ADMS-STAR

Model for the analysis of
short-term accidental releases

ADMS-STAR screenshot showing the Source tab and ADMS Line Plotter window

User Guide CERC

ADMS-STAR

Model for the analysis
of short-term accidental releases

User Guide

Version 1.1

July 2016

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

Telephone: +44 (0)1223 357773
Facsimile: +44 (0)1223 357492
Email: help@cerc.co.uk
Website: www.cerc.co.uk

Contents

SECTION 1 INTRODUCTION TO ADMS-STAR

1.1 Introduction

ADMS-STAR is a computer code for the analysis of **S**hort-**T**erm **A**ccidental **R**eleases combining the **ADMS** description of the atmospheric boundary layer and spreading of releases with a Lagrangian puff approach for modelling explosive and short-term releases. ADMS-STAR can use spatially and temporally varying meteorological data and nested flow and turbulence fields from the FLOWSTAR model (CERC 2016) for the effects of local terrain and local variations in surface roughness.

ADMS-STAR has been developed specifically to estimate air activity/concentration and accumulated deposition for radiological or chemical emissions to the atmosphere from short-term accidental releases.

ADMS-STAR is capable of modelling two different release types: continuous releases, where the release occurs over a finite time period; and explosive releases, where the release is defined by an initial cloud, the properties of which are defined from the strength of the explosion.

ADMS-STAR can be used in two different ways:

- If details of the release are known, or can be estimated, then the user can enter the source details directly; or

- In an emergency situation the source information is often not well known and may need to be estimated. Samples of air and/or soil may be available, and these sample data may be input into ADMS-STAR, which will then estimate the source strength.

In both cases ADMS-STAR will calculate air activity/concentrations and accumulated deposition. These can be compared against regulatory or other levels (e.g. EU maximum permitted levels[1] in foodstuffs, MPLs).

ADMS-STAR may be linked to ESRI's ArcGIS (Geographical Information System) to display the source and sample locations along with contours of accumulated deposition or air activity/concentration, overlaid on a map of the area where the incident occurred. There is also a link for contour plotting of results using Golden Software's Surfer and a facility for line plotting variables of continuous releases.

---

[1] http://europa.eu/legislation_summaries/food_safety/contamination_environmental_factors/l21109_en.htm

1.2   About this User Guide

This *ADMS-STAR User Guide* provides both guidance in using the model and a technical summary.

Section 2 Provides information about model installation, system requirements and navigating around the ADMS-STAR interface.

Section 3 Gives an introduction to using ADMS-STAR.

Section 4 Explains the user-defined input parameters for each screen.

Section 5 Discusses the advanced modelling features of ADMS-STAR.

Section 6 Describes the ways in which results of ADMS-STAR model runs can be displayed.

Section 7 Describes four worked examples using ADMS-STAR.

Section 8 Summarises the technical details of the ADMS-STAR dispersion model.

Appendix A Provides instructions on how to use ADMS-STAR with ArcGIS.

Appendix B References.

1.2.1 Conventions

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

- ADMS-STAR interface controls are shown in **Arial** font, e.g. from the **Output** screen, click **Plot**.

- Keyboard keys are shown in **bold**, e.g. press **Enter**.

- Directory and file names are shown in *italics*, e.g. *adms-star.exe*, *<install_path>\Documents*.

- Tips and other notes are shown like this:

  CERC is located in Cambridge, a thriving University City.

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

SECTION 2   GETTING STARTED

2.1   System requirements

ADMS-STAR has been developed to operate on a PC running Microsoft Windows 7, Windows 8 or Windows 10. The PC should be a high specification PC in terms of processor speed and memory. The ADMS-STAR model cannot currently take advantage of multi-processor capabilities.

To install ADMS-STAR there must be a minimum of 200 MB of disk space available and a recommended 1 GB further for running the model and saving output files.

2.2   Installation

2.2.1   Installing ADMS-STAR

Please check with your own IT personnel for your organisation's procedures for installing software before following these steps.

Users should be logged on as Local Administrator for the PC to install ADMS-STAR.

Step 1   Log on as Local Administrator for the PC.

Step 2   Unzip the downloaded *.zip* file to a local directory. In Explorer, browse to this directory and double-click on the file '*setup.exe*'.

This will open the **Welcome** screen shown below in **Figure 2.1**.

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

Step 4   Enter your username and organisation in the designated places. Click **Next>** to go to the **Destination Folder** screen (**Figure 2.3**).
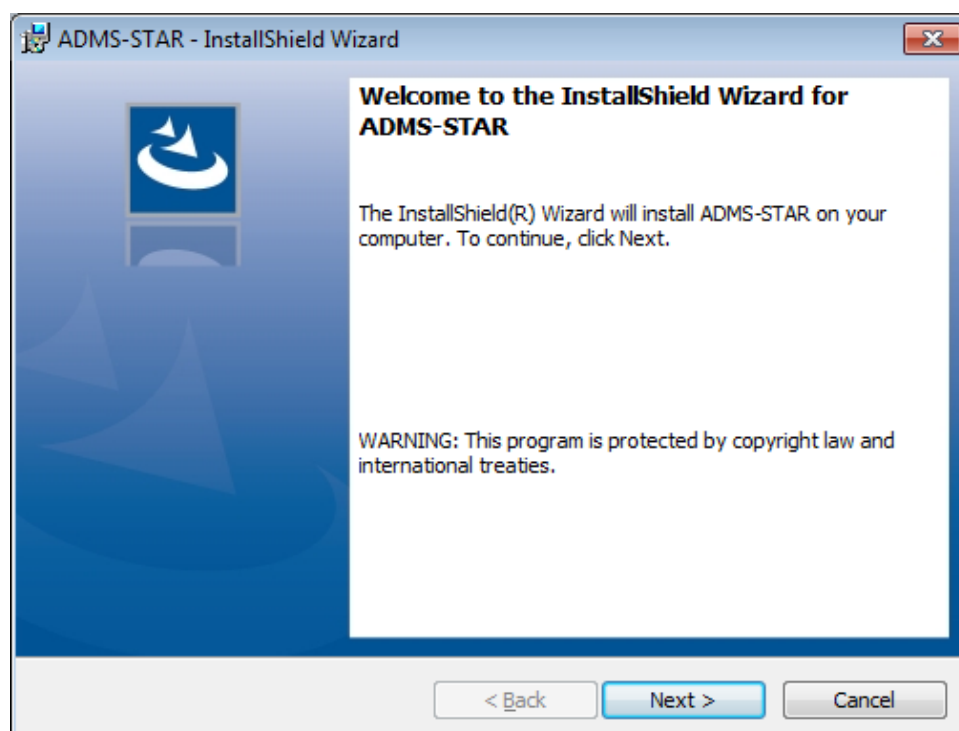
Figure 2.1 - The ADMS-STAR Installation **Welcome** screen
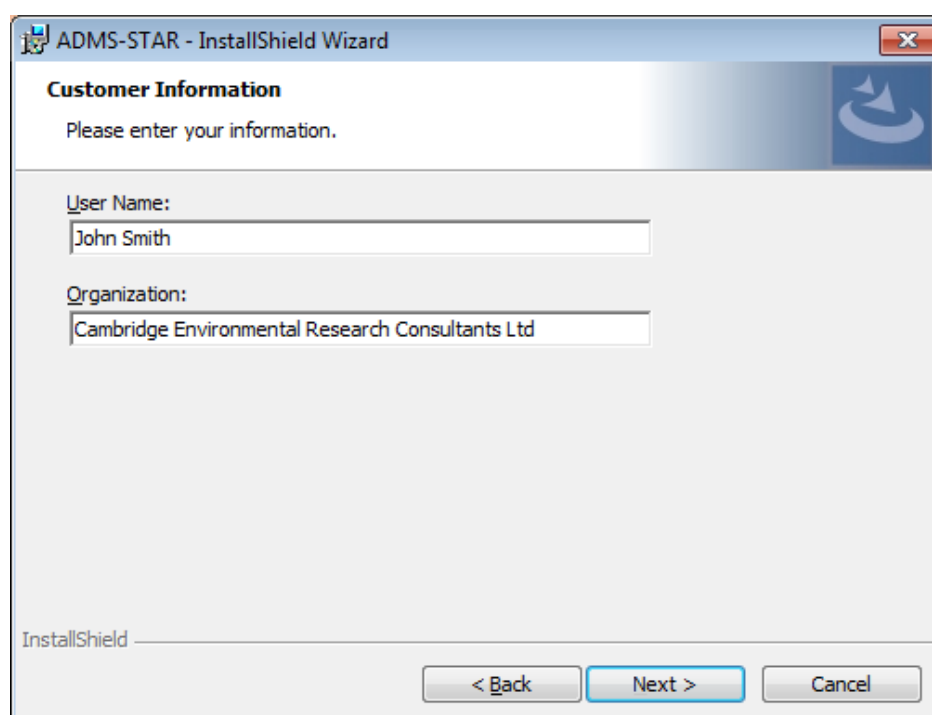
Figure 2.2 - The ADMS-STAR **Customer Information** screen

Step 5 You should select a drive with at least 200MB of available disk for the installation. The default installation directory is *C:\Program Files\CERC \ADMS-STAR*, as shown in **Figure 2.3**. If required, use **Change**... to select your own installation directory. Click **OK** to return to the **Destination Folder** screen.
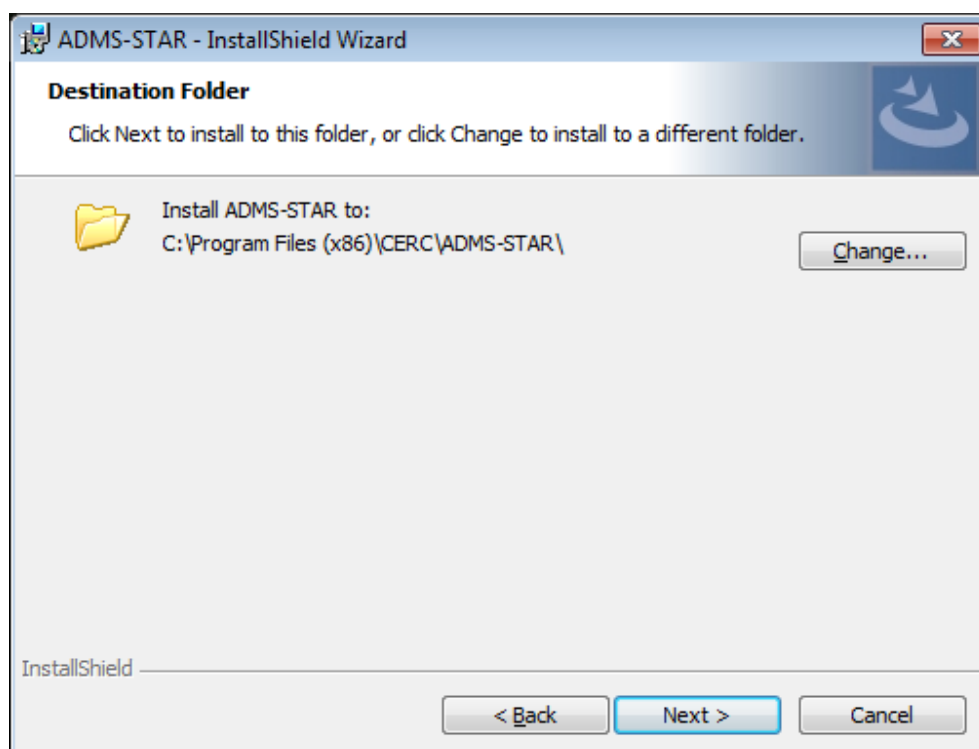
Figure 2.3 - The ADMS-STAR **Destination Folder** screen

Step 6 Choose whether to install for all users or just the current user.

If the local administrator profile is not the usual user profile, choose to install for all users.
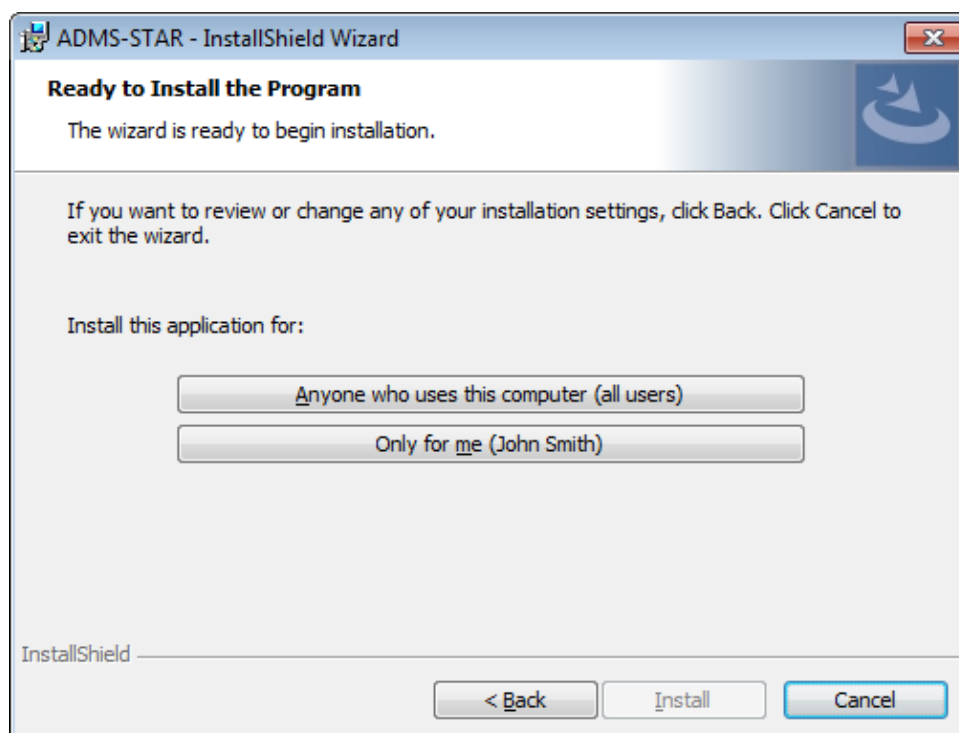
Figure 2.4 - The ADMS-STAR **Current Settings** screen

Step 7   Once the ADMS-STAR files have been successfully installed, the final screen will appear, as shown below in **Figure 2.5**.
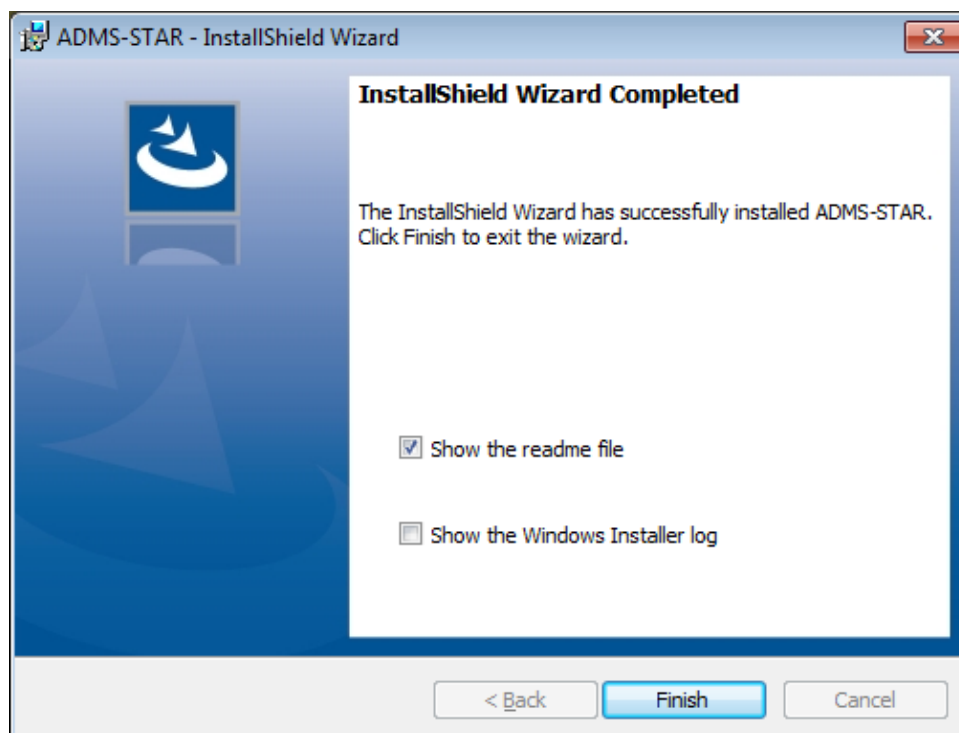
Figure 2.5 - The final screen of the installation.

Step 8   Click **Finish** to complete the installation.

The installation is now complete.

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

Step 9   To install the ADMS-STAR licence, copy the file *ADMSSTAR.lic* to the installation directory chosen in Step 5.

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

Step 10   Restart your computer.

Note that an application icon or shortcut to ADMS-STAR on the desktop is automatically created by the install program, for the users you selected in Step 6.

2.2.2 Installing the ADMS-STAR–ArcGIS extension

The ADMS-STAR–ArcGIS link is automatically installed as part of the ADMS-STAR installation. ArcGIS must, of course, be installed for the link to run!

2.2.3 Repairing ADMS-STAR

If any of the ADMS-STAR files are accidentally deleted from your computer, it is possible to **Repair** your installation of the model. To do this, log on as Local Administrator for the PC and unzip the downloaded ADMS-STAR installation *.zip* file to a local directory. Browse to this directory in Explorer and double click '*setup.exe*' to display the **Program Maintenance** screen shown in **Figure 2.6**.

Select **Repair** and click **Next>** to repair your installation.

The **Modify** option is intended for applications where the user can choose which elements will be installed. In ADMS-STAR there is no such choice and clicking **Modify** will have no effect.

The **Remove** option can be used to uninstall ADMS-STAR, or it can be uninstalled by following the instructions given in Section 2.2.4.
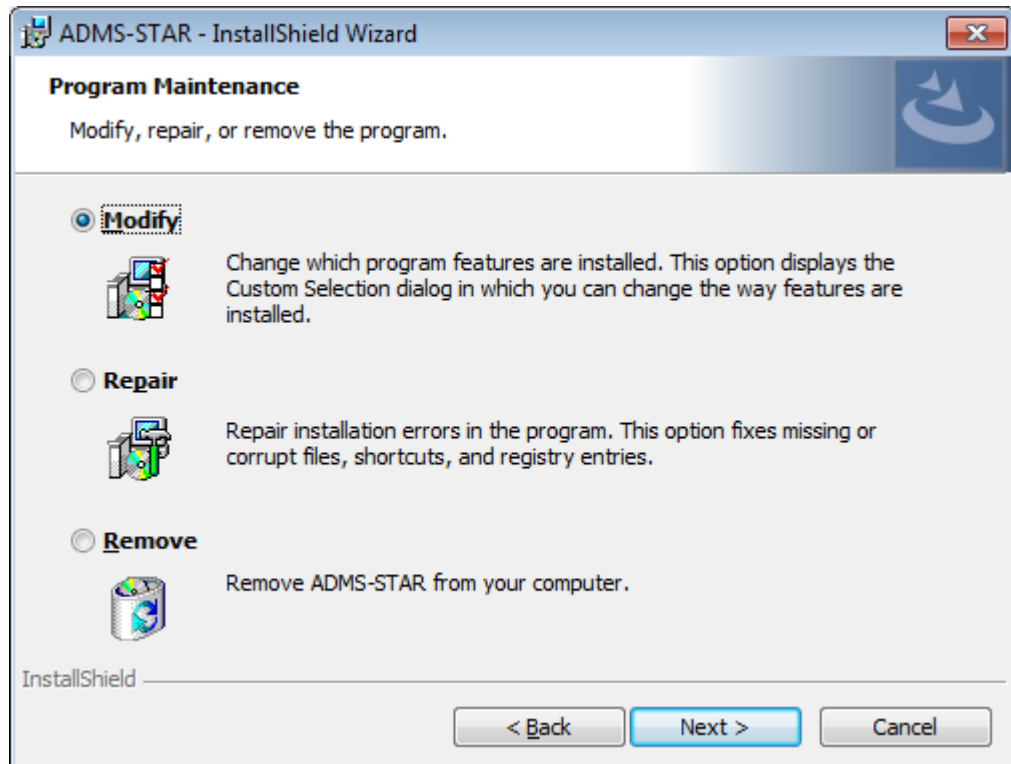
Figure 2.6 - The ADMS-STAR **Program Maintenance** screen

2.2.4 Uninstalling ADMS-STAR

To uninstall ADMS-STAR, log on as Local Administrator for the PC and click the Windows **START** button, and then click **Control Panel**. Click the **Uninstall program** option, select ADMS-STAR from the list and click **Uninstall**.

2.3 Navigating around the interface

2.3.1 Mouse buttons

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

2.3.2 Keyboard access

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

Moving the cursor between data entry boxes

TAB	Moves the cursor forward through data entry boxes or buttons
SHIFT + **TAB**	Moves the cursor backwards through data entry boxes and buttons
RETURN	'Enters' or accepts the current data page or executes the action of a highlighted button
SPACEBAR	Selects or deselects the highlighted option

Entering data in a box

DELETE	Deletes the character immediately to the right of the cursor
BACKSPACE	Deletes the character immediately to the left of the cursor
← arrow	Moves the cursor one space to the left in the current box
→ arrow	Moves the cursor one space to the right in the current box
SHIFT + arrow	Begins highlighting characters in the direction of the arrow (see above)

Highlighted text

DELETE	Deletes all highlighted characters
(Type)	Typing text replaces the highlighted text with new text

Radio buttons

← arrow	Moves the cursor up through the radio buttons for the current item
→ arrow	Moves the cursor down through the radio buttons for the current item

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

Access Keys

Also known as shortcut keys, these are combinations of keys that perform some of the main commands. For example, tabbed screens whose name contains one letter underlined will be accessible by typing **ALT** and the underlined letter together, e.g. the **Set̲up** screen may be accessed by typing **ALT** + **U**.

SECTION 3 USING ADMS-STAR

This section guides you around the ADMS-STAR interface and explains each menu option.

3.1 Starting ADMS-STAR

The ADMS-STAR interface can be launched in two ways: in "standalone" mode, or in conjunction with ArcGIS. APPENDIX A describes the use of ADMS-STAR with ArcGIS.

To start ADMS-STAR in standalone mode, double-click the ADMS-STAR icon on your desktop.

3.2 The ADMS-STAR screens

When ADMS-STAR is launched, you will see the ADMS-STAR model interface with a title bar across the top, reading "**ADMS-STAR 1 - (untitled)**". The word "untitled" refers to the current model run file name, which has not yet been defined (**Figure 3.1**).
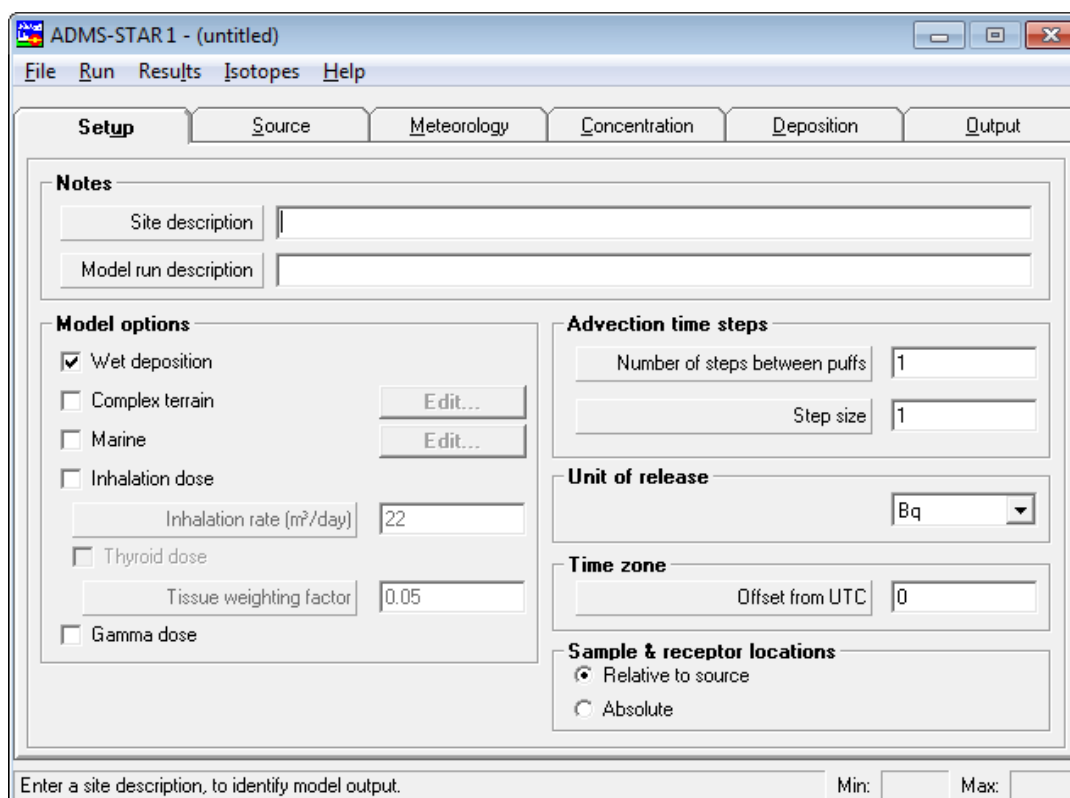
Figure 3.1 - The ADMS-STAR model interface

The ADMS-STAR model interface comprises six tabbed screens where you enter data. Clicking on the appropriate tab brings that screen to the front for viewing or editing. The six screens are as follows:

Setup   This is where model options such as complex terrain or inhalation dose can be selected. The parameters controlling the modelling of the dispersion can also be altered from this screen. This is also where the offset from UTC to local time is entered and where the choice is made whether the receptor and sample locations will be specified in coordinates relative to the source or in absolute coordinates defined on a local Cartesian grid. You can also add notes to identify the model file and model output.

Source   This is where you specify whether to use a continuous or explosive release and enter the parameters describing the release. These can include the source location, source height, start and end time of the release, cloud top height. You state here whether you would like to enter source data directly or would like the model to estimate source strengths.

Meteorology   This is where you enter parameters describing the meteorological conditions.

Concentration   This is where you specify the details of air activity/concentration samples that are available.

Deposition   This is where you specify the details of ground deposition samples that are available.

Output   This is where you select the options for the model output. This includes the model output time, size of output grid, location of specified output points and contour options, for instance MPL or ERL contours.

3.2.1 Entering information

When the ADMS-STAR model interface is started or when you select the **New** option from the **File** menu, a new model file, or scenario, is created and default values are loaded into the screens for you to edit. Alternatively, you may wish to load an existing model file and edit those values.

Each screen may contain:

- *text boxes* for you to enter a numerical value or text string

- *radio* buttons to indicate which option has been selected

- *buttons* such as **OK**, **Cancel**, to accept or reject the values in the screen, or buttons to take you to a further sub-screen

- a *help bar* containing useful tips about completing the screen (see below)

- *tables* of user-supplied data e.g. locations of output points. Information is added to or removed from the tables using the **Add** and **Delete** buttons next to the tables.

Throughout the ADMS-STAR interface, values or options that are greyed out are not used in the current model calculation.

3.2.2 The help bar

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

For example, in the "**Output**" screen when the **Number of grid lines** box is selected, the help bar will display the line:

```
Number of lines to use in x and y. Min: 3 Max: 101.
```

3.3 The ADMS-STAR menus

The menu bar has five menus: **File**, **Run**, **Results**, **Isotopes** and **Help**. Some of the menu headings have drop-down lists of options. The menus have the following options and roles.

File			
	New		Reset the data parameters to their default values
	Open...		Open a previously saved model file
	Save		Save the current parameters under the current file name
	Save As...		Save the current parameters and prompts you for a new file name
	Verify		Show advice on the current time step size and puff release rate and estimates the run time
	Run		Run the model using the current parameter file (equivalent to the **Run** main menu item)
	Preferences	**Model Execution**	Access options for the run time window state and exit mode
	Preferences	**Viewing Output**	Choose the application for viewing numerical results files in ADMS-STAR (the default application is Microsoft Excel if it is installed)
	Exit		Quit ADMS-STAR
	1, ...		Show the names of the most recently opened parameter files in ADMS-STAR (click on the desired file name to open that file in ADMS-STAR)
Run			Run the model using the current parameter file
Results	**Output**		Open output files generated by ADMS-STAR
	X-Y Plot		Launch the line-plotting facility
	Contour plot in Surfer		Launch the contour plotting facility to plot contours using Surfer (if installed)
	Automate Surfer Contour Plots		Launch an Excel template containing macros for the automation of contour plotting in Surfer[2]

---

[2] Please refer to the Surfer Automation and Slideshow Creator Tools User Guide for more details

	Make Slideshow	Launch an Excel template containing macros for creating a slideshow[2]
Isotopes	**Palette**	Open the isotope palette
Help	**User Guide**	Opens the ADMS-STAR User Guide
	What's New	Opens the What's New in ADMS-STAR 1.1 document
	Map	Display a map of the coordinate system selected for describing the source location
	Met. Conditions	Display a list of combinations of parameters describing a variety of meteorological conditions
	MPL Data	Display the maximum permitted levels (MPLs) used by ADMS-STAR for each standard isotope and foodstuff combination
	Email CERC	Auto-address a new email to the ADMS-STAR helpdesk in the user's default email client
	CERC website	Open the CERC homepage in the user's default internet browser
	Licence Details	Show details of the current licence
	About ADMS-STAR	Show the version number of your copy of ADMS-STAR

3.4 Creating a model file and running the model

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

- create a new model file or open an existing model file;
- enter or edit data to define the problem;
- save the model file;
- verify the model file;
- run the model; and
- display the output.

3.4.1 Opening an existing ADMS-STAR model file

To open an existing model file, choose **Open** from the **File** menu. By default, ADMS-STAR will display only files ending in *.hpl*. Alternatively, the names of the model

files most recently opened in ADMS-STAR will appear in the **File** menu. One of these can be opened by clicking on the required file name.

The first time either the "Open", "Save" or "Save As" dialogue boxes appear, the default drive and directory will be the directory where ADMS-STAR is installed. If the directory is changed during the opening or saving of a file, this new directory will become the default, until it is changed again or you exit from the ADMS-STAR interface.

3.4.2 Changing values in the input screens

If you enter a value outside the permitted range for a parameter, ADMS-STAR will display a warning message box that tells you the appropriate limits. The message may be displayed either when you move to another input box or when you save the file. A similar message appears if you forget to enter a value and leave a cell blank.

Note: Usually a blank cell does not denote a value of zero.

3.4.3 Saving the model file

Once you have entered data but before you verify or run the model, you must save the model file as

\\PATHNAME\filename.hpl

User files may be kept in any (sub-)directory with any file name, but must have the extension *.hpl*.

If you are working with an existing file and do not wish to change the file name then select **Save** from the **File** menu. The file will be saved with its original name, overwriting the previously saved file.

If you started with the default information (the file "untitled") and you are saving the input data for the first time then use **Save As**... from the **File** menu and select a suitable directory and file name with which to save the model file. You should also use **File**, **Save As**... if you are working with an existing file but do not wish to overwrite the original model file.

3.4.4 Verifying the model file

Once you have saved a model file, an option exists to verify its contents. This produces a *.log* file (in the same directory as the *.hpl* file) which gives advice on the current time step size and puff release rate, and an estimate of the run time. It also carries out all of the model data checks. The model will carry on past as many errors as it can so as to produce as comprehensive a list of errors as possible.

To verify the model file, select **Verify** from the **File** menu. If you have changed anything in the interface since the model file was last saved, the **Verify** option will appear disabled. As soon as you re-save the model file, the **Verify** option will again become

---

available.

3.4.5 Running ADMS-STAR

When you have verified the model file, you can go to the menu bar and click **Run** or **File**, **Run** to run the program.

When you run the model, if you have changed anything in the interface since the scenario was last saved then you will be prompted with:

```
Save current data?
```

Click **Yes** to continue, or **Cancel** to abort the save and allow the file to be saved under an alternative name.

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

Runtime preferences

The user can edit the runtime options from the **File**, **Preferences**, **Model Execution** menu. This will display the **Runtime Preferences** dialogue box shown in **Figure 3.2**, divided into two sections:

The **Window State Options** control the size and state of the run window: normal, minimized or maximised, with focus (active window) or without focus.

The **Exit Mode Options** determine if the run window closes after the run has completed. If **Normal termination box** is selected, a dialogue box appears at the end of the run asking whether you want the window to close. If **No termination box, window open** is selected, no dialogue box is displayed and the progress window is kept open at the end of the run. If **No termination box, window closes** is selected, no dialogue box is displayed and the progress window is closed at the end of the run.
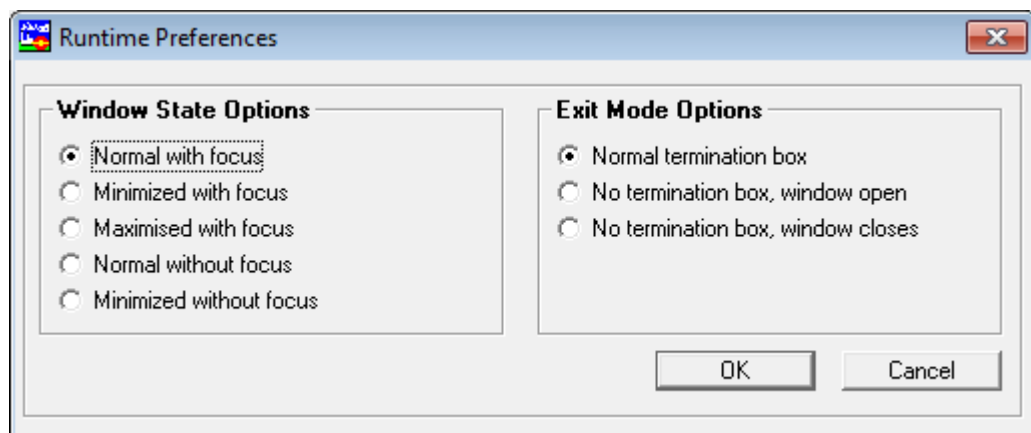

Figure 3.2 – The **Runtime Preferences** screen

SECTION 4 DATA INPUT

4.1 Input screens

When ADMS-STAR is started, the **Setup** screen shown in **Figure 4.1** appears. "Untitled" in the menu bar indicates that this is a new file and contains default data.

You may now enter the parameters in each of the screens that are required to run the model. Parameter values may only be entered in boxes that are active, that is, where the title appears in black. If a box is not active, the title appears in grey and the model does not use the values within it. Selecting certain options causes other options to be switched off, shown by the text becoming grey.

There are six data input screens: **Setup**, **Source**, **Meteorology**, **Concentration**, **Deposition** and **Output**. Move between them by clicking the tabs along the top of each screen.

Although data input to these screens are not completely independent of each other, they may be entered in any sequence. However, it is advisable to complete the input screens from left to right (**Setup** to **Output**), to ensure that all parameters are defined for each model run.

4.2 Time in ADMS-STAR

The ADMS-STAR interface, output and user guide refers to "time" quite frequently. This section explains what the values of time are used for and what you need to remember to make sure you use the correct data.

- ADMS-STAR needs to know when sunrise and sunset occur. The *local solar time* required for this is calculated by ADMS-STAR from the value of longitude you enter in the **Define location** sub-screen of the **Source** screen, so be sure to enter the correct longitude for the release location.

- If you are using spatially varying meteorology, the times given in the WRF files are in UTC (Coordinated Universal Time) which is the same as GMT (Greenwich Mean Time).

- The times you enter for the source release, spatially homogenous met data, specifying the WRF files, observed met data at the source location, air concentration and ground deposition samples, and modelling end time must all be in the same time reference, which in the model interface is referred to as *local time*. Your local time could be anything you wish but it is likely to be the time given on clocks and watches at the release site. All model output will also be given in this local time.

- You must know the difference between your local time and UTC. The difference is entered in the **Time zone** section of the **Setup** screen. For locations east of 0° longitude the offset is likely to be positive. For locations west of 0° longitude the offset is likely to be negative.

4.3   Setup screen

The **Setup** screen is shown in **Figure 4.1** and includes sections for **Notes**, **Model options**, **Unit of release**, **Advection time steps**, **Time zone** and **Sample & receptor locations**.

Figure 4.1 - The **Setup** screen shown with the Complex terrain option selected

Notes

In the **Notes** text boxes you have the option to enter information about your site and the model run. These notes are saved with the model file and written to output files, to identify output with a particular model set-up. Each note may be up to 256 characters long.

Model options

This section of the **Setup** screen allows certain model options to be switched on and off. Specifically:

- **Wet Deposition** can be switched off.   If **Wet Deposition** is deselected, then the washout coefficient for all isotopes is set to 0.

- Either the **Complex terrain** or **Marine** options can be selected.   They cannot both be selected together and the **Marine** option is not available when using spatially varying met data. After selecting the option required, click **Edit**... to enter the parameters for this option.   More details on these model options are given in Sections 4.3.1 and 4.3.2 respectively.

- **Inhalation dose** due to any isotope and **Thyroid dose** due to iodine can be selected. More details on these model options are given in Section 4.3.3.

- **Gamma dose** can be selected, more details on this option are given in Section 4.3.4.

Unit of release

Select the unit of the release as **Bq** or **g**. This option changes the units displayed in the ADMS-STAR interface and in the output files generated. Changing this option does not transform the data entered elsewhere in the interface. If **g** is selected radioactive decay calculations will still be carried out by assuming that the concentration levels represent an activity.

Advection time steps

Enter the size of the advection time step and, for continuous releases, the number of time steps between puff releases. The **Verify** option from the **File** menu can be used to give advice on the choice of advection time step, the number of time steps between puff releases and the estimated run time with the selected parameters.

Time zone

Enter the number of hours offset from UTC to local time. Refer to Section 4.2 for a description of the meaning of 'local time' in ADMS-STAR.

Sample & receptor locations

Select whether to specify the sample and receptor locations in coordinates relative to the source or in absolute coordinates defined on a local Cartesian grid, e.g. UK Ordinance Survey National Grid. This must be the same local coordinate system used to specify the source location in the **Define location** sub-screen of the **Source** screen (see Section 4.4). Gridded model output is always given in absolute coordinates, whereas specified point output is given in both coordinates relative to the source and in absolute coordinates.

4.3.1 Complex terrain option

Selecting the **Complex terrain** option means that the FLOWSTAR model, an integral part of ADMS-STAR, will be run to calculate the wind speed and turbulence due to the terrain elevation entered via the *.ter* file and/or the spatially varying surface roughness entered via the *.ruf* file. It is possible to use a *.ter* or *.ruf* file or both.

When to use the Complex terrain option

The **Complex terrain** option should be used if the modelling region contains an area of interest, typically around the source, with complex topographical and/or surface

features that are likely to significantly modify the local flow field from the upstream condition (if using homogeneous met data) or that are unlikely to be well resolved by the coarser 3D mesoscale flow field grid (if using spatially varying met data). The *.ter* and *.ruf* files can contain data for a domain smaller than the output domain, thus allowing the FLOWSTAR solution to be nested within the homogeneous solution or the 3D mesoscale model solution.

How to use the Complex terrain option

To calculate the effects of complex terrain, select the **Complex terrain** option from the **Setup** screen and then click **Edit...** to display the **Complex terrain** options screen, **Figure 4.2**.
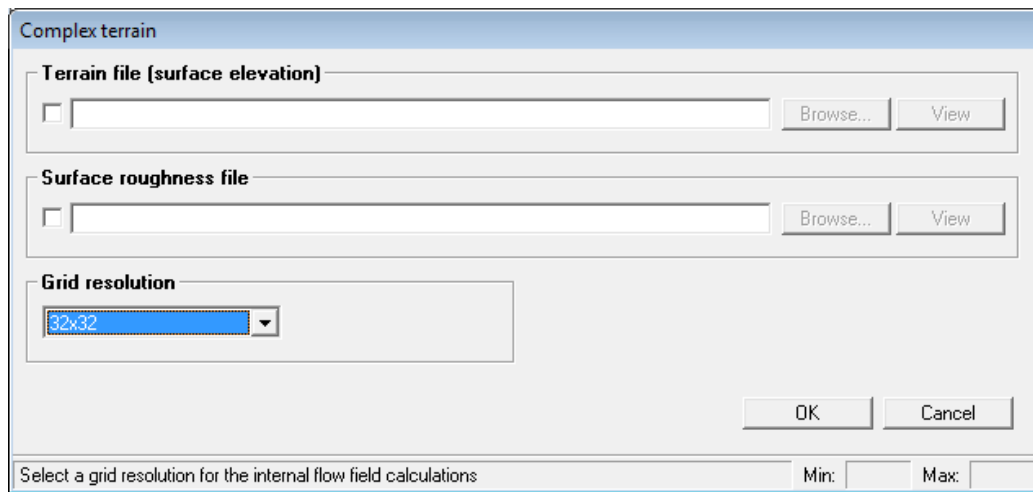
Figure 4.2 - The **Complex terrain** screen shown with the default options

Terrain File (surface elevation)

If a file of variable terrain height is to be entered select the check box and enter the file name of the variable terrain file *(.ter)* to be used.

Surface roughness file

If a file of spatially varying roughness is to be entered select the check box and enter the file name of the spatially varying roughness file *(.ruf)* to be used.

Grid resolution

The internal grid resolution should be selected from **16x16**, **32x32**, **64x64**, **128x128** or **256x256.**

The lowest grid resolution (**16x16**) is very coarse and should only be used for testing model runs.

Format of the terrain and surface roughness files

Both the variable terrain (*.ter*) file and variable surface roughness file (*.ruf*) consist of a series of lines of data in the format:

```
n, x, y, z
```

where `n` is an incrementing counter for each line, `x` is the east-west location of the point (in meters) in absolute coordinates (i.e. on a local Cartesian grid), `y` is the north-south location of the point (in meters) in absolute coordinates and `z` is the height of the terrain in meters (for a variable terrain file) or the surface roughness value in meters (for a variable surface roughness file) at that (x,y) location.

The maximum number of data points that may be included in a .ter/.ruf file is 66000.

The data points do not need to be regularly spaced, but the area described should be rectangular. The surface roughness file should cover the same area as the terrain file. An easy way to create a surface roughness file is to edit the `z` values of the terrain file being used so that the same domain, with the same `x` and `y` coordinates, is used for both files. This can be carried out using a spreadsheet package such as Excel.

Two example files, *terrain.ter* and *roughness.ruf*, describing the terrain and surface roughness of a bell-shaped hill are supplied in *<install_path>\Data*.

4.3.2 Marine option

ADMS-STAR includes a marine boundary layer scheme for calculating surface roughness and heat fluxes over the sea.

When to use the Marine option

The **Marine** option could be used, for example, for dispersion modelling of releases on oil extraction platforms. To use this option the source and all receptors should be over the sea; it is not suitable for coastal modelling.

How to use the Marine option

To use the Marine boundary layer option, select the **Marine** option on the **Setup** screen and then click **Edit...** to display the Marine boundary layer screen, shown in **Figure 4.3**.

Figure 4.3 - The **Marine boundary layer** screen with the default data

On the **Marine boundary layer** screen the **Height of temperature measurements** above sea level and the **Charnock parameter** must be entered. The **Charnock parameter** is a constant used in the heat flux calculations, with typical values ranging from 0.018 to 0.08, refer to Section 8.9 for more details.

As well as entering the data on the **Marine boundary layer** screen, if the marine option is

---

selected the **Sea surface temperature (ºC)** for each met period should be entered in the **Homogeneous met. data** screen. If sea surface temperatures are not known then the **Marine boundary layer assumed to be neutral** option should be checked on the **Marine boundary layer** screen.

4.3.3 Inhalation dose and Thyroid dose

The **Inhalation dose** option allows the inhalation dose due to the emitted isotopes to be calculated. If the **Inhalation dose** option is selected then additional input parameters are required:

- the **Inhalation rate (m3/day)** which is entered on the **Setup** screen, a value of 22 m^3/day which is suitable for adults is used by default; and

- the **Inhalation dose coefficient** (Sv/Bq) for each isotope which is entered in the isotope palette, refer to Section 4.9. Default values for all isotopes, except the user defined isotope, are supplied. These values are suitable for adults, refer to Section 8.10 for more details.

Inhalation dose output is calculated for receptor points (in the *.rec* file) and for gridded output (in the *.glt* file). In both cases the inhalation dose is given per isotope and as a total over all isotopes.

If **Inhalation dose** is selected the option to model **Thyroid dose** due to iodine also becomes available. With this option the **Tissue weighting factor** also needs to be entered, by default a value of 0.05 is used. Like the **Inhalation dose** the **Thyroid dose** output is also given for receptor points (in the *.rec* file) and for gridded output (in the *.glt* file) and is given per iodine isotope and as a total over all iodine isotopes.

In addition to the inhalation dose and thyroid dose output, when these options are used, it is also possible to output Emergency Response Level contours, refer to Section 4.8 for more details.

4.3.4 Gamma dose

Selecting the **Gamma dose** option allows the gamma dose due to concentration and deposition to be calculated up to the modelling end time. The gamma dose can be calculated at up to 5 receptor points, the choice of receptor points is made on the **Output** screen, refer to Section 4.8 for more details on selecting the output points. The gamma dose for each of the selected receptor points is written out to the *.rec* file. If gamma dose output is required for the user-defined isotope the energy levels and branching ratios need to be entered for the user-defined isotope, refer to Section 4.9 for more details.

4.4 Source screen

The **Source** screen contains the source details. A single source can be included in each model run. ADMS-STAR includes default values for all input parameters on the **Source** screen; these are shown in **Figure 4.4** and **Figure 4.8**. Edit these default values as required. Two types of source can be modelled in ADMS-STAR: a **Continuous release**, described in Section 4.4.1; or an **Explosive release**, described in Section 4.4.2. The release type is selected from the radio buttons at the top of the **Source** screen.

4.4.1 Continuous releases

Figure 4.4 - The **Source** screen showing default input data for a continuous release.

Source Details

The **Source location** is entered by clicking ✏ to display the **Define location** screen (**Figure 4.5**). The following information must then be entered:

- **Longitude (decimal)** and **Latitude (decimal)**: Longitude and latitude of the source location entered as decimal degrees.

- **X datum (km)** and **Y datum (km):** East-west and north-south location of the source in absolute coordinates defined on a local Cartesian grid, e.g. UK Ordinance Survey National Grid, entered in kilometres. East and north are positive.

  Gridded model output is always given in absolute coordinates. If you would prefer all output to be in coordinates relative to the source, this can be achieved by setting the X, Y location of the source to (0,0).

The **Define location** screen shows the approximate location of the source on a world map using the latitude and longitude provided. The ![globe button] button in the lower left corner of the map can be used to display the current source location in Google Maps[3] using your default internet browser. Note that if invalid location data are entered, they are displayed in red.
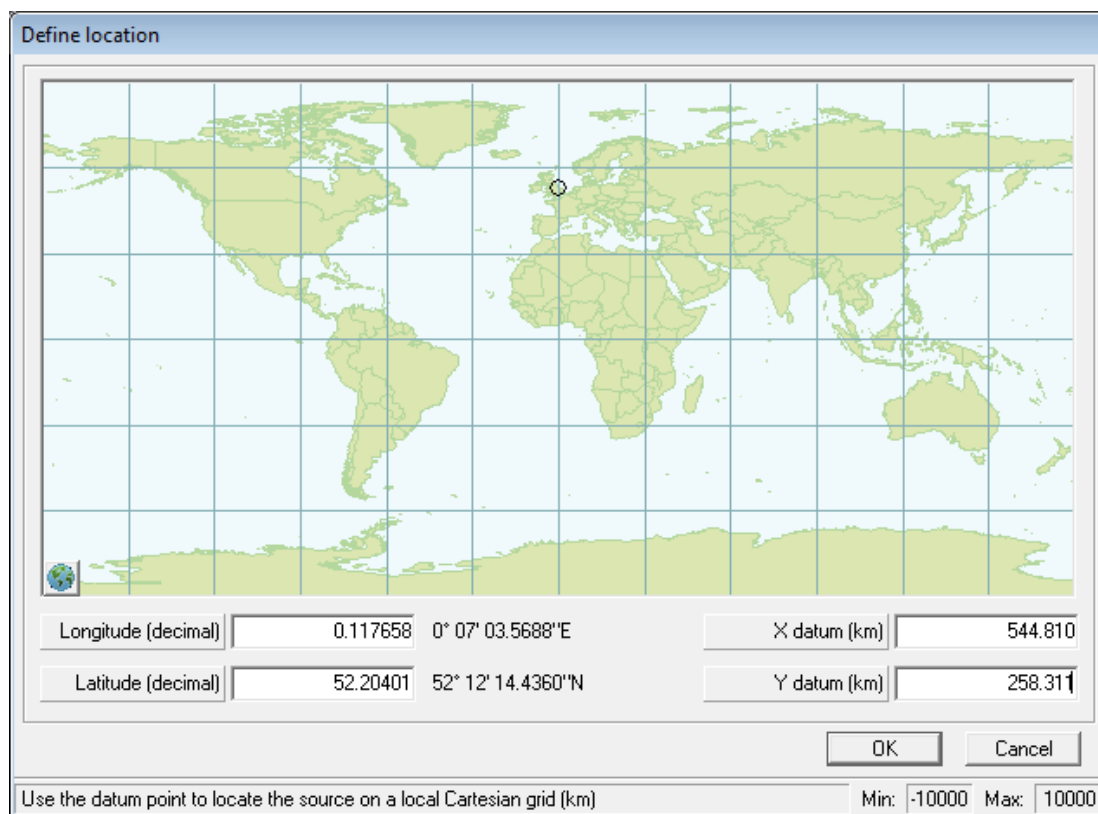

Figure 4.5 - The **Define location** screen

The **Release start time** and **Release start date**, and the **Release end time** and **Release end date** are the time/date at which the release started and the time/date at which the release ended. Values must be entered for these parameters: the default start values are the time and date at which the *.hpl* file is opened, while the default end values correspond to 4 hours later. The date should be in the form day/month/year, e.g. 30/01/2001, or in the form 30 Jan 2001 (i.e. 3-letter abbreviation for month name). The time should be entered in local time (see Section 4.2) in the form hours:minutes.

It is worth checking that the date of the release is shown correctly. For example, if the actual date is 2nd January 2001 then the default values for the Release start date and the Release end date should be 2 Jan 2001. If the date is shown incorrectly e.g. as 1 Feb 2001, then change the regional settings on the computer to English (United Kingdom).

The **Height** is the source height. This should be in metres above the local terrain.

The **Diameter** is the source diameter. Click the check box and enter a value of the diameter in metres. In the box is unchecked a default of 1 m is assumed.

---

[3] http://maps.google.co.uk/maps

The **Temperature** is the temperature of the release in °C.

The **Exit Velocity** can either be specified in meters per second or calculated from the plume top height. If the exit velocity is set to be calculated from the plume top height then the **Plume top height** must be entered in meters.

The exit velocity can only be calculated from the plume top height if spatially homogeneous meteorological data are used. Only the first meteorological condition will be used to calculate exit velocity from the plume top height.

Time-varying source data

Time-varying emissions factors can be entered by selecting the **Enter time-varying emissions** check box and then clicking **Edit**.... This will display the **Time-varying emission factors** screen shown in **Figure 4.6**.
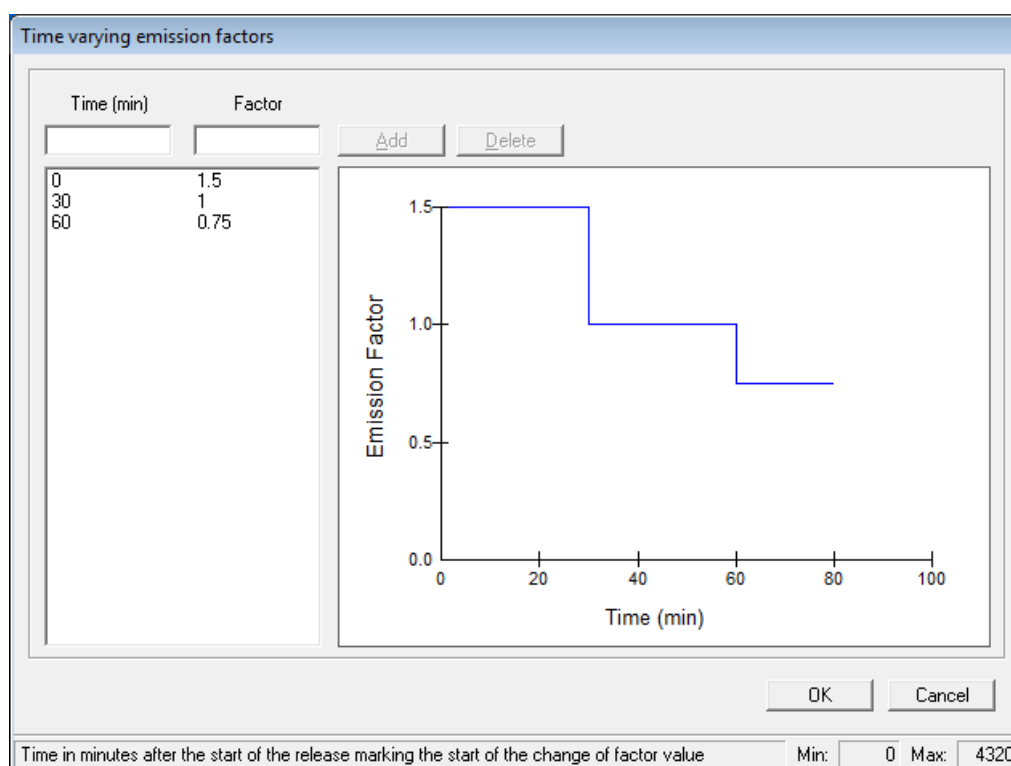
Figure 4.6 - The **time varying emission factors** screen with three sets of emissions factors entered

The emission factors are entered as a sequence of up to 60 pairs of time since release in minutes, and emissions factor for that time. The factor applies to all isotopes and applies from the time given until the time of the next emission factor. If no emission factor is given for the start of the release a default value of 1 is used.

New factors are added by entering the data in the **Time (min)** and **Factor** boxes and clicking **Add**. The **Delete** button can be used to remove unwanted emission factors. The graph on the right shows how the emissions factors will be applied and automatically updates as emissions factors are added or removed.

Source strengths

ADMS-STAR models a single source but the source may emit multiple species. The user has two options for defining the source strength; "**The model will estimate source strengths",** or "**Specify some source strengths**".

If the quantities of material being released are not known but there are data available from air activity/concentration or soil spectra samples, then you should use the option "**The model will estimate source strengths**". The sample details must be entered into the **Concentration** and/or **Deposition** screens as appropriate (refer to Sections 4.6 and 4.7). Ground deposition samples supersede air activity/concentration measurements, as the deposition measurements are likely to be more reliable, representing the impact of the release over a longer period than the air measurements.

However, if sample data are not available or if the source strength is known then you should use the option "**Specify some source strengths**". When this option is selected, you will not be able to edit the values in the **Concentration** and **Deposition** screens, because they are not used by the model. Click **Source isotopes** to display the "**Source isotopes**" input screen and enter the appropriate data for the isotopes/chemicals being released (**Figure 4.7**).

Figure 4.7 - The "**Source isotopes**" sub-screen showing default input data.

Select the isotope name from the drop-down list of isotopes (33 standard isotopes and one user-defined). Specify the release rate (**Strength**) in *Bq/s* (or *g/s* depending on the choice of release unit).

*To view the list of standard isotopes, their half-lives and standard deposition velocities, select the **Palette** from the **Isotopes** menu. The **Palette** also allows for the deposition properties of each isotope and the half-life of the use- defined isotope to be altered.*

Select **Add** to include the isotope details you have just entered; the details will be transferred to the large table below.

Repeat until you have a list of all isotopes being released. To remove an isotope from the list, select it in the table then click **Delete**.

Click **OK** to return to the Source screen.

4.4.2 Explosive releases

Figure 4.8 - The **Source** screen showing default input data for an explosive release.

Source Details

The **Source location** is entered by clicking ✏ to display the **Define Location** screen (**Figure 4.9**). It is then necessary to enter the following information:

- **Longitude (decimal)** and **Latitude (decimal)**: Longitude and latitude of the source location entered as decimal degrees.

- **X datum (km)** and **Y datum (km):** East-west and north-south location of the source in absolute coordinates defined on a local Cartesian grid, e.g. UK Ordinance Survey National Grid, entered in kilometres. East and north are positive.

> *Gridded model output is always given in absolute coordinates. If you would prefer all output to be in coordinates relative to the source, this can be achieved by setting the X, Y location of the source to (0,0).*

The **Define location** screen shows the approximate location of the source on a world map using the latitude and longitude provided. The [globe] button in the lower left corner of the map can be used to display the current source location in Google Maps[3] using your default internet browser. Note that if invalid location data are entered, they are displayed in red.

Figure 4.9 - The **Define location** screen

The **Release start time** and **Release start date** are the time/date at which the release occurred. Values must be entered for these parameters: the default values are the time and date at which the *.hpl* file is opened. The date should be in the form day/month/year, e.g. 30/01/2001, or in the form 30 Jan 2001 (i.e. 3-letter abbreviation for month name). The time should be entered in local time (see Section 4.2) in the form hours:minutes.

> *It is worth checking that the date of the release is shown correctly. For example, if the actual date is 2nd January 2001 then the default value for the Release start date should be 2 Jan 2001. If the date is shown incorrectly e.g. as 1 Feb 2001, then change the regional settings on the computer to English (United Kingdom).*

The cloud top height must be entered. This can either be specified directly in meters or set to be calculated from the mass of explosive (TNT equivalent) in kilograms.

Cloud properties

An explosive release is split by ADMS-STAR into five clouds, see **Figure 4.10**. The height of the top of each cloud as a fraction of the cloud top height and the percentage of material in each of these clouds should be specified. The top of the highest cloud is forced to be at the **Cloud top height**. The percentage of mass in the top cloud is set automatically to ensure the total is 100%. The **Sort** button can be used to ensure the properties are listed in descending height order.
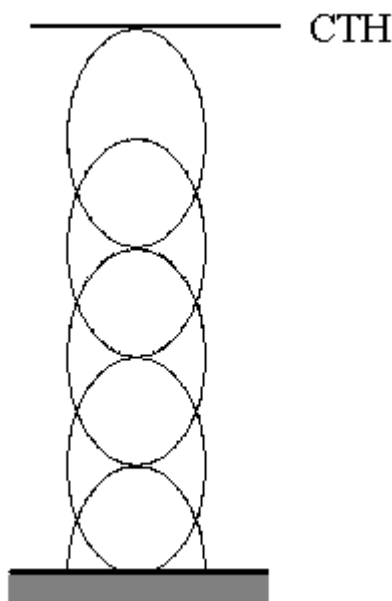

Figure 4.10 - The split of clouds for an explosive release

Source strengths

ADMS-STAR models a single source but the source may emit multiple species. The user has two options for defining the source strength; "**The model will estimate source strengths**", or "**Specify some source strengths**".

If the quantities of material being released are not known but there are data available from air activity/concentration or soil spectra samples, then you should use the option "**The model will estimate source strengths**". The sample details must be entered into the **Concentration** and/or **Deposition** screens as appropriate (refer to Sections 4.6 and 4.7). Ground deposition samples supersede air activity/concentration measurements, as the deposition measurements are likely to be more reliable, representing the impact of the release over a longer period than the air measurements.

However, if sample data are not available or if the source strength is known then you should use the option "**Specify some source strengths**". When this option is selected, you will not be able to edit the values in the **Concentration** and **Deposition** screens, because they are not used by the model. Click **Source isotopes** to display the "**Source isotopes**" input screen and enter the appropriate data for the isotopes/chemicals being released (**Figure 4.11**).

Figure 4.11 - The "**Source Isotopes**" sub-screen showing default input data.

Select the isotope name from the drop-down list of isotopes (33 standard isotopes and one user-defined). Specify the release rate (**Strength**) in Bq (or g depending on the choice of release unit).

*To view the list of standard isotopes, their half-lives and standard deposition velocities, select the **Palette** from the **Isotopes** menu. The **Palette** also allows for the deposition properties of each isotope and the half-life of the user-defined isotope to be altered.*

Select **Add** to include the isotope details you have just entered; the details will be transferred to the large table below.

Repeat until you have a list of all isotopes being released. To remove an isotope from the list, select it in the table then click **Delete**.

Click **OK** to return to the **Source** screen.

4.5 Meteorology screen

The **Meteorology** screen contains the details of the meteorological conditions for the incident (**Figure 4.12**). This screen gives the site data as well as providing the choice of what type of meteorological data are to be used and access to the sub-screens to define the meteorological data.

Figure 4.12 - The **Meteorology** screen showing default input data.

Site data

The Site data box allows for details of both the dispersion and meteorological measurement site to be entered.

In the **Dispersion site** box, enter the **Surface Roughness (m)** at the dispersion site (in meters). Either enter a value directly into the box or use the list to select a value based on land use.

The **Met. measurement site** box allows the surface roughness and other parameters of the meteorological measurement site to be entered. For the meteorological measurement site there is the option to use the dispersion site values.

For both the dispersion and met measurement site, **Advanced Meteorology** can be selected to allow the user to alter values of surface albedo, Priestley-Taylor parameter and minimum Monin-Obukhov length. In the **Advanced dispersion site data** screen you can also enter a factor to account for differences in precipitation between the dispersion and meteorological measurement site. Details of how to use these options are given in Section 5.1.

If spatially varying meteorology is entered, the dispersion site is assumed to be representative of the meteorological site for all parameters.

Met output

To generate a file of meteorological output at each receptor point, check the **Create met. output file** option. The format of the meteorological output file (**.mop*) created is described in more detail in Section 6.2.6.

Met data

Choose whether **Spatially homogeneous met. data** or **Spatially varying met. data from files** should be used by selecting the appropriate option and then clicking **Edit**... to enter the parameters. If spatially homogeneous met data is selected there is the option to specify a boundary layer profile file and if spatially varying meteorology is selected there is the option to enter observed meteorological data at the source. Details of these options are given below.

4.5.1 Spatially homogeneous meteorological data

To enter spatially homogeneous meteorological data select **Spatially homogeneous met. data** on the **Meteorology** screen and then click **Edit**... to display the **Homogeneous met. data** screen, **Figure 4.13**.

Figure 4.13 - The **Homogeneous met data** screen showing all possible columns, with one line of default data added

This screen allows up to 72 sets of meteorological conditions to be entered. New lines of data are added by clicking **Add** and unwanted lines of data can be removed by first

selecting them and then clicking **Delete**. The data can be sorted into chronological order using the **Sort** button.

The following data should be entered.

The **Local date and time** of the measurement. The date should either be entered in the form day/month/year, e.g. 30/01/2001, or in the form 30 Jan 2001 (i.e. 3-letter abbreviation for month name). The time should be entered in local time (see Section 4.2) in the form hours:minutes. The minimum time between meteorology measurement is 30 minutes.

The **Wind speed (m/s)** is the wind speed in m/s. The height (m) at which the wind speed measurement has been taken should be entered in the **Height of wind measurements** box.

The **Wind direction (deg)** is the direction from which the wind is blowing. This should be in *degrees clockwise from north*, between 0° and 360°, e.g. a wind direction of 270° is blowing from the west, i.e. a westerly wind.

The **Air temperature (ºC)** is the temperature near the ground (at the screen height of 1.22 m), measured in degrees centigrade.

The **Cloud cover (oktas)** is the amount of cloud cover at the source. This should be between 0 and 8 *oktas*, where 0 is a clear sky and 8 is totally overcast.

The **Precipitation rate (mm/hr)** is the precipitation rate in mm/hr. A value can either be entered directly or selected from a list of Dry (0.0 mm/hr), Light (0.25 mm/hr), Medium (2.25 mm/hr) or Heavy (5.5 mm/hr).

The **Boundary layer height (m)** is the height of the part of the atmosphere where the mixing is affected by the earth's surface, i.e. the region in which turbulence is generated by the effects of friction and the heating or cooling of the surface. In general, the boundary layer height is calculated by ADMS-STAR and is not entered by the user. However, you may use your own measured or estimated value by first clicking the **Use boundary layer height** check box then entering the boundary layer height in metres.

The **Albedo** is the surface albedo; the fraction of direct solar radiation reflected at the Earth's surface. This column is only available if **Specify values on the met form** is selected for **Albedo** on the **Advanced meteorology** screen for the dispersion site.

Priestley-Taylor is the Priestley-Taylor parameter; a parameter representing the surface moisture available for evaporation. This column is only available if **Specify values on the met form** is selected for **Priestley-Taylor parameter** on the **Advanced meteorology screen** for the dispersion site.

The **Sea surface temperature (ºC)** is the sea surface temperature in °C. This column is only available if the **Marine** option is selected on the **Setup** screen.

Vertical profiles of boundary layer parameters
If spatially homogeneous meteorological data are being modelled, then it is possible to specify vertical profiles of some of the boundary layer meteorological parameters.

This can be done by ticking **Vertical profiles**, and browsing to locate a text file (*.prf*) that contains the boundary layer profile data. Further details regarding this option are given in Section 5.2.

4.5.2 Spatially varying meteorological data

To enter spatially varying meteorological data select **Spatially varying met. data from files** on the **Meteorology** screen and then click **Edit...** to display the **Spatially varying met. data** screen, shown in **Figure 4.14**.

Figure 4.14 - The **Spatially varying met. data** screen with all possible columns showing with one line of default data entered.

This screen allows up to 72 sets of meteorological conditions to be selected by specifying up to 73 data files. New lines of data are added by clicking **Add** and unwanted lines of data can be removed by first selecting them and then clicking **Delete**. The data can be sorted into chronological order using the **Sort** button.

The height of the wind measurement to be used should be entered in meters. This height is the height of wind data from the file that will be used to determine boundary layer properties.

Also select whether to **Use boundary layer height from file** or whether ADMS-STAR should calculate the boundary layer height.

The following data should be entered.

The **Local date and time** of the measurement. The date should either be entered in the form day/month/year, e.g. 30/01/2001, or in the form 30 Jan 2001 (i.e. 3-letter abbreviation for month name). The time should be entered in local time (see Section 4.2) in the form hours:minutes. The minimum time between entries is 30 minutes.

The **Albedo** is the surface albedo; the fraction of direct solar radiation reflected at the Earth's surface. This column is only available if **Specify values on the met form** is selected for **Albedo** on the **Advanced meteorology** screen for the dispersion site.

Priestley-Taylor is for the Priestley-Taylor parameter; a parameter representing the surface moisture available for evaporation. This column is only available if **Specify values on the met form** is selected for **Priestley-Taylor parameter** on the Advanced meteorology screen for the dispersion site.

The **Filename** is the file name of the spatially varying meteorological data file for this time period. This must be an output file from the WRF (Weather Research & Forecasting) model[4]. To enter the filename either type the full path and filename or click the arrow to display a **Browse** dialogue box. The WRF files must be in the appropriate format and satisfy certain conditions for ADMS-STAR to run correctly, see Section 8.7 for details.

---

*The times given in the WRF files are always in UTC. When ADMS-STAR is run (in verification or full calculation mode), the model checks that the difference between the time in a WRF file and the local time as specified by the user in the **Local date and time** column of the **Spatially varying met. data** screen is consistent with the offset from UTC entered in the **Sample & receptor locations** section of the **Setup** screen. An error message will be given if this check fails.*

WRF outputs precipitation data as accumulated precipitation since the beginning of the WRF run. ADMS-STAR calculates a precipitation rate using the difference between the accumulated precipitation in one WRF file and the next. It is therefore necessary to ensure that there is one WRF file for a time after the modelling end time.

Observed site met data

If spatially varying meteorology is selected it is possible to enter values of wind speed and direction, air temperature and precipitation rate measured at the source to improve the calculations there. To enter this site observed data, with **Spatially varying met. data from files** selected, select the **Use observed data for the source location** option and then click **Edit...** to display the **Observed met. data at the source location** screen, **Figure 4.15**.

---

[4] http://www.wrf-model.org/index.php

Figure 4.15 - The **Observed met. data at the source location** screen with one line of default data added.

The site observed data are applied at the source and then relaxed back to the WRF meteorological data over a relaxation radius. The values of the horizontal and vertical relaxation radii are entered in the **Effective range of observed met. data** section. The height at which the wind measurements were recorded should also be specified in this section.

Each set of observed met data is applied from the local date and time entered by the user in this screen until the next set of observed met data or the end of the met period in which the observation time falls, as defined on the **Spatially varying met. data** screen, whichever is the sooner.

*To use the same set of observed met data for several met periods the observed met data line should be repeated with times corresponding to the start of each met period as entered in the **Spatially varying met. data** screen.*

The **Observed met. data at the source location** screen allows up to 72 sets of observed met data to be entered. New lines of data are added by clicking **Add** and unwanted lines of data can be removed by first selecting them and then clicking **Delete**. The data can be sorted into chronological order using the **Sort** button. The following data should be entered.

The **Local date and time** of the measurement. The date should either be entered in the form day/month/year, e.g. 30/01/2001, or in the form 30 Jan 2001 (i.e. 3-letter abbreviation for month name). The time should be entered in local time (see Section 4.2) in the form hours:minutes. The times and dates entered do not have to match the

meteorological periods specified for the spatially varying meteorological data.

The **Wind speed (m/s)** is the wind velocity in m/s at the height (m) entered in the **Height of recorded wind** box.

The **Wind direction (deg)** is the direction from which the wind is blowing. This should be in *degrees clockwise from north*, between 0° and 360°, e.g. a wind direction of 270° is blowing from the west, i.e. a westerly wind.

The **Air temperature (ºC)** is the temperature near the ground (at the screen height of 1.22 m), measured in degrees centigrade.

The **Precipitation rate (mm/hr)** is the precipitation rate in mm/hr. A value can either be entered directly or selected from a list of Dry (0.0 mm/hr), Light (0.25 mm/hr), Medium (2.25 mm/hr) or Heavy (5.5 mm/hr).

4.6 Concentration screen

Field measurements of air activity/concentration are entered in the **Concentration** screen (**Figure 4.16**). Up to 25 activity/concentration measurements sampled at different times or locations can be entered. Each activity/concentration sample can include up to 34 isotope-strength pairs of data; one for each of the 34 ADMS-STAR isotopes (including the user-defined isotope/chemical). At least one data pair must be entered per sample.

Figure 4.16 - The **Concentration** screen showing default input data.

The east-west, **X (km)**, and north-south, **Y (km)**, location of each sample should be entered in coordinates relative to the source or in absolute coordinates on a local Cartesian grid depending on the choice made in the **Sample & receptor locations** section of the **Setup** screen. East and north are positive.

The sampling **Date** should be entered in the form day/month/year, e.g. 30/01/2001, or in the form 30 Jan 2001 (i.e. 3-letter abbreviation for month name). The sampling **Time** should be entered in local time (see §4.2) in the form hours:minutes. The default entries will be the date and time the *.hpl* file was opened.

Samples may be included for multiple isotopes at one or more locations. The table at the right hand side of the **Concentration** screen controls the specification of the isotope strengths for each measurement location. You may select from the list of the 34 built-in isotopes, including the user-defined isotope/chemical. Enter the sample **Strength** in Bq/m^3 (or g/m^3).

*If the isotope/chemical **User defined** is chosen then the appropriate half-life and deposition parameters must be defined in the **Isotope palette***

Include each isotope in the table of **Isotopes for Location** by clicking **Add** next to the table. Use

the **Delete** button next to the table to remove any isotopes for the selected location.

A list of all the activity/concentration sample locations and their associated isotopes that will be used in estimating source strengths is displayed in the large table at the bottom of the **Concentration** screen. Once you have entered all isotope samples for your selected location click **Add** to the right of the large sample table to include the activity/concentration data in the table. Repeat for all locations for which data are available. To remove a set of data, select the appropriate row in the table and click **Delete** to the right of the sample table.

*All the isotopes sampled at a particular location should first be added to the table "**Isotopes for Location**", then the whole activity/concentration sample added to the large "**Samples**" table at the bottom of the screen.*

4.7 Deposition screen

The **Deposition** screen may be used when soil samples have been taken and analysed (**Figure 4.17**). If deposition and air samples are both entered the deposition samples are used in preference to the air activity/concentration measurements, as they are likely to be more reliable.

This screen allows the user to enter up to 25 deposition measurements sampled at different times or locations. Each deposition sample can include up to 34 isotope-strength pairs of data; one for each of the 34 ADMS-STAR isotopes (including the user-defined isotope/chemical). At least one data pair must be entered per sample.

Figure 4.17 - The **Deposition** screen showing default input data.

The east-west, **X (km)**, and north-south, **Y (km)**, location of each sample should be entered in coordinates relative to the source or in absolute coordinates on a local Cartesian grid depending on the choice made in the **Sample & receptor locations** section of the **Setup** screen. East and north are positive. Each location must be downwind of the source.

The sampling **Date** should be entered in the form day/month/year, e.g. 30/01/2001, or in the form 30 Jan 2001 (i.e. 3-letter abbreviation for month name). The sampling **Time** should be entered in local time (see Section 4.2) in the form hours:minutes. The default entries will be the date and time the *.hpl* file was opened.

Samples may be included for multiple isotopes at one or more locations. The table at the right hand side of the **Deposition** screen controls the specification of the isotope deposition for each measurement location. You may select from the list of the 34 built-in isotopes, including the user-defined isotope/chemical. Enter the sample **Strength** in Bq/m^2 (or g/m^2).

*If the isotope/chemical **User defined** is chosen then the appropriate half-life and deposition parameters must be entered in the **Isotope palette**.*

Include each isotope in the table of **Isotopes for Location** by clicking **Add** next to the table. Use the **Delete** button next to the table to remove any isotopes for the selected location.

A list of all the deposition sample locations and their associated isotopes that will be used in estimating source strengths is displayed in the large table at the bottom of the **Deposition** screen. Once you have entered all isotope samples for your selected location click **Add** to the right of the large sample table to include the deposition data. Repeat for all locations for which data are available. To remove a set of data, select the appropriate row in the table and click **Delete** to the right of the sample table.

*All the isotopes sampled at a particular location should first be added to the table "**Isotopes for Location**", then the whole deposition sample added to the large "**Samples**" table at the bottom of the screen.*

4.8 Output screen

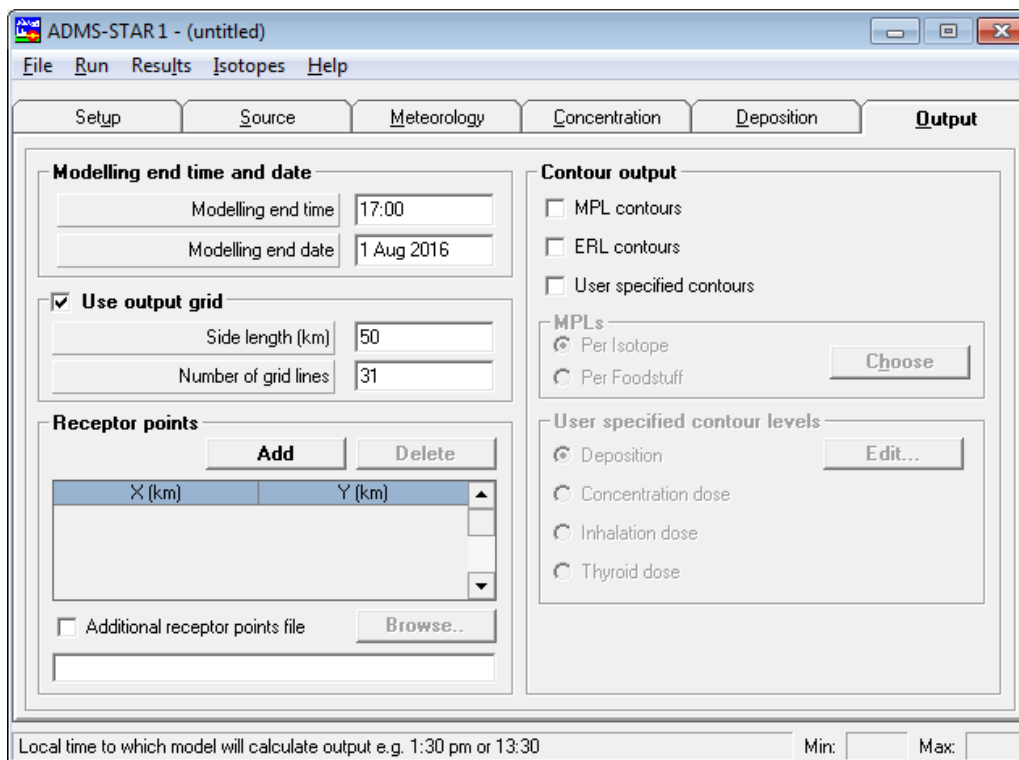The **Output** screen (shown in **Figure 4.18**) is where the output options are selected.

Figure 4.18 - The **Output** screen showing the default options

Modelling end time and date

This is the time and date at which the model calculation will stop. It is the time at which the gridded total accumulated deposition will be calculated. The date should be entered in the form day/month/year, e.g. 30/01/2001, or in the form 30 Jan 2001 (i.e. 3-letter abbreviation for month name). The time should be entered in local time (see Section 4.2) in the form hours:minutes. The default entries will be five hours after the date and time the *.hpl* file was opened.

A range of warning messages will be issued to the .log file (upon running the model in verification mode or full calculation mode) and to the run screen (if running in full calculation mode only) if there is the possibility of deposition levels being under-predicted due to the release not being modelled for the full duration within the modelling domain. Specifically, a warning message will be issued:

a)   *if the* **Modelling end time** *is before or within one hour of the end of the release;*

b)   *if the model's estimate of the travel distance of the end of the release based on the meteorological conditions at the source at the start of the release is less than the distance to the furthest output point.*

A warning message will also be issued upon running the model in full calculation mode only:

c)   *if any part of the release is within the modelling region at the end of the calculation period, where the modelling region is defined to be the region encompassing the source and all output points.*

Output Grid

To calculate gridded output **Use output grid** should be selected. The **Side length (km)**, in kilometres, and **Number of grid lines** (assumed to be the same in the east-west and north-south directions) should then be specified. The grid created will be square and centred on the source.

Receptor points

Enter the location of the points at which you would like activity/concentration and deposition to be calculated. There are no receptor points defined in the default input file, you must enter at least one receptor point before running the model. Up to 25 receptor points can be entered in the interface and a further 200 can be specified in an *.arp* file. The receptor points need not be specified in any order.

The east-west and north-south location of each receptor point should be entered in coordinates relative to the source or in absolute coordinates on a local Cartesian grid depending on the choice made in the **Sample & receptor locations** section of the **Setup** screen. Both values should be given in kilometres. East and north are positive.

The table lists all the receptor points at which the activity/concentration and deposition will be calculated. Click **Add** for a new row to be added to the table and enter the location. To remove a value from the list select the appropriate value so that it is highlighted and then click **Delete**.

If **Gamma dose** is being modelled (refer to Section 4.3.4) then the **Gamma** column indicates at which receptor points the gamma dose rate will be calculated. Click in the column to select a receptor point and click again to deselect the receptor point, up to 5 receptor points can be selected for **Gamma dose** output.

To specify the path of an additional receptor points (*.arp)* file containing up to 200 receptor locations, check **Additional receptor points file** and browse to locate the path to the *.arp* file. The *.arp* file should contain either:

*   a list of easterly and northerly distances relative to the source (in km), or

*   a list of easterly and northerly distances in absolute coordinates (in km),

depending on the choice made in the **Sample & receptor locations** section of the **Setup** screen. **Figure 4.19** shows an example *.arp* file called *Distance.arp*, which is also supplied in *<install_path>\Data*.
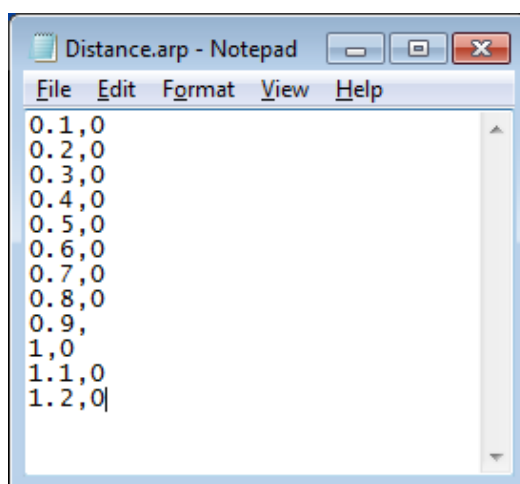
Figure 4.19 - Example of an *.arp* file for specifying additional receptor points

Contour output

If the use of an output grid is selected, then for each emitted isotope the instantaneous activity/concentration will be calculated at the end of each met period, and the total accumulated deposition and concentration dose will be calculated at the modelling end time. If the **Inhalation dose** and **Thyroid dose** modelling options are selected these will also be calculated at the modelling end time for each emitted isotope and as a total over all isotopes.

There are three further options that allow for the calculation of MPL, ERL and/or user defined contour levels:

1. **MPL contours**

    (a) **Per Isotope** (or isotope group)

    (b) **Per Foodstuff**

2. **ERL contours**

3. **User specified contours**

The model can calculate and display contours of MPLs (Maximum Permitted Levels) for particular isotopes/foodstuffs of interest, ERLs (Emergency Response Levels) for inhalation/thyroid dose and/or up to five user-specified contour levels for a particular isotope, group of isotopes, or user-defined chemical.

For MPL contours you can either specify up to five foodstuffs for a chosen isotope or isotope group (**Figure 4.20**) or you can specify up to five individual isotopes or four isotope groups for a single foodstuff (**Figure 4.21**).

Figure 4.20 - The **Per Isotope Data** sub-screen showing default input data.

So, for example, choosing a single isotope and, say, five foodstuffs on the **Per Isotope Data** screen means that contours for deposition and concentration are calculated corresponding to the MPL for that isotope for each of the five foodstuffs.

Figure 4.21 - The **Per Foodstuff Data** sub-screen showing default input data.

It is possible for the user to view the MPLs for each of the 33 standard isotopes and 10 foodstuffs by means of the **MPL Data** option from the **Help** menu on the ADMS-STAR menu bar, illustrated in **Figure 4.22**. To view the MPLs for a particular isotope and foodstuff combination, simply select the appropriate isotope and foodstuff from the drop-down lists.

Figure 4.22 - The **View Foodstuffs Data** sub-screen showing MPL data for I-131 for herbaceous fruit.

For details of the calculation of MPLs for isotope groups, including a list of the isotopes in each group see Section 8.14.

If **Inhalation dose** or **Thyroid dose** is being modelled the model can display **ERL contours** (Emergency Response Level contours). With inhalation dose the ERL contours are displayed for both sheltering and evacuation, and with thyroid dose the ERL contours are displayed for sheltering, evacuation and application of stable iodine tables. In all cases both the lower and upper ERL can be displayed. For more details regarding the ERLs used refer to Section 8.10.

For user entered contours, the choice should first be made between **Deposition**, **Concentration dose**, **Inhalation dose** and **Thyroid dose**. The required levels for that choice can then be entered by clicking on the **Edit**... button to bring up the **User specified contour levels** sub-screen, **Figure 4.23**. On this screen a choice should be made between **Isotope**, **Isotope group** or **Total** using the radio buttons and then the particular isotope or isotope group selected from the appropriate list. Up to five contour levels can be entered. Contour levels are entered by typing the contour value into the box and then pressing SPACE. To remove a level, first select it from the list and then click **Delete**.

*For **Thyroid dose** only the Iodine group may be selected for user defined contours.*
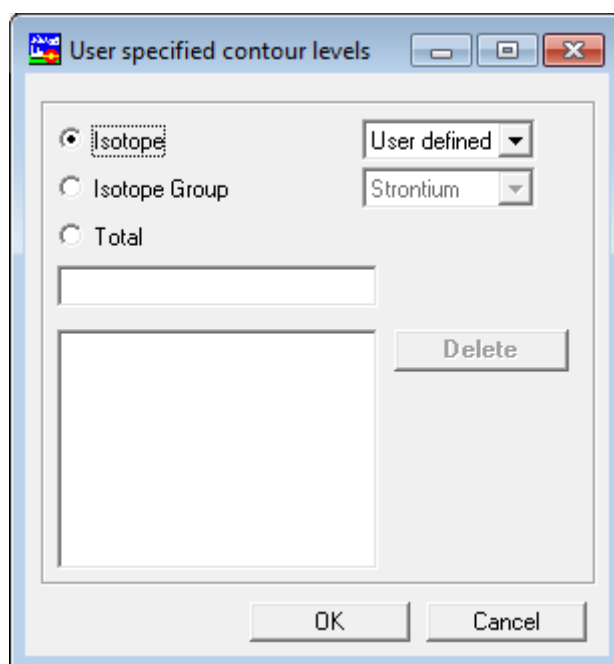
Figure 4.23 - The **User specified contour levels** sub-screen

4.9   Isotope Palette

ADMS-STAR contains a palette of isotopes (**Figure 4.24**). This isotope palette allows for the half-life and standard deposition velocities of the 33 standard isotopes to be viewed and for the half-life of the user-defined isotope to be edited. The isotope palette also allows for more advanced deposition parameters to be entered for all isotopes. The isotope palette is accessed by selecting **Palette** from the **Isotopes** menu.

Figure 4.24 - The isotope palette with the user-defined isotope set to be particulate with two lines of particle properties entered.

The **Isotope name** is used to identify the isotope and is not editable by the user.

The **Type** is the type of isotope; either **Gas** (for gaseous isotopes) or **Particle** (for particulate isotopes).

The **Half life** and **Half life units** give the half life of the isotopes. Only the half life of the user defined isotope is editable.

The **Deposition velocity type** gives the choice of how the deposition velocity is determined for each isotope. There are three options:

*   **Standard** – When this option is selected, ADMS-STAR will modify the **Standard deposition velocity** (which is *not* editable) according to the local meteorological conditions (see Section 8.4.2). The deposition velocity therefore changes temporally (with each met period) and may also change spatially if modelling the effects of complex terrain and/or using spatially varying met data. This is the default option for

all but the user defined isotope.

- **Specified** – When this option is selected, the **User deposition velocity** (which *is* editable) is used. This value is not modified according to the local meteorological conditions. This is the default option for the user defined isotope.

- **Calculated** – When this option is selected, the deposition velocity will be calculated by ADMS-STAR. The method of calculation depends on the isotope **Type**:

    - If the isotope is a **Gas**, the user must specify the **Gas nature**. The choices are:

        o **Reactive**: gas expected to undergo significant chemical reaction with the surface;

        o **Non-reactive**: gas not undergoing significant chemical reaction with the surface; and

        o **Inert**: Noble gas.

    - If the isotope is a **Particle**, the user must enter the particle properties in the bottom right table. New lines of data are added with the **Add** button and unwanted lines can be removed by first selecting them and then clicking **Delete**. Up to 10 sets of particle data can be added for each isotope. The following data are required:

        o **Particle diameter** – the diameter of the particle in microns

        o **Mass fraction** – the fraction of the mass of the particles with the specified diameter. The total mass fraction must add up to 1.

        o **Density** – the density of the material of which the particles are composed in kg/m^3.

This information is then used to calculate the deposition velocity, as detailed in Section 8.4.1.

The **Inhalation dose coefficient** column appears if the **Inhalation Dose** modelling option is selected. Refer to Section 8.10 for more details regarding the inhalation dose coefficient, including the source of the default values.

The table in the bottom left shows the energy levels and branching ratios used for each isotope if the **Gamma Dose** modelling option is selected. Only the values for the user-defined isotope are editable. For each required energy level place a tick in the **Include** column and enter the **Branching ratio**.

SECTION 5 ADVANCED MODELLING

This section describes how to use the advanced model features available in ADMS-STAR.

1. The **Advanced meteorology** parameters are found on the **Meteorology** screen; additional parameters may be specified as described in Section 5.1.

2. User-input vertical profiles of wind speed, turbulence and temperature can be entered into ADMS-STAR, this is described in Section 5.2.

3. Instantaneous activity/concentrations at the end of each meteorological period are by default output on the model grid at ground level, but they can instead be output at a specified height above ground level, this is described in Section 5.3.

5.1 Advanced Meteorological Parameters

Advanced meteorological data can be entered for the dispersion site and/or the meteorological measurement site. To enter advanced meteorological data go to the **Meteorology** screen, check the **Advance meteorology** option in either the **Dispersion site** or **Met. measurement site** boxes, then click **Edit....**

The **Advanced dispersion site data** screen is shown in **Figure 5.1**; the **Advanced met. site data** screen is shown in **Figure 5.2**.

Figure 5.1 - The **Advanced dispersion site data** screen

Figure 5.2 - The **Advanced met. site data** screen

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

This option to alter these parameters at either the dispersion or meteorological measurement site is described further in Sections 5.1.1 to 5.1.3. Additionally there is an option to account for differences in precipitation levels between the dispersion site and the meteorological measurement site; this option is described in more detail in Section 5.1.4.

5.1.1 Surface albedo

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

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

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

  o **Snow-covered ground** = 0.6

  o **Not snow-covered** = 0.23

- **Specify values on the met. form**: Use hourly varying values of surface albedo entered on the **Homogeneous met. data** screen or **Spatially varying met. data** screen.

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

5.1.2 Priestley-Taylor parameter

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

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

- **Specify value**: Enter a constant value. Either enter a value directly into the box or select a value from the list:
  - **Desert** $= 0$
  - **Dry grassland** $= 0.45$
  - **Moist grassland** $= 1$

- **Specify values on the met. form**: Use hourly varying values of Priestley-Taylor parameter entered on the **Homogeneous met data** screen or **Spatially varying met data** screen.

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

5.1.3 Minimum Monin-Obukhov length

This is an option to specify the minimum value of the Monin-Obukhov length to allow for the heat island effect in cities. The minimum Monin-Obukhov length provides a measure of the maximum stability of the atmosphere (see Section 8.2). In urban areas, there is a significant amount of heat stored and later released from buildings and traffic, which warms the air above the town/city. For large urban areas this is known as the urban heat island. It has the effect of preventing the atmosphere from becoming very stable. In general, the larger the area the more heat is released and the stronger this effect becomes. This means that in stable conditions the Monin-Obukhov length will never fall below some minimum value. The minimum Monin-Obukhov length should be between 1 and 200 m; the default value is 1 m corresponding to a rural area. The following options are available for the minimum Monin-Obukhov length:

- **Use model default**: use the model default value; this is the default option for the dispersion site.

- **Specify value**: Enter a value, either directly into the box or select from the list:
  - **Large conurbations > 1 million** $= 100$ m
  - **Cities and large towns** $= 30$ m
  - **Mixed urban/industrial** $= 30$ m
  - **Small towns < 50,000** $= 10$ m

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

5.1.4 Precipitation

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

The precipitation factor is the ratio of the annual precipitation at the dispersion site to that at the meteorological site and is applied to the hourly rates, e.g. if the precipitation factor is 0.5 then when the precipitation rate at the meteorological site is 2 mm/h it will be 1 mm/h at the dispersion site. The following options appear:

- **Same as at meteorology site**: the precipitation rate at the meteorological site is representative of that at the dispersion site. This is the default option.

- **Precipitation factor**: Enter a value for the precipitation factor. The precipitation factor should be between 0 and 20; the default value is 1.

5.2 User-input vertical profiles of meteorological data

If spatially homogeneous met data are being used then user-input vertical profiles of wind speed, turbulence and temperature can be entered into ADMS-STAR using a *.prf* file. To use this option check **Vertical profiles** on the **Meteorology** screen and enter the path of the *.prf* file.
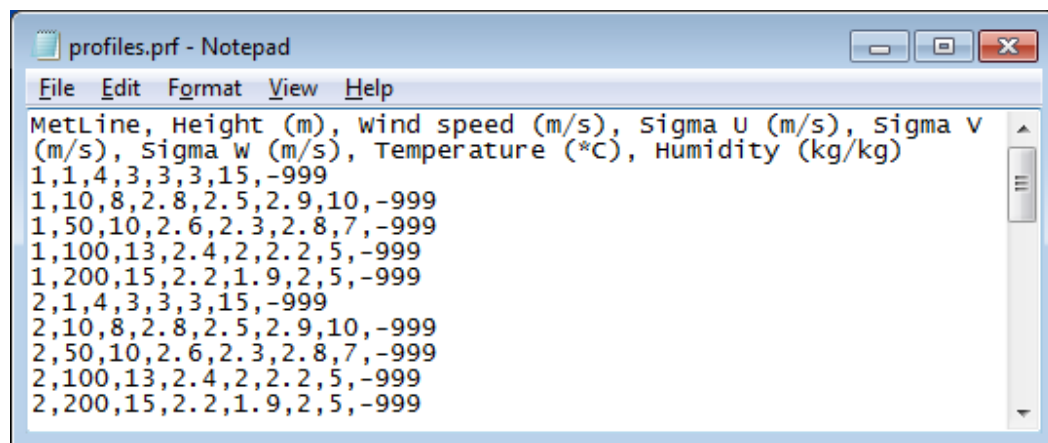
5.2.1 Format of the *.prf* file

Figure 5.3 - Example *.prf* file

The *.prf* file should be comma separated. An example *.prf* file called *profiles.prf* is shown in **Figure 5.3** to illustrate the format of the file. This file is also supplied in *<install_path>\Data*. The first line of the file is a header line; the rest of the lines then contain data in the following format:

```
Met line, Height, Wind speed, Sigma U, Sigma V, Sigma W, Temp., Spec.
humidity
```

Data should be entered for at least two heights for every line of data entered on the **Homogeneous met. data** screen. The lines should be in meteorological data line order with the heights for each met line in ascending order. All of the columns must be present for each line of data; if values are unknown for any parameter then a value of -999 should be entered.

The variables entered in the meteorological file are as follows:

1. `Met line`: the meteorological data line number corresponding to this line of profile data

2. `Height`: the height in metres (must be greater than 0 m)

3. `Wind speed`: the wind speed in m/s

   The wind speeds entered should be consistent with the wind speeds on the **Homogeneous met data** screen, as the wind speed on the **Homogeneous met data** screen is used in combination with the data from the *.prf* file to produce a

modified profile.

Minimum > 0 m/s

Maximum = 100 m/s

4. **Sigma U**: The along-wind turbulence in m/s

    Minimum > 0 m/s

    Maximum = 1000 m/s

5. **Sigma V**: The cross-wind turbulence in m/s

    Minimum > 0 m/s

    Maximum = 1000 m/s

6. **Sigma W**: The vertical turbulence in m/s

    Minimum > 0 m/s

    Maximum = 1000 m/s

7. **Temperature**: the temperature in °C

    Minimum = -100°C

    Maximum = 60°C

8. **Specific humidity**: this is not used in ADMS-STAR and should be set to -999.

5.3 Gridded values of instantaneous activity/concentration above ground level

Gridded values of instantaneous activity/concentration at the end of each meteorological period are output at ground level by default. It is possible to change this so that the activity/concentration at a specified height above ground level is output. The data at ground level or above ground level are output to a *.gst* file. The grid used is that specified in the **Output Grid** panel of the **Output** screen of the ADMS-STAR interface; an output grid must be included in the run for this option to be available.

The option to output gridded values of instantaneous activity/concentration at a height other than at ground level is controlled using an *.hai* file. The *.hai* file is a text file which should be given the same name as the corresponding *.hpl* file and saved in the same directory. The following text should be included in the *.hai* file:

- **HAIFileVersion1**: the version number for the *.hai* file

- **GRIDDEDCONCHEIGHT:** the keyword for this section.

- **GriddedConcAboveGL: Y** if the gridded values of instantaneous activity/concentration should be calculated above ground level

    If **GriddConcAboveGL=Y**, then provide:

- **GridConcHeight**: the height (m) at which to calculate the values of instantaneous activity/concentration.

An example of the *.hai* file for outputting gridded values of instantaneous activity/concentration at a height of 21 m above ground level is given in **Figure 5.4**.
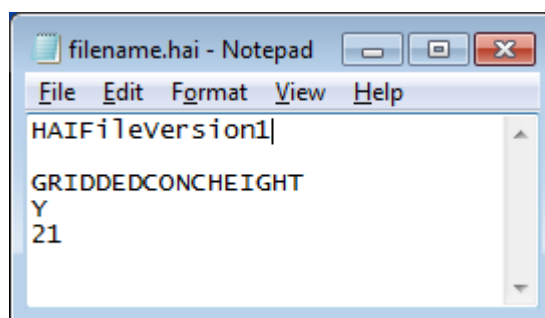

Figure 5.4 - Example *.hai* file used for specifying that gridded values of instantaneous activity/concentration should be output for a height of 21 m.

SECTION 6 OUTPUT RESULTS

The model is run by selecting **Run** or **File**, **Run** from the menu, which runs the model with the currently loaded data (*.hpl* file). Once a model run is complete, you can choose how to display the output results. The various display options use one or more of the output files that are produced during the model run.

6.1 Output files

During the model run, output files are created in the same directory as the input *.hpl* file. The output files are, *.rec*, *.glt*, *.rlt*, *.!01*, *.gst* and *.mop*.

The output files have the same file stem as the input file. For instance, for the input *.hpl* file *example.hpl*, the output files produced could be

- *example.rec*
- *example.glt*
- *example.rlt*
- *example.!01*
- *example.gst*
- *example.mop*

The file extensions indicate the different types of output data. These are outlined below.

.rec   Contains instantaneous activity or concentrations (Bq/m^3 or g/m^3), total accumulated deposition (Bq/m^2 or g/m^2) and concentration dose ($Bq.s/m^3$ or $g.s/m^3$) at each of the receptor points at the end of each met period within the modelling period and at the modelling end time. If the **Inhalation dose** or **Thyroid dose** modelling options are selected it also contains the inhalation dose (Sv) and thyroid dose (Sv) per isotope and as a total over all isotopes at the modelling end time. If the **Gamma dose** modelling option is selected it also contains the gamma dose due to concentration (Sv) and gamma dose due to deposited material (Sv) at the modelling end time for the selected receptor points. The *.rec* file also contains a description of the run set-up. Further details of the *.rec* file are given in Section 6.2.2.

.glt   Contains gridded data for the total accumulated deposition, concentration dose, inhalation dose and thyroid dose, as well as for any MPL, ERL or user-defined levels selected in the **Output** screen (described in Section 4.8). The contours may be displayed using a contour plotting system. A full explanation of the data in the *.glt* file is given in Section 6.2.3.

.gst   Contains gridded data for the instantaneous activity or concentrations (Bq/m^3 or g/m^3) at the end of each met period within the modelling period and at the modelling end time. The contours may be displayed using a contour plotting system. A full explanation of the data in the *.gst* file is given in Section 6.2.4.

.rlt   Contains data for the total accumulated deposition and concentration dose along radii at 10° intervals. A full explanation of the data in the *.rlt* file is given in Section 6.2.5.

.mop   Contains meteorological parameter data at each of the receptor points specified by the user on the **Output** screen or in the *.arp* file (see Section 4.8). A full explanation of the data in the *.mop* file is given in Section 6.2.6.

.!01   Is used by the x-y plotting facility and is not meant to be read by the user. The x-y plotting facility is described fully in Section 6.3.

6.2 Viewing numerical data in ADMS-STAR

The numerical output files may be viewed by selecting **Results**, **Output** from the ADMS-STAR menu bar, and choosing the required file. The output file is then opened with the application selected in the **File Viewing Preferences** screen as described in Section 6.2.1.

6.2.1 Choosing application to view output

To change the preferred viewing application for input data and output files, select **File**, **Preferences**, **Viewing Output** from the menu bar of the ADMS-STAR interface. This will display the **File Viewing Preferences** screen shown in **Figure 6.1**.
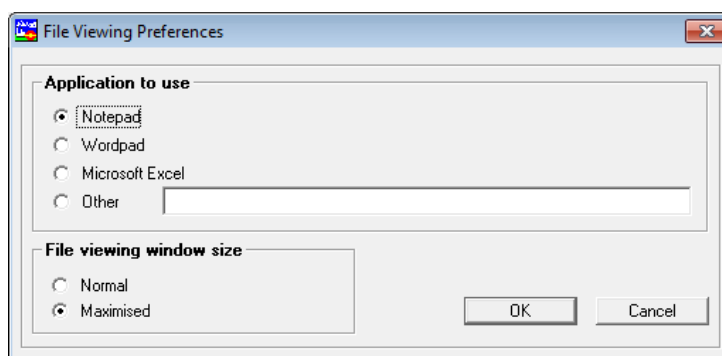
Figure 6.1 - The **File Viewing Preferences** window

Application to use allows the user to select which application to use to view files. If an application other than Notepad, WordPad or Microsoft Excel is to be used, then select **Other** and enter the path of the application.

File viewing window size defines the size of the viewing window when it is opened. Choices are either **Normal** (size of the window last used) or **Maximised** (full screen).

Restore Defaults returns the current settings to the default settings, and **Save Defaults** saves the current settings as the default settings so that the selected options are used the next time ADMS-STAR is opened.

6.2.2 .rec file

Figure 6.2 - An example *.rec* numerical output file

Figure 6.2 shows an example of the instantaneous concentration, concentration dose and total accumulated deposition output in the *.rec* file. In this case the output is for a single isotope and a single specified output point. The *.rec* file also contains details of the source set-up, isotope set-up, meteorological data and any sample data. If sample data are used the estimated source strength is also given in the *.rec* file. For particulate isotopes with the deposition velocity set to be **Calculated**, the concentration and total accumulated deposition is given for each particle size as well as the total amount for that isotope. The location of each receptor point is given both in absolute coordinates on a local Cartesian grid and in coordinates relative to the source (in kilometres).

6.2.3 .glt file

Figure 6.3 - Example of a *.glt* numerical output file

If gridded output is selected then a *.glt* file will be produced containing the total accumulated deposition and concentration dose at the modelling end time for each isotope emitted. For particulate isotopes with the deposition velocity set to be **Calculated**, the total accumulated deposition is given for each particle size as well as the total amount for that isotope.

If the **Inhalation dose** or **Thyroid dose** modelling options are selected then the inhalation dose and thyroid dose for each isotope as well as the total over all isotopes are given.
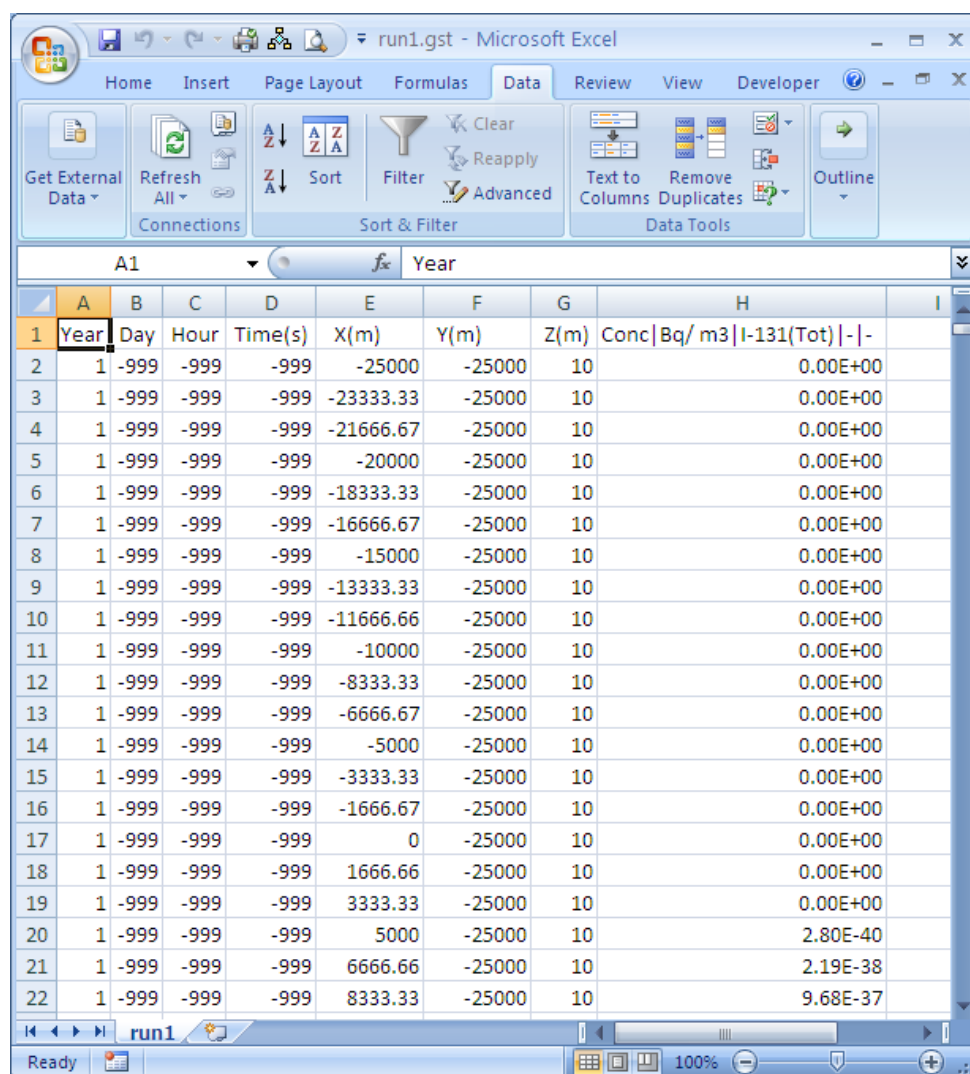
If the user selects MPL, ERL and/or user-defined contour output on the **Output** screen there are also columns for each of the combinations selected. These columns give the appropriate value divided by the reference level, e.g. for MPL values are given for the total accumulated deposition divided by the deposition MPL and for the concentration dose divided by the concentration MPL. Therefore, for these contours if the reference level is reached it is given by the contour with value 1.

Details for each column are given in the header of the *.glt* output file.

- `TotDep` indicates the total accumulated deposition

- `ConcDos` indicates the concentration dose

- `InhlDos` indicates the inhalation dose

- `ThyrDos` indicates the thyroid dose

- For MPL contours the Foodstuff and isotope are given along with the MPL level.

- For ERL contours, the type is given as `Whole body` (for inhalation dose) or `Thyroid` (for thyroid dose). The response measure is given as `Shelt` (for sheltering), `Evac` (for evacuation) or `StabI` (for application of stable iodine tablets). Also the level is indicated as `L` (lower) or `U` (upper).

- For the User-defined contours the type is given by `Dose` (concentration dose), `Dep` (accumulated deposition), `Inhl` (inhalation dose) or `Thyr` (thyroid dose)

Output points are given in absolute coordinates on a local Cartesian grid (in metres).

6.2.4 .gst file

Figure 6.4 - Example of a *.gst* numerical output file

If gridded output is selected then a *.gst* file will be produced containing the instantaneous activity or concentrations (Bq/m^3 or g/m^3) at the end of each met period within the modelling period and at the modelling end time for each isotope emitted. Output is given at ground level by default, but can be given at a specified height above ground level using an *.hai* file (refer to Section 5.3 for details).

Details for each column are given in the header of the *.gst* output file. The value in the `Year` column refers to the met period for which the instantaneous activity/concentrations are given, apart from the highest value of `Year` which indicates that the instantaneous activity/concentrations given are for the modelling end time.

Output points are given in absolute coordinates on a local Cartesian grid (in metres).

6.2.5 .rlt file

Figure 6.5 - Example of an *.rlt* numerical output file

If gridded output is selected then an *.rlt* file will be produced containing the total accumulated deposition and the concentration dose at the modelling end time for each isotope emitted. The *.rlt* file contains values along radii at 10° intervals. The distance covered and the spacing between points along a radius is determined by the properties of the output grid. The points along the radius that runs from west to east from the source to the edge of the output grid are coincident with the points on the output grid.

The *.rlt* file consists of a header row followed a separate block of data for each output type for each isotope. For each isotope, output type combination the output value is then given for each angle and distance from the source. The angle for each measurement is given as the angle measured in degrees anti-clockwise from the x-axis (i.e. east). For particulate isotopes with the deposition velocity set to be **Calculated** blocks of data are given for each particle size as well as the total amount for that isotope.

6.2.6 .mop file

Figure 6.6 - Example of a *.mop* numerical output file

If selected on the **Meteorology** screen then a *.mop* file will be produced. The *.mop* file contains the meteorological parameters for each met period at each of the receptor points defined on the **Output** screen and in the *.arp* file. The *.mop* file lists the meteorological data at each location for each met period up to the modelling end time. Meteorological data calculated by the model is given alongside the input data. The location of each receptor point is given both in absolute coordinates on a local Cartesian grid and in coordinates relative to the source (in kilometres).

6.3 X-Y plots

The x-y plotting facility can be used to display results for runs modelling continuous releases. Plume dispersion data can be plotted along a line of points extending in the downstream direction from the source, i.e. along the 'plume centreline', as far as the release travels during the first met period.

A specific type of file is used by the graphical interface to produce x-y plots. These files have the extension *.!01*. They have been formatted so that they can be read by the graphical interface and are not intended to be read or interpreted by the user.

To use the x-y plot facility, select **Results**, **X-Y Plot**... from the ADMS-STAR menu bar to obtain the **ADMS Line Plotter** screen as shown in **Figure 6.7**.

Select the required output from the **Definition** screen.

*More than one output file from the list displayed in the screen can be plotted at the same time. Use the **Shift** and **Control** keys to select multiple files to plot.*

For ADMS-STAR, the **Type of Modelling Run** will always be the **Plume or Time integrated puff** option. Make sure this option is selected. All files of the form **.!01* in the current directory are then displayed in the lower centre box of the screen.
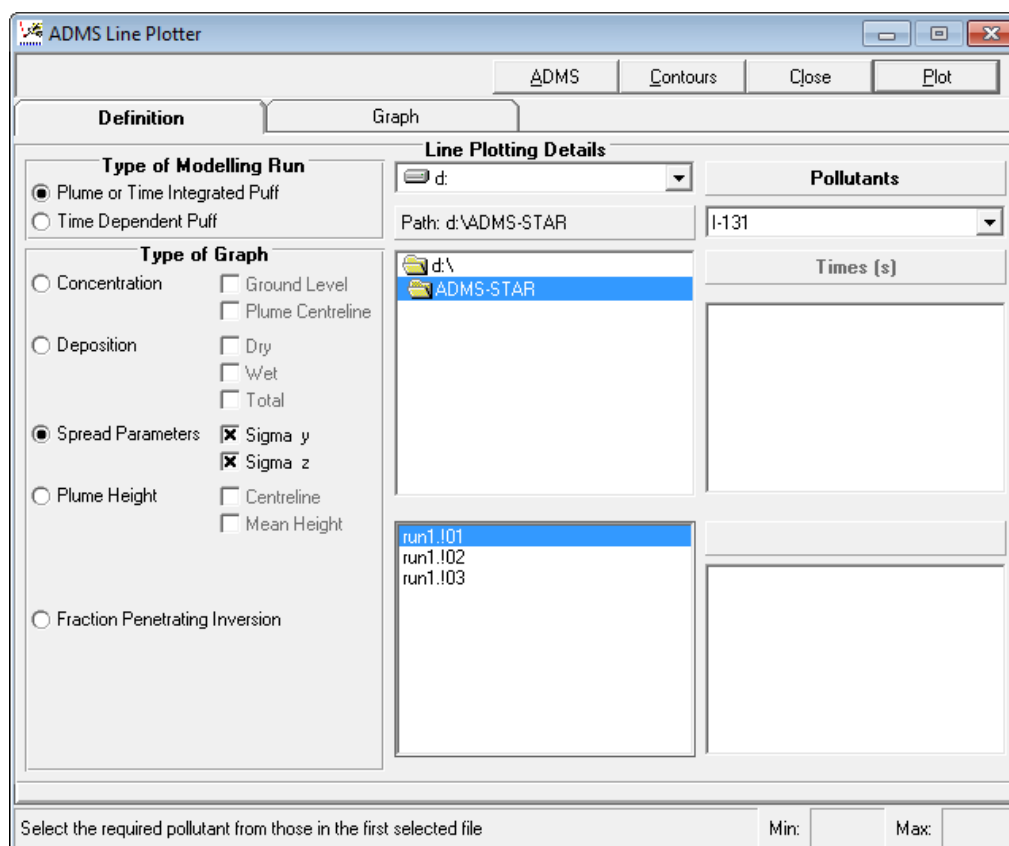
Figure 6.7 - The ADMS Line Plotter screen.

Next, in the **Type of Graph** part of the screen, you can choose one of the following options by

selecting the appropriate radio button or check box.

- **Spread Parameters** – **Sigma y** and/or **Sigma z**. These are the lateral spread and the vertical spread of the release about the plume centreline, respectively. If there is an inversion at the top of the boundary layer, and part of the plume penetrates the inversion, the output is for the part of the plume below the inversion.

- **Plume Height** – **Centreline** and/or **Mean Height**. If there is an inversion at the top of the boundary layer, and part of the plume penetrates the inversion, the output is for the part of the plume below the inversion.

- **Fraction Penetrating Inversion** – The fraction of the plume that has penetrated above the top of the boundary layer.

*The other options; **Concentration** and **Deposition** are not available with ADMS-STAR and are given as –999 at all downstream locations.*

Finally, click **Plot** and the selected x-y plot is drawn.

The appearance of the plot can be altered from the **Graph** screen by clicking **Graph Setup** and using the **Graph Design** dialogue box shown in **Figure 6.8**.
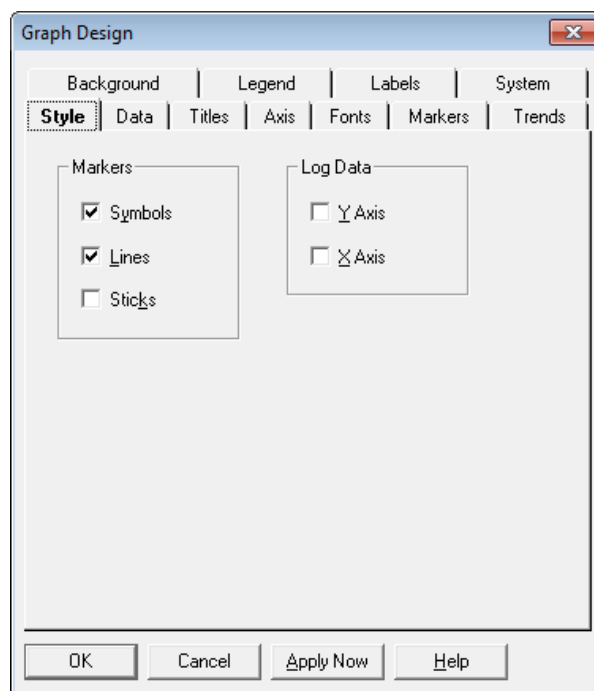
Figure 6.8 - The x-y plotting facility Graph Design box.

The **Graph Design** dialogue box has 11 options, the purpose of which can be briefly summarised as follows.

- **Legend** – allows the user to select the legend text, size and position.

- **Labels** – allows the user to turn axis labels and data point labels on and off and change their format.

- **System** – allows the user to print a graph and save a graph (data and/or graph layout) to file.

- **Style** – allows the user to select how the data are to be displayed on the graph (e.g. just points or with lines joining the points), and change the scale of the axes to a log scale.

- **Data** – allows the user to add or remove data points and control the range of data points plotted.

- **Titles** – allows the user to write/edit titles and change the orientation of titles. The limit of 80 characters in the title is set by the graphics software.

- **Axis** – allows the user to alter the minimum value, maximum value and number of ticks on the axes. To do this, click on the axis in the **Apply to Axis** box. Click **User-defined** in the **Scale** box and then enter the appropriate minimum value, maximum value and number of ticks in the **Range** box. Click **Apply Now** and if the graph is correct, click **OK**. Labels and their orientation are set on the **Labels** screen.

- **Fonts** – allows the user to set the font of the graph title, axes labels and legends.

- **Markers** – allows the user to alter the symbol used in the graph to denote the data points, change the size and colour of these symbols, and adjust the colour and thickness of the line and whether it is dashed or solid.

- **Trends** – allows the user to add limit lines, statistical lines and fit a curve to data points.

- **Background** – allows the user to alter the background colour of the graph and the style of borders drawn around titles, the legend and the graph.

6.4 Contour plots in Surfer (ADMS Contour Plotter)

This section outlines how to produce contour plots in Surfer from gridded ADMS-STAR output data using the **ADMS Contour Plotter** utility launched from the ADMS-STAR interface.

To launch the **ADMS Contour Plotter (Plotting in Surfer)** utility, select **Results**, **Contour Plot in Surfer**... from the top of the ADMS-STAR interface. This displays the screen shown in **Figure 6.9**.

*The **Flow field** and **Show specified points files** options are not used with ADMS-STAR.*
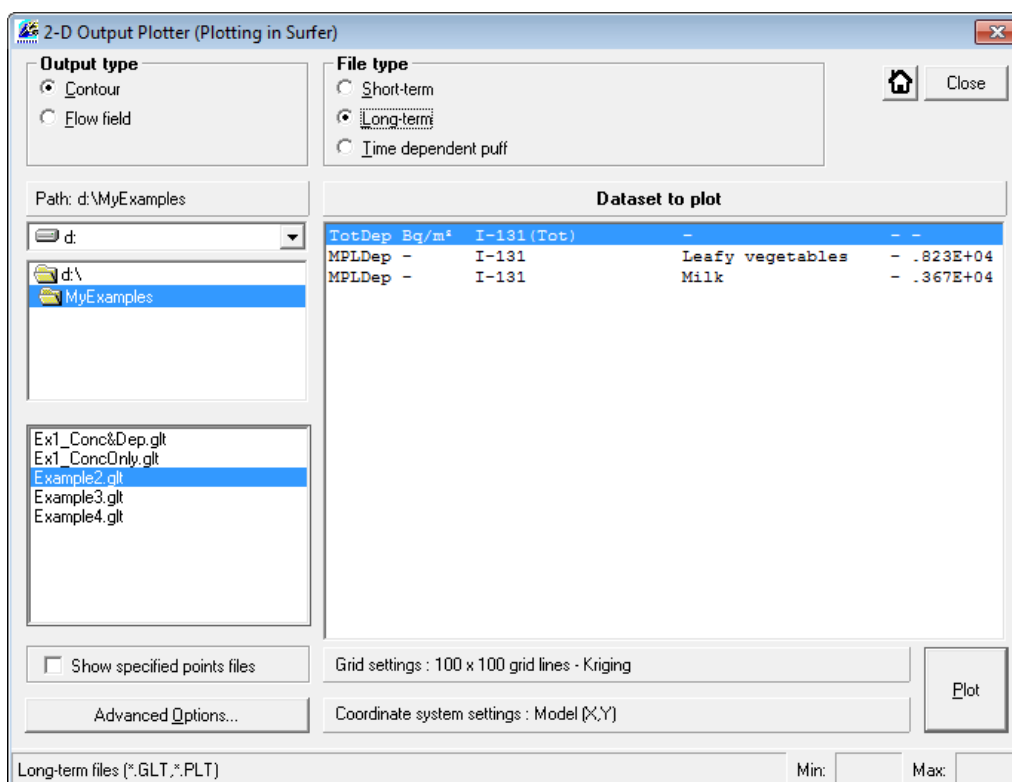
Figure 6.9 − The **ADMS Contour Plotter** screen.

6.4.1 Main interface

Here are the steps to follow in order to create a contour plot of data:

Step 1 Ensure the **Contour** radio button is selected.

Step 2 Select the **Long term** radio button to plot the data in the *.glt* file (i.e. total accumulated deposition) or the **Short term** radio button to plot the data in the *.gst* file (i.e. instantaneous activity/concentrations).

Step 3 Select the appropriate folder and then click on the name of the file containing the data to plot.

Step 4 Select the dataset to plot (in the **Dataset to Plot** box), and if relevant select also the time for which the data are to be plotted (**Time (year, day, hour)** box).

The **Dataset to Plot** box shows a list of all the variables that can be used for the plot. Each record in the list shows the type of output (e.g. 'TotDep' for total accumulated deposition or 'MPLDep' for MPL levels), the units of output (e.g. 'Bq/m^2'), the isotope or isotope group name (e.g. 'I-131'), the foodstuff (e.g. 'Milk' or 'User defined' for user defined contour levels), and the MPL level.

Step 5 Click **Advanced Options**... to set some properties of the plot (optional, see Section 6.4.2).

Step 6 Click **Plot** to plot the selected data in Surfer.

The ADMS-STAR output file is converted to a grid file (.grd*) for Surfer to plot. Save this in the directory where the* .hpl *file is located. Once the Surfer drawing has been saved, you can delete the* .grd *file as it is no longer needed.*

In the upper right-hand corner, the **Close** button closes the **ADMS Contour Plotter** and the **ADMS** button brings the ADMS-STAR interface to the front without closing the utility.

6.4.2 Advanced options

The **Advanced Options**... button gives the user a number of options to enhance the contour plot. Clicking it displays the **Advanced Contour Options** screen shown in **Figure 6.10**. From that screen, you can set the contour levels, the coordinate system of the grid, the Surfer gridding options and whether to overlay the terrain file.

Figure 6.10 − The **Advanced Contour Options** screen.

Contour levels

To choose the contour levels to plot, click **(User-specified)** from the list. For each level

---

you want to plot (up to a maximum of 8 levels), type the level value in the small box above the **Delete** button and press the space bar to enter it into the larger box. If you want to delete any level, select that number in the larger box and click **Delete**.

If you are plotting MPL or ERL contours, it is often useful to specify a contour level with value 1. This is because the MPL data in the *.glt* file give the total accumulated deposition value (or concentration dose) divided by the MPL level. Therefore, for these contours, if the MPL level is reached it is given by the contour with value 1. A similar principle applies with the ERL contours for inhalation dose or thyroid dose.

Contour plot coordinate system

To convert output from X-Y coordinates (in metres) on a local Cartesian grid to longitude-latitude coordinates (in decimal degrees), select **Longitude-Latitude** and then click **Add/Edit reference points**. You must supply the X-Y and longitude-latitude coordinates of a reference point, as shown in **Figure 6.11** (left). Longitude and latitude should typically be given to 6 decimal places.

Figure 6.11 − Left: entering coordinates of a reference point. Right: revising and updating coordinates of a reference point.

It is recommended that the X, Y and longitude-latitude coordinates of the reference point be the same as those used to define the source location in the **Source** *screen of the interface. Note however that here the X, Y values should be given in metres rather than in kilometres.*

Click **Add** to store the coordinate data and on **Close** to leave the screen. Reference point information can be deleted by clicking **Remove** or revised by entering the new data and then clicking **Update** as shown in **Figure 6.11** (right).

If you have entered several reference points, because you are studying sites in different locations that use different local co-ordinate systems, choose the appropriate reference point from the drop down list.

Gridding options

A number of different gridding methods can be selected from the **Gridding method** drop-down list. Information about the available gridding methods can be found in the Surfer user guide.

To specify the grid resolution, tick the **Specify number of grid lines** box. Select the

appropriate option to specify the number of grid lines in both the X and Y directions, or in one direction only. If the number of lines is specified in one direction only, the contouring interface will ensure that the resolution is the same in the X and Y directions. Enter the number of grid lines in the **X:** and/or **Y:** boxes.

*If the **Specify number of grid lines** option is not selected, the **ADMS Contour Plotter** will choose the resolution based on the resolution of the data to be plotted.*

Overlay terrain file

If the effects of complex terrain have been included in the model run, it may be appropriate to overlay the activity/concentrations on the terrain data. To do this, go to the **Overlay terrain file** box, then click **Browse** to locate the *.ter* file. When you click **Close** and then **Plot**, the activity/concentration contours will be overlaid on contours of the terrain. You should ensure that the terrain file is the right one for the contours being displayed.

The **Grid settings** and **Coordinate system settings** selected on the **Advanced Contour Options** screen are displayed at the bottom of the main screen, as shown at the bottom of **Figure 6.9**.

When you have finished with the **Advanced Contour Options** screen, click **Close** to return to the **ADMS Contour Plotter (Plotting in Surfer)** main screen. Once the required information has been selected in this screen, click **Plot** to create the plot.

SECTION 7 WORKED EXAMPLES

In this section, worked examples are described to guide you through setting up some typical ADMS-STAR model runs and to demonstrate how results may be presented. It is recommended that you work through these examples when starting to use ADMS-STAR. Note that a complete set of the resulting ADMS-STAR input and output files can be found in the *<install_path>\Examples* directory.

You can launch ADMS-STAR in several different ways:

- double-click the icon for the shortcut created earlier (see §2.2);

- use the Windows **Start** menu and select **Programs**, **ADMS-STAR**; or

- go to the main ADMS-STAR directory *<install_path>* and double-click the file *ADMS-STAR.exe*.

It is strongly recommended to create a directory for setting up and running these worked examples, for instance D:\MyExamples, in order to keep them separate from the examples provided in <install_path>\Examples directory supplied with the model.

7.1 Example 1: Explosive release and estimated source strength

The purpose of this example is to demonstrate how ADMS-STAR might be set up and run to model the explosive release of a chemical. In this example it will be assumed that the source strength is not known and must therefore be estimated from concentration and deposition samples collected after the explosion.

7.1.1 Setting up the run

Work through the following steps in order to set up the run

Step 1 Open ADMS-STAR and select **File**, **New** to create a new input file.

Step 2 In the **Setup** screen, enter a **Site description** and **Model run description** of your choice.

These are optional but a meaningful title can help to differentiate between model runs. The titles will be written out to the *.rec* file.

Step 3 In this example we will be modelling the release of a radioactively stable chemical. The **Unit of release** should therefore be set to grams (g) rather than Becquerels (Bq).

All other options in the **Setup** screen can be left as their default setting for this example, i.e. the effects of wet deposition will be modelled but not the effects of complex terrain or the marine boundary layer; the local time zone is already in UTC;

---

the sample and receptor locations are to be defined in coordinates relative to the source. Compare your screen to that shown in **Figure 7.1**.

Figure 7.1 - Example 1: the **Setup** screen.

Step 4   Go to the **Source** screen. Use the radio buttons at the top of the screen to select that you wish to model an **Explosive release**.

Step 5   To define the source location, click the 🖉 button to display the **Define location** sub-screen.

The source location should be entered in both longitude-latitude decimal degrees and in coordinates on a local Cartesian grid (in kilometres). Here, we will use the UK Ordinance Survey National Grid to define a fictitious source with a longitude and latitude of -1.5° and 52.5° respectively. In OS National Grid coordinates, the easting and northing of this location (in metres) are approximately (434000, 289000), which must be entered in the interface as 434km for the X datum and 289km for the Y datum.

Step 6   Enter the longitude, latitude and X,Y datum values shown in **Figure 7.2**. You can click the 🌐 button to view this location in Google Maps[3]. Click **OK** to return to the **Source** screen.
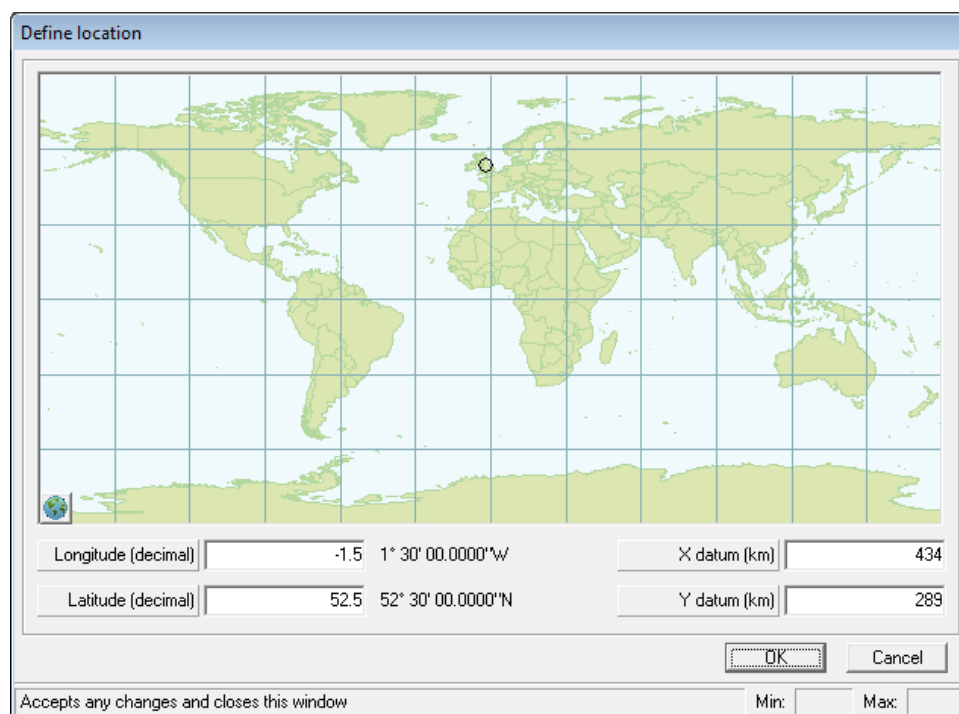
Figure 7.2 - Example 1: the **Define location** sub-screen.

Step 7 Enter the release start time and date, cloud top height and cloud properties as shown in **Figure 7.3**.

Step 8 Select the appropriate radio button so that **The model will estimate source strengths** from air concentration and soil deposition samples.
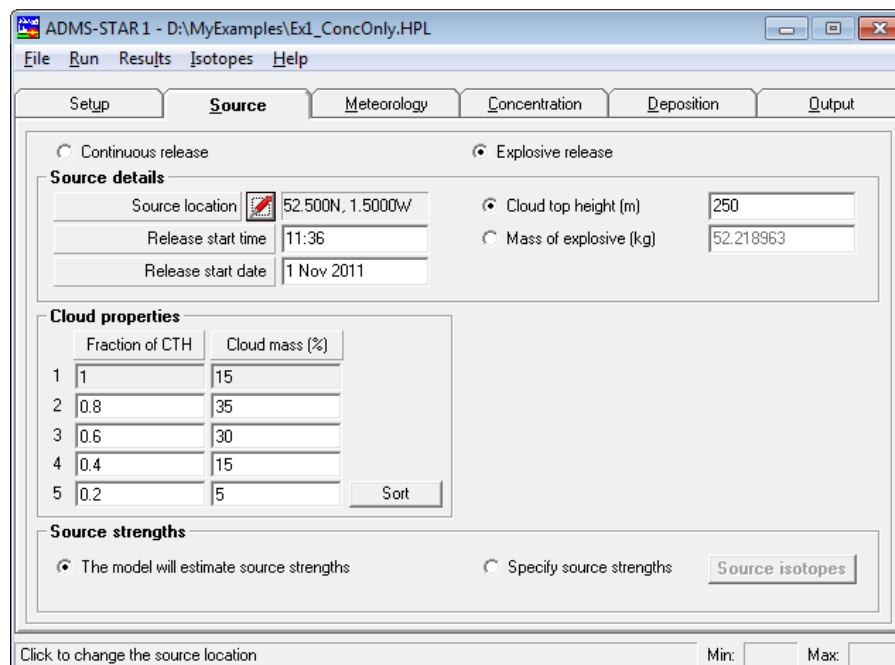
Figure 7.3 - Example 1: the **Source** screen.

Now move to the **Meteorology** screen to enter the meteorological data.

Step 9 Leave the **Surface Roughness** at the dispersion site as 0.3m and leave the option ticked to **Use dispersion site roughness value** at the met site.

Step 10 In the **Met. data** box, select the **Spatially homogeneous met. data** radio button and click **Edit...** to display the **Homogeneous met. data** sub screen. Enter the data as shown in **Figure 7.4**. Click **OK** and return to the **Meteorology** screen.

Figure 7.4 - Example 1: the **Homogeneous met. data** sub-screen.

Now move to the **Concentration** screen. This is where field measurements of air concentration are input in order to estimate the source strength when it is not known. Typically after an accidental release, downwind air concentration measurements are the first samples gathered. Later, soil samples are collected but must be processed before the deposition data are available. This data would be entered in the **Deposition** screen. If deposition and air samples are both entered in the interface, the deposition samples are used in preference to the air concentration measurements, as they are likely to be more reliable, representing a sample over a longer duration than an air sample. In this example we will run the model twice. In the first instance only air samples will be provided, in the second instance deposition samples will also be provided. This demonstrates how, in an emergency situation, the model may be run more than once as more data becomes available.

In this example, we are modelling the release of a radioactively stable chemical with a known particle distribution. We will define these properties before entering the sample data.

Step 11 Select the **Palette** option from the **Isotopes** menu to display the **Isotope palette** sub-screen. For the 'User defined' isotope, change the property in the **Type** column to 'Particle', the **Half life units** column to 'Stable' (the value in the **Half life** column is then ignored) and the **Deposition velocity type** column to 'Calculated'.

Step 12 To define the particle distribution, enter values of **Particle diameter, Mass fraction** and **Density** in the particle properties table. For this example, enter the values as shown in **Figure 7.5**. New lines of data are added with the **Add** button. After the data has been entered, click **OK** and return to the **Concentration** screen.

Figure 7.5 - Example 1: the **Isotope palette** sub-screen.

Let us assume that approximately one and a half hours after the initial explosion, three air concentration samples were obtained at three separate locations downstream of the source; at 23.2km east, 1.5km north of the source at 13:01 with a measured concentration of 71µg/m^3; at 25.7km east, 3.6km south of the source at 13:05 with a measured concentration of 26µg/m^3 and; at 28.1km east, 1.3km north of the source at 13:11 with a measured concentration of 54µg/m^3.

Step 13 Enter this data in the **Air concentration sample details** box. For each sample location, the concentration data should first be added to table of **Isotopes for Location** using the **Add** button next to the table. Note that data should be entered in g/m^3. Once all samples have been defined for a given location, use the **Add** button to the right of the large sample table to include the concentration data in the table. Compare your screen with **Figure 7.6**.

Sample locations have been entered in coordinates relative to the source (in km) to reflect the choice made in the **Sample & receptor locations** section of the **Setup** screen.

Figure 7.6 - Example 1: the **Concentration** screen

Now move to the **Output** screen to select the output options for this run.

Step 14 Ensure that the **Use output grid** option is ticked so that gridded deposition and concentration output is given. Enter a modelling end time and date, output grid side length and number of grid lines as shown in **Figure 7.7**.

It is also necessary to define at least one receptor point before running the model. Receptor locations should be entered in coordinates relative to the source (in km) to reflect the choice made in the **Sample & receptor locations** section of the **Setup** screen.

Figure 7.7 - Example 1: the **Output** screen

Step 15 Select **File**, **Save As...** and save the file as *Ex1_ConcOnly.hpl*.

Step 16 Click **Run** to run the model.

The run will only take a few seconds to complete. A run window will appear. The file name is shown in the title bar of the window. The first messages to appear on the screen are related to the licence details for the model, followed by messages relating to the current run. When the run has finished, a dialog box may appear depending on the user's choice of runtime options made from the **File**, **Preferences**, **Model Execution** menu. Click **Yes** to close this window.

We will now add deposition sample data, re-save the run file and re-run the model. Move to the **Deposition** screen. We will now assume that approximately 3 hours after the initial explosion, after the majority of the material had been advected out of the modelling region of interest by the prevailing wind, two deposition samples were obtained at two separate locations downstream of the source; at 10.3km east, 2.7km south of the source at 14:38 with a measured deposition rate of 3260μg/m^2 and; at 18.8km east, 3.9km north of the source at 14:41 with a measured deposition rate of 2530μg/m^2.

Step 17 Enter this data in the **Ground deposition sample details** box, in the same manner that concentration data were entered in the **Concentration** screen. Compare your screen with **Figure 7.8**.

Sample locations have been entered in coordinates relative to the source (in km) to reflect the choice made in the **Sample & receptor locations** section of the **Setup** screen

Figure 7.8 - Example 1: the **Deposition** screen

Step 18 Select **File**, **Save As...** and save the file as *Ex1_Conc&Dep.hpl*. Then **Run** the model.

Remember that when deposition and air samples are both entered the deposition samples are used in preference to the air concentration measurements, as they represent a sample over a longer duration than an air sample. Thus in this run, the data in the **Concentration** screen will be ignored by the model.

7.1.2 Viewing output results

In ADMS-STAR, the results can be viewed in up to three ways, namely numerically, X-Y plots and contour plots. Go to the main **Results** menu at the top of the interface to see these three options.

Numerical results

There are a number of numerical output files created for each run. These numerical files can be viewed by selecting **Results, Output** from the menu and choosing the required file. A description of the information in each type of file is given in §6.2.

If sample data are used, the estimated source strength is given in the *.rec* file. Open *Ex1_ConcOnly.rec* and *Ex1_Conc&Dep.rec* using a text editor such as Notepad and compare the source strength estimated from concentration sample data with the source strength estimated from the deposition sample data.

Part of the output file *Ex1_Conc&Dep.glt* is shown in **Figure 7.9**. It has been opened in Excel and converted to columns using the 'Text to Columns' tool for ease of viewing. The user can then read off the accumulated deposition at the modelling end time at any particular grid point. Note that deposition is given for each particle size as well as the total amount for the modelled chemical.

Figure 7.9 - Example 1: *.glt* file viewed numerically

The X-Y plotting facility

The x-y plotting facility can be used to display results for runs modelling continuous releases, and is described in §6.3. No x-y plotting output files are produced for this explosive release example.

Contour Plotting

The gridded output results can also be viewed as contour plots over the output grid specified on the **Output** screen. The contour plots are produced in Surfer with the **ADMS Contour Plotter** utility (see full details on the utility in §6.4).
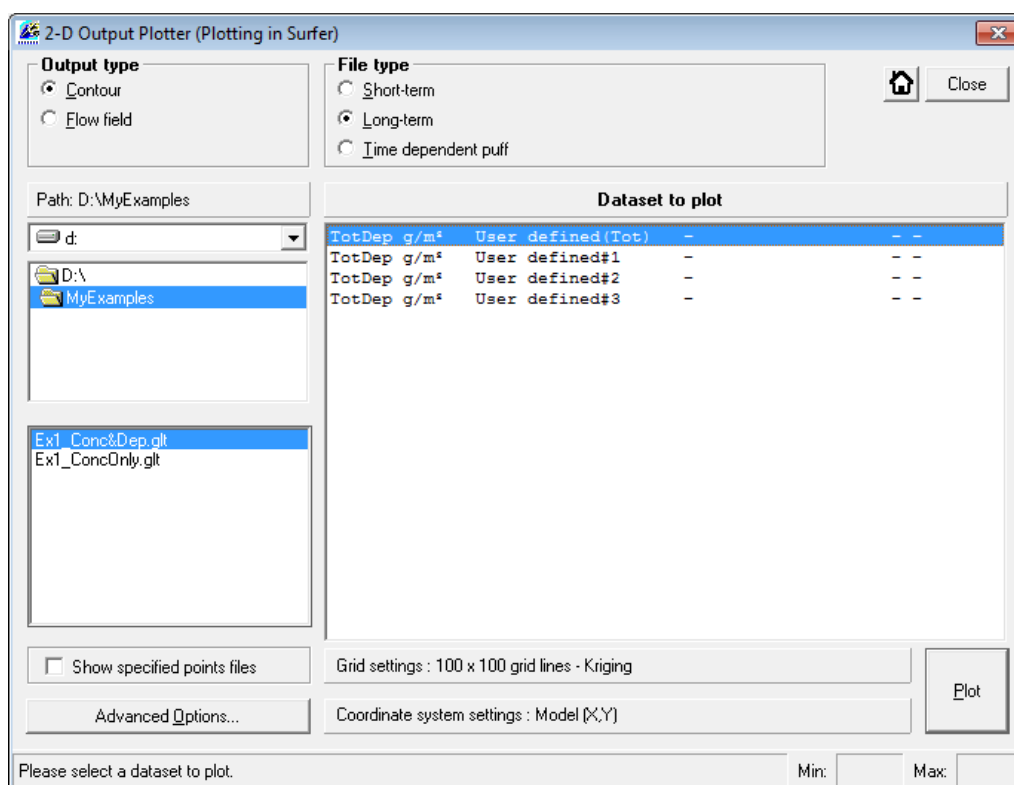
Figure 7.10 - Example 1: the **ADMS Contour Plotter** utility.

To create a contour plot of total accumulated deposition over all particle sizes for the run utilising deposition samples, proceed as follows.

Step 1 Go to **Results, Contour Plot in Surfer**... to launch the **ADMS Contour Plotter** utility. Select the **Long term** radio button to produce a screen like that shown in **Figure 7.10**.

Step 2 Browse to and select the file *Ex1_Conc&Dep.glt* and select "*TotDep g/m^2 User defined(Tot) - --*" as the dataset to plot.

Step 3 Click **Plot**.

This will first launch the **Save Surfer Grid File As**... dialogue box. Save with the default name, in this case *Ex1_Conc&Dep.grd*. Surfer is then launched and a contour plot of the total accumulated deposition created in a separate window. The title of the plot clearly indicates to which file and isotope the results correspond. The plot can be edited to show more contour levels, change colours, etc. **Figure 7.11** shows the contour plot with edited contour levels.
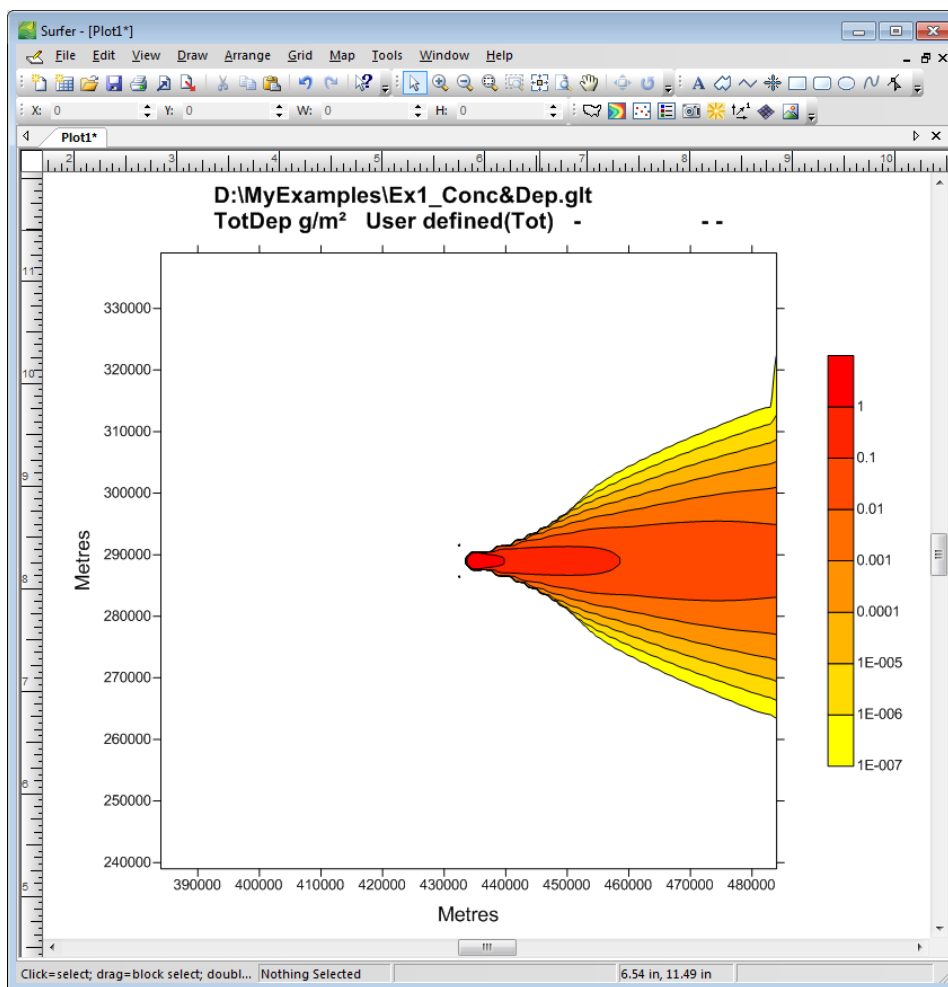
Figure 7.11 - Example 1: Total accumulated deposition contour plot in Surfer.

7.2   Example 2: Continuous release and MPL contours

The purpose of this example is to demonstrate how ADMS-STAR might be set up and run to model the continuous release of an isotope. In this example it will be assumed that the source strength is known. How to plot contours of MPLs (Maximum Permitted Levels) to allow comparison of estimated deposition with regulatory limits will be demonstrated.

7.2.1   Setting up the run

To set up this example, proceed as follows.

Step 1   Open ADMS-STAR and select **File**, **New** to create a new input file

Step 2   In the **Setup** screen, enter a **Site description** and **Model run description** of your choice.

In this example we will model a release in the UK during British Summer Time. Thus the number of hours offset from UTC to local time is +1 hour.

Step 3   Enter an **Offset from UTC** of 1 hour. Choose to define **Sample & receptor locations** in **Absolute** coordinates. Leave the rest of the screen as it is.

Step 4   In the **Source** screen, select the **Continuous release** radio button at the top of the screen and enter the source details as shown in **Figure 7.12**.

Step 5   In the **Define location** sub-screen (accessed by clicking the ✎ button) specify the longitude and latitude of the source as -1.8° and 53.2° respectively and the X and Y datum as 413km and 367km respectively.

Again, we are using the OS National Grid as the local Cartesian grid with which to reference the source location.
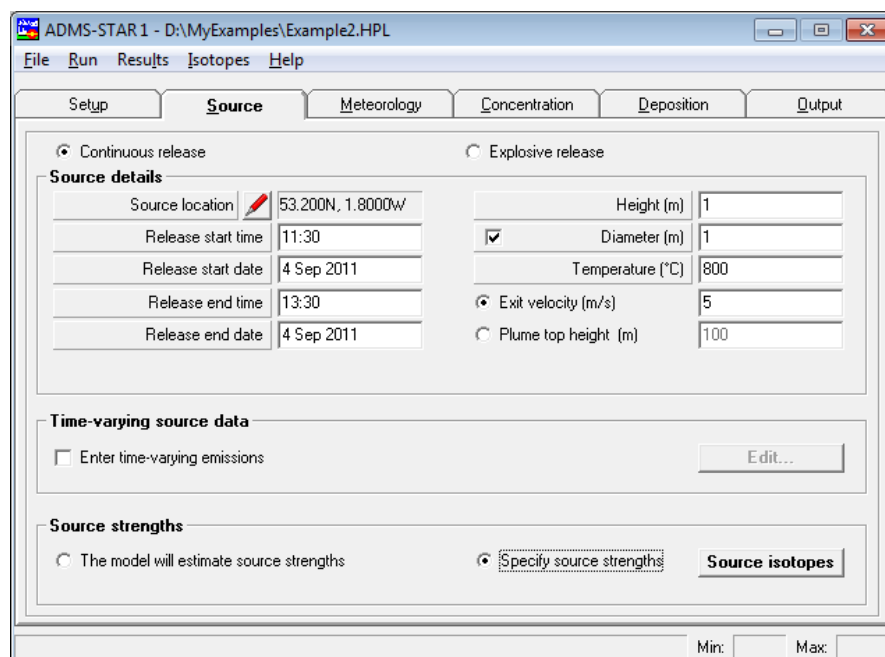
Figure 7.12 - Example 2: the **Source** screen

Step 6 Select the **Specify source strengths** radio button and click **Source isotopes** to display the sub-screen shown in **Figure 7.13**. Specify a source strength of $50x10^6$Bq/s for the isotope I-131 (radioiodine).

Figure 7.13 - Example 2: the **Source Isotopes** sub-screen

Step 7 Move to the **Meteorology** screen. Keep the default settings in the **Site data** box. Select **Spatially homogeneous met. data** and click **Edit...** to display the **Homogeneous met. data** sub-screen. Enter the met. data shown in **Figure 7.14**.

Note that the **Use boundary layer height** option box has been ticked and boundary layer height data has been entered directly in the table. This means that the model will not estimate the boundary layer height from the other variables.

Figure 7.14 - Example 2: the **Homogeneous met. data** sub-screen

Step 8 Move to the **Output** screen. Ensure that the **Use output grid** option is ticked.

Enter a modelling end time and date, output grid side length, number of grid lines and receptor point location as shown in **Figure 7.15**.

In this example, the receptor location is defined on the local Cartesian grid (i.e. OS National Grid) as the choice to use absolute coordinates was made in the **Setup** screen.

Figure 7.15 - Example 2: the **Output** screen

Step 9   Tick the **MPL contours** option and select the **Per Isotope** radio button. Click **Choose** to display the **Per Isotope Data** sub-screen (**Figure 7.16**).

Step 10   Ensure the **Choose a single isotope** radio button is selected and select I-131 from the drop-down list. In the **Choose 1 to 5 foodstuffs** box, tick **Leafy vegetables** and **Milk** only. Click **OK** to return to the main interface.

Figure 7.16 - Example 2: the **Per Isotope Data** sub-screen

Step 11 Select **File**, **Save As...** and save the file as *Example2.hpl*. Then **Run** the model.

7.2.2 Viewing output results

The X-Y plotting facility

As this is a continuous release, the X-Y plotting facility can be used to display results. Plume dispersion data are plotted along a line of points extending in the downstream direction from the source, i.e. along the 'plume centreline', as far as the release travels during the first met period.

Step 1 Go to **Results**, **X-Y Plot**... to launch the **ADMS Line Plotter** utility (**Figure 7.17**).
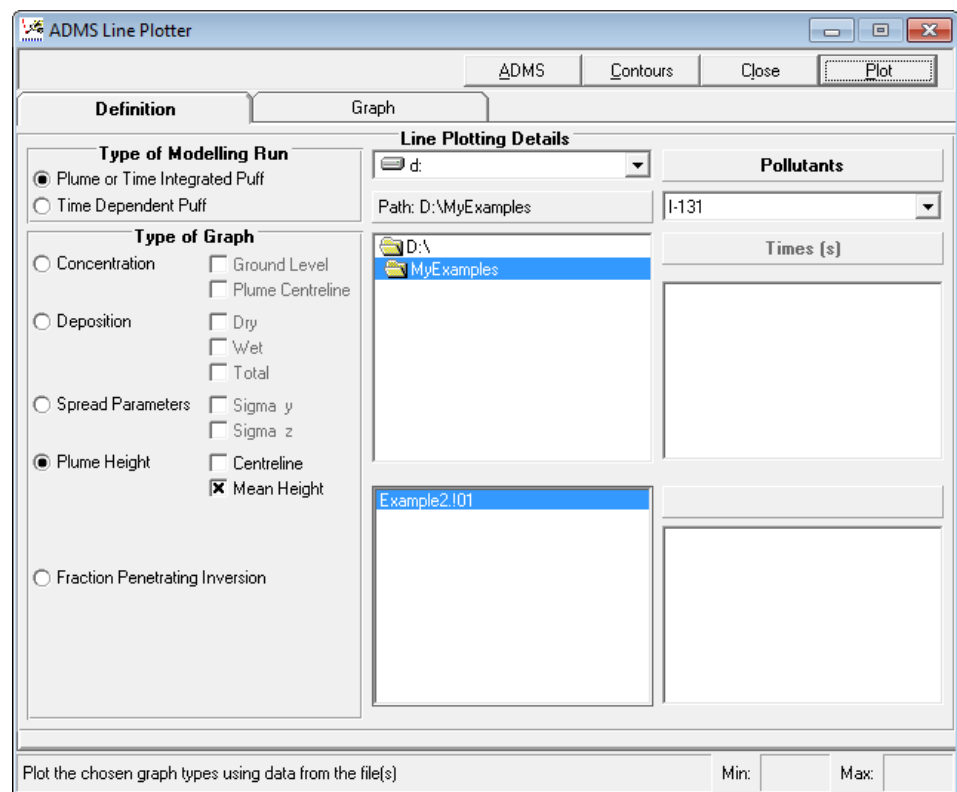
Figure 7.17 - Example 2: the **ADMS Line Plotter** facility.

Step 2 Ensure the **Plume or Time Integrated Puff** radio button is selected and use the boxes in the middle of the screen to browse to and select the file *Example2.!01*.

Step 3 Select a plume parameter from the **Type of Graph** section and click **Plot**.

> *The **Concentration** and **Deposition** options are not available with ADMS-STAR and are given as –999 at all downstream locations.*

Figure 7.18 shows the x-y plot of mean plume height.
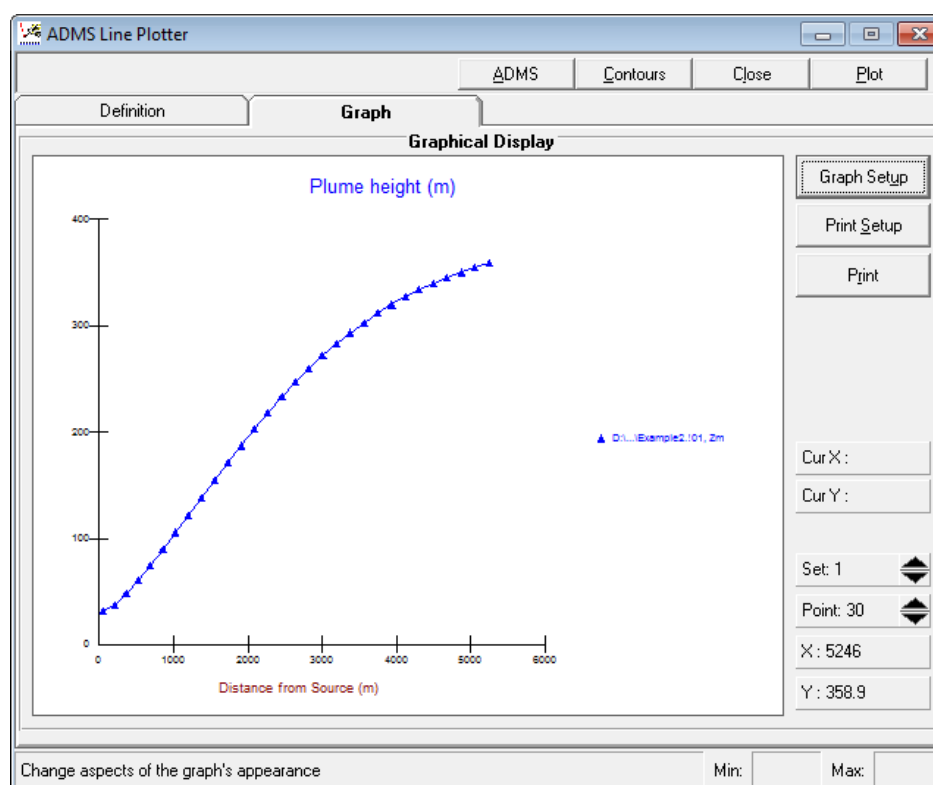
Figure 7.18 - Example 2: x-y plot of mean plume height

Contour Plotting

Step 1   Go to **Results, Contour Plot in Surfer**... and select **Long term**.

Step 2   Browse to and select the file *Example2.glt* and, in turn, plot the MPL deposition (`MPLDep`) dataset for Leafy Vegetables and for Milk.

For a particular MPL level, the data in the *.glt* file give the total accumulated deposition value divided by the MPL level. Therefore the MPL level is given by the contour with value 1. For each plot in turn:

Step 3   In **Surfer**, in the **General** tab of the **Properties Manager** un-tick **Fill Contours** (**Figure 7.19**). If you cannot see the **Properties Manager** go to **View, Managers, Property Manager**.
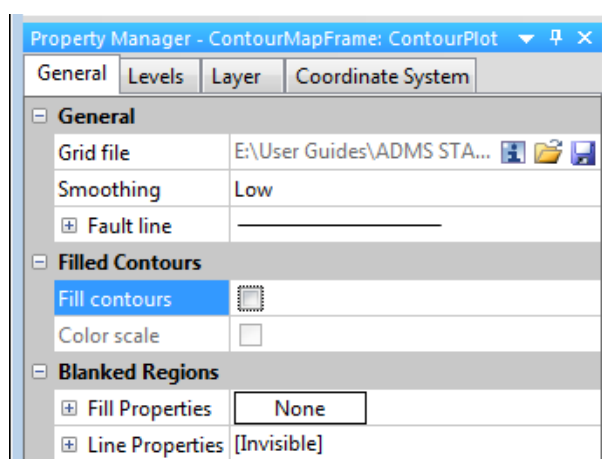
Figure 7.19 - Example 2: the **General** tab in the Surfer **Properties Manager** window

Step 4 In the **Levels** tab change the **Level Method** from **Simple** to **Advanced** and select **Edit Levels....** In the **Properties** window use the **Delete** button to delete all but one level. By double-clicking on each field in turn, set the **Level** of the contour to 1, choose an appropriate **Line** thickness and colour and set the contour **Fill** colour to None (**Figure 7.20**).
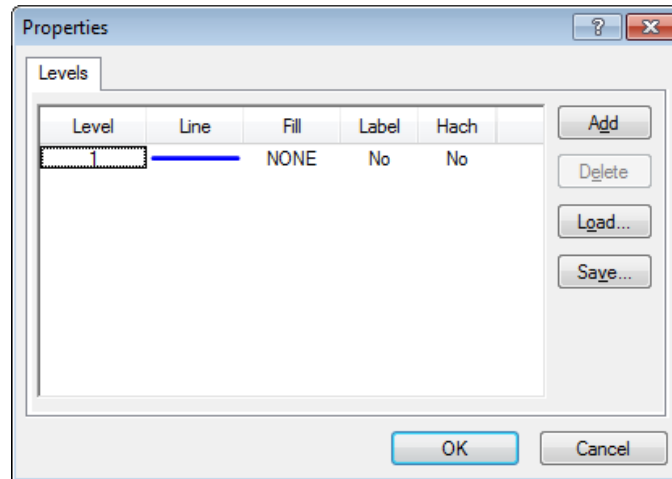
Figure 7.20 - Example 2: the Levels **Properties** window in the Surfer **Properties Manager**

Step 5 Overlay one contour plot on-top of the other. To do this copy and paste one contour plot into the same plot document as the other, select both plots by holding down Shift and clicking on each plot in turn, and select **Map**, **Overlay Maps** from the menu bar.

The plot should look similar to that shown in **Figure 7.21**. Plots of this type give a useful visualisation of area within which the deposition levels are predicted to exceed a given MPL.
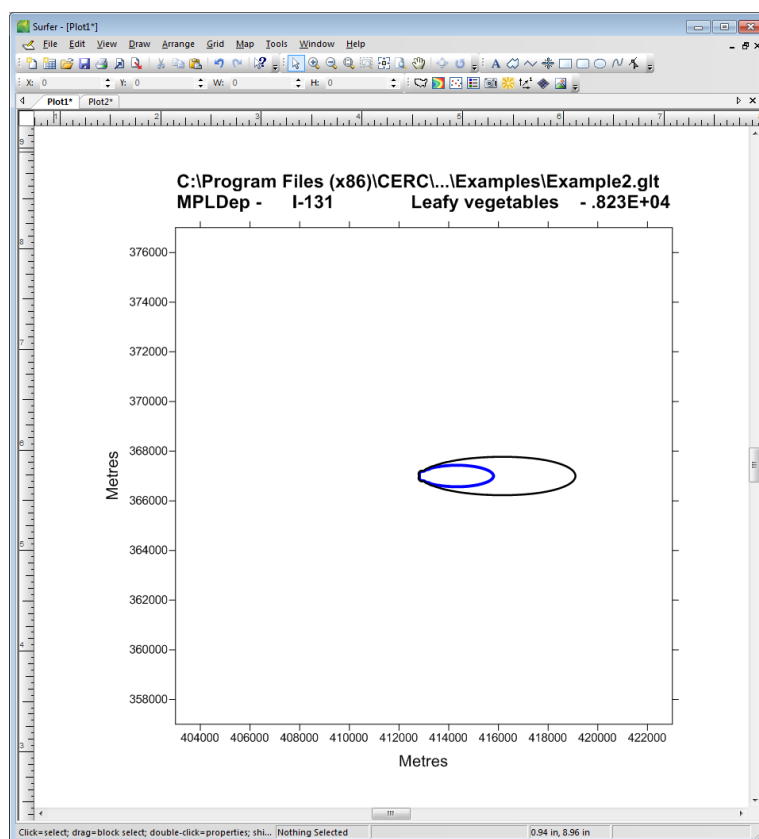
Figure 7.21 - Example 2: MPL contour for isotope I-131 for Leafy vegetables (blue line) and Milk (bslck line), plotted in Surfer.

7.3   Example 3: Spatially varying meteorological data

This example demonstrates the use of spatially varying meteorological data from output files of the WRF (Weather Research & Forecasting) model, a numerical mesoscale meteorological model.

7.3.1 Setting up the run

To set up this example, proceed as follows.

Step 1 Open ADMS-STAR and select **File**, **New** to create a new input file.

Step 2 In the **Setup** screen, enter a **Site description** and **Model run description** of your choice.

In this example we will model a release in the UK during British Summer Time. Thus the number of hours offset from UTC to local time is +1 hour.

Step 3 Enter an **Offset from UTC** of 1 hour. Change the **Unit of release** to grams. Leave the rest of the screen as it is.

Step 4 In the **Source** screen, choose to model a **Continuous release** and enter the source details as shown in **Figure 7.22**. Click the 🖊 button and locate the source at a longitude and latitude of -1.5° and 51.5° respectively and

an X, Y location of 435km and 178km respectively.

Step 5   Change the **Release start time** to 09:00 and the **Release start date** to 10 Jul 2008. Change the **Release end time** to 09:10 and the **Release end date** to 10 Jul 2008.

Figure 7.22 - Example 3: the **Source** screen

Step 6   Choose to **Specify source strengths** directly rather than estimate them from sample data. Click **Source isotopes** and in the resulting sub-screen enter a source strength for the 'User defined' chemical of 1000g/s and click **OK**.

Step 7   Select the **Palette** option from the **Isotopes** menu to display the **Isotope palette** sub-screen. For the 'User defined' isotope, change the **Half life units** to 'Stable' (the **Half life** is then ignored). Click **OK** to return to the **Source** screen.

Step 8   Move to the **Meteorology** screen. Keep the default settings in the **Site data** box. Select **Spatially varying met. data from files** and click **Edit...** to display the **Spatially varying met. data** sub-screen. Click **Add** to add a new line of data to the table and click the drop-down arrow in the **Filename** field to display a window that allows you to browse to a data file.

Step 9   Browse to the *Data* sub-directory of your ADMS-STAR install directory, select the WRF output file named *wrfout_d02_2008-07-10_08_00_00.nc* and click **Open** to add it to the table. Add further data files to the table, *wrfout_d02_2008-07-10_11_00_00.nc* and *wrfout_d02_2008-07-10_14_00_00.nc*, both of which are located in the same directory.

These WRF output files contain spatially varying met data over a grid covering the UK with horizontal resolution 9km. The date of the file is 10 Jul 2008 and the time of each file is 08:00, 11:00 and 14:00 respectively. The time parameter in a WRF output file is always given in UTC. The time often appears in the file name. Thus in BST the time of each file is 09:00, 12:00 and 15:00 respectively.

Step 10 Enter the **Local date and time** for each file as shown in **Figure 7.23**.

*When ADMS-STAR is run (in verification or full calculation mode), the model checks that the difference between the time in a WRF file and the local time as specified by the user in the **Local date and time** column of the **Spatially varying met. data** screen is consistent with the offset from UTC entered in the **Sample & receptor locations** section of the **Setup** screen. An error message will be given if this check fails.*

Figure 7.23 - Example 3: the **Spatially varying met. data** sub-screen

Step 11 Tick the **Use boundary layer height from file** option. This means that the model will use the boundary layer height data contained in the WRF files directly rather than estimate its value from other variables. Click **OK** to return to the **Meteorology** screen.

Step 12 Tick the **Use observed data for the source location** box and click **Edit...** to display the **Observed met. data at the source location** sub-screen.

This option allows the user to enter values of some meteorological variables at the source, possibly recorded by a weather station at, or close to, the source in order to improve the calculations. The site observed data are applied at the source and then relax back to the WRF meteorological data over a relaxation radius. The values for these relaxation radii are entered in the **Effective range of observed met. data** section.

Step 13 Click **Add** to add a new line of data to the table and enter the values as shown in **Figure 7.24**. Set the **Height of recorded wind** to 10m. Leave the horizontal and vertical radius values as they are. Click **OK.**

Figure 7.24 - Example 3: the **Observed met. data at the source location** sub-screen

Step 14 Move to the **Output** screen. Enter a modelling end time and date of 15:00 and 10 Jul 2008 respectively. Select to **Use output grid** and specify a **Side length** of 500km and the **Number of grid lines** as 10101. Add a receptor point in coordinates relative to the source so that the model will run.

Step 15 Select **File**, **Save As...** and save the file as *Example3.hpl*. Then **Run** the model.

7.3.2 Viewing output results

Contour Plotting

For this example, we will use ArcMap to plot contours of the total accumulated deposition. However Surfer could also be used if preferred. APPENDIX A gives a brief introduction to ArcMap and a detailed description of the various functionalities of the ADMS-STAR–ArcGIS link.

Step 1 Launch ArcMap and open a new empty map. Ensure that the ADMS-STAR toolbar is displayed.

Step 2 Click the **Generate Contours** button from the ADMS-STAR toolbar to display the **ADMS Contour Plotter** utility.

Step 3 Browse to and select the file *Example3.glt*, select the dataset for the total accumulated deposition and click **Plot**.

This will first launch the **Save Output Raster** dialogue box. Save with the default name. The contour plot of the total accumulated deposition is then added to the plot window as a new layer. This layer can be edited to show more contour levels, change colours,

etc. **Figure 7.25** shows an edited contour plot. A *.shp* file showing a base map of the outline of the UK has also been added.

In this case, the spatially varying meteorology has caused the trajectory of the released material to curve slightly as it moves downstream. There is also an area of higher values downstream of the source, which is the result of increased wet deposition caused by higher precipitation in that region.
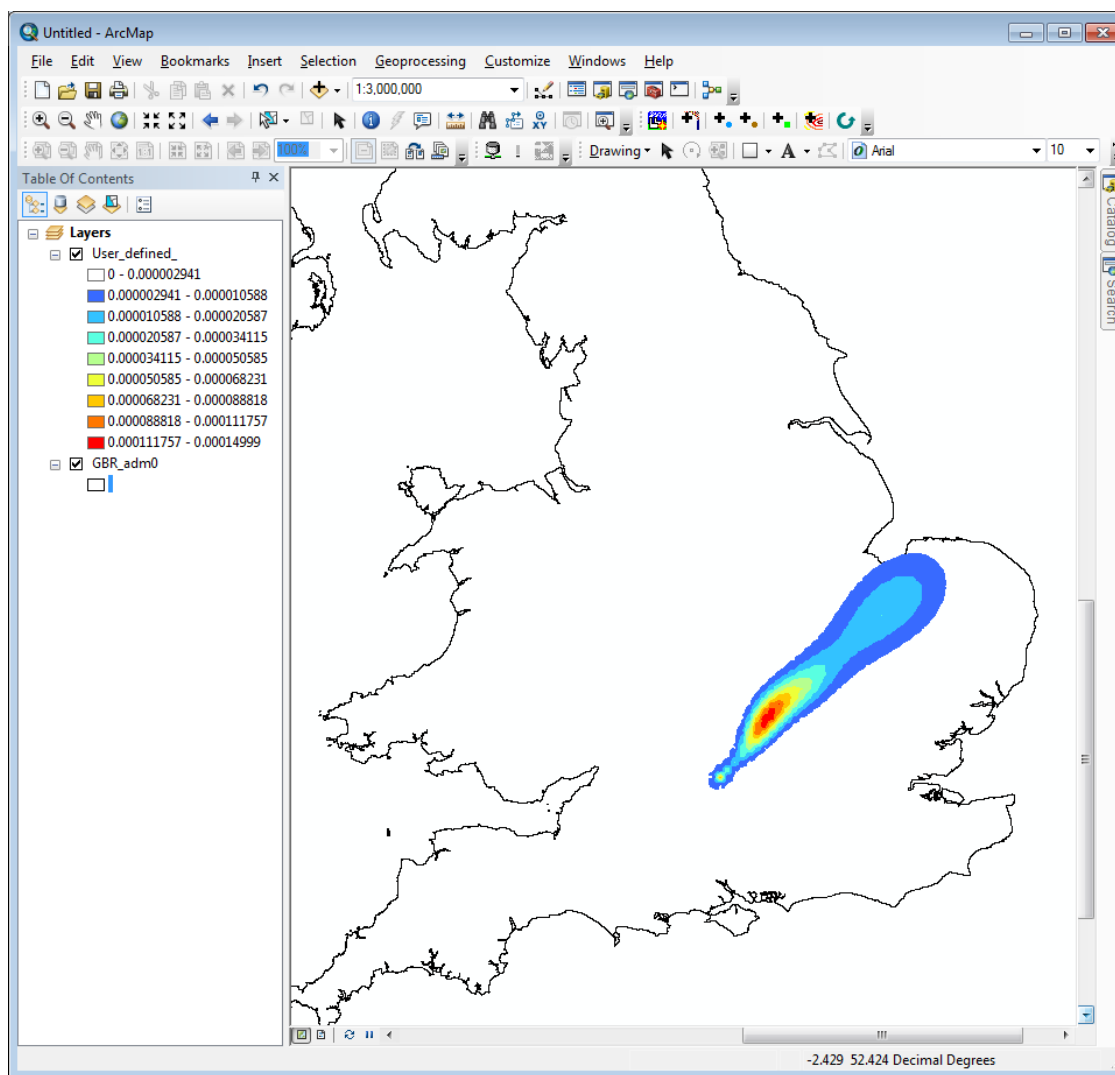
Figure 7.25 - Example 3: Total accumulated deposition contour plot in ArcMap.

7.4 Example 4: Using the ADMS-STAR–ArcGIS link

The purpose of this example is to demonstrate how the ADMS-STAR–ArcGIS link can be used to assist in setting up an ADMS-STAR run and to plot results. If you do not have ArcGIS installed on your computer, it is not necessary to work through this example.

If you are unfamiliar with ArcMap and the ArcGIS - ADMS-STAR link, then you should read APPENDIX A which gives a brief introduction to ArcMap and a detailed description of the various functionalities of the ADMS-STAR–ArcGIS link.

7.4.1 Setting up the run

To set up this example, proceed as follows.

Step 1 Launch ArcMap and open a new empty map.

Step 2 Ensure that the ADMS-STAR toolbar is displayed and open the ADMS-STAR interface by clicking the **Show ADMS-STAR** button.

Step 3 In the **Setup** screen, enter a **Site description** and **Model run description** of your choice. Enter an **Offset from UTC** of 1 hour. Leave the rest of the screen as it is.

Step 4 In ArcMap, click the **Add Data** button. Use the browser to browse to the *Data* sub-directory of your ADMS-STAR install directory, select the file *TL.tif* and click **Add**.

If the following warning message appears (**Figure 7.26**), click **OK** to continue. This message appears because the data do not include information about the conversion from the projected to the geographic coordinate system. This is not important, since we will always be working in the projected system.
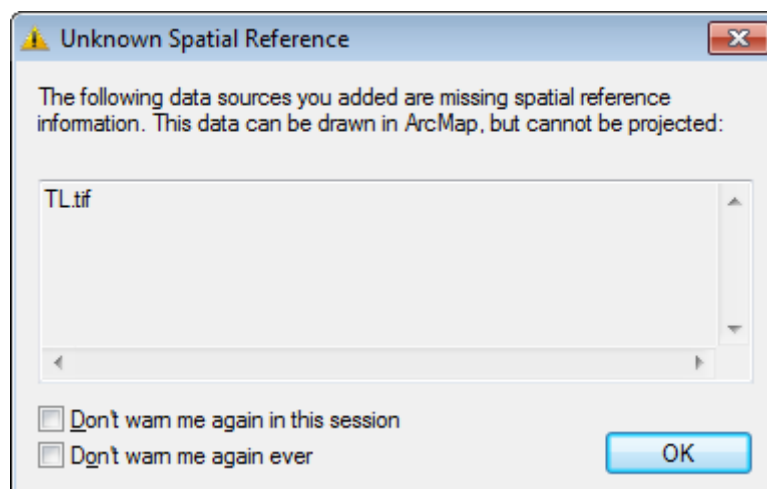
Figure 7.26 - Example 4: Warning message in ArcMap.

A new layer should now be visible showing a UK Ordnance Survey map

tile[5], which is 100km x 100km in size (**Figure 7.27**). If the whole map is not visible, click the **Full Extent** 🌐 tool, or right click on the name of the layer (i.e. *TL.tif*) and select **Zoom to Layer**.
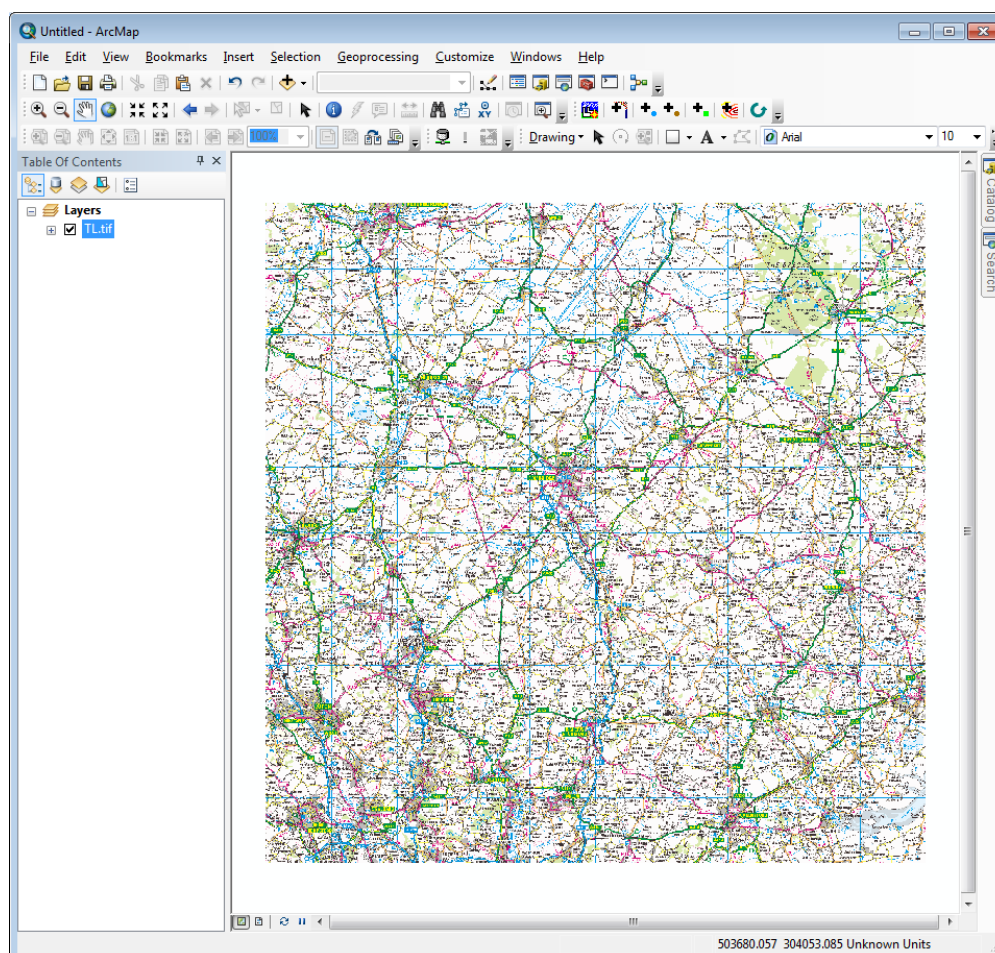
Figure 7.27 - Example 4: Base map in ArcMap.

As you move the cursor over the map, the coordinates and units are displayed in the status bar at the bottom of the ArcGIS window. ArcMap uses the data in the file *TL.tfw* (which is located in the same directory as *TL.tif*) to set the map coordinates. The units are in metres (as required by the ADMS-STAR–ArcGIS link) but this information has not been given with the file so ArcMap displays the units as *Unknown*. The units will automatically be set to metres when the update 🔄 button is clicked from the ADMS-STAR toolbar or the source is located using the ➕ button.

Step 5   Double click the layer *TL.tif* in the left hand pane to display the **Layer Properties** window. From the **Display** tab, increase the **Transparency** to 50%, say, and click **OK** to return to the plot window.

Increasing the transparency of the base map makes it easier to see the symbols used for the source, sample and receptor locations.

Step 6   Zoom in on the area of the map shown in **Figure 7.28** using the **Zoom In** 🔍 tool. Click the **Define the source datum** ➕ button from the ADMS-

---

[5] Contains Ordinance Survey data © Crown copyright and database right 2012

STAR toolbar and then click on the map to add the source at a similar location to that shown in **Figure 7.28**.

The source will be added to the plot window as a new layer called *ADMS-STAR Source*. The source will be displayed on the map as a red dot.

The source X and Y datum will automatically be updated in the ADMS-STAR interface on the **Define location** sub-screen of the **Source** screen. The existing source datum will be overwritten. The coordinates are automatically converted from metres to kilometres.

The source can be relocated by either re-using the **Define the source datum** *tool in ArcMap or by modifying the source X and Y datum directly in the ADMS-STAR interface and clicking the update* ↻ *button on the ADMS-STAR toolbar in ArcMap.*
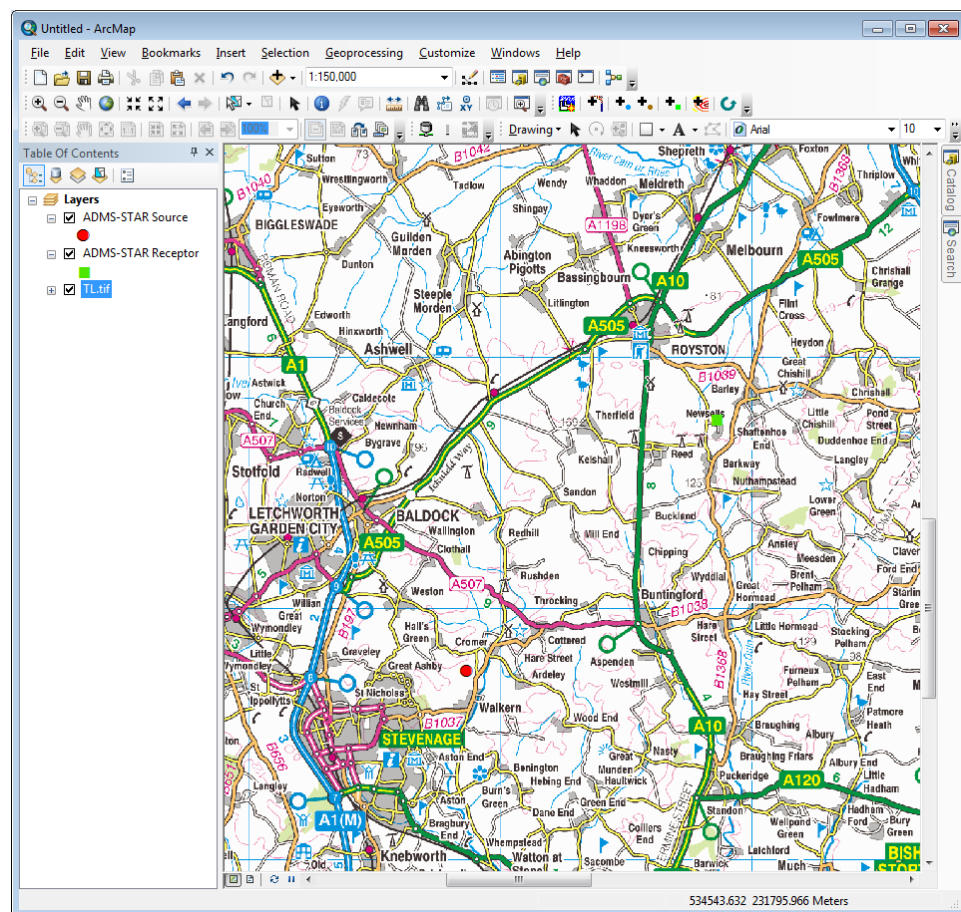
Figure 7.28 - Example 4: defining the source location in ArcMap. Source location shown by a red dot.

Step 7 In the ADMS-STAR interface, click the button from the **Source** screen to display the **Define location** sub-screen. Enter the longitude and latitude of the source location in decimal degrees.

The ADMS-STAR–ArcGIS link *does not* automatically set the source longitude-latitude as it does not know the local coordinate system being used and so cannot apply the appropriate conversion. Typically, the source

longitude-latitude will either be known or can be obtained from the source X and Y datum using a coordinate conversion calculation. Online conversion tools exist for many of the world's commonly used coordinate systems. The longitude and latitude of the source shown in **Figure 7.28** are 0.208152° and 52.147569° respectively.

*It is important that the source longitude and latitude are entered correctly as this information is used in the calculation of incoming solar radiation, a key parameter in determining the boundary layer meteorology. If the **Spatially varying met. data from files** option is used, the source longitude and latitude are also used to locate the source on the met grid.*

Step 8 From the **Source** screen, choose to model an **Explosive release** and set the source details and cloud properties to those shown in **Figure 7.29**. Select the option to have the model estimate source strengths.

Figure 7.29 - Example 4: the **Source** screen

Step 9 Move to the **Meteorology** screen. Choose the **Spatially varying met. data from files** option and click **Edit...** to display the **Spatially varying met. data** sub-screen. In turn, use the **Add** button and the drop-down arrow in the **Filename** field to add the WRF output files *wrfout_d02_2008-07-10_08_00_00.nc*, *wrfout_d02_2008-07-10_11_00_00.nc* and *wrfout_d02_2008-07-10_14_00_00.nc* to the table. All three files can be found in the *Data* sub-directory of your ADMS-STAR install directory. Enter the **Local date and time** for each file as shown in **Figure 7.30**. Tick the **Use boundary layer height from file** option. Click **OK** to return to the **Meteorology** screen.
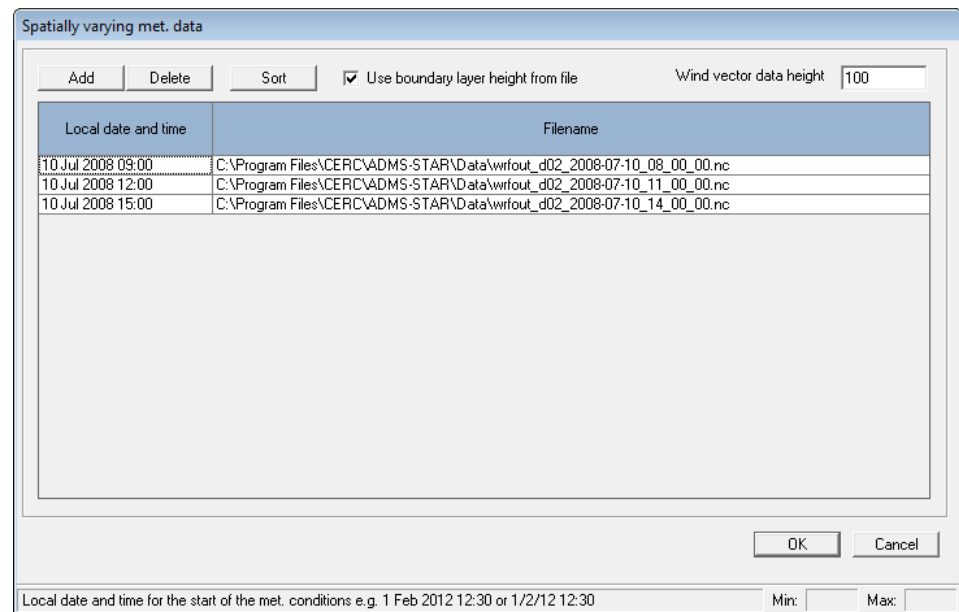
Figure 7.30 - Example 4: the **Spatially varying met. data** sub-screen

Step 10 In ArcMap, click the **Add a deposition sample** ✚• button from the ADMS-
STAR toolbar. Click on the map to add a sample at a similar location to
Sample 1 shown in **Figure 7.31**.

This will display the **Deposition** screen of the ADMS-STAR interface. The
coordinates of the sample location will automatically be converted from
metres to kilometres and added to the **X (km)** and **Y (km)** fields. They will be
relative to the source to reflect the choice made in the **Sample & receptor
locations** section of the **Setup** screen.
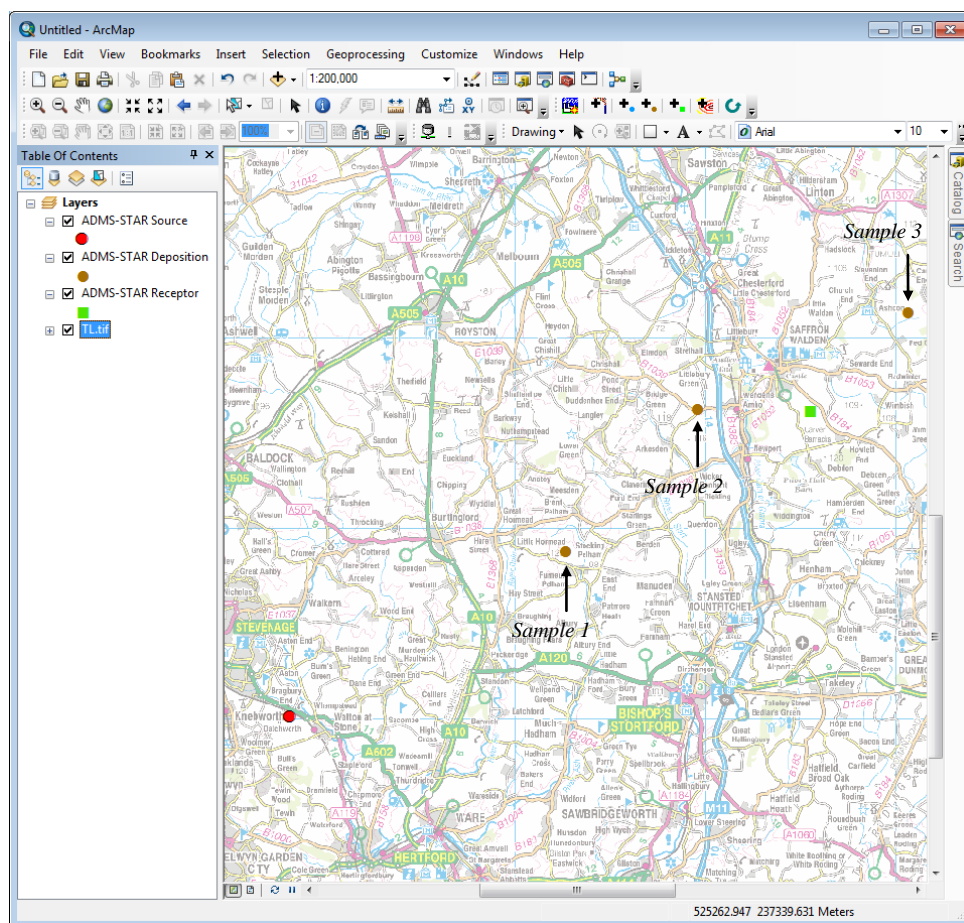
Figure 7.31 - Example 4: defining the deposition sample locations in ArcMap. Sample location shown by brown dots and labelled.

Step 11 Define the sample **Date** and **Time** as 10 Jul 2008 and 12:20, respectively, and set the sample strength as 39900Bq/m^2 for isotope I-131. Use the **Add** button to the right of the large sample table to add the sample.

Step 12 Return to ArcMap and click the update ↻ button from the ADMS-STAR toolbar. The sample location should now be visible as a brown dot.

Step 13 Repeat Step 10 to Step 12 for the next two sample locations shown in **Figure 7.31**. The date, time and strength of each of these two samples are given in the table below:

Sample	**Date**	**Time**	**Strength (Bq/m^2) for isotope I-131**
Sample 2	10 Jul 2008	12:25	14200
Sample 3	10 Jul 2008	12:15	19200

Table 7.1 - Example 4: deposition sample information

Step 14 In ArcMap, click the **Add a receptor point** ✚ button from the ADMS-STAR toolbar. Click on the map to add a receptor point at a location of your choice.

This will display the **Output** screen of the ADMS-STAR interface. The coordinates of the sample location will automatically be converted from

metres to kilometres and added to the upper boxes in the **Receptor points** field. They will be relative to the source to reflect the choice made in the **Sample & receptor locations** section of the **Setup** screen.

Step 15 Click **Add** next to the receptor points table to add the receptor location to the table.

Step 16 Return to ArcMap and click the update ↻ button from the ADMS-STAR toolbar. The receptor location should now be visible as a green square.

Step 17 Return to the **Output** screen of the ADMS-STAR interface. Enter a modelling end time and date of 15:00 and 10 Jul 2008 respectively. Select to **Use output grid** and specify a **Side length** of 100km and the **Number of grid lines** as 101.

Step 18 Select **File**, **Save As...** and save the file as *Example4.hpl*. Then **Run** the model.

7.4.2 Viewing output results

Contour Plotting

Step 1 In ArcMap, click the **Generate Contours** button from the ADMS-STAR toolbar to display the **ADMS Contour Plotter** utility.

Step 2 Browse to and select the file *Example4.glt*, select the dataset for the total accumulated deposition and click **Plot**.

This will first launch the **Save Output Raster** dialogue box. Save with the default name. The contour plot of the total accumulated deposition is then added to the plot window as a new layer. This layer can be edited to show more contour levels, change colours etc.

Step 3 Double click the colour box of the lowest contour level the left hand pane to display the **Symbol Selector** window. Change the contour level colour to *Hollow* and click **OK** to return to the plot window.

You can now clearly see the map where the activity was either zero or very low.

Step 4 Double click the layer containing all the contour levels in the left hand pane to display the **Layer Properties** window. From the **Display** tab, set the **Transparency** to 25% and click **OK** to return to the plot window.

The plot should now look similar to that shown in **Figure 7.32**.
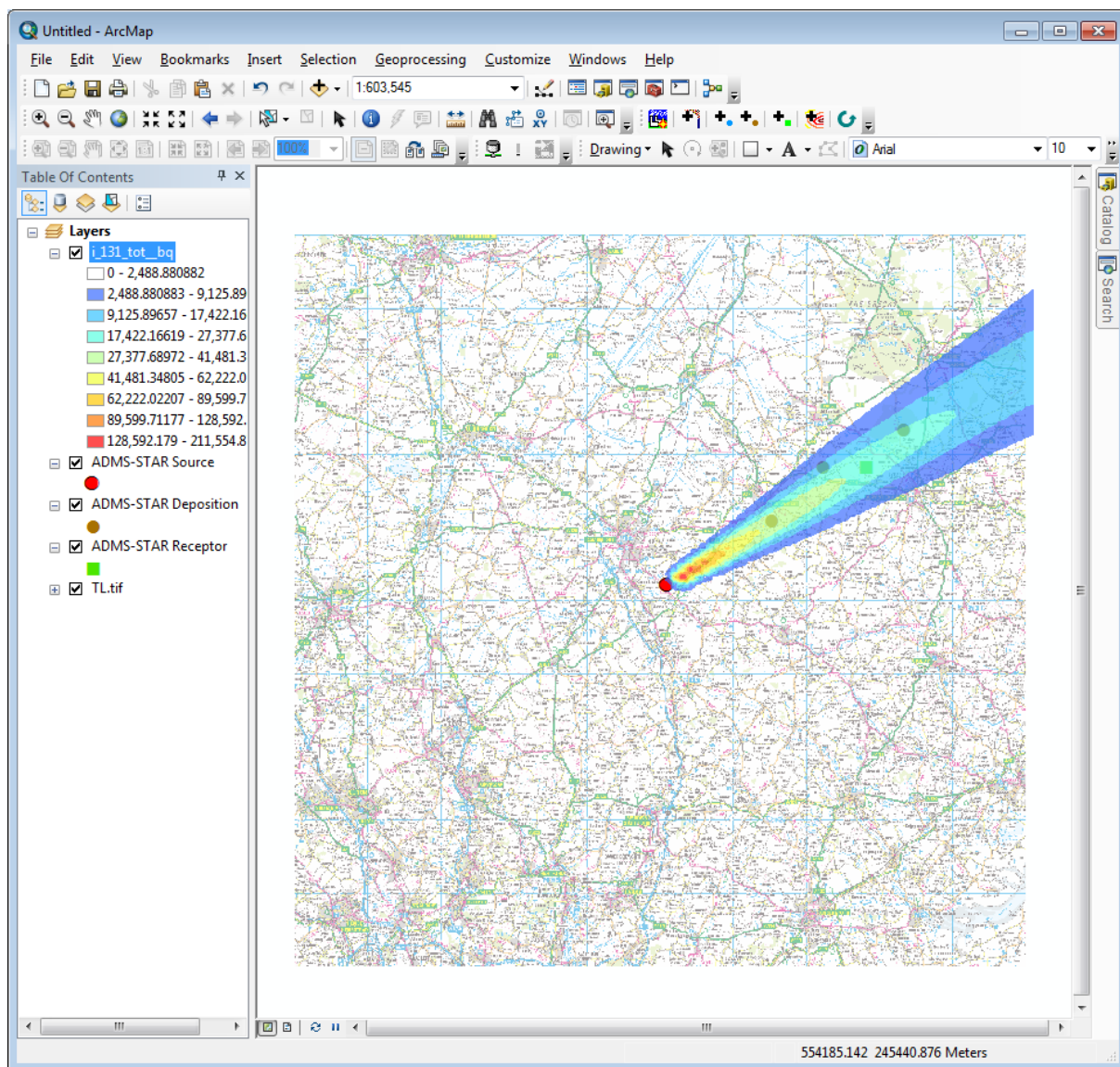
Figure 7.32 - Example 4: total accumulated deposition contour plot in ArcMap.

SECTION 8 TECHNICAL SUMMARY

This section is intended to provide a summary of the mathematical and physical background to ADMS-STAR and is an abbreviated form of the ADMS Technical Specification documentation (CERC, 2016), to which readers should refer for full details. In this Technical Summary, the following topics are covered:

(1) Met input;

(2) Parameterisation of the boundary layer;

(3) Puff dispersion model;

(4) Dry deposition;

(5) Wet deposition;

(6) Radioactive decay;

(7) Spatially varying meteorology;

(8) Complex terrain;

(9) Marine boundary layer;

(10) Inhalation dose and thyroid dose;

(11) Gamma dose;

(12) Puff distribution;

(13) Use of sample measurements;

(14) Isotope properties and groups.

8.1   Met input module

8.1.1   Introduction

The Met Input module reads the data and uses pre-processing algorithms to estimate values of the various meteorological quantities required for running the dispersion model.

8.1.2   Input parameters

The input data consists of wind speed, direction, cloud cover, time of day, time of year, temperature, precipitation rate and, optionally, boundary layer depth.

Note that Pasquill-Gifford stability categories cannot be directly input into the model, nor are they output. In ADMS-STAR the boundary structure is characterised by the two parameters, boundary layer height, h, and Monin-Obukhov length, L_{MO}. Values of these parameters corresponding approximately to the Pasquill-Gifford categories are shown in **Table 8.1**.

U (m/s)	L_{MO} (m)	$1/L_{MO}$ (m^{-1})	h (m)	h/L_{MO}	P-G Category
1	-2	-0.5	1300	-650	A
2	-10	-0.1	900	-90	B
5	-100	-0.01	850	-8.5	C
5	∞	0	800	0	D
3	100	0.01	400	4	E
2	20	0.05	100	5	F
1	5	0.2	100	20	G

Table 8.1 - Values of wind speed, Monin-Obukhov length (L_{MO}) and boundary layer height (h) which may be used to represent Pasquill-Gifford categories A-G.

There is no exact correspondence between the boundary layer parameters (h, L_{MO}) and the Pasquill-Gifford categories since many different values of h and L_{MO} may correspond to one Pasquill-Gifford category.

Stability: Stable $h/L_{MO} > 1$

                    Neutral $-0.3 \leq h/L_{MO} \leq 1$

                    Convective $h/L_{MO} < -0.3$

In general, the boundary layer depth should always be entered if you think you can provide a better estimate than the met input module algorithms.

In addition to the data in the met input data, the module also requires a surface roughness which should be entered on the **Meteorology** screen.

Further parameters may, optionally, be specified via the interface. These include a minimum value of L_{MO} (for urban areas) and default values of modified Priestley-Taylor parameter (which apportions the surface sensible and latent heat fluxes) and surface albedo.

If the met site is distant from the area of dispersion, the met input module includes the option to modify the wind profile at the source by taking account of the surface roughness both at the met site and the source. It is also possible to enter a precipitation factor to account for differences in precipitation between the sites (described in Section 5.1.4).

8.1.3 Met pre-processing

The module is called once for each met condition and uses ADMS's standard algorithms to calculate the boundary layer meteorological parameters required by the dispersion model. Full details can be found in Holtslag and van Ulden (1983) and the ADMS 5 Technical Specification (CERC, 2016). Details of how this is done for spatially varying meteorology read in from WRF files is given in Section 8.7.

8.1.4 Limitations

In calculating the boundary layer parameters, it is assumed that the boundary layer is self-similar for a given value of h/L_{MO}. However, users of the model should be aware that there are some situations, such as latitudes near the equator or the poles, where the approximation can lead to significant errors. Full details are given in the ADMS 5 Technical Specification (CERC 2016).

8.2 Parameterisation of the boundary layer

In ADMS-STAR the boundary layer is characterised by the boundary layer height h and the Monin-Obukhov length L_{MO} and not by a Pasquill-Gifford stability category. The Monin-Obukhov length is defined as

$$(8.1) \qquad L_{MO} = \frac{-u_*^3}{\kappa g F_{\theta_0} / (\rho c_p T_0)}$$

in which u_* is the friction velocity at the Earth's surface, κ (= 0.4) is the von Karman constant, g is the acceleration due to gravity, F_{θ_0} is the surface sensible heat flux, ρ and c_p are, respectively, the density and specific heat capacity of air and T_0 is the near surface temperature.

In unstable or convective atmospheric conditions, the Monin-Obukhov length is negative. Then, the magnitude of the length is a measure of the height above which convective turbulence, i.e. turbulent motions caused by thermal convection, is more important than mechanical turbulence, generated by friction at the Earth's surface.

In stable conditions the Monin-Obukhov length is positive. Then it is a measure of the height above which vertical turbulent motion is significantly inhibited by the stable stratification.

Figure 8.1 shows the different regions of the boundary layer in terms of the parameters h/L_{MO} and z/h where z is height above the ground. **Figure 8.2** shows the same information but with a dimensional vertical scale, z, the height above the ground. In the different regions of the boundary layer different mechanisms are important in generating turbulence. These are:

1. Surface heating or, convectively generated turbulence. The convective eddies increase in energy as they rise through the boundary layer.

2. Turbulence mechanically generated by shearing at the surface.

3. Local shear, for instance at the top of the boundary layer, which can be a weak source of turbulence.

This approach to boundary layer stability, whereby the boundary layer structure is defined in terms of two variables, z/L_{MO} and z/h, supersedes the Pasquill-Gifford formulation, and differs crucially from the Pasquill formulation in allowing the variation of boundary layer properties with height to be included. It is difficult to make exact comparisons between the two schemes, however, and **Figures 8.1** and **8.2** show Pasquill-Gifford stability categories corresponding approximately to ranges of h/L_{MO}. Note that, particularly in stable met conditions, the Pasquill class is not a simple function of h/L_{MO}.
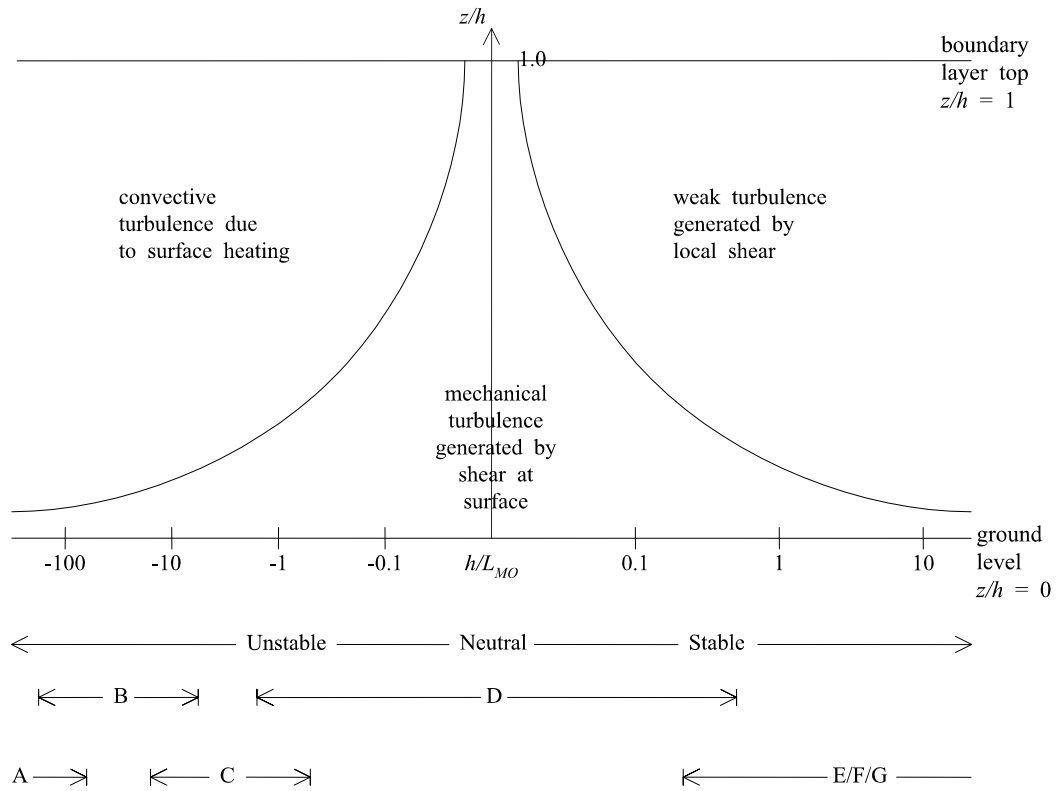
Figure 8.1 - Non-dimensional schematic representation of variation of Monin-Obukhov length with atmospheric stability.
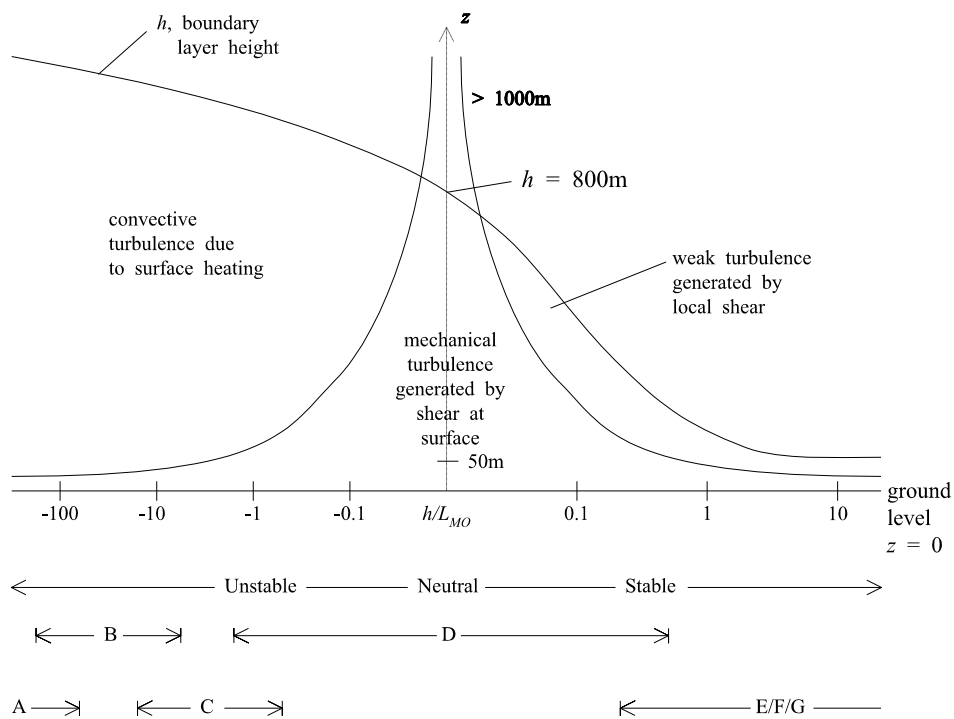
Figure 8.2 - Dimensional schematic representation of variation of Monin-Obukhov length with atmospheric stability.

8.2.1 Boundary layer structure

ADMS calculates boundary layer variables, listed in **Table 8.2**, at different heights. Vertical profiles are expressed as functions of z/L_{MO} and z/h and have been derived from experimental data (Caughey and Palmer, 1979; van Ulden and Holtslag, 1985; Hunt et al., 1988).   These variables are used, in turn, by other modules.

Variable	Description and units
$U(z), \dfrac{dU}{dz}, \dfrac{d^2U}{dz^2}$	Mean wind speed (m/s) and its first (/s) and second derivatives (/s^2) with height above ground
$\sigma_u(z), \sigma_v(z), \sigma_w(z)$	Root mean square (r.m.s.) turbulent velocities (m/s)
$\Lambda_v(z), \Lambda_w(z)$	Turbulent length scales (m)
$\varepsilon(z)$	Energy dissipation rate (m^2/s^3)
$T_L(z)$	Lagrangian time scale (s)
$N(z)$	Buoyancy frequency (/s)
$T(z)$	Temperature (K)
$\rho(z)$	Density (kg/m^3)

Table 8.2 Boundary layer variables calculated by ADMS.

Note that the puff spread parameters σ_x, σ_y and σ_z are calculated using these boundary layer variables and hence vary with source height and plume height. This contrasts with the approach adopted in models using Pasquill categories, in which values of σ_x, σ_y and σ_z are obtained from measured profiles of σ_x, σ_y and σ_z and are independent of plume height.

8.3 Puff dispersion model

8.3.1 Overview

ADMS-STAR uses a puff model to model the dispersion of the release. The release is characterised by a set of instantaneous puffs, which may increase in number over time as the release continues. These puffs are advected independently by the local meteorological conditions. The activity/concentration field at any particular time is then given by the combination of the activity/concentration fields from each individual puff at that time.

The instantaneous puffs are characterised by their position and size, given as spread parameters. These properties are updated on a time scale shorter than that on which the meteorology changes. As the puff's properties are updated at each time step, the individual puff's history is taken into account. This history will be different for each puff.

8.3.2 Concentration

Each individual puff is described by its centre position (x_c, y_c, z_p) and spread parameters σ_x, σ_y, σ_z. The activity/concentration distribution for each individual puff is taken to be Gaussian in the along-wind and cross-wind directions and Gaussian or skewed-Gaussian in the vertical. **Figure 8.3** shows a representation of an individual puff.
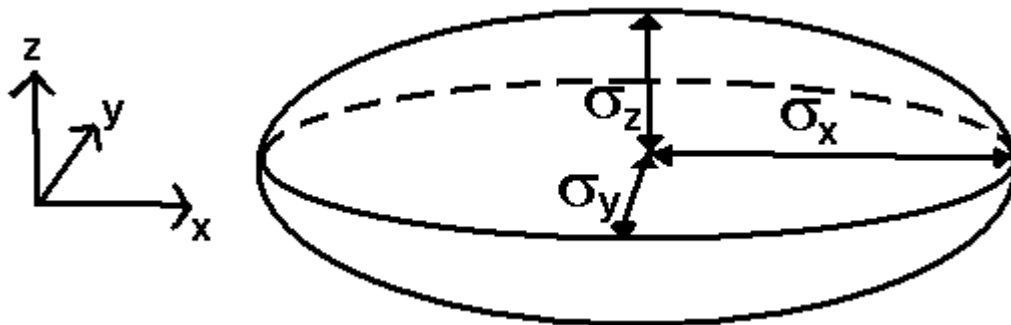
Figure 8.3 - Diagrammatic representation of a puff showing the spread parameters

The activity/concentration at a point due to this puff is then given by

$$C(x, y, z, t) = \frac{M_s}{(2\pi)^{3/2} \sigma_x \sigma_y \sigma_z} \exp\left(\frac{-(x - x_c)^2}{2\sigma_x^2}\right) \exp\left(\frac{-(y - y_c)^2}{2\sigma_y^2}\right)$$

(8.2)
$$\left\{ \exp\left(\frac{-(z - z_p)^2}{2\sigma_z^2}\right) + \exp\left(\frac{-(z + z_p)^2}{2\sigma_z^2}\right) + \exp\left(\frac{-(z - 2h + z_p)^2}{2\sigma_z^2}\right)\right.$$

$$\left. + \exp\left(\frac{-(z + 2h - z_p)^2}{2\sigma_z^2}\right) + \exp\left(\frac{-(z - 2h - z_p)^2}{2\sigma_z^2}\right)\right\}$$

in neutral and stable conditions in the presence of an inversion, where here M_s is the total mass in the puff. For convective conditions, stable conditions with no inversion or for a puff above the boundary layer the term in curly brackets is replaced with each appropriate vertical term for a plume in ADMS (for instance in convective conditions a skewed Gaussian form is used), see the ADMS 5 Technical Specification (CERC 2016).

8.3.3 Advection

The puff properties are updated based on the meteorological conditions at the position of the puff at that time at the mean puff height. The mean puff height z_m is calculated in the presence of an inversion as follows

(8.3)
$$z_m(t) = \frac{\int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \int_0^h z\, C(x, y, z, t)\, dz\, dy\, dx}{\int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \int_0^h C(x, y, z, t)\, dz\, dy\, dx}$$

the upper limit of the integral in z is h since material passing above the boundary layer is dealt with separately. In stable conditions with no inversion the upper limit of the vertical integral is ∞.

Position
The position of the centre of the puff is updated using wind speed components at the mean puff height for that time.

$$x_c(t + \Delta t) = x_c(t) + \Delta t U_x(z_m, t)$$

(8.4)
$$y_c(t + \Delta t) = y_c(t) + \Delta t U_y(z_m, t)$$

$$z_c(t + \Delta t) = z_c(t) + \Delta t\left(U_z(z_m, t) - v_s + w_{pr}\right)$$

where the vertical term contains components for gravitational settling, v_s, and plume rise, w_{pr}, (discussed in Sections 8.4 and 8.3.4 respectively). The vertical position is limited to always be at least $1.5z_0$ where z_0 is the local roughness length.

Plume height

In convective or neutral conditions, or in stable conditions in the presence of an inversion, the plume heights of the two parts of the puff within and above the boundary layer, z_{pl} and z_{pu}, are given by:

(8.5)
$$z_{pl} = \min(z_c, h)$$
$$z_{pu} = \max(z_c, h)$$

and for stable conditions with no inversion $z_p = z_c$.

Spread parameters

The spread parameters are updated using the change in the standard ADMS spread parameters that would have occurred during the same time step assuming the current meteorological conditions experienced by the puff had been experienced since the start of the release. Again these are the meteorological conditions at the mean puff height.

(8.6)
$$\sigma_x^2(t + \Delta t) = \sigma_x^2(t) + \Delta t \frac{\partial \sigma_x^2}{\partial t} + \Delta \sigma_{pr}^2$$

$$\sigma_y^2(t + \Delta t) = \sigma_y^2(t) + \Delta t \frac{\partial \sigma_y^2}{\partial t} + \Delta \sigma_{pr}^2$$

$$\sigma_z^2(t + \Delta t) = \sigma_z^2(t) + \Delta t \frac{\partial \sigma_z^2}{\partial t} + \Delta \sigma_{pr}^2$$

where $\Delta \sigma_{pr}$ is the change due to plume rise, which is discussed in §8.3.4, and the derivative is approximated using

(8.7)
$$\frac{\partial \sigma_i^2}{\partial t} = \frac{\overline{\sigma}_i^2(z_m, t + \Delta t) - \overline{\sigma}_i^2(z_m, t)}{\Delta t}$$

where here the $\overline{\sigma}$ quantities are the standard ADMS spread parameters assuming the current meteorological conditions had persisted since the start of the release. These quantities are given by

The longitudinal spread:

(8.8)
$$\overline{\sigma}_x^2 = \sigma_{xt}^2 + \sigma_{xs}^2$$

where the longitudinal spread due to turbulence, σ_{xt}, is given by

(8.9)
$$\sigma_{xt} = \sigma_u t \left\{ 1 + (15.6)^{1/3} \frac{u_* t}{h} \right\}^{-1/2}$$

and the longitudinal spread due to shear, σ_{xs}, is given by

(8.10)
$$\sigma_{xs} = \frac{1}{2} \sigma_z t \frac{\partial U}{\partial Z}$$

The transverse spread:

(8.11)
$$\overline{\sigma}_y^2 = \sigma_{y_t}^2 + \sigma_{y_w}^2$$

where the crosswind spread due to turbulence, σ_{yt}, is given by

(8.12)
$$\sigma_{y_t} = \sigma_v t \left\{ 1 + (15.6)^{1/3} u_* t / h \right\}^{-1/2}$$

and the crosswind spread due to meandering of the mean wind, σ_{yw}, is given by

(8.13)
$$\sigma_{y_w} = 0.065 x \sqrt{7T / U_{10}}$$

The vertical spread:

(8.14)
$$\overline{\sigma}_z = \sigma_w t \left\{ \frac{1}{b^2} + \frac{N^2 t^2}{1 + 2Nt} \right\}^{-1/2}$$

See the ADMS 5 technical specification (CERC 2016) for full details.

An upper limit is placed on the longitudinal spread due to shear, σ_{xs}.

Far field

If the puff grows sufficiently in the vertical direction in the presence of an inversion the activity/concentration becomes nearly uniform in the vertical. A puff is defined to be in the far field if $\sigma_z \geq 1.5h$. If this condition is reached then the mean plume height is set to be half boundary layer height, the activity/concentration calculations assume uniform activity/concentration in the vertical and σ_z is set to be 1.5*h*. This last condition is particularly important as the calculation of the longitudinal spread depends on the vertical spread. If the boundary layer height changes, for instance due to a change in meteorological period, then the puff may no longer be in the far field.

8.3.4 Plume rise

In ADMS-STAR plume rise is calculated for a puff using the same plume rise formulation as ADMS. This is a top-hat integral model, requiring the solution of conservation equations for mass, momentum, enthalpy and emitted material.

Entrainment is described by an entrainment velocity with separate components due to the plume motion relative to the ambient air and due to ambient turbulence. The model also includes the penetration of inversions and the trapping of plumes beneath inversions. The output from this model is the change in height, additional spread and amount of material that has penetrated the inversion due to the plume rise for that time step. Full details of this method are given in the ADMS 5 Technical Specification document (CERC 2016).

Lofting

Lofting of the puff occurs if the puff centre position is in the upper half of the boundary layer, if the geometric condition for plume penetration of the inversion is satisfied (h-z_p<b, where b is the plume radius) and if a velocity scale dependent on the buoyancy of the puff is greater than the turbulent velocity i.e.

(8.15)
$$bg\frac{\Delta\rho}{\rho} > \sigma_w^2$$

where ρ and $\Delta\rho$ are the ambient density and the difference between the puff and ambient density, respectively.

If the conditions for lofting are satisfied, the vertical spread of the puff is reduced because its vertical meandering or 'flapping' is reduced. Under these circumstances, the skewness of the puff is removed and the puff spread parameters are calculated at the puff centre height z_p instead of the mean puff height, z_m. In ADMS-STAR these modifications are only made while the lofting conditions hold. This differs from ADMS 5, as in ADMS 5 if a plume is found to loft, the dispersion calculations are restarted from the source and these modifications applied throughout the dispersion calculations for that met line.

8.4 Dry deposition module

Dry deposition modifies the airborne activity/concentration in two ways: (i) a reduction in plume strength (integrated flux of isotope) *Q* with distance as material is removed from the plume at the surface and (ii) adjustment of the vertical profile because removal occurs of material at the surface. This is illustrated in **Figure 8.4**.
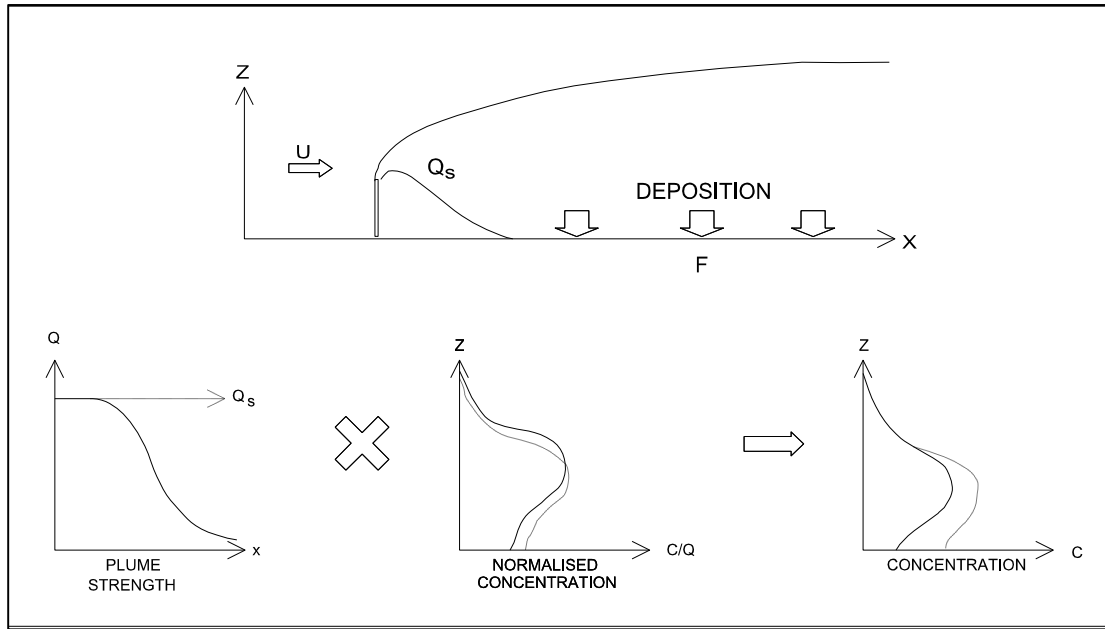
Figure 8.4 - Dry deposition modifies the vertical profile of activity/concentration. Profiles of concentration and normalised concentration are shown.

The rate of dry deposition is assumed to be proportional to the near-surface activity/concentration, i.e.

(8.16)
$$F_{dry} = v_d C(x, y, 0)$$

where F_{dry} is the rate of dry deposition per unit area per unit time, $C(x, y, 0)$ is the predicted airborne activity/concentration at ground level and v_d is the deposition velocity. This velocity contains a diffusive part, v_d', commonly referred to as the deposition velocity itself, and an element due to the gravitational settling, v_s, the terminal velocity of a particle. They are related to the overall deposition velocity, v_d by the equation

(8.17)
$$v_d = \frac{v_s}{1 - \exp(-v_s / v_d')}$$

When v_s is zero, $v_d = v_d'$ and when v_d' is zero $v_d = v_s$.

There are three ways in which the deposition velocity may be determined:

- **Specified** – A constant deposition velocity (v_d) is entered by the user and v_s is taken to be zero. This deposition velocity is spatially and temporally constant and is not modified according to the local meteorological conditions.

- **Standard** – For each isotope, a fixed reference deposition velocity is modified according to the local meteorological conditions, as described in Section 8.4.2. The deposition velocity changes temporally and spatially if modelling the effects of complex terrain and/or using spatially varying met data.

- **Calculated** – The deposition velocity will be calculated by ADMS-STAR. The calculation depends on the properties of the isotope and the meteorological conditions. Details of these calculations are given in Section 8.4.1.

8.4.1 Calculated deposition velocity

If v_d' is estimated by the model it is expressed as the reciprocal of the sum of three resistances,

(8.18) $$1/v_d' = r_a + r_b + r_s$$

where r_a is the aerodynamic resistance, r_b is the sub-layer resistance and r_s the surface layer resistance. Details of these parameters are given in the ADMS 5 Technical Specification documents (CERC 2016).

These resistances depend on the isotope species, nature of the surface and the wind speed. For particles $r_s = 0$, while for gases r_s is estimated as 30 s/m for reactive gases (e.g. SO_2), 1000 s/m for unreactive gases (e.g. O_2) and is infinite for inert gases.

The terminal velocity, v_s, is always zero for gaseous isotopes, while for particles it is estimated from the properties of the particle. Details of this calculation are given in the ADMS 5 Technical Specification documents (CERC 2016).

8.4.2 Standard deposition velocity

If the **Standard** deposition velocity type is selected, then the reference deposition velocity for that isotope (as shown in the **Standard deposition velocity** column of the **Isotope palette**) is modified to take into account the local meteorology, both spatially and temporally. This modification is carried out by multiplying the reference deposition velocity by a ratio of the deposition velocities for a reference isotope of the same type (i.e. gaseous or particulate) at the current and reference meteorological conditions:

- For a gaseous isotope the reference isotope is taken to be an unreactive gas. For a particulate isotope the reference isotope is taken to have a particle diameter of 2.5μm, a density is not needed as the settling velocity is taken to be zero.

- The reference meteorological conditions used are a neutral boundary layer ($h/L_{MO} = 0$), with a wind speed of 4 m/s at 10 m and surface roughness of 0.3 m.

The deposition velocity used at a point (x,y,z) at time, t, is given by

(8.19)
$$v_d(x, y, t) = v_{d\ st} \frac{v_{d\ calc}(x, y, t)}{v_{d\ ref}}$$

where:

- v_d is the deposition velocity to be used

- $v_{d\ st}$ is the reference deposition velocity for that isotope

- $v_{d\ calc}$ is the deposition velocity calculated for the reference isotope for the current meteorological conditions using the equation given in Section 8.4.1.

- $v_{d\ ref}$ is the deposition velocity calculated for the reference isotope for the reference meteorological conditions using the equation given in Section 8.4.1.

8.5 Wet deposition module
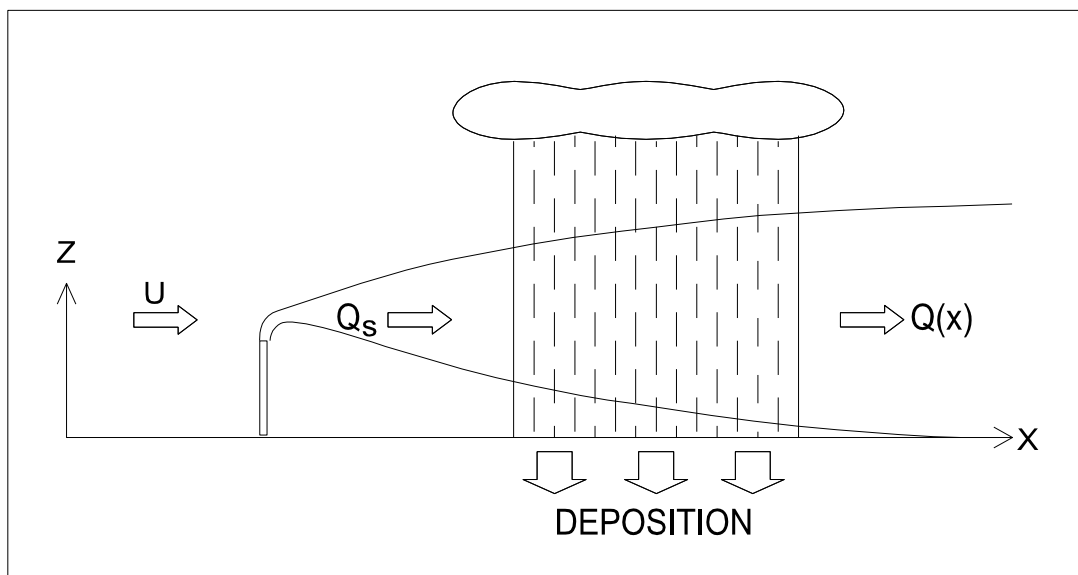
Figure 8.5 - Wet Deposition model.

Wet deposition is modelled through a washout coefficient Λ, such that the amount of material incorporated into any falling precipitation is ΛC per unit area per unit vertical distance per unit time, where C is the local airborne activity/concentration.

Assuming irreversible uptake, the total wet deposition rate per unit horizontal area per unit time, F_{wet}, is found by integrating through a vertical column of air:

(8.20)
$$F_{wet} = \int_0^\infty \Lambda C \, dz$$

The amount of isotope remaining in the puff, Q, therefore decreases with downwind distance.

The following simplifications are made.

1. Uptake of isotopes is irreversible; uptake in precipitation does not lead to a *redistribution* of material in the plume.

2. All puff material lies in or below rain cloud; furthermore no distinction is made between in-cloud scavenging (*rainout*) or below-cloud scavenging (*washout*).

3. The precipitation rate may vary spatially over the area of calculation (see Section 8.7).

The washout coefficient Λ is dependent on a large number of parameters, including the nature of the isotope, precipitation rate, droplet size distribution and the isotope activity/concentrations in the air and in the raindrops. A value for Λ is estimated by the system, as follows:

(8.21)
$$\Lambda = AP^B$$

In ADMS-STAR, the values of A and B are $A = 0.0001$ and $B = 0.8$ for all isotopes.

8.6 Radioactive decay

The Radioactive Decay module solves the coupled ordinary differential equations governing the transformation of radioactive isotopes

(8.22)
$$\frac{dN_i}{dt} = -\lambda_i N_i + \sum_{j \neq i} f_{ij} \lambda_j N_j$$

where

N_i = number density (mol/kg) of isotope i in the emission,

λ_i = decay constant of isotope i ($\lambda_i = \dfrac{\ln 2}{T_{1/2,i}}$, where $T_{1/2,i}$ = half life of isotope i),

f_{ij} = fraction of isotope j decaying into isotope i.

8.7 Spatially varying meteorology

8.7.1 Input data and calculation of meteorological parameters

Spatially varying met data are input through WRF (Weather Research & Forecasting) met data files. These files contain spatially varying fields of the meteorological parameters required for input into ADMS-STAR. It is important to ensure that the following conditions are met:

- The data should be in standard WRF output format (netCDF format).

- WRF must have been run using a Lambert-Conformal projection

- All data files used in an ADMS-STAR run must be from the same WRF run. This is to ensure that the modelling domain and grid are consistent in all files and that accumulative precipitation data has not been reset.

- The first output file from a WRF run should not be used, as certain variables such as cloud cover are not properly set up until the second output file.

With spatially varying meteorology the ADMS met processor is run using data for each of the spatially varying grid points in turn, taking into account the latitude and longitude of that grid point. This then provides the meteorological parameters required for ADMS-STAR on each of these grid points. During the dispersion calculations the meteorological parameters at the current puff location are then calculated by interpolating from this grid of meteorological parameters.

8.7.2 Effect on dispersion calculations

With spatially varying meteorology the dispersion calculations are carried out in the same way as with homogeneous meteorological data, except that now the meteorology parameters depend on the puff's horizontal position as well as the mean puff height and the time. For instance the advection of the puff centre position becomes:

$$x_c(t + \Delta t) = x_c(t) + \Delta t U_x(x_c, y_c, z_m, t)$$

(8.23) $$y_c(t + \Delta t) = y_c(t) + \Delta t U_y(x_c, y_c, z_m, t)$$

$$z_c(t + \Delta t) = z_c(t) + \Delta t \left(U_z(x_c, y_c, z_m, t) - v_s + w_{pr} \right)$$

and, similarly, for the spread parameters the calculation of the standard ADMS spread parameters now uses the meteorology at that position.

8.7.3 Use of site observed data

If spatially varying meteorology is used it is possible to, in addition, specify values of wind speed and direction, temperature and precipitation rate at the source. These values are then used in preference to the spatially varying meteorological data at the source. The observed data then relaxes back to the spatially varying meteorology over

a relaxation radius. This relaxation occurs in both the vertical and horizontal, so for instance for the easterly wind speed component, U_e, at point (x, y, z) we define

(8.24)
$$\zeta = \sqrt{\left(\frac{(x-x_s)^2 + (y-y_s)^2}{R_h^2}\right) + \frac{(z-z_{ob})^2}{R_v^2}}$$

where (x_s, y_s) is the horizontal location of the source, z_{ob} is the height of the observed data and R_h and R_v are the horizontal and vertical relaxation radii. Then the wind speed component used is:

(8.25)
$$U_e(x, y, z) = \begin{cases} (1-\zeta)U_{OBSe} + \zeta U_{NWPe}(x, y, z) & \zeta < 1 \\ U_{NWPe}(x, y, z) & \zeta \geq 1 \end{cases}$$

where U_{OBSe} is the observed wind speed and U_{NWPe} is the wind speed from the WRF file. The observed data are used from the time given until the time of the next observed data or the end of the current met period; whichever is the sooner.

8.8 Complex terrain

8.8.1 Use of FLOWSTAR in ADMS-STAR

In ADMS-STAR files of spatially varying terrain height or surface roughness can be entered. Flow and turbulence fields are calculated from the variable terrain height and surface roughness data using FLOWSTAR. Details of how the FLOWSTAR calculations are carried out can be found in the ADMS 5 Technical Specification documents (CERC 2016).

The wind field generated by FLOWSTAR is nested within, and used in preference of, the spatially varying or homogeneous meteorology. Outside of the terrain region, the flow field and turbulence parameters calculated by FLOWSTAR are relaxed back to the spatially varying or homogeneous meteorology. Further details of using complex terrain with spatially varying meteorology are given in Section 8.8.3.

The puff model used in ADMS-STAR is able to take into account reverse flow regions and will follow the path of the puffs correctly through these regions. However, slopes should be limited to be less than 1 in 3 as the FLOWSTAR algorithms are less valid outside of this range.

8.8.2 Use of spatially varying turbulence parameters

The old and new spread parameters based on the meteorological conditions at the current time and location are calculated using the flat terrain turbulence algorithms. A correction factor is then applied utilising the ratio of the turbulence parameter generated by FLOWSTAR, e.g. σ_{uhill}, to the flat terrain turbulence parameter, e.g. σ_{uflat}. So for example for the longitudinal spreading the expression becomes:

$$(8.26) \qquad \sigma_x^2(t + \Delta t) = \sigma_x^2(t) + \left(\frac{\sigma_{uhill}}{\sigma_{uflat}} \right)^2 \left(\overline{\sigma}_x^2(t + \Delta t) - \overline{\sigma}_x^2(t) \right) + \Delta \sigma_{pr}^2$$

8.8.3 Complex terrain and spatially varying meteorology

For spatially homogeneous meteorology, the upwind condition used by FLOWSTAR for a given met period is the condition specified by the user for that met period. For spatially varying meteorology, the upwind condition for a given met period is calculated as follows. The model first calculates the average wind direction across the whole of the terrain (surface roughness) region. This wind direction is then used to define an upstream edge of the terrain region and the meteorological condition calculated as the average of conditions along this edge.

Inside the terrain region the flow field output from FLOWSTAR is used and the other meteorological conditions, except for the precipitation rate, are set to be that of the upstream edge for consistency. Outside the terrain region the spatially varying meteorological data are used with a relaxation region between the two. If observed

site data are specified then this is used in preference to both the spatially varying meteorological data and the flow field data output from FLOWSTAR.

8.9 Marine boundary layer

ADMS-STAR includes a marine boundary layer scheme for calculating surface roughness and heat fluxes over the sea, which could be used, for example, for dispersion modelling of stacks on oil extraction platforms. To use this option the source and all receptors should be over the sea; it is not suitable for coastal modelling. This option cannot be used with complex terrain or spatially varying meteorology.

8.9.1 Wind profile

For surface roughness we adopt the formula of Beljaars (1994), used by the European Centre for Medium range Weather Forecasts (ECMWF):

(8.27)
$$z_0 = \alpha_m \frac{\nu}{u_*} + \alpha_{Ch} \frac{u_*^2}{g}$$

where u_* (m/s) is the friction velocity, ν (m²/s) is the kinematic viscosity of air, g is the acceleration due to gravity (m/s²), $\alpha_m = 0.11$ and α_{Ch} is the Charnock parameter entered by the user.

The velocity profiles used are then the same as those used over the land, described in the ADMS 5 Technical Specification (CERC, 2016). The equations are solved by iteration.

8.9.2 Heat fluxes

Over sea the surface roughnesses for sensible heat (z_{0H}) and moisture (z_{0q}) are given by Beljaars (1994) as

(8.28)
$$z_{0H} = \alpha_H \frac{\nu}{u_*} \quad and \quad z_{0q} = \alpha_q \frac{\nu}{u_*}$$

where $\alpha_H = 0.4$ and $\alpha_q = 0.62$.

Then the sensible heat flux F_{θ_0} is given by

(8.29)
$$F_{\theta 0} = \frac{-c_p \rho \kappa^2 (\theta(z) - \theta_0) u(z)}{\left[\ln\left(\frac{z + z_{0H}}{z_{0H}} \right) - \psi_H \left(\frac{z + z_{0H}}{L_{MO}} \right) \right]\left[\ln\left(\frac{z + z_0}{z_0} \right) - \psi \left(\frac{z + z_0}{L_{MO}} \right) \right]}$$

where c_p is the specific heat capacity of air (J/kg/K), ρ is the density of air (kg/m3), κ is von Karman's constant (0.4), θ is potential temperature (K), θ_0 is the potential temperature corresponding to the temperature of the sea surface, and L_{MO} is the Monin-Obukhov length (Panofsky and Dutton, 1984).

For stable and neutral conditions ($1/L_{MO} \geq 0$), $\psi_H = \psi$ (as defined in the ADMS 4 Technical Specification (CERC, 2010)), and for convective conditions ($1/L_{MO} < 0$) ψ_H is given by

(8.30)
$$\psi_H = \ln \frac{(1+y)^2}{(1+y_{surface})^2}$$

where

(8.31)
$$y = \sqrt{1 - 16 \frac{z + z_{0H}}{L_{MO}}} \quad and \quad y_{surface} = \sqrt{1 - 16 \frac{z_{0H}}{L_{MO}}}$$

The latent heat flux λ_E is given similarly by

(8.32)
$$\lambda E = \frac{-\lambda \rho \kappa (q(z) - q_{sat0}) u(z)}{\left[\ln\left(\frac{z + z_{0q}}{z_{0q}}\right) - \psi_q\left(\frac{z + z_{0q}}{L_{MO}}\right) \right]\left[\ln\left(\frac{z + z_0}{z_0}\right) - \psi\left(\frac{z + z_0}{L_{MO}}\right) \right]}$$

where λ is the specific latent heat of vaporization of water (J/kg), q is the specific humidity, q_{sat0} is the saturation specific humidity at the sea surface, and ψ_q is equivalent to ψ_H but using z_{0q} rather than z_{0H}. Again the equations are solved by iteration (Panofsky and Dutton, 1984).

8.9.3 Input data requirements

Some additional data are required to use the marine boundary layer option, all of which are important in determining the structure of the marine boundary layer:

- sea surface temperature (°C),

- Charnock parameter,

  This is a constant used in the heat flux calculations. Typical values range from 0.018 to 0.08, but in the marine boundary layer this may depend on, for example, distance from the coast. Validation suggests that a value of 0.08 is most appropriate in ADMS-STAR for regions not altogether remote from land (e.g. North Sea).

- height above sea level of wind speed and direction measurements (in metres)

- height above sea level of temperature measurements (in metres)

8.10 Inhalation dose and thyroid dose

ADMS-STAR includes options to calculate the inhalation dose and thyroid dose for each isotope and as a total over all isotopes. The inhalation dose (ID) for a specific isotope is calculated from:

$$ID = C * ir * DCC$$

where *C* is the concentration dose from that isotope, *ir* is the inhalation rate and *DCC* is the inhalation dose coefficient for that isotope.

The default inhalation rate is taken to be 22 m^3/day, which is a value suitable for adults taken from ICRP 119 (ICRP (2012)). .

Default values of the inhalation dose coefficient are taken from ICRP 119. In most cases the values are taken from Table G.1 - the effective dose coefficients for inhalation (activity median aerodynamic diameter = 1 μm) of radionuclides for members of the public up to 70 years of age, with the worst case for adults being used. The exceptions to this are:

- S-35 (org) which is taken from Table B.1 – effective dose coefficients for inhalation of soluble or reactive gases for workers; and

- Ar-41 which is calculated from Table C.1 – effective dose rate coefficients for exposure of workers or adult members of the public to airborne concentration of inert gases.

The thyroid dose is calculated in the same way as the inhalation dose except the inhalation dose coefficient is divided by the thyroid tissue weighting factor. The default value of tissue weighting factor is taken to be 0.05 from ICRP 119. Only concentrations of iodine isotopes contribute to thyroid dose.

8.10.1 Emergency Response Levels

If inhalation dose or thyroid dose are selected for output contours of Emergency Response Levels may be output. Values are given for the whole body, based on the inhalation dose, and for the thyroid based on the thyroid dose.

		Dose equivalent level (mSv)	
Countermeasure	**Organ**	**Lower**	**Upper**
Sheltering	Whole body	3	30
	Thyroid	30	300
Evacuation	Whole body	30	300
	Thyroid	300	3000
Stable iodine	Thyroid	30	300

Table 8.3 Emergency Response Levels used in ADMS-STAR, values are from NRPB Vol 1 No 4.

8.11 Gamma dose

Gamma dose rate due to both airborne and deposited radioactive material may be calculated at up to 5 receptor points.

8.11.1 Gamma ray dose rate due to airborne radioactive material

The effective flux of gamma rays $\Phi(r)$ at a point distance r from a monoenergetic source dispersed in air for a particular energy level, i, is

$$(8.1) \qquad \Phi_i(r) = \iiint \frac{f(E_i) C(E_i, r) B(E_i, \mu_i | r - r'|) e^{-\mu_i |r-r'|}}{4\pi (r - r')^2} \, d^3 r'$$

where $f(E)$ is the branching ratio to the specified energy E, C the concentration in Bq/m^3, B the build-up factor, and μ the linear attenuation coefficient.

The build-up factor is calculated from Berger's analytic expression

$$(8.2) \qquad B(E_i, \mu_i r) = 1 + a(E_i) \mu r \, e^{b(E_i) \mu r} .$$

Coefficients $a(E_i)$ and $b(E_i)$ are obtained from tabulated data.

The effective body dose rate is obtained by multiplying the flux at energy Φ_i by an absorption coefficient M_{ai}, a conversion coefficient C_{bi}, and the energy E_i, and summing over all n energy levels so that the effective body dose rate due to airborne material, D_{air}, is

$$(8.3) \qquad D_{\text{air}} = \sum_{i=1}^{n} C_{bi} M_{ai} E_i \Phi_i .$$

The gamma dose is calculated by summing the gamma dose rate across all time steps.

The input data are the concentration fields for the different isotopes, which are obtained from the radioactive decay calculations, and the gamma dose information for each isotope, i.e. the number of energy levels, the energy of each branch (MeV) and the branching ratios.

In evaluating the integral for $\Phi(r)$, a hemisphere of 15*15*15 calculation points centred on the receptor point of interest is used.

8.11.2 Gamma ray dose rate due to deposited radioactive material

The gamma dose rate due to deposited material, D_{dep}, can be calculated by once again summing over energy levels:

$$(8.4) \qquad D_{\text{dep}} = \sum_{i=1}^{n} C_{bi} M_{ai} E_i F_i ,$$

where for each energy level

$$(8.5) \qquad F_i = \frac{1}{4\pi} \int_0^{2\pi} \int_0^\infty \frac{f(E_i) \times F_{dr}(E_i, r) \times B(E_i, \mu_i r) e^{-\mu_i r}}{r} dr d\theta$$

and F_{dr} is the rate of activity of deposited material (Bq/m^2).

The gamma dose is calculated by summing the gamma dose rate across all time steps.

8.12 Puff distribution

8.12.1 Continuous releases

For a continuous release one initial puff is released for each isotope/particle size combination. As the release continues further puffs are added at a rate governed by the parameters entered by the user in the interface. For these puffs the amount of material in the puff relates to the length of time covered by its release.

The initial height of this puff is the source height, with the initial spread given by the source diameter. The temperature is as specified by the user and the exit velocity is either as specified by the user or calculated from the maximum plume height as described below.

Plume top height calculation

In ADMS-STAR instead of entering the exit velocity for a continuous release the maximum plume height can be entered. This, along with the meteorological conditions for the first meteorology period, is then used to determine the exit velocity of the release. The exit velocity is initially estimated and the first puff modelled to determine the maximum height from this estimated exit velocity. The maximum height achieved is then compared to the required maximum height and the estimate of the exit velocity improved. This process is repeated until the maximum plume height obtained for the estimate of exit velocity matches that entered. The full dispersion calculations are then carried out using this calculated exit velocity.

8.12.2 Explosive releases

For an explosive release five initial puffs are released for each isotope/particle size combination. However, as an explosion is an instantaneous event no further puffs are released. The amount of material in each puff is the appropriate fraction of the release as defined by the fraction within the puff.

The five puffs released are all of the same size and are stacked vertically, the heights of the puffs and the amount of material in each puff is entered by the user. Also entered by the user, either directly or to be calculated from the mass of explosive, is the cloud top height. The initial spread parameters are defined to be 0.2 times the cloud top height in the vertical and 0.1 times the cloud top height in the horizontal. For an explosive release the puffs are all at ambient temperature and no plume rise calculation is carried out.

Calculation of cloud top height

The cloud top height may either be entered directly or calculated from the mass of explosive (TNT equivalent in kg). The cloud top height, *CTH*, is calculated from the mass of explosive, M_{kg}, as

(8.33)
$$CTH = 76\beta^{0.25}M_{kg}^{\ 0.25}$$

where β is a mass conversion factor ($\beta = 2.20462$ lb/kg).

8.13 Use of sample measurements for inverse calculation

ADMS-STAR can use field measurement data of the level of radioactivity in the air close to the ground and/or the level of activity of isotopes deposited on the ground. It is assumed that at first only measurements of the concentration of radioactivity in the air close to the ground are available. These are derived from sampling the air at a point for a short period of time and calculating an average activity (concentration) level in Bq/m^3 (g/m^3). Some measurements of air concentration will not contain the isotopic composition of the sample and in this case values for the half-life and deposition velocity of a representative isotope (which may be fictitious) can be supplied by the user entering the air concentration samples as the User defined isotope/chemical.

Whereas *air concentration measurements* must take place while some part of the release is above the measurement station, deposition measurements can also be made once the plume has moved on downwind. As soon as deposition data become available they supersede the air concentration data.

Deposition data are derived from soil samples, which are analysed to give spectra of the radio-isotopes present in each sample and their deposition levels in Bq/m^2 or g/m^2. Material is deposited at a location whilst the release is overhead and there is zero deposition at other times. After the material is deposited the radioactive isotopes continue to decay.

8.14 Isotope properties and groups

8.14.1 Isotope properties

Values for the half-lives and representative deposition velocities of 33 isotopes are included in ADMS-STAR; these may be viewed from the isotope palette in the ADMS-STAR interface by selecting **Isotopes**, **Palette**. While the former are usually well-known, experimentally determined quantities, the deposition velocity can vary with the type of terrain, state of the isotope (i.e. gaseous or particulate) and, if particulate, on the average size of the particles. Note, for instance, that there are two entries for S_{35}, as different deposition velocities are associated with organic and inorganic forms of S_{35}.

Of the 33 isotopes, some are isomeric pairs (i.e. one decays to a more stable isotope of the same element) and, in addition, there are some that are the decay products of others, namely Te_{132}, which decays to isomeric pairs I_{132}^{m} and I_{132}, Ba_{140}, which decays to La_{140}, and Np_{239}, which decays to Pu_{239}. All these decay chains are taken into account in ADMS-STAR.

The concentration dose, inhalation dose, thyroid dose and gamma dose calculations don't take into account the full decay chain – only the decay of the parent isotope.

8.14.2 Isotope groups

As an alternative to calculating contour output for single isotopes, ADMS-STAR can calculate contour output for any of four EU groups of isotopes. These are the 'Iodine' group (containing the Iodine isotopes plus Tellurium), the 'Strontium' group (containing the Strontium isotopes), the 'Alpha-emitting' group (containing alpha-emitting isotopes of Pu and Transplutonium) and the 'Other' group (containing all other isotopes). The isotopes in each group are listed in **Table 8.4**.

Group	Isotopes
Iodine	Te_{132}, I_{129}, I_{131}, I_{132}, I_{133}, I_{135}
Strontium	Sr_{89}, Sr_{90}
Alpha-emitting	Pu_{238}, Pu_{239}, Pu_{240}, Am_{241}, Cm_{242}
Other	H_3, C_{14}, S_{35} (organic), S_{35} (inorganic), Ar_{41}, Zr_{95}, Tc_{99}, $Tc_{99}m$, Ru_{103}, Ru_{106}, Sb_{125}, Cs_{134}, Cs_{137}, Ba_{140}, La_{140}, Ce_{144}, U_{235}, Np_{237}, Np_{239}, Pu_{241}

Table 8.4 - Members of each isotope group

The group activity calculated by ADMS-STAR is a weighted sum of the individual isotope concentrations, where the activity of each isotope is weighted by a factor equal to the 'group MPL' divided by the individual isotope MPL. (The 'group MPLs' are listed in **Table 8.5**.)

Only the isotopes in the iodine group contribute to the calculated thyroid dose.

Group	MPL for all foodstuffs except milk	MPL for milk
Strontium	750	125
Iodine	2000	500
Alpha	80	20
Other	1250	1000

Table 8.5 - Group MPLs (Bq/kg)

8.14.3 Isotopes in the Iodine group

For isotopes of the iodine group only, the deposition MPL values are calculated assuming the time for the maximum activity to be reached in the different foodstuffs has elapsed.

APPENDIX A   USING ADMS-STAR WITH ARCGIS

ADMS-STAR may be run in tandem with ArcGIS. This enables you to use background map data to locate sources and samples and readily display the total accumulated deposition and instantaneous activity/concentration contours.

If you have not used ArcMap before, we recommend that you complete the introductory ArcGIS ArcMap tutorial supplied with ArcMap and familiarise yourself with the ArcMap documentation. You will then have a good understanding of its terms and conventions and so be able to get the most out of your modelling system.

The features of ArcGIS that are relevant to ADMS-STAR users are introduced and explained later on in this section.

A.1 Setting up the ADMS-STAR toolbar with ArcGIS

ArcGIS should be installed before ADMS-STAR, as the ADMS-STAR–ArcGIS link is set up as part of the ADMS-STAR installation.

If ArcGIS, ADMS-STAR and the link have been successfully installed, the ADMS-STAR toolbar should be displayed when you start ArcGIS. If it is not, perform the following steps:

1. Double-click the ArcMap icon.

2. Once ArcMap has been launched, right click anywhere on the toolbar and select the ADMS-STAR toolbar from the list. The toolbar should then be added to the screen.

   If the toolbar does not appear in the list,

   (a) select **Customize**... at the bottom of the list. A dialogue box is displayed, as shown in **Figure A.1**.
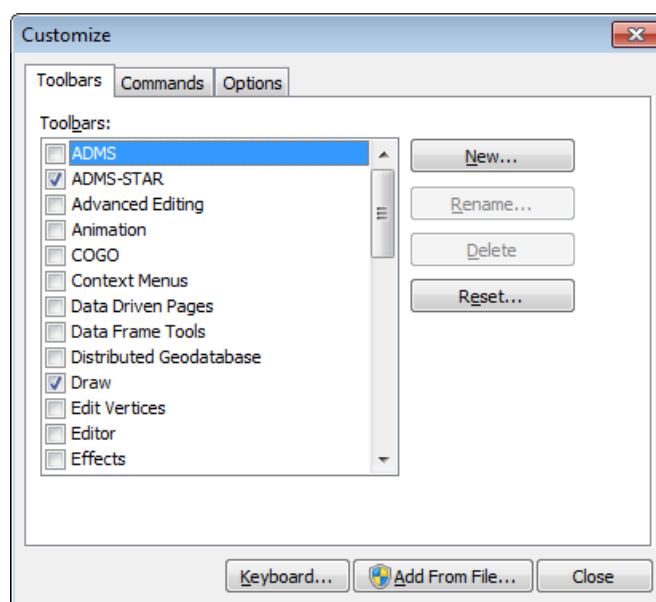
Figure A.1 - The **Customize** toolbar dialogue box.

(b) Click **Add from file...** and browse for the file ADMS-STAR_ArcGISLink.dll, which should be located in the ArcGIS folder of the ADMS-STAR application directory (*<install directory>\ArcGIS*) then select Open.

(c) You will be shown an **Added Objects**... dialog, click **OK**. The ADMS-STAR toolbar will now have been added to the list in the **Customize** toolbar dialog, tick the box so that it is displayed and select **Close**.

A.2 A brief introduction to ArcMap

The "Untitled" project window appears when ArcMap is first launched (**Figure A.2**). All work in ArcMap is saved in *Map Documents*. A Map Document is a collection of *Layers* within *Data Frames, Tables, Charts, Layouts* and *Scripts*. Of these, only layers, data frames and tables are relevant to the use of ArcMap with ADMS-STAR.
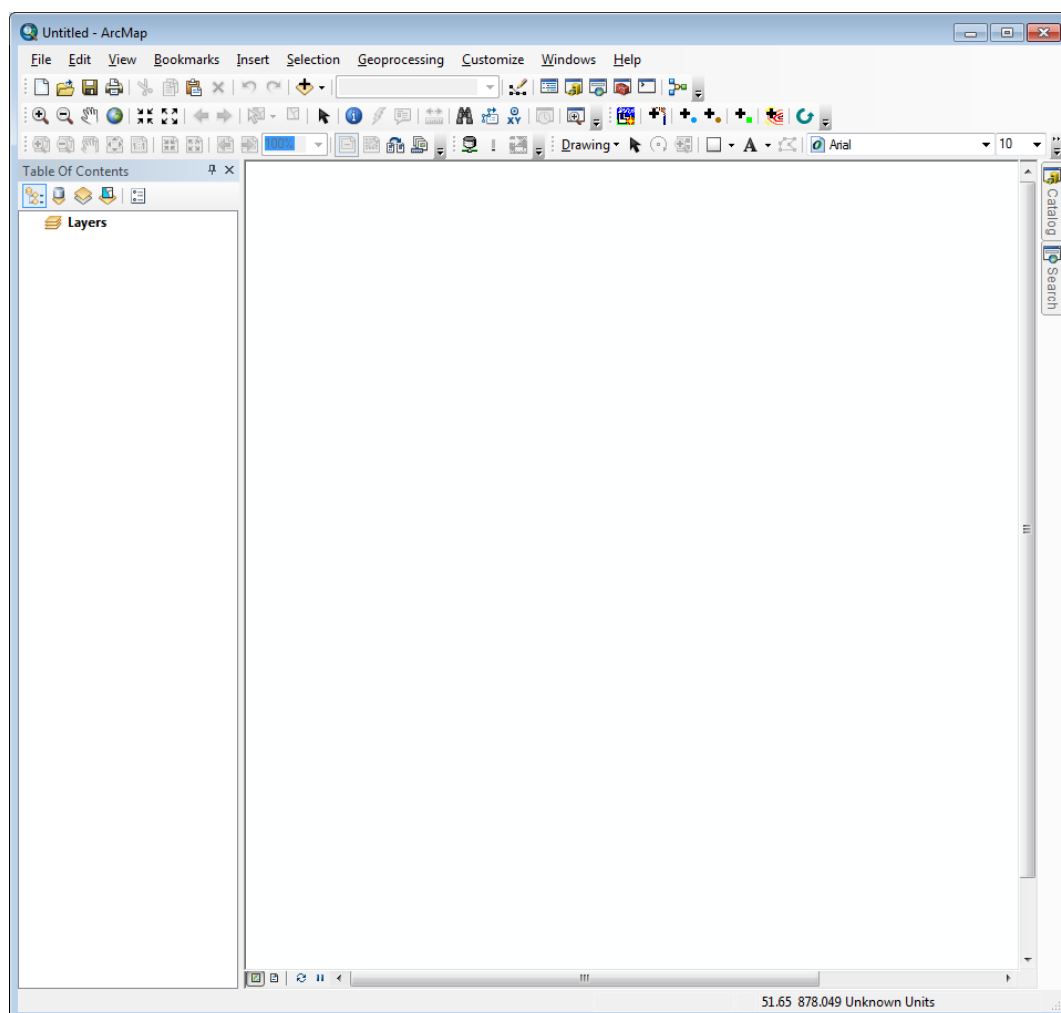
Figure A.2 - The default project screen in ArcMap.

A *map document* is an interactive map that lets you display, explore, query and analyse geographic data in ArcMap. A *data frame* contains a collection of map *layers*. Each layer represents a set of geographic features. For example, a data frame showing a country might have one *layer* representing cities, one *layer* representing roads, one representing rivers, etc.

By default the new map document will have a data frame called Layers (**Figure A.2**). To create another data frame, select **Insert**, **Data Frame** from the menu.

A *Data frame* is a collection of map *layers*, which may include

- raster data such as aerial photographs and/or Ordnance Survey raster maps,

- vector data such as a road network or an administrative boundary (these may be ArcMap Shape files, CAD drawings, or an ARC/INFO library),

- modelled activity/concentration or deposition data.

A.3 Map units in ArcMap

The map units are the units in which the spatial data in the data frame are drawn. The map units are determined by the coordinate system of the data frame. The coordinates and units at the position of the cursor are displayed in the status bar at the bottom of the window (see **Figure A.2**). When a new map is loaded in ArcMap, the units are displayed as *Unknown*.

*To use the ADMS-STAR – Arc-GIS link, the map units **must** be in metres.*

Listed below are three ways to set the map units to metres:

1. If the map units are displayed as *Unknown*, they will be automatically set to metres when the update button is clicked from the ADMS-STAR toolbar or the source is located using the button.

2. The map units can be set to metres directly in ArcMap by clicking **View, Data Frame Properties...** and in the **General** tab, using the dropdown menus to choose metres as the Map and Display units (**Figure A.3**).
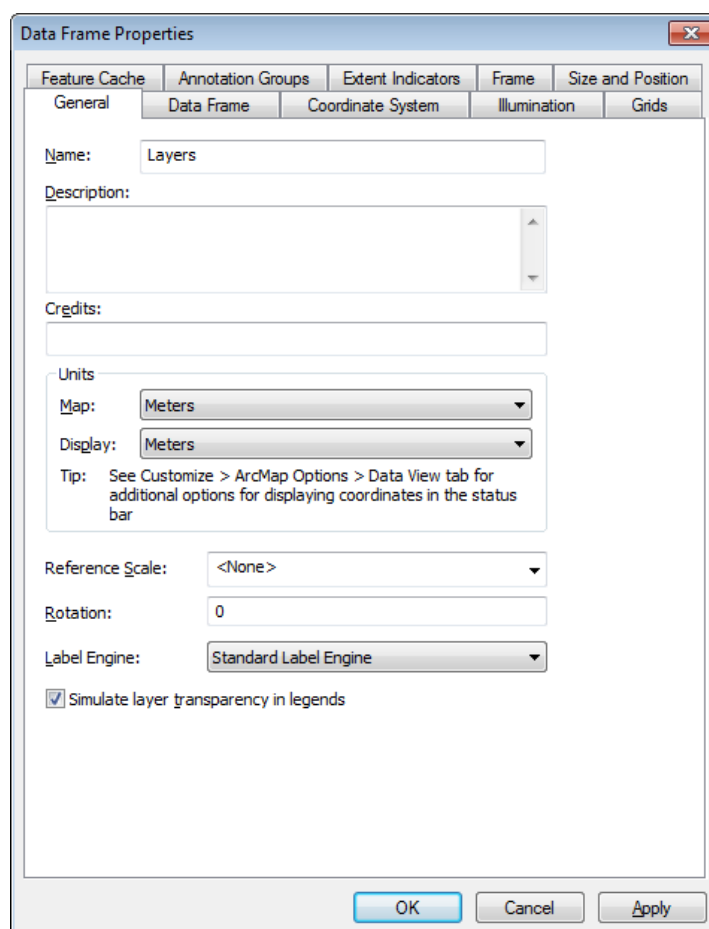
Figure A.3 - General tab of the **Data Frame Properties** window in ArcMap.

3. ArcMap can be set up to use a predefined projected coordinate system, e.g. the UK Ordinance Survey National Grid, in which metres are the measurement unit. This is done by clicking **View, Data Frame Properties...** and selecting the **Coordinate System** tab (**Figure A.4**). The explorer window can be used to navigate to and select the required predefined projected coordinate system. The units of the selected coordinate system are included as part of the information in the **Current coordinate system** box. After returning to the main plot window, the correct units will be displayed.
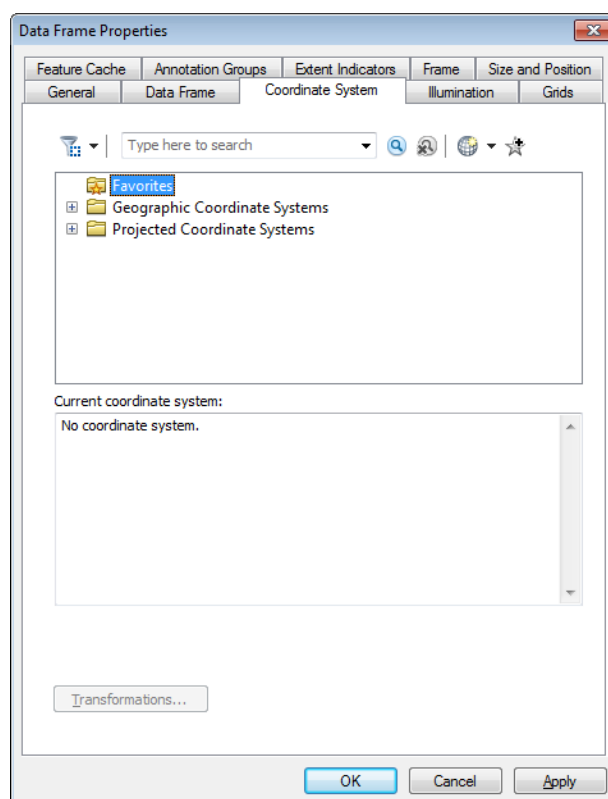
Figure A.4 - Coordinate System tab of the **Data Frame Properties** window in ArcMap.

If a coordinate system with a different unit of measurement is chosen, e.g. feet, the following warning message will be issued the next time the update ⟳ button on the ADMS-STAR toolbar is used:
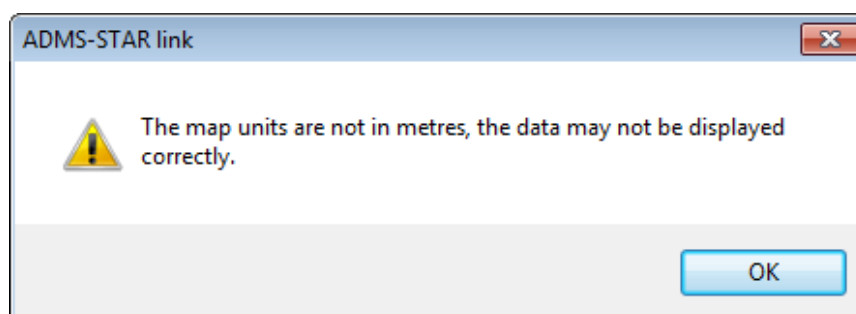

Figure A.5 - Coordinate System tab of the **Data Frame Properties** window in ArcMap.

If you are adding a base map of the modelling region to the plot window, you must again make sure that the map coordinates are in metres. If the map data does not contain the unit of measurement, ArcMap will display the units as *Unknown*. The units can then be changed to metres using method 1 or 2 in the list above.

In the ADMS-STAR interface, the source X,Y location and sample/receptor must be given in kilometres rather than metres. When information is passed from ArcMap to the ADMS-STAR interface (or vice versa) using the link, the conversion from metres to kilometres (or kilometres to metres) is carried out automatically.

A.4 A guide to the ADMS-STAR controls in ArcMap

This section describes the action of the ADMS-STAR controls that are added to the ArcMap interface. For information on other controls, please refer to the ArcMap documentation.

There are seven buttons that control the ADMS-STAR functions available from ArcMap. The buttons are shown in an ADMS-STAR toolbar.

Figure A.6 - The ADMS-STAR toolbar.

The buttons are used for opening the ADMS-STAR interface and transferring data between the interface and ArcMap. There are also buttons for plotting data from an ADMS-STAR output file and refreshing the ADMS-STAR data in the map view.

For each button there are tool tips and information shown in the help line (at the bottom left of the screen) when the cursor is positioned over the corresponding ADMS-STAR buttons.

A.4.1   ADMS-STAR link

	Brings up the interface to the ADMS-STAR dispersion model. The ADMS-STAR window appears on the screen. The user may set up a new problem, open or edit an existing input data (*.hpl*) file or view results (*.rec*, *.glt* etc. files) within ADMS-STAR in the usual way. To return to the ArcMap interface, minimise the ADMS-STAR window or click on the map.

A.4.2   Adding points directly in ArcMap

	Enter the source location by clicking on the map. This button allows the user to add the source datum by clicking with the mouse at the required location on the map. The source X and Y datum will automatically be updated in the ADMS-STAR interface in the **Define location** sub-screen of the **Source** screen. Existing source coordinates will be overwritten. Coordinates are automatically converted from metres (used in ArcMap) to kilometres (used in ADMS-STAR). The source longitude and latitude should then be added in ADMS-STAR by the user. The user can view the new source datum in ArcMap by selecting the update ↻ button.
	Add a new activity/concentration reading to ADMS-STAR at the location specified. This button allows the user to add the location of an air activity/concentration sample by clicking with the mouse at the required location on the map. A new set of sample coordinates will automatically be added to the ADMS-STAR interface on the **Concentration** screen, in either absolute or relative (to the source) coordinates depending on the choice made in the **Sample & receptor locations** section of the **Setup** screen. Coordinates are automatically converted from metres to kilometres. The sample date/time and strength MUST then be added in ADMS-STAR by the user. The location of the new sample may be viewed on the map by returning to ArcMap and selecting the update ↻ button.
	Add a new deposition reading to ADMS-STAR at the location specified. This button allows the user to add the location of a deposition sample by clicking with the mouse at the required location on the map. A new set of sample coordinates will automatically be added to the ADMS-STAR interface on the **Deposition** screen, in either absolute or relative (to the source) coordinates depending on the choice made in the **Sample & receptor locations** section of the **Setup** screen. Coordinates are automatically converted from metres to kilometres. The sample date/time and isotope strength MUST then be added in ADMS-STAR by the user. The location of the new sample may be viewed on the map by returning to ArcMap and selecting the update ↻ button.

	Add a new receptor point to ADMS-STAR at the location specified. This button allows the user to add the location of a receptor point by clicking with the mouse at the required location on the map. A new set of receptor point coordinates will automatically be added to the ADMS-STAR interface on the **Output** screen, in either absolute or relative (to the source) coordinates depending on the choice made in the **Sample & receptor locations** section of the **Setup** screen. Coordinates are automatically converted from metres to kilometres. The user MUST then click **Add** to add the receptor point to the table. The location of the new receptor point may be viewed on the map by returning to ArcMap and selecting the update ↻ button.

A.4.3 Plotting ADMS-STAR output in ArcMap

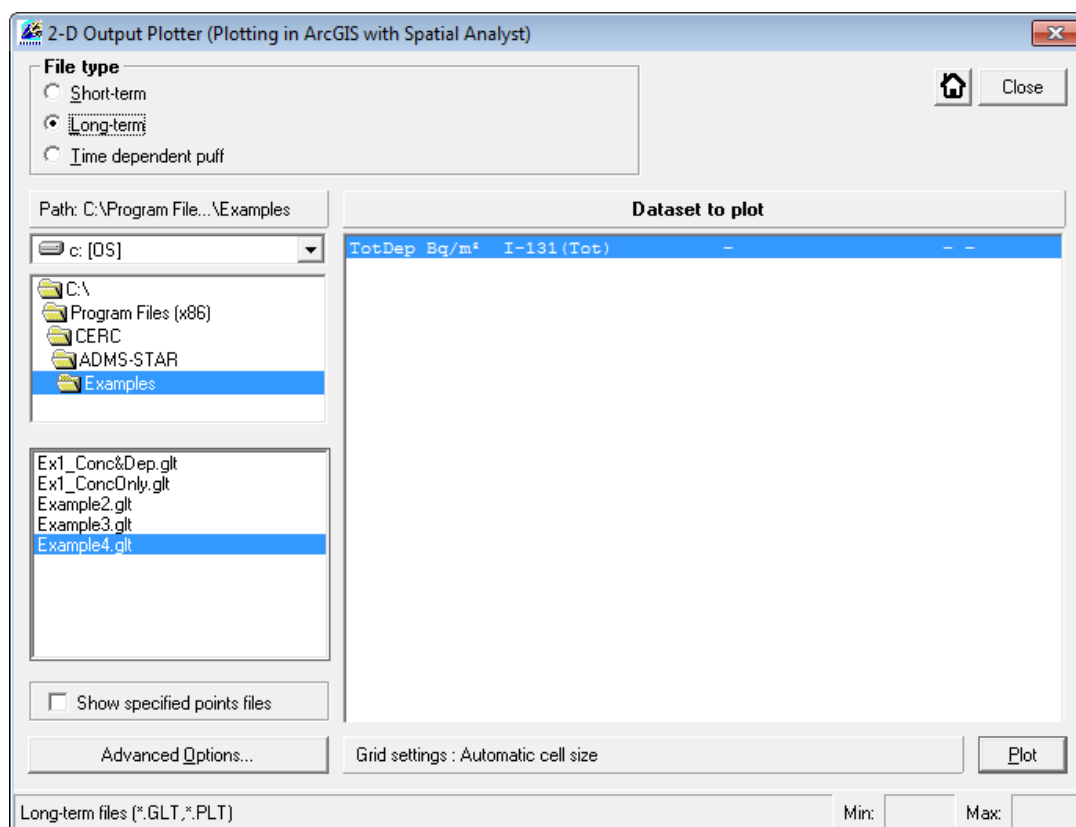|	**Create contour layer from ADMS-STAR output file.** Launches the **ADMS Contour Plotter (Plotting with ArcGIS and Spatial Analyst)**, see **Figure A.7**.

Figure A.7 - ADMS Contour Plotter for plotting contours in ArcGis.

Here are the steps to follow in order to create a contour plot of data:

Step 5 Select the **Long term** radio button to plot the data in the *.glt* file (i.e. total accumulated deposition) or the **Short term** radio button to plot the data in the *.gst* file (i.e. instantaneous activity/concentrations).

Step 6 Select the appropriate folder and click on the name of the file containing the data to plot.

Step 7 Select the dataset to plot (in the **Dataset to Plot** box), and if relevant select also the time for which the data are to be plotted (**Time (year, day, hour)** box).

The **Dataset to Plot** box shows a list of all the variables that can be used for the plot. Each record in the list shows the type of output (e.g. 'TotDep' for total accumulated deposition or 'MPLDep' for MPL levels), the units of output (e.g. 'Bq/m^2'), the isotope or isotope group name (e.g. 'I-131'), the foodstuff (e.g. 'Milk' or 'User defined' for user defined contour levels), and the MPL level.

Step 8 Click **Advanced Options**... to set some properties of the plot (optional, see below).

Step 9 Click **Plot** to plot the selected data in ArcMap.

> *The ADMS-STAR output file is converted to a raster file (*.aux*) for ArcMap to plot. Save this in the directory where the* .hpl *file is located. Once the ArcMap drawing has been saved, you can delete the* .aux *file as it has become redundant.*

In the upper right-hand corner, the **Close** button closes the **ADMS Contour Plotter** and the **ADMS** button brings the ADMS-STAR interface to the front without closing the utility.

Advanced options

Clicking **Advanced Options** displays the **Advanced Contour Options** screen (**Figure A.8**). This allows you to set the grid cell size in metres. Click **Close** to return to the main **ADMS Contour plotter** screen.

Figure A.8 - Advanced Contour Options screen for the ADMS Contour Plotter for plotting contours in ArcGIS.

A.4.4   Updating information in ArcMap

Update the display to reflect the current information in the ADMS-STAR model interface. This button will update the locations of the source, concentration samples and/or deposition samples for the layers that have been added to the open view in ArcMap to those corresponding to the current ADMS-STAR interface. Note that for sample locations some sample data must have been added to the bottom table in the **Concentration** or **Deposition** screen before the location is displayed.

A.4.5   Saving your work in ArcMap

When you have displayed your data in ArcMap you should save your data to a user-named shape file. To do this right click on the layer you want to save, select **Data** and then **Export Data...** and specify a new user-defined name when prompted. If you save your work as an *.mxd* file, when you re-open the *.mxd*, ArcGIS will display the *data last viewed* in the link, not necessarily the data originally saved. This is because the data displayed continuously overwrite the data saved.

A.5 Switching between ADMS-STAR and ArcMap

If you have started both ADMS-STAR and ArcMap, you will need to switch between them during your work.

There are several ways of switching from ArcMap to ADMS-STAR:

- Click the **Show ADMS-STAR** button on the ADMS-STAR tool bar in ArcMap.

- Click on ADMS-STAR on the taskbar (if visible)

- Use **ALT** + **TAB** to step through the open programs on your computer

- Click on the title bars of ADMS-STAR (if visible). When switching back to ArcMap always click on the window title or on a grey part of the ArcMap interface, e.g. on a space in the button bar, and not directly on a map otherwise an ArcMap command may be implemented unintentionally.

When you are entering data in ADMS-STAR, you can switch to ArcMap in order to position sources and sample points on maps.

APPENDIX B   REFERENCES

Beljaars, A. C. M. (1994): The parameterisation of surface fluxes in large-scale models under free convection. In *Quart. J. R. Met. Soc.*, **121**, pp. 255-270.

Caughey, S.J. and Palmer, S.G. (1979) Some aspects of turbulence structure through the depth of the convective boundary layer. Quart. J. R. Met. Soc. **105**, 811-827.

CERC (2016) ADMS 5 Technical Specification, http://www.cerc.co.uk/TechSpec

Holtslag, A.A.M. and van Ulden, A.P. (1983) A simple scheme for daytime estimates of the surface fluxes from routine weather data. J. Clim. Appl. Met. **22**, 517-529.

Hunt, J.C.R., Stretch, D.D. and Britter, R.E. (1988) Length scales in stably stratified turbulent flows and their use in turbulence models In Proceedings IMA Conference on Stably Stratified Flow and Dense Gas Dispersion (ed. J.S. Puttock), pp. 285-322. Clarendon Press.

ICRP (2012) Compendium of Dose Coefficients based on ICRP Publication 60. ICRP Publication 119. Ann. ICRP 41(Suppl.)

Oke, T.R. (1987) Boundary layer climates. Second edition. Methuen.

Panofsky, H.A. and Dutton, J.A. (1984): Atmospheric Turbulence, Models and Methods for Engineers and Scientists. John Wiley and Sons, New York.

van Ulden, A.P. and Holtslag, A.A.M. (1985): Estimation of atmospheric boundary layer parameters for diffusion applications. In *J. Clim. Appl. Met.*, **24**, pp. 1194-1207.

B.1 Further Reading

Briggs, G.A. (1985) Analytical parameterisations of diffusion: the convective boundary layer. J. Clim. Appl. Met. 14, 1167-1186.

Carruthers, D.J., Edmunds, H.A., Bennett, M., Woods, P.T., Milton, M.J.T., Robinson, R., Underwood, B.Y. and Franklyn, C.J. (1995). Validation of the UK-ADMS Dispersion Model and Assessment of its Performance Relative to R91 and ISC using Archived LIDAR Data. Study commissioned by Her Majesty's Inspectorate of Pollution (published by DoE). DoE/HMP/RR/95/022.

Clarke, R.H. (1979) A model for short and medium range dispersion of radionuclides released into the atmosphere. NRPB Report NRPB-R91.

Hanna, S.R. and Paine, R.J. (1989) Hybrid plume dispersion model (HPDM) development and evaluation. J. Appl. Meteorol. **28**, 206-224.

Hunt, J.C.R. (1985) Turbulent diffusion from sources in complex flows. Ann. Rev. Fluid

Mech. **17**, 447-458.

Hunt, J.C.R., Holroyd, R.J and Carruthers, D.J. (1988) Preparatory studies for a complex dispersion model. CERC Report HB9/88.

Lamb, R.G. (1982) Diffusion in the convective boundary layer. In Atmospheric Turbulence and Air Pollution Modelling (eds. F.T.M. Nieuwstadt and H. van Dop), pp. 159-230. D. Reidel, Dordrecht.

Moore, D.J. and Lee, B.Y. (1982) An asymmetrical Gaussian plume model. Central Electricity Generating Board Report RD/L/2224N81.

Perry, S.G. (1991) CTDMPLUS: A dispersion model for sources near complex topography. Part I: Technical formulations. J. Appl. Meteor. **31**, 633-645.

Sehmel, G.A. (1980) Particle and gas dry deposition: a review. Atmospheric Environment **14**, 983-1011.

Weil, J.C. (1985) Updating applied diffusion models. J. Clim. Appl. Met. **24**, 1111-1130.

CERC