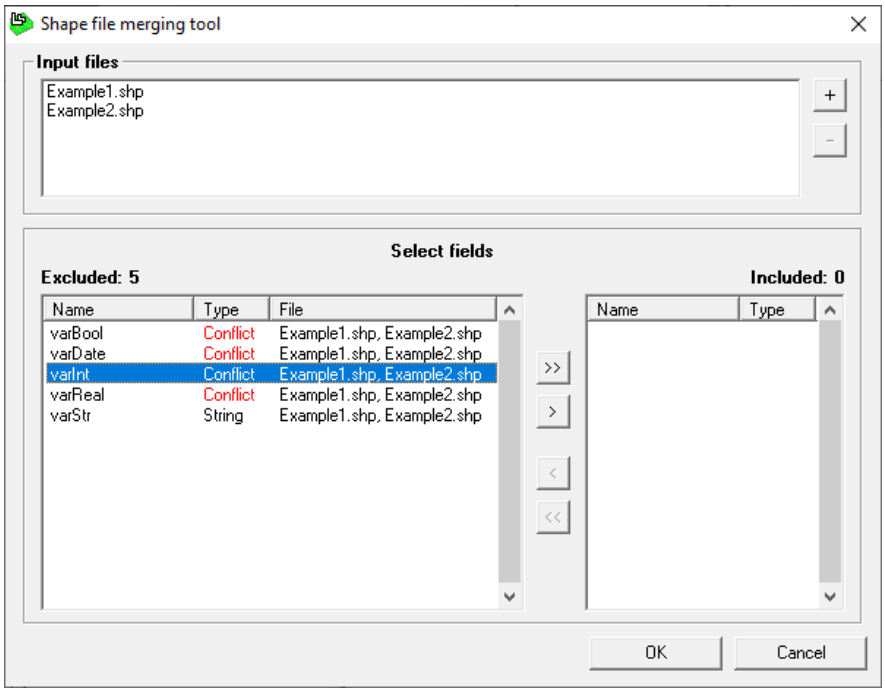


Figure 84 - Fields defined as varying types are marked as **Conflict**

Step 2 Resolve a **Conflict** field by double-clicking on it. The **Resolve conflict** screen is then displayed where the following actions are available:

- Select the checkbox next to the **Type** to derive the data type from the input **File** the field is defined in.
- Navigate to other items, if any, in the **Excluded** list by clicking on the arrow buttons.

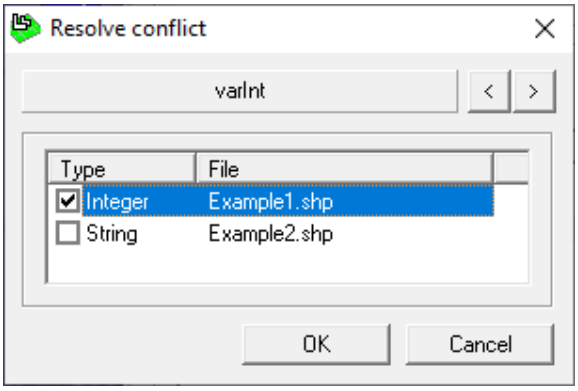


Figure 85 - Resolve conflict screen

Step 3 Once you have ensured all necessary fields have been resolved, click the **OK** button. The **Excluded** list will be updated with the selected data types. Fields that have been resolved can now be moved to the **Included** list.

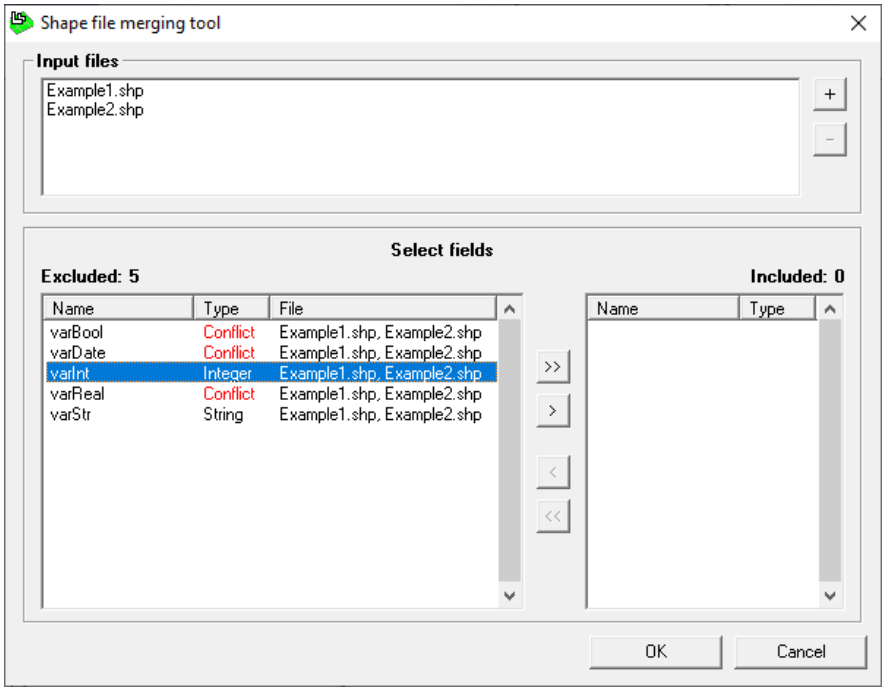


Figure 86 - Updated type after resolving conflict

When copying the field values of differing data types, simple conversion is done for compatible data types. Otherwise, incompatible types are set to default values (see Section 6.5.2). Some examples of acceptable conversion between dissimilar types are:

- Numerical string to float or integer
- Integer to Boolean: "1" = True; any other value = False

- String to Boolean: "`True`" or "`Yes`" = True; any other string = False
- Date to string, and vice-versa: Follows the system regional settings
- Any type to string

APPENDIX A Advanced topics

A.1 Label formatting

This section[2] outlines how to format the display labels using some of the more advanced features available in the Mapper.

A.1.1 Overview

The Mapper provides a simple markup language for custom label formatting. You can apply this in the **Label** section of the layer properties dialog for the layer.

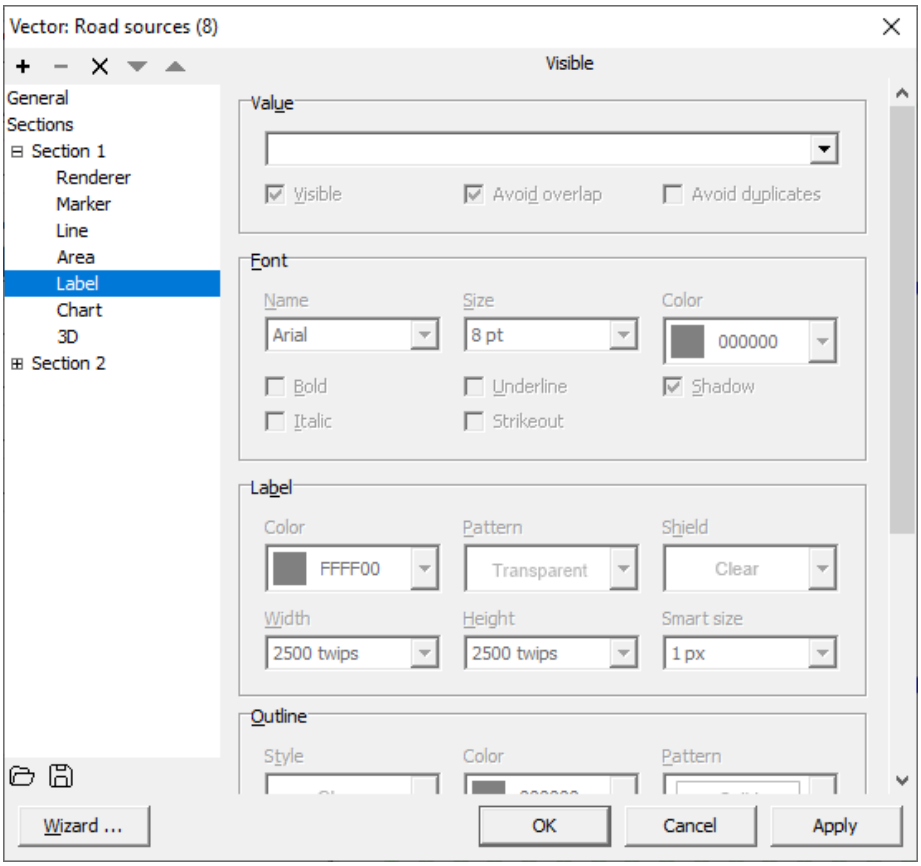


Figure 87 - Layer label properties

You can use the **Value** to both define and format the label using format descriptors described in the rest of this section. The label can be any combination of fixed text and fields defined in the current layer. A simple example would be to have the fixed text "Canyon" and then use the CanyonHeight field to display the canyon height for all roads using the specified formatting.

[2] This information is taken from the following site:
https://docs.tatukgis.com/EDT/ENU/Help/index.htm?mnulayerpropertiesvectorlabelformatting.htm

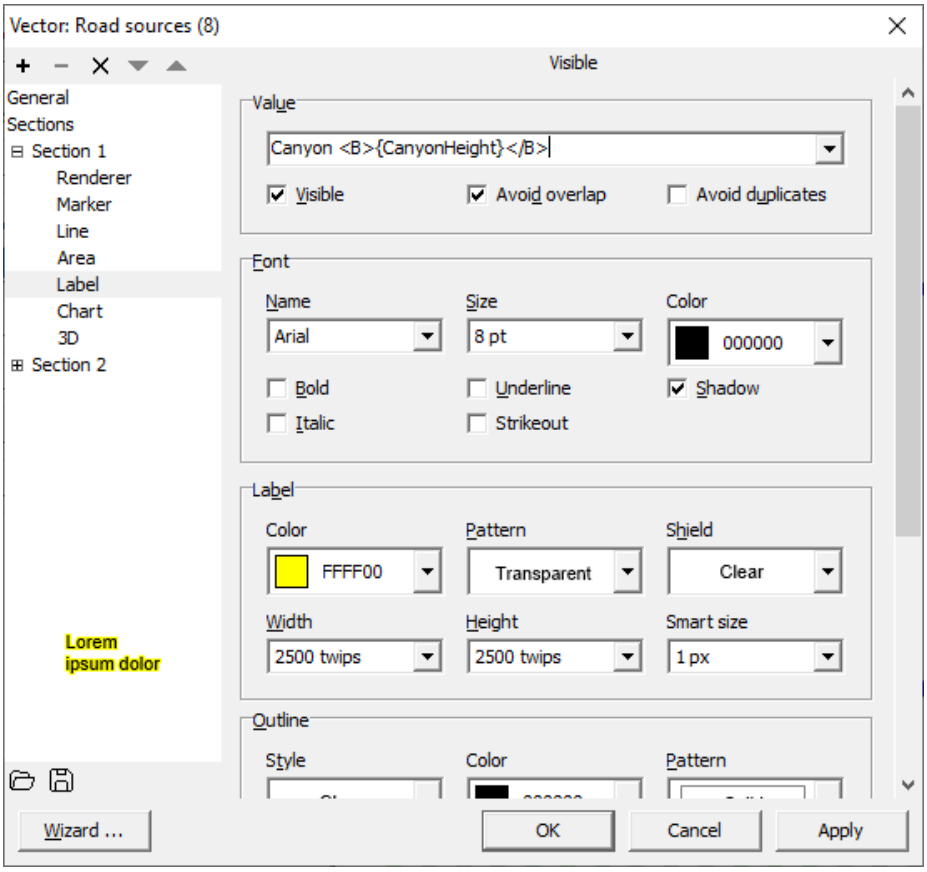


Figure 88 - Setting the label value option

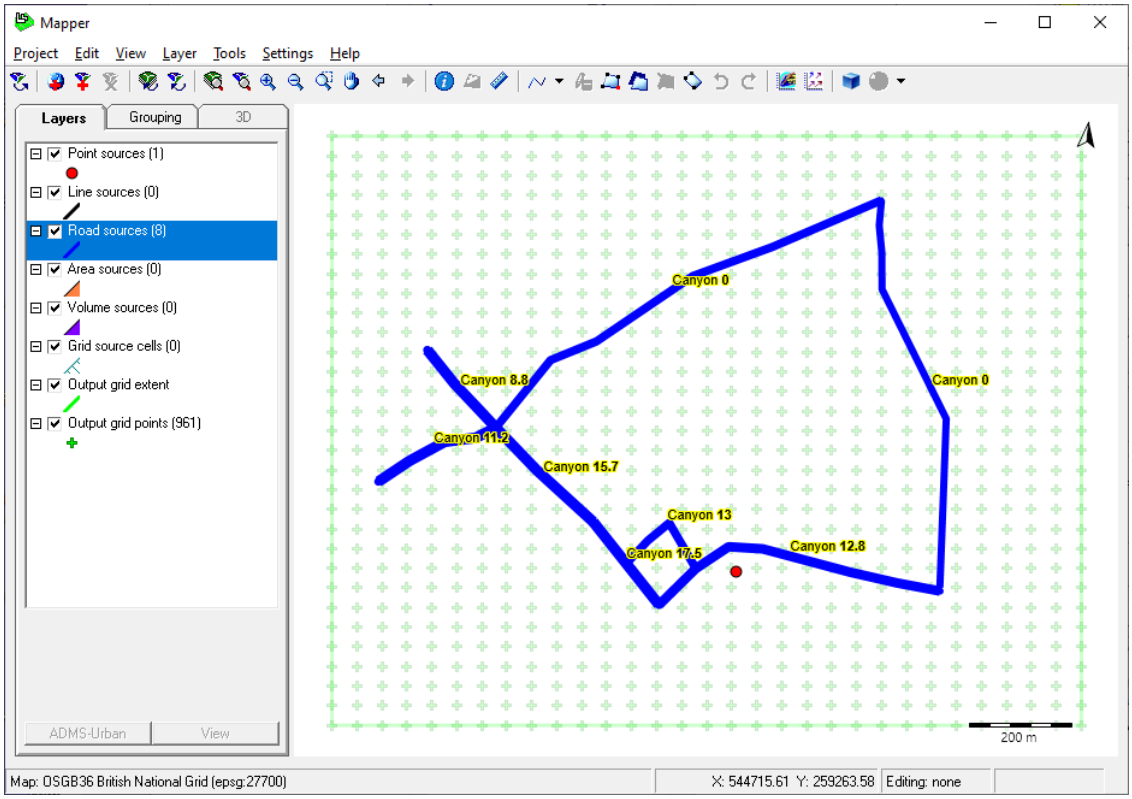


Figure 89 - Map with formatted labels

A.1.2 Visual formatting

The **Value** input supports some of the standard HTML markers for visual formatting:

Markup	Description
<B></B>	Text inside is displayed in bold
<I></I>	Text inside is displayed in italic
<U></U>	Text inside is underlined
<HR>	Horizontal line
 	Line break
<P></P>	Text inside is treated as a complete paragraph
<STRONG></STRONG>	Text inside is displayed in extra bold font
<FONT SIZE="S" NAME="F" COLOR="C"></FONT>	Text inside is displayed in the F font of the size S and colour C F: ARIAL, TIMES, TAHOMA, etc. C: BLACK, BLUE, FUCHSIA, GRAY, GREEEN, LIME, MAROON, NAVY, OLIVE, PURPLE, RED, SILVER, TEAL, WHITE, YELLOW The C colour can be also given in the RGB representation #RRGGBB, where RR, GG and BB are two digit hexadecimal number representations of the red, green and blue colour intensities
&	The ampersand (&) sign
	Forced insertion of space
>	The greater-than (>) sign
<	The less-than (<) sign

Table 9 - Simple markup

Example

```
 <U><FONT NAME="ARIAL" COLOR=#FF0000><I>value1</I></FONT> <FONT
NAME="ARIAL" COLOR=#0000FF><B>&lt;</B></FONT> <FONT NAME="ARIAL"
COLOR=#00FF00><BOL>value2</BOL></FONT></U>
```

The above code will result in the following label: *value1* **< value2** .

A.1.3 Inserting fields

The code **{FIELD_NAME:FORMAT}** inserts the value of a field or attribute named **FIELD_NAME** with formatting defined by the format string **FORMAT**. The format string is attribute type specific.

A.1.4 Formatting of NUMBER and FLOAT type attributes

Automatic formatting:

Format character	Format name	Description
C or c	Currency	Integer following the character defines the number of digits displayed after the decimal point (default is 2). Uses the currency marker from the system settings.
D or d	Digit (integer)	Integer next to the character defines the minimum number of digits to be displayed; number will be padded with zeroes if necessary; non-integer value will be rounded first.
E or e	Scientific (exponential)	Integer next to the character defines the number of digits displayed after the decimal point (default is 2); number formatted as *Scientific* is always displayed in exponential form;
F or f	Fixed-point	Integer next to the character defines the number of digits displayed after the decimal point (default is 2).
G or g	General	Displays the number in a format best suited for the value; Integer next to the character defines the number of significant digits.
N or n	Number	Integer following the character defines the number of digits displayed after the decimal point (default is 2); inserts thousand separators accordingly to the system settings.
P or p	Percent	The number is multiplied by 100; Integer following the character defines the number of digits displayed after the decimal point (default is 2)
R or r	Round-trip	Equivalent of *General*; Integer next to the character is ignored.
X or x	Hexadecimal	Integer next to the character defines the minimum number of hexadecimal digits to be displayed; number will be padded with zeroes if necessary; non-integer values will be rounded first.

Table 10 - Automatic number formatting

Custom formatting:

Format character	Name	Description
#	Digit placeholder	Defines optional position of a digit; if corresponding digit in the value does not exist, the digit placeholder is neglected.
0	Zero placeholder	Defines obligatory position of a digit; if corresponding digit in the value does not exist, the zero placeholder is replaced by '0'.
.	Decimal point	Defines the position of the decimal point.
[other]		Other characters are displayed without any change.

Table 11 - Custom number formatting

Examples:

Code	Attribute value	Result string	Note
{FIELD_NAME:#.###}	123.59	'123.59'	
{FIELD_NAME:#.000}	123.59	'123.590'	
{FIELD_NAME:#.####0}	123.59	'123.59000'	
{FIELD_NAME:#####.###}	-123.59	'-123.59'	
{FIELD_NAME:00000.###}	123.59	'00123.59'	
{FIELD_NAME:0-0-0-0-0}	123.59	'0-0-1-2-4'	rounding
{FIELD_NAME:over #;below #;zero #}	123.59	'over 124'	rounding
{FIELD_NAME:over #;below #;zero #}	-123.59	'below 124'	rounding, no minus sign
{FIELD_NAME:over #;below #;zero #}	0	'zero '	no zero sign
{FIELD_NAME:over #;below #;zero 0}	0	'zero 0'	
{FIELD_NAME:;below #;zero#}	123.59	''	empty string
{FIELD_NAME:over #;;zero#}	-123.59	''	empty string
{FIELD_NAME:over #;below #;}	0	''	empty string

Table 12 - Number formatting examples

A.1.5 Formatting of BOOLEAN type attributes

Automatic formatting:

Code	Attribute value	Result
{FIELD_NAME}	True	'True'
{FIELD_NAME}	False	'False'

Table 13 - Automatic Boolean formatting

Examples:

Code	Attribute value	Result	Note
{FIELD_NAME:yes;no}	True	'yes'	
{FIELD_NAME:yes;no}	False	'no'	
{FIELD_NAME:yes}	True	'yes'	
{FIELD_NAME:yes}	False	''	empty string
{FIELD_NAME:;no}	True	''	empty string
{FIELD_NAME:yes;}	False	''	empty string
{FIELD_NAME:yes;no1;no2}	False	'no1;no2'	

Table 14 - Boolean formatting examples

A.1.6 Formatting of DATE type attributes

Note: character in brackets represents a string consisting of any number of this character.

Format string	Format name	Description
d	Day	Day as a number 1 to 31
dd	Day	Day as a number 01 to 31
ddd	Day	Day as an abbreviated name of the day of the week; depends on system regional settings
dddd(d)	Day	Day as a full name of the day of the week; depends on system regional settings
f	Second fraction	Most significant digit of the second fraction
ff	Second fraction	Two most significant digits of the second fraction
fff	Second fraction	Three most significant digits of the second fraction
ffff	Second fraction	Four most significant digits of the second fraction
fffff	Second fraction	Five most significant digits of the second fraction
ffffff	Second fraction	Six most significant digits of the second fraction
fffffff(f)	Second fraction	Seven most significant digits of the second fraction
F	Second fraction	Most significant digit of the second fraction; if zero then nothing will be displayed
FF	Second fraction	Two most significant digits of the second fraction; trailing zeros will be truncated
FFF	Second fraction	Three most significant digits of the second fraction; trailing zeros will be truncated
FFFF	Second fraction	Four most significant digits of the second fraction; trailing zeros will be truncated
FFFFF	Second fraction	Five most significant digits of the second fraction; trailing zeros will be truncated
FFFFFF	Second fraction	Six most significant digits of the second fraction; trailing zeros will be truncated
FFFFFF(F)	Second fraction	Seven most significant digits of the second fraction; trailing zeros will be truncated
h	Hour	Hour as a number 1 to 12
hh(h)	Hour	Hour as a number 01 to 12
H	Hour	Hour as a number 1 to 24
HH(H)	Hour	Hour as a number 01 to 24
m	Minute	Minute as a number 0 to 59
mm(m)	Minute	Minute as a number 00 to 59
M	Month	Month as a number 1 to 12
MM	Month	Month as a number 01 to 12
MMM	Month	Month as an abbreviated name of the month; depends on

		system regional settings
MMMM(M)	Month	Month as a full name of the month; depends on system regional settings
s	Seconds	Seconds as a number 0 to 59
ss(s)	Seconds	Seconds as a number 00 to 59
t	AM/PM	First character of the 'AM' or 'PM' string; depends on system regional settings
tt(t)	AM/PM	'AM' or 'PM' string; depends on system regional settings
y	Year	Year as a two digit number; if year has less than two digits then one digit number will be displayed
yy	Year	Year as a two digit number; if year has less than two digits then result will be padded with zero
yyy	Year	Year as a three digit number; if year has less than three digits then result will be padded with zeros
yyyy	Year	Year as a four digit number; if year has less than four digits then result will be padded with zeros
yyyy(y)	Year	Year as a five digit number; if year has less than five digits then result will be padded with zeros
[other]		Other characters are displayed without any change

Table 15 - Custom date formatting

Examples:

Code	Attribute value	Result
{FIELD_NAME:d}	3rd of June 2008: 17:09:03.650, US English	'3'
{FIELD_NAME:dd}	3rd of June 2008: 17:09:03.650, US English	'03'
{FIELD_NAME:ddd}	3rd of June 2008: 17:09:03.650, US English	'Tue'
{FIELD_NAME:dddd}	3rd of June 2008: 17:09:03.650, US English	'Tuesday'
{FIELD_NAME:M}	3rd of June 2008: 17:09:03.650, US English	'6'
{FIELD_NAME:MM}	3rd of June 2008: 17:09:03.650, US English	'06'
{FIELD_NAME:MMM}	3rd of June 2008: 17:09:03.650, US English	'Jun'
{FIELD_NAME:MMMM}	3rd of June 2008: 17:09:03.650, US English	'June'
{FIELD_NAME:y}	3rd of June 2008: 17:09:03.650, US English	'08'
{FIELD_NAME:yy}	3rd of June 2008: 17:09:03.650, US English	'08'
{FIELD_NAME:yyy}	3rd of June 2008: 17:09:03.650, US English	'008'
{FIELD_NAME:yyyy}	3rd of June 2008: 17:09:03.650, US English	'2008'
{FIELD_NAME:yyyyy}	3rd of June 2008: 17:09:03.650, US English	'02008'
{FIELD_NAME:h}	3rd of June 2008: 17:09:03.650, US English	'5'
{FIELD_NAME:hh}	3rd of June 2008: 17:09:03.650, US English	'05'
{FIELD_NAME:H}	3rd of June 2008: 17:09:03.650, US English	'17'
{FIELD_NAME:HH}	3rd of June 2008: 17:09:03.650, US English	'17'
{FIELD_NAME:t}	3rd of June 2008: 17:09:03.650, US English	'P'
{FIELD_NAME:tt}	3rd of June 2008: 17:09:03.650, US English	'PM'
{FIELD_NAME:m}	3rd of June 2008: 17:09:03.650, US English	'9'
{FIELD_NAME:mm}	3rd of June 2008: 17:09:03.650, US English	'09'
{FIELD_NAME:s}	3rd of June 2008: 17:09:03.650, US English	'3'
{FIELD_NAME:ss}	3rd of June 2008: 17:09:03.650, US English	'03'
{FIELD_NAME:f}	3rd of June 2008: 17:09:03.650, US English	'6'
{FIELD_NAME:ffff}	3rd of June 2008: 17:09:03.650, US English	'6500'
{FIELD_NAME:FFFF}	3rd of June 2008: 17:09:03.650, US English	'65'
{FIELD_NAME:yyyy/MM/dd HH/mm/ss.ffff}	3rd of June 2008: 17:09:03.650, US English	'2008/06/03 17:09:03.6500'

Table 16 - Date formatting examples

A.1.7 Formatting of STRING type attributes

Note: i is a placeholder for an integer number.

Format character	Name	Description
`$`	normal text	Text displayed without any changes.
`S`	uppercase	Text displayed in uppercase.
`s`	lowercase	Text displayed in lowercase.
i *[at the beginning]*	first character	**i**>0 defines from which character (counting from the beginning of the string) should be the string displayed; if **i**<0 then the characters are counted from the end of the string.
i *[at the end]*	last character	**i**>0 defines up to which character (counting from the beginning of the string) should be the string displayed; if **i**<0 then the characters are counted from the end of the string.
`H`	force HTML	force HTML interpretation of <> symbols

Table 17 - Custom string formatting

Examples:

Code	Attribute value	Result string	Note
{FIELD_NAME:$}	'Abcdefghij'	'Abcdefghij'	
{FIELD_NAME:S}	'Abcdefghij'	'ABCDEFGHIJ'	
{FIELD_NAME:s}	'Abcdefghij'	'abcdefghij'	
{FIELD_NAME:1$-1}	'Abcdefghij'	'Abcdefghij'	
{FIELD_NAME:1S3}	'Abcdefghij'	'ABC'	
{FIELD_NAME:1s3}	'Abcdefghij'	'abc'	
{FIELD_NAME:1$-3}	'Abcdefghij'	'Abcdefgh'	
{FIELD_NAME:-3$-1}	'Abcdefghij'	'hij'	
{FIELD_NAME:4$-4}	'Abcdefghij'	'defg'	
{FIELD_NAME:8$-8}	'Abcdefghij'	''	empty string
{FIELD_NAME:H}	'A<b>c'	'A<b;>c'	
{FIELD_NAME:H$}	'A<b>c'	'A<b;>c'	
{FIELD_NAME:HS}	'A<b>c'	'A<B;>C'	
{FIELD_NAME:Hs}	'A<b>c'	'a<b;>c'	
{FIELD_NAME:H1s3}	'A<b>c'	'a<b'	

Table 18 - String formatting examples

A.2 Using symbology files

The Mapper supports symbology (SYM) files, which use a series of command instructions to create custom line styles. When drawing the layer using a symbology file, the Mapper will follow the rendering instructions to draw custom lines. The "mini-language" used to code the instructions is set out in **Table 19**.

A.2.1 Transparent roads example

There are times when you wish to see the concentration contours but when the source is displayed the contours are obscured. This can be the case for **Road sources** where the highest concentrations are along the road. Hiding the road layer (unticking it in the layer panel) is a possibility, but then you lose the visual information about the road layout. On solution would be to display road sources with the contour plot showing through – i.e. delineated by their borders alone so that the main body of the polyline would be transparent.

The following example demonstrates how a symbology file can be applied to the **Road sources** layer in order to produce the transparency effect described above.

Step 1 Double click on the layer of interest in the layer panel to bring up the layer properties window – in this case bring up the properties for the **Road sources** layer as shown in **Figure 90**.

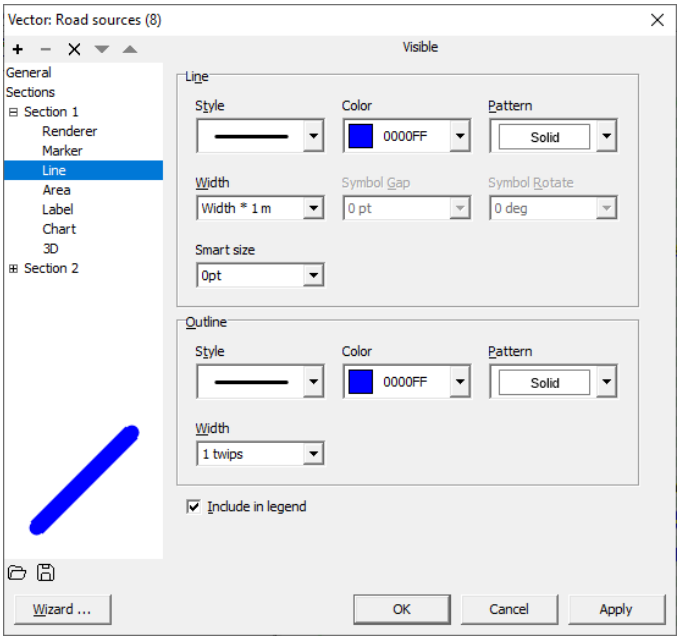


Figure 90 - Layer properties window

Step 2 Choose **Line** from the left panel, and in the **Line** section, use the drop-down menu under **Style** to choose the **Symbol**... button in order to bring up the **Symbol** dialog, see **Figure 91**.

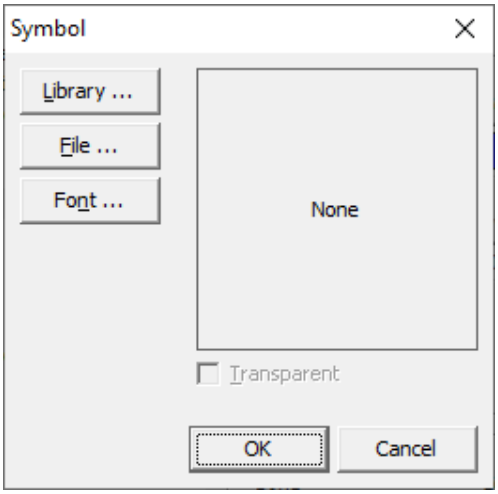


Figure 91 - Symbol dialog

Step 3 Click the **File** button in the **Symbol** dialog and browse for the example *.sym* file supplied with the installation, **Figure 92**.

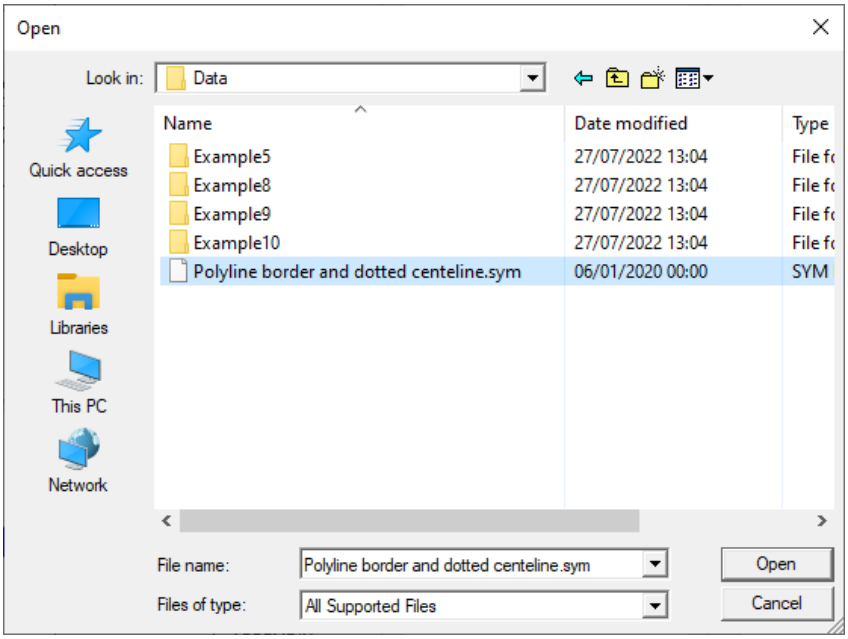


Figure 92 - Selecting a symbology file

Step 4 Click **OK** to return to the Layer properties window (**Figure 93**) and either click **Apply** or **OK** to apply the chosen symbology to the **Road sources** layer, **Figure 94**.

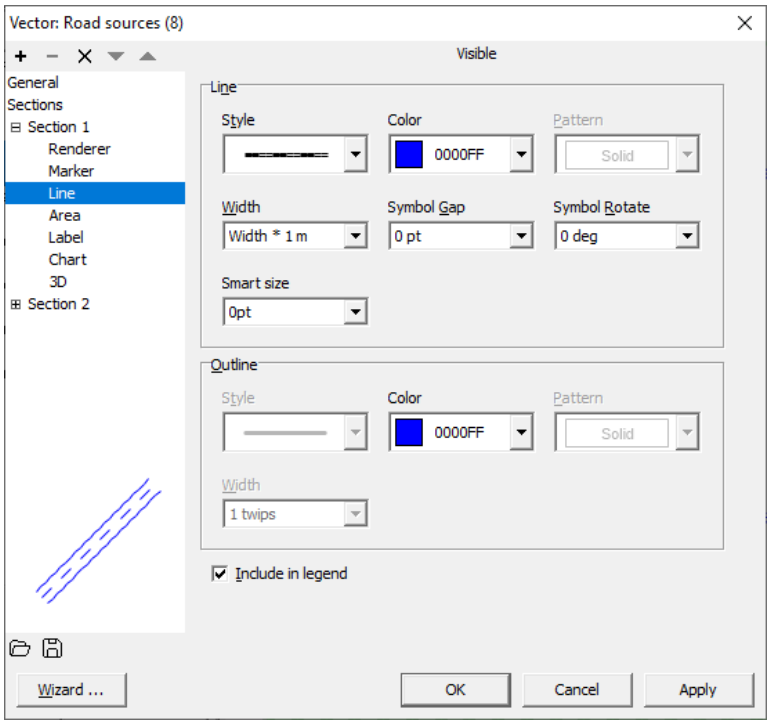


Figure 93 - Layer properties window with symbology selected

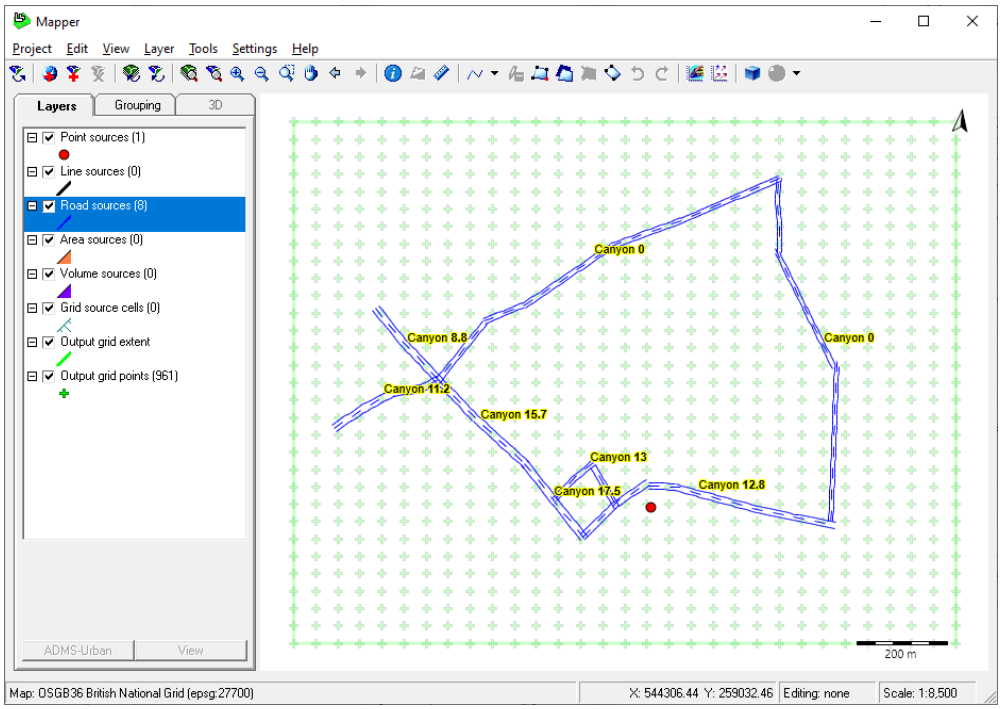


Figure 94 - Roads layer with symbology applied.

Step 5 Load a contour layer to see the effects of the new rendering, **Figure 95** and **Figure 96**.

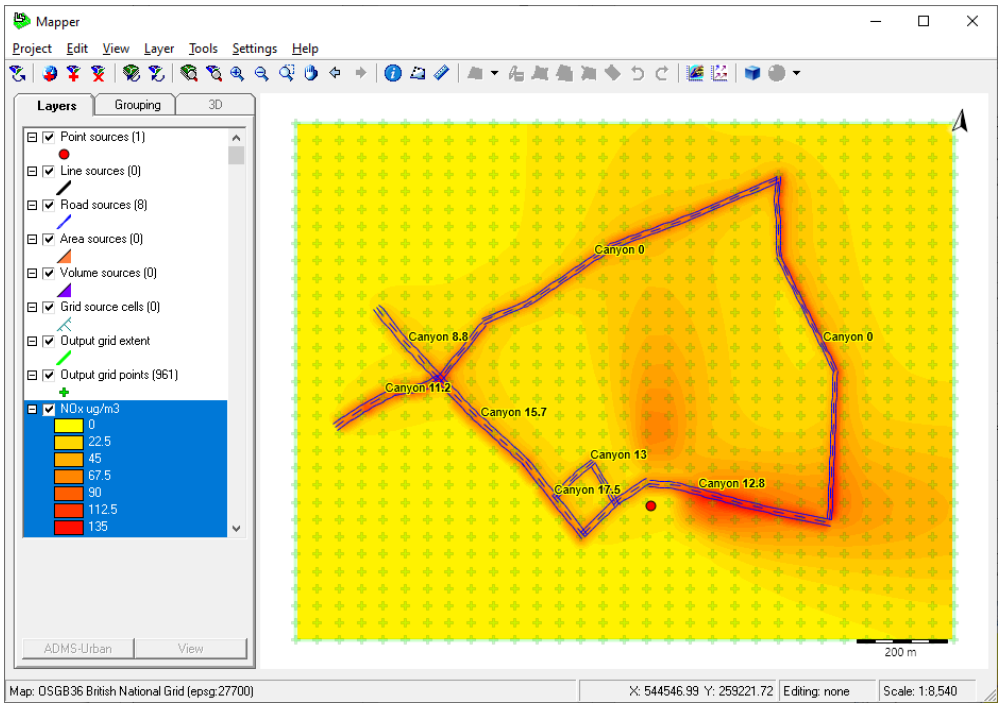


Figure 95 - Roads layer with symbology applied over a contour layer.

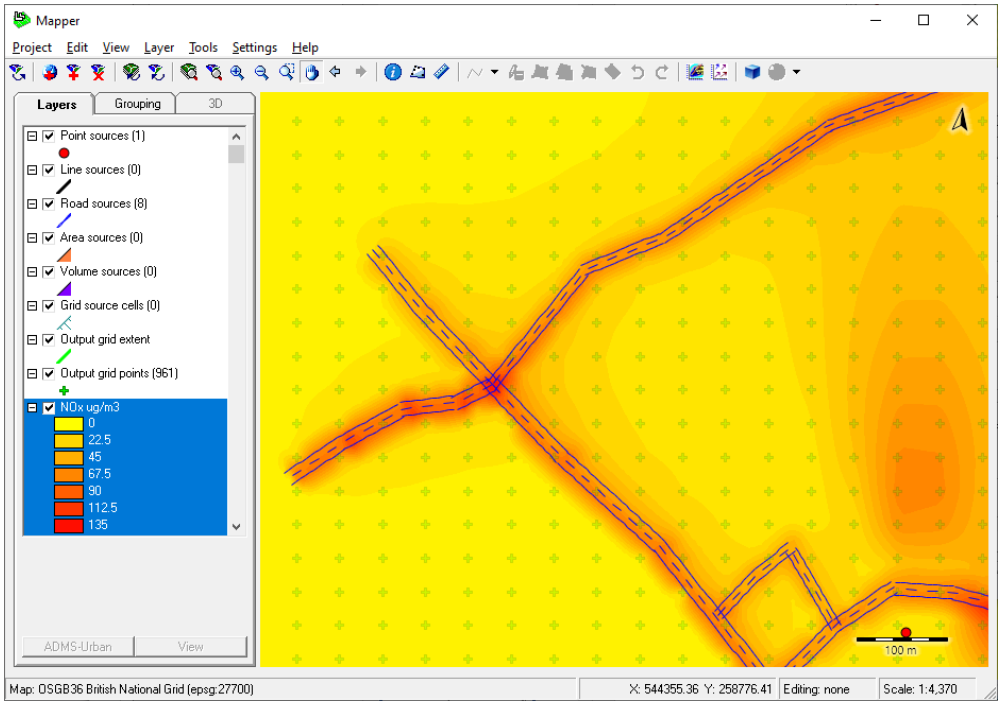


Figure 96 - Close up of Roads layer with symbology applied over a contour layer.

A.2.2 Making symbology files

The following list summarises the key elements of using symbology files

- All parameters (except GOTO) are relative and move the "drawing position" cursor along the shape.

- The X coordinate moves cursor along the shape. The Y coordinate moves cursor perpendicularly.
- Drawing can contain as many commands as desired.
- Using GOTO(0) enables drawing over an already drawn line.
- FOR..END loop cannot be embedded into another loop.
- The more complicated the symbol, the slower the drawing speed.

Commands

Full	Short	Description	Sample
`;`		Comment out whole line	`;A comment`
`GOTO(x)`	`G`	Move drawing position along the shape to an arbitrary position.	`GOTO(10%)` `G(10%)`
`MOVE(dx,dy)`	`M`	Move drawing along line by a delta.	`MOVE(10,-10)` `M(+10-10)`
`DRAW(x1,y1,x2,y2,..,xn,yn)`	`D`	Draw a vector segment.	`DRAW(5,5 5 -5)` `D(+5+5+5-5)`
`OUTLINE(x1,y1,x2,y2,..,xn,yn)`	`O`	Draw a line symbol with the rotation point at the place of the current "drawing position". Whole symbol will be rotated with the same parameters. First set of coordinates means the starting point of the shape.	`OUTLINE(5,5 -10,0 0,-10 10,0 0,-10)` `O(+5+5-10+0+0-10+10+0+0-10)`
`FILL(x1,y1,x2,y2,..,xn,yn)`	`I`	Draw a filled symbol with the rotation point at the place of the current "drawing position". Whole symbol will be rotated with the same parameters. First set of coordinates means the starting point of the shape.	`FILL(5,5 -10,0 0,-10 10,0 0,-10)` `I(+5+5-10+0+0-10+10+0+0-10)`
`FOR(x)`	`F`	Start of the loop. Parameters for the distance the loop will be repeated. Default is 100%. A negative value means calculation starting from the end of the line. For example, FOR(-45%) means repeat until 45% of the distance of the entire line length, starting from the end of the line.	`FOR(50%)` `F(50%)`
`END()`	`E`	End of the loop.	`END()` `E()`
`WIDTH(x)`	`W`	Change line width.	`WIDTH(10W)` `W(10W)`
`COLOR(r,g,b)`	`C`	Change drawing color.	`COLOR(255,0,0)` `C(255,0,0)`
`COLOR(mode)`	`C`	Change drawing color to: 0 - line/area color 1 - outline color	`COLOR(1)` `C(1)`

Table 19 - Symbology commands summary

Values option

Option	Description	Sample
None	Default – pixels	GOTO(10)
X	Device independent pixels - 1 pixel in 96dpi. Multiplied by 10 so 10X means one "pixel"	GOTO(10X)
T	Twips (1/1440 of inch)	GOTO(700T)
W	Line width (as set for Line.Width) expressed multiplied by 10. So 15T means 1.5 line widths.	GOTO(15T)
S	Line with scaled device independent pixels (similar to 'X'). Scales means that wider lines will result no proportional scaling: 10X for thin lines means really same as 10X but for wider line it will be represented by smaller value for better visual effect	GOTO(10S)

Table 20 - Symbology measuring settings

A.3 Supported file types

This section details the file types that are supported by the Mapper.

A.3.1 Recognised file types

The Mapper supports loading the following recognised file types and formats natively. This allows them to be displayed in the map view as new non-editable layers. The list is not comprehensive, but gives the most commonly supported file types.

Format	*File extension*
ArcExplorer Project File	*.aep
Arcinfo Ascii Grid	*.asc
Arcinfo Binary Grid	*.adf
Arcinfo Export Format	*.e00
Arcinfo Float Grid	*.flt
ArcView 3.xx Project File	*.apr
ArcView Shape Files	*.shp
AutoCAD DWG	*.dwg
AutoCAD DXF	*.dxf
Binary Terrian Grid	*.bt
CADRG Compressed ARC Digitized Raster Graphics	*.toc
Census 2000 TIGER/Line	*.rt1
Comma Separated Values point layer (CSV)	*.csv and others
Digital Elevation Model (ASCII GRID or SPOT)	*.dem
Digital Line Graphs (DLG)	*.opt;*.dlg
Digital Terrain Elevation Data	*.dt0;*.dt1;*.dt2
Enhanced Compressed Wavelet	*.ecw
ERDAS IMAGINE Image File Format	*.img
ESRI File Geodatabase API for vectors	*.ttkls; *.ttklayer
FME Feature Store	*.ffs
GDAL Wrapper for raster formats	(multiple)
Geographic Data Files	*.gdf
Geographic Markup Language	*.gml;*.xml
GeoJSON	*.json;*.geojson
Global Self-consistent Hierarchical High-res Shorelines GSHHS	*.rim;*.b
Golden Software Surfer Grid	*.grd
Google Earth Keyhole Markup Language	*.kml;*.kmz
GPS Exchange Format	*.gpx
Graphic Interchange Format	*.gif
IFC	*.ifc and others
IHO S-57 ENC	*.000
JPEG 2000 via ECW	*.jp2
JPEG 2000 via MRSID	*.jp2
JPEG File Interchange Format	*.jpg;*.jpeg
LandXML	*.xml

• LiDAR LAS (ASPRS)	*.las;*.laz
• Lizardtech MrSID	*.sid
• Mapbox Vector Tile Format	*.mvt;*.pbf
• MapInfo Interchange Format	*.mif
• MapInfo Native Format	*.tab
• MapInfo WorkFile	*.wor
• MapInfoX Project File	*.gst
• MicroStation Native Format	*.dgn
• OGR Wrapper for vector formats	(multiple)
• Open Asset Import Library	(multiple)
• OpenStreetMap Format	*.osm;*.pbf
• Pictometry Oblique Images	*.psi
• PLY Polygon File Format	*.ply
• Portable Network Graphic	*.png
• Protocol Layer Connector	*.ttkwp; *.ttklayer
• QGIS Project File	*.qgs;*.qgz
• SAGA GIS Binary Grid	*.sdat
• SDTS Raster Profile and Extensions	*.ddf
• SDTS Topological Vector Profile	*.ddf
• SPOT Binary In-line Format	*.bil;*.dem
• SQL Layer Connector	*.ttkls;*.ttklayer
• SRTM HGT Grid	*.hgt
• Standard Tessallation Language	*.stl;*.stla
• Tag Image File Format	*.tif;*.tiff
• TatukGIS PixelStore	*.ttkps;*.ttklayer
• TatukGIS Project File	*.ttkproject;*.ttkgp
• Vector Product Format (VPF)	*.dht; *.lht
• WexBIM	*.wexbim
• Window Bitmap	*.bmp
• XYZ Point Cloud data	*.xyz; *.asc

A.3.2 Overriding recognised file types

Files containing numerical data may share extensions with one of the recognised file types. The `ShowForm` setting under the `[ExtendedFileTypes]` section in the ADMS model interface **.ini* file specifies a list of file extensions that should be handled as a general numerical data file. This allows displaying of the file as in Section 3.4. The value for this setting is a semicolon-delimited string of file extensions.

Example

```
ShowForm=*.csv;*.ext
```

A.3.3 Adding bespoke text file types

The `KnownTypes` setting under the `[ExtendedFileTypes]` section in the ADMS model interface **.ini* file expands the Mapper's file handling for known file types. This setting defines a list of custom behaviours for specific file extensions by pointing to a layer configuration (**.ttkstyle*) file stored in the Config directories in the ADMS model install. The value for this setting is a semicolon-delimited string of known file types. Each file type section includes up to three additional parameters: the configuration file, the header alias and the data start line, separated by a pipe ("|").

The general form of a file type section is as follows, where `[]` denote optional entries

```
extension|[style]|[header]|[start];
```

A.3.3.1 Extension

The extension is required in order to identify the file type. It may be written

```
*.ext
.ext
ext
```

A.3.3.2 Style

The style file reference is optional, but the file will need to exist for the layer to be customised. If it is omitted, the Mapper assumes that the file name matches the extension, e.g. *ext.ttkstyle* for **.ext* files.

There are predefined configuration files using an enumeration number for the standard ADMS layers. You can reference these if you wish your added layers to look similar to one of these.

If a section omits a parameter, you must include the appropriate pipes if you wish to define a later parameter, e.g.

```
*.ext||index,X,Y,Z|2;
```

A.3.3.3 Header alias

The header alias is an optional parameter. If the file type always contains a header, you do not need to set it here. There is an assumption that the first non-blank line will be the header if you don't define the header alias.

Even if the file type does always contain a header, you can use the alias to

- Skip columns using a dash or space or nothing

  ```
  -,x,y,-,-,z
   ,x,y, , ,z
  ,x,y,,,z
  ```

- Redefine columns, e.g. set a column to Z and it will be used as elevation

  ```
  -,x,y,z,-,-
  ```

- Define polylines using X1, Y1 etc., as described in 5.5.3

A.3.3.4 Start line

The start line is optional, although there are times when it is useful. For example, if the file type always has a header but you have defined a header alias, set the start line to skip over the file header. With a header alias defined, the first non-blank line from the start line will be considered as data.

Example

```
KnownTypes=*.ex1|bespoke.ttkstyle|X,Y,Z;*.ex2;*.ex3||X,Y,Val|2
```

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