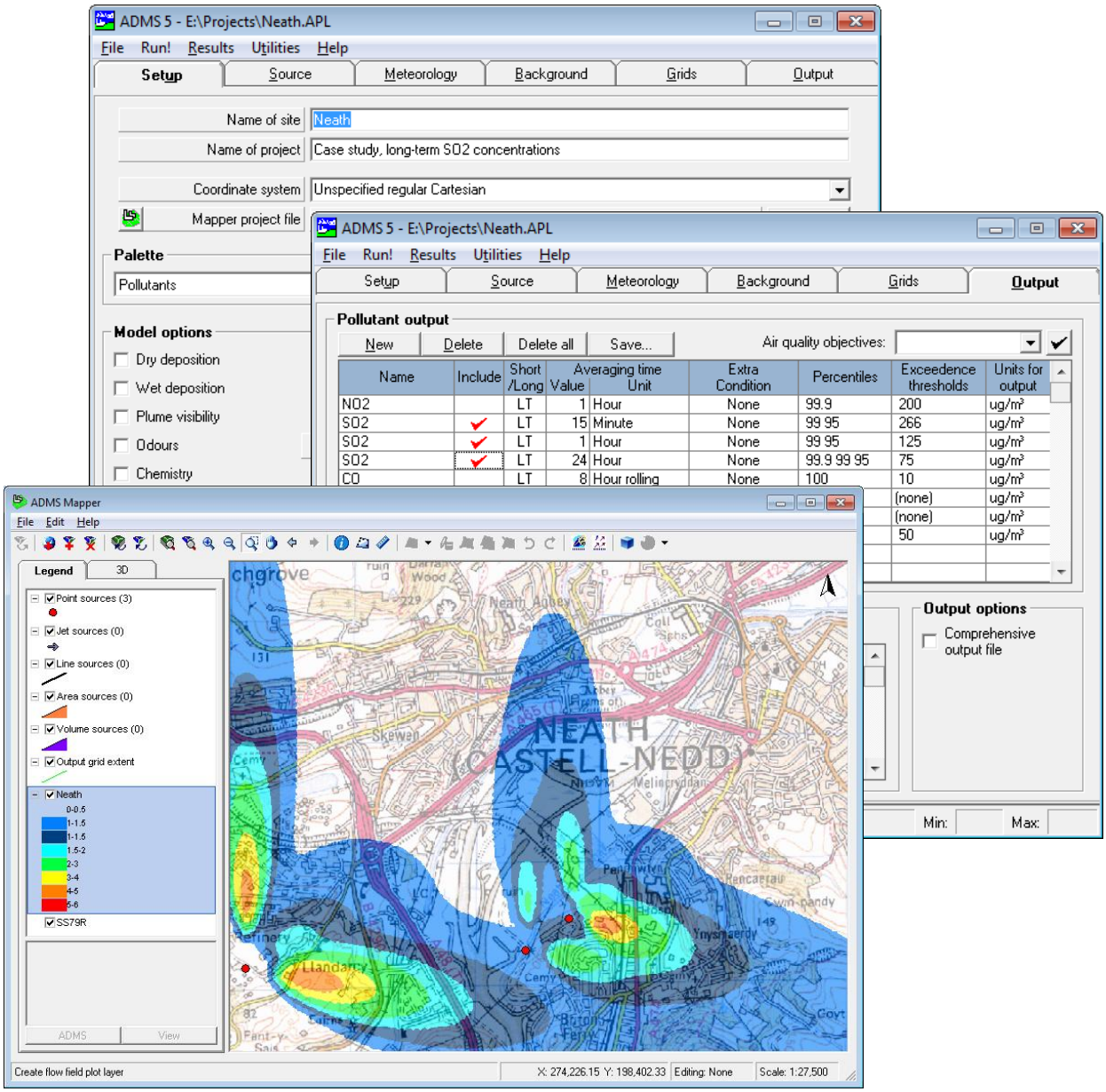
ADMS 5

Atmospheric Dispersion Modelling System



User Guide CERC

Copyright © Cambridge Environmental Research Consultants Limited, 2016

ADMS 5

Atmospheric Dispersion Modelling System

User Guide

Version 5.2

November 2016

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

Telephone: +44 (0)1223 357 773
Facsimile: +44 (0)1223 357 492
Email: help@cerc.co.uk
Website: www.cerc.co.uk

Contents

SECTION 1 Introduction 1

1.1 About ADMS 5 1

1.2 Model features 3

1.3 About this User Guide 5

SECTION 2 Getting Started 7

2.1 System requirements 7

2.2 Installation 7

2.2.1 ADMS 5 as an upgrade 7
2.2.2 Installing ADMS 5 7
2.2.3 Installing the ADMS GIS extension 11
2.2.4 Setting up ADMS 5 in Run Manager 11

2.3 Getting around the interface 13

2.3.1 Mouse buttons 13
2.3.2 Keyboard access 13

2.4 Main menu options 14

2.5 Creating a model file and running the model 17

2.5.1 Creating a model file 17
2.5.2 Entering information 17
2.5.3 Saving input data to a model file 18
2.5.4 Additional model input files 18
2.5.5 Verifying the model file 18
2.5.6 Running ADMS 5 19
2.5.7 Running AERMOD 21
2.5.8 Displaying model output 23

SECTION 3 Model Input 25

3.1 Setup screen 26

3.1.1 Name of site, Name of project 26
3.1.2 Coordinate system 26
3.1.3 Mapper project file 27
3.1.4 Palette 27
3.1.5 Radioactivity options 29
3.1.6 Model options 29
3.1.7 Additional input file (.aai) 29

3.2 Source screen 31

3.2.1 Sources and groups 31
3.2.2 Source data 34
3.2.3 Source geometry 37
3.2.4 Source emissions 39
3.2.5 Time-varying emissions 41

3.3 Meteorology screen 42

3.3.1 Site data 43
3.3.2 Met. data 44
3.3.3 Entering meteorological data from a file 45
3.3.4 Entering meteorological data on screen 49
3.3.5 Interpolation of meteorological data 52

3.4 Background screen 53

3.4.1 Entering background data from a file 54
3.4.2 Entering background data by hand 55

3.5 Grids screen 56
3.5.1 Gridded output 57
3.5.2 Specified points output 59

3.6 Output screen 61
3.6.1 Pollutant output 61
3.6.2 Group and source output 65
3.6.3 Comprehensive output file 66
3.6.4 Air quality objectives 67

SECTION 4 Additional Model Options 71

4.1 Time-varying source data 72
4.1.1 Source data from a .var file 72
4.1.2 Emission factors from a .fac file 79
4.1.3 Emission factors from screen 82
4.1.4 Daylight saving time 82
4.1.5 Time-varying emission history 83

4.2 Dry deposition 86
4.2.1 Modelling dry deposition 86
4.2.2 Additional input options 90
4.2.3 Modelling without plume depletion 97
4.2.4 Output 98
4.2.5 Guidance on dry deposition parameters 99

4.3 Wet deposition 100
4.3.1 Modelling wet deposition using a washout coefficient 102
4.3.2 Additional input options 103
4.3.3 Modelling wet deposition using the ‘falling drop’ method 103
4.3.4 Modelling without plume depletion 104
4.3.5 Output 105
4.3.6 Guidance on wet deposition parameters 105

4.4 Radioactive emissions 107
4.4.1 Modelling isotopes without radioactive decay 107
4.4.2 Modelling isotopes with radioactive decay 107
4.4.3 Gamma dose 110

4.5 Plume visibility 111
4.5.1 Threshold for ambient cloud 112
4.5.2 Plume visibility parameters 113
4.5.3 Restrictions 114

4.6 Temperature and humidity output 115
4.6.1 Temperature and humidity output for point sources 115
4.6.2 Temperature output for downstream jet sources 117
4.6.3 Output 117
4.6.4 Restrictions 118

4.7 Odours 119
4.7.1 Input data 119
4.7.2 Restrictions 119

4.8 Chemistry 120
4.8.1 Background concentrations 120
4.8.2 Pollutant parameters 120
4.8.3 Meteorology 120
4.8.4 Averaging times 120
4.8.5 Emissions data 120
4.8.6 Local night-time chemistry 122

4.9 Buildings 123

4.9.1 Building effects 123
4.9.2 Defining buildings 123
4.9.3 Defining the 'main building' for each source 125
4.9.4 Guidance 125

4.10 Complex terrain 127
4.10.1 Terrain height and surface roughness files 127
4.10.2 Modelling the effects of complex terrain 128
4.10.3 Restrictions 129

4.11 User-input 3D flow field data 130
4.11.1 Format of the user-input 3D flow field file 130
4.11.2 Additional input file 131
4.11.3 Guidance 132
4.11.4 Restrictions 132

4.12 Coastline 133

4.13 Puff 135

4.14 Fluctuations 138
4.14.1 Restrictions 140

4.15 Advanced meteorological parameters 141
4.15.1 Surface albedo 142
4.15.2 Priestley-Taylor parameter 142
4.15.3 Minimum Monin-Obukhov length 143
4.15.4 Precipitation 144

4.16 User-input vertical profiles of meteorological data 145

4.17 Boundary layer profile output 147

4.18 Calm conditions 148
4.18.1 Notes on selecting the appropriate value of MinU10 149
4.18.2 Restrictions 149

4.19 Marine boundary layer 150

4.20 Combine flues into a single stack 152

4.21 Switch off stack downwash 154

4.22 User-defined minimum turbulence 155

4.23 Wind turbine effects 156
4.23.1 Wind turbine data file 158
4.23.2 Wind farm data file 159
4.23.3 Wind turbine output 161
4.23.4 Restrictions 161

4.24 Urban canopy flow 162
4.24.1 Urban canopy flow input data file format 163
4.24.2 Restrictions 165

4.25 Extend output files 166

SECTION 5 Import and Export 167

5.1 File formats 167
5.1.1 .spt file 167
5.1.2 .vgt file 169
5.1.3 .eit file 169
5.1.4 .gpt file 169
5.1.5 .ptt file 170
5.1.6 .bpt file 171
5.1.7 Import templates 171

5.2 Import 172

5.2.1 Select files to import 172
5.2.2 Filter sources by type 173
5.2.3 Select sources 174
5.2.4 Source settings 175
5.2.5 Pollutant settings 176
5.2.6 Building settings 177
5.2.7 Group settings 178
5.2.8 Check import 179

5.3 Export 180

SECTION 6 Model Output 181

6.1 ADMS 5 output files 182
6.1.1 .err, .wng, .log and .rpt files 182
6.1.2 .cen file 182
6.1.3 .glt, .levels.glt and .plt files 183
6.1.4 .gst, .levels.gst, .gtd, .levels.gtd, .pst and .ptd files 184
6.1.5 .max file 186
6.1.6 .mop file 187
6.1.7 .nc file 187
6.1.8 .out.met file 190
6.1.9 Boundary layer profile: .pro file 191
6.1.10 Buildings: .bef, .bld and .bwk files 191
6.1.11 Deposition: .dep file 191
6.1.12 Fluctuations: .fst file 192
6.1.13 Flowfield: .W01 files 192
6.1.14 Flowfield: .wlt file 192
6.1.15 Flowfield: .T01 files 193
6.1.16 Flowfield: .tlt file 193
6.1.17 Flowfield: .zst file 194
6.1.18 Flowfield: .zlt file 195
6.1.19 Plume visibility: .vlt and .vst files 195
6.1.20 Radioactivity: .rad file 196
6.1.21 .!01 and .~01 extensions 196
6.1.22 Increasing the length of output files 196

6.2 AERMOD files 198
6.2.1 Meteorological settings file 198
6.2.2 Error, warning and log files 198
6.2.3 Concentration and deposition output files 199
6.2.4 ADMS meteorological data file 199
6.2.5 Intermediate files 199
6.2.6 Complex terrain files 200
6.2.7 Buildings files 200
6.2.8 AERMET meteorology files 201

6.3 Plots in Surfer (2-D Output Plotter) 202
6.3.1 Contour 202
6.3.2 Flow field 204
6.3.3 Advanced options 208

6.4 Line plots (ADMS Line Plotter) 211
6.4.1 Main interface 211
6.4.2 Graph customisation 214

6.5 Viewing numerical data 216
6.5.1 Choosing application to view output 216
6.5.2 Viewing output from the ADMS 5 interface 216
6.5.3 Use of Microsoft Excel to view numerical output 217
6.5.4 Use of Microsoft Excel to create a time series graph 219

6.6 Model run reports 221

SECTION 7 Utilities 223

7.1 ADMS Mapper 224

7.2 Visualising input data in Surfer 225
7.2.1 Display and/or save the visualisation 225
7.2.2 Layout of visualised features 226

7.3 Converting meteorological data 230
7.3.1 Running the utility 230
7.3.2 Format of the output meteorological data file (.met) 232
7.3.3 Format of the report file (.rpt) 233

7.4 Making a terrain file 234
7.4.1 Creating an ADMS 5 terrain file from OS Landform NTF (GB) data 235
7.4.2 Creating an ADMS 5 terrain file from IGN XYZ (FR) data 238
7.4.3 Creating an ADMS 5 terrain file from OS Northern Ireland DTM (NI) data 240
7.4.4 Creating an ADMS 5 terrain file from OS Northern Ireland XYZ (NI) data 243
7.4.5 Creating an ADMS 5 terrain file from OS Ireland XYZ (EIRE) data 243

7.5 Making and running a batch file 244
7.5.1 Format of batch files 244
7.5.2 Batch File Creator utility 244

7.6 Viewing a wind rose 246
7.6.1 Categories for the data 246
7.6.2 Using the wind rose utility 247

SECTION 8 Worked Examples 249

8.1 Example 1: Single point source under different meteorological conditions 249
8.1.1 Setting up the run 249
8.1.2 Viewing output results 257

8.2 Example 2: Long-term average and percentile concentrations 264
8.2.1 Setting up the run 264
8.2.2 Viewing output results 269

8.3 Example 3: Time-varying emissions with sequential meteorological data 272
8.3.1 Setting up the run 272
8.3.2 Viewing output results 279

8.4 Example 4: Radioactive releases 283
8.4.1 Example 4a: radioactive emissions and deposition 283
8.4.2 Example 4b: radioactive emissions and gamma dose 299

8.5 Data files supplied with ADMS 5 305

SECTION 9 Technical Summary 307

9.1 Meteorological input and output 307
9.1.1 Input 307
9.1.2 User-input vertical profile data 311
9.1.3 Meteorological data pre-processing 311
9.1.4 Output 312
9.1.5 Limitations 312

9.2 Parameterisation of the boundary layer 314
9.2.1 Boundary layer structure 316

9.3 Dispersion over flat terrain 317
9.3.1 Dispersion parameters 317
9.3.2 Stable and neutral boundary layers 317
9.3.3 Convective boundary layer 318
9.3.4 Special treatment of ‘calm’ wind conditions 320

9.4 Plume rise 322

9.4.1 Stack-induced downwash 322

9.5 Source types 324
9.5.1 Point sources 324
9.5.2 Area, volume and line sources 324
9.5.3 Jets and directional releases 326

9.6 Multiple sources 328
9.6.1 Groups 328
9.6.2 Sources 328
9.6.3 Pollutants 328
9.6.4 Particle sizes and gases 328
9.6.5 Combining multiple sources into stacks 328

9.7 Output grids and points 331

9.8 Averaging times and statistics 332
9.8.1 Averaging times of one hour or longer 332
9.8.2 Averaging times shorter than one hour 333
9.8.3 Maximum daily output 333
9.8.4 Finite duration releases 333
9.8.5 Limitations 334
9.8.6 Long-term statistics 334

9.9 Marine boundary layer 335
9.9.1 Wind profile 335
9.9.2 Heat fluxes 335
9.9.3 Input data requirements 336

9.10 Dry deposition 337

9.11 Wet deposition 339
9.11.1 Washout coefficient method 339
9.11.2 ‘Falling drop’ method 341

9.12 Radioactive decay 342
9.12.1 Limitations 342

9.13 Gamma ray dose rate 343
9.13.1 Gamma ray dose rate due to airborne radioactive material 343
9.13.2 Gamma ray dose rate due to deposited radioactive material 343
9.13.3 Limitations 344

9.14 Plume visibility 345
9.14.1 Visibility criteria 345
9.14.2 Temperature and humidity output 346
9.14.3 Limitations 346

9.15 Chemistry 347
9.15.1 Limitations 347

9.16 Buildings 348
9.16.1 Determination of the ‘effective building’ 349
9.16.2 Limitations 351

9.17 Complex terrain 352
9.17.1 FLOWSTAR-D solution 352
9.17.2 Very stable flow solution 352
9.17.3 Regions of reverse flow 353
9.17.4 Running the complex terrain module 353

9.18 User-input 3D flow field 355

9.19 Coastline 357
9.19.1 Limitations 357

9.20 Puffs 358

9.21 Fluctuations 359

9.21.1 Calculation procedure 359
9.21.2 Short-term output 360
9.21.3 Long-term output 361
9.21.4 Limitations 361

9.22 Fires and flares 362
9.22.1 Source height 363
9.22.2 Heat release Fb 364
9.22.3 Momentum flux Fm 365

9.23 Wind Turbine Effects 366
9.23.1 Representing a wind turbine as an effective volume source 367
9.23.2 Shear-induced turbulence 369
9.23.3 Treatment of wind turbine interaction 370

9.24 Urban canopy flow 372
9.24.1 Input data 372
9.24.2 Velocity profile 372
9.24.3 Turbulence profile 373
9.24.4 Flow regimes 373

9.25 Running AERMOD 374
9.25.1 Meteorological data 374
9.25.2 Output 375
9.25.3 Complex terrain 375
9.25.4 Buildings 375

APPENDIX A Model Limits 377

A.1 Sources, groups and pollutants 377

A.2 Concentration output 379

A.3 Using multiple model options in the same run 380

A.4 Restrictions on running AERMOD 381
A.4.1 Sources and groups 381
A.4.2 Output types and locations 381
A.4.3 Model options 382

APPENDIX B NOx in ADMS 5 385

B.1 What is NOx? 385

B.2 What does 'NOx as NO2' mean? 385

B.3 Treatment of NOx in ADMS 5 386

B.4 NO2 emissions in ADMS 5 386

B.5 Converting between 'NOx as NO2' and 'True NOx' 386

APPENDIX C Air Quality Limits and Guidelines 389

C.1 UK Air Quality Strategy and Regulations 390

C.2 EU limit values 393

C.3 Lithuanian limit values 395

C.4 French Limit Values 396

C.5 US National Ambient Air Quality Standards 399

C.6 World Health Organisation guidelines 401

APPENDIX D Surfer Tips 403

D.1 Contour maps (2-dimensional) 403

D.2 Surface maps (3-dimensional) 406
D.2.1 Create the grid file 406
D.2.2 Create the 2-D contour map 406
D.2.3 Create the 3-D surface map 406

D.3 Overlay on a digital or surface map 409
D.3.1 Prepare the map 410
D.3.2 Prepare the item to overlay 411
D.3.3 Overlay the item on the map 412

D.4 Customise a map 415
D.4.1 Levels 415
D.4.2 Contour colour and pattern 416
D.4.3 Line colour, thickness and pattern 419
D.4.4 Contour labels 419
D.4.5 Colour scale 420
D.4.6 Axes and grid lines 420
D.4.7 Scale and extent 421

D.5 Save or print a map with transparency properties 422

APPENDIX E Useful Contacts 423

E.1 ADMS 5 contact information 423

E.2 Contact details for ADMS 5 input data 424

E.3 Output visualisation tools 425

E.4 Official organisations 426

APPENDIX F References 427

SECTION 1 Introduction

1.1 About ADMS 5

What is ADMS 5?

ADMS 5 is a practical, short-range dispersion model that simulates a wide range of buoyant and passive releases to the atmosphere, either individually or in combination. It is a "new generation" dispersion model which uses two parameters to describe the atmospheric boundary layer, namely the boundary layer height *h* and the Monin-Obukhov length $L_{MO,}$ and a skewed Gaussian concentration distribution to calculate dispersion under convective conditions. The model is applicable up to about 60 km downwind of the source and provides useful information for distances up to 100 km.

ADMS 5 is supplied with a Mapper that can be used to visualise, add and edit sources, buildings and output points and to view model output. It also links to other software packages, such as Surfer, a contour plotting package for easy and effective display of results, and ArcGIS and MapInfo GIS (Geographical Information System) software, for display of results and easy data entry. Separate User Guides are provided for using the ADMS Mapper and the links to ArcGIS and MapInfo. More information about using Surfer with ADMS 5 is provided in Appendix D.

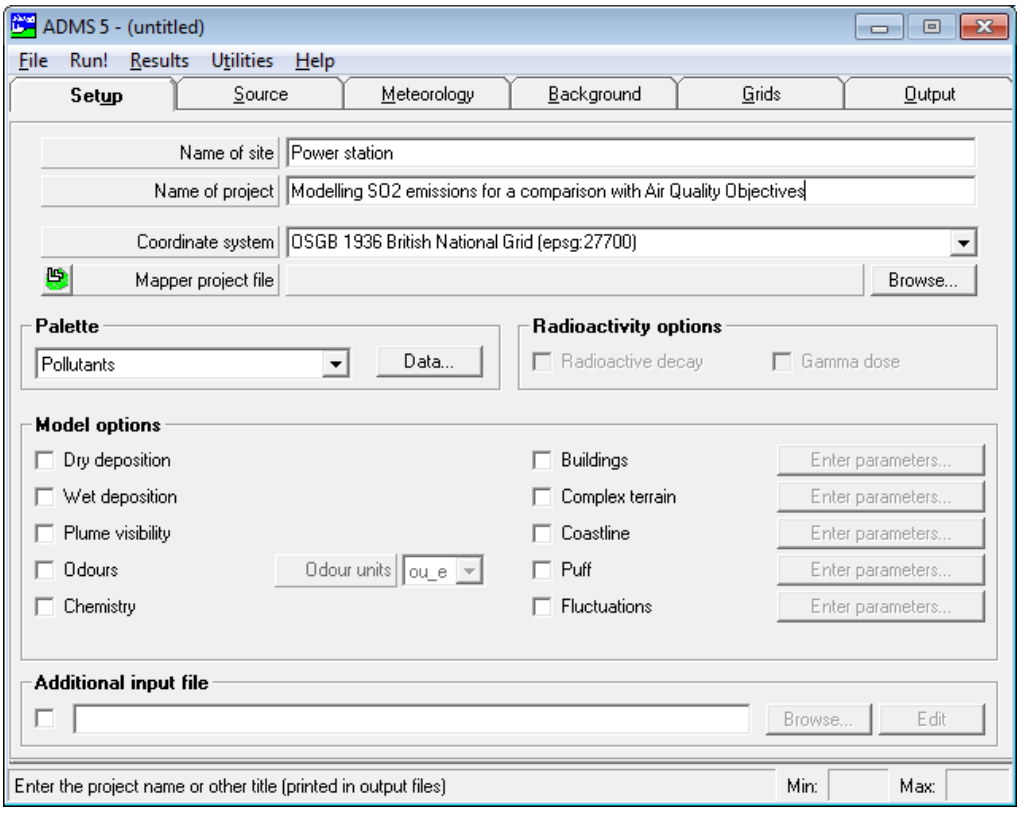


Figure 1.1 − The **Setup** screen of ADMS 5 showing model options.

What are the main applications of ADMS 5?

Typical applications include planning and permitting, stack height assessment, odour modelling, environmental impact assessments and safety planning. ADMS 5 is used by regulators, government departments, industrial companies and consultancies.

Model validation

The model has been extensively validated against field data sets. Since 1992 CERC have been key participants in the series of "Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes" workshops, hosting the 11th Conference in the UK in 2007. The workshops have included validation of models against field data sets, analysis of the validation results and discussion of validation techniques.

Appendix F gives details of validation for ADMS 5 as well as other publications relevant to ADMS 5. Documents describing the latest model validation are available on the CERC website (www.cerc.co.uk).

1.2 Model features

The basic ADMS 5 dispersion model calculates long-term and short-term concentrations and deposition fluxes from continuous point, jet (directional release), line, area and volume sources. Long-term output can be in terms of averages, rolling averages, percentiles or number of exceedences for direct comparison with air quality objectives and limits. Sources can be time-varying on an hourly, diurnal or monthly basis. If background ambient concentration data are supplied by the user the output will be the combined concentration due to the process emissions and the ambient concentration, sometimes called the PEC (Predicted Environmental Concentration).

A meteorological pre-processor developed by the UK Met Office is part of the model and calculates values of the meteorological parameters in the boundary-layer required by ADMS 5 from the input meteorological data.

In addition to the basic model of dispersion over flat terrain ADMS 5 can model:

- the effect of plume rise;
- the effect of buildings;
- the effect of wind turbines;
- the effect of hills (complex terrain) and spatial variations in surface roughness;
- the effect of buildings in an urban area (urban canopy);
- the effect of a coastline;
- NO_X chemistry using a simple reaction scheme for NO_X, NO_2 and ozone;
- the kinetics of the uptake of gases, and the thermodynamics and chemistry of the dissolution of gases in raindrops for wet deposition of SO_2 and HCl;
- short-term fluctuations in concentration due to atmospheric turbulence, particularly important for the modelling of odours and concentrations for averaging times less than one hour; and
- radioactive decay and gamma dose.

ADMS 5 also includes options to calculate:

- the effect of the sea or ocean on the atmospheric boundary layer;
- dispersion in a 3-D wind and turbulence field, for instance one generated by a CFD code;
- plume visibility due to condensed water;
- concentrations in units of ou_e for odour studies; and
- concentrations and deposition fluxes due to an instantaneous or finite duration release (puffs).

ADMS 5 has a number of utilities:

- a verification mode: the *.apl* file created can be verified before it is run to warn the user of errors in input before the full calculations are performed;
- a tool to visualise in Surfer the location of sources, buildings, receptors, output grid, area of terrain and coastline;
- a wind rose plotter;
- a meteorological data converter to convert data supplied in a U.S. data format to ADMS 5 format meteorological data files;
- a terrain data utility to create terrain data files for ADMS 5 from commonly available data formats; and
- a utility to import and export source, pollutant and building data from or to a set of comma-separated files.

ADMS 5 also has a facility to run the main model options of AERMOD using ADMS meteorological data from the ADMS 5 interface. These options include the main dispersion module as well as the effects of spatially varying terrain height and buildings.

1.3 About this User Guide

This *ADMS 5 User Guide* is both a manual and a technical summary of the model.

Conventions

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

- ADMS 5 interface controls are shown in **Arial** font, e.g. the **Grids** screen, click on the **Plot** button.
- Keyboard keys are shown in **bold**, e.g. press **Enter**.
- Directory and file names are shown in *italics*, e.g. *adms.exe*, *<install_path>\Data*.
- Tips and other notes are shown thus:

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

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

SECTION 2 Getting Started

2.1 System requirements

ADMS 5 will run in Windows 7, Windows 8 and Windows 10 environments. The minimum memory specification is 1 GB of RAM and 10 GB of hard disk space.

However, it is recommended that the highest specification machine available should be used for running simulations. Higher processor speeds and larger amounts of RAM will be particularly beneficial. As ADMS 5 does not currently take advantage of dual processing capabilities, there is no benefit in using a dual processor PC for a single run. However, up to four runs can be run simultaneously on a single, multi-processor PC, providing the computer has sufficient RAM.

2.2 Installation

The installation of ADMS 5 is straightforward. It uses an Installation Wizard, which guides the user through a short series of screens, collecting information on the user and installation parameters, before installing the software.

For users who already have ADMS installed in an earlier version, this version should be uninstalled before installing ADMS 5. Please proceed as explained in Section 2.2.1 and then follow instructions given in Section 2.2.2.

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

2.2.1 ADMS 5 as an upgrade

If you have a previous version of ADMS installed on your computer, this should be uninstalled before installing ADMS 5.

To uninstall a previous version, proceed as follows. Click on the Windows **Start** button. From the **Start** menu, select **Control Panel**. Select **Programs and features** and then select the program name (ADMS) from the list of installed applications and then click **Uninstall**.

2.2.2 Installing ADMS 5

The following steps lead you through the ADMS 5 installation process.

Step 1 Log on as Local Administrator for the PC.

Step 2 ADMS 5.2 will either have been supplied by download link or on CD. Follow the appropriate instructions:

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

CD: Insert the installation CD and the install program should automatically start. If it does not, browse to locate the CD in Explorer and double-click on the file '*setup.exe*'.

In both cases, the screen shown in **Figure 2.1** will be launched.

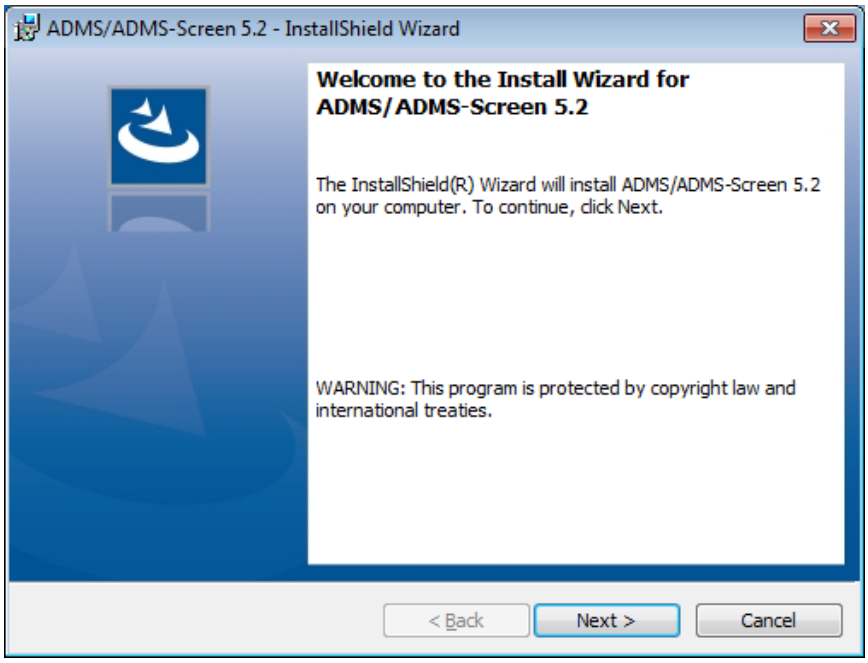


Figure 2.1 – The ADMS 5.2 installation **Welcome** screen.

Step 3 Click **Next >** on the **Welcome** screen and then 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 cancel the installation process.

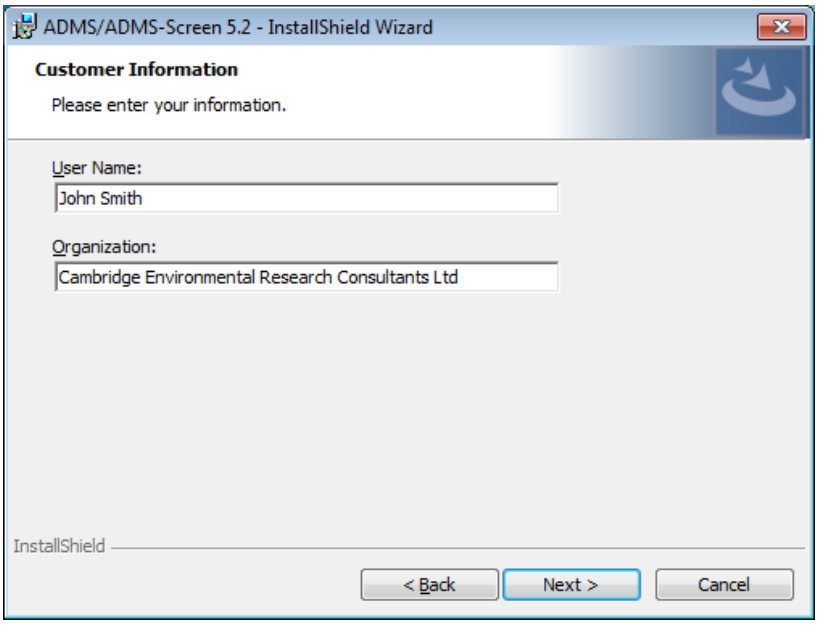


Figure 2.2 – The ADMS 5.2 **Customer Information** screen.

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

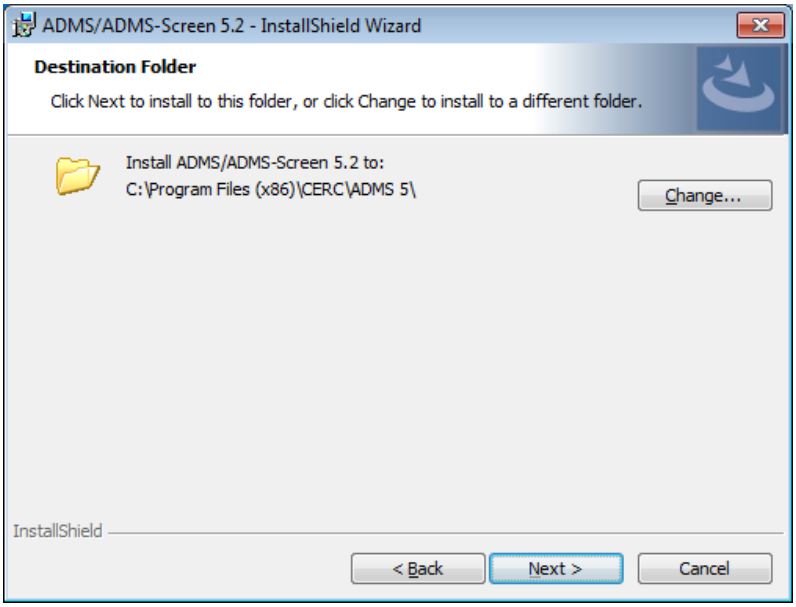


Figure 2.3 − The ADMS 5.2 **Destination Folder** screen.

Step 5 You should select a drive with at least 1 GB of available disk space. The default installation directory is *C:\Program Files (x86)\CERC\ADMS 5*. If required, use the **Change**... button to select your own installation directory. Click **OK** to return to the **Destination Folder** screen.

The abbreviation <install_path> *will be used in the rest of the User Guide to denote the installation directory you have chosen, for example* C:\Program Files (x86)\CERC\ADMS 5.

Click **Next >** to choose who should be able to use ADMS 5, as shown in **Figure 2.4**.

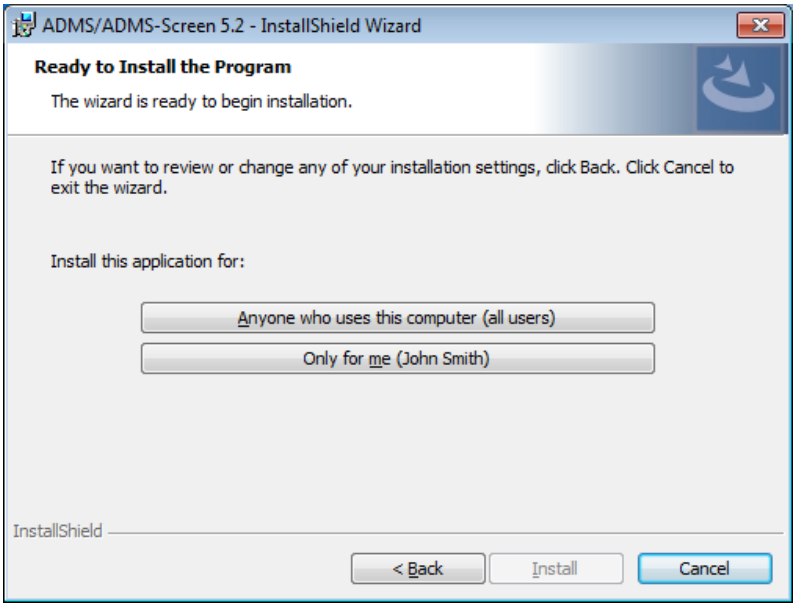


Figure 2.4 − The ADMS 5.2 **Ready to Install** screen.

Step 6 Chose 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.

If you wish to amend any details, press the **< Back** and **Next >** buttons as appropriate. Once the **Install** button has been pressed, and the ADMS 5 files have been successfully installed, the final screen will appear, as shown in **Figure 2.5**.

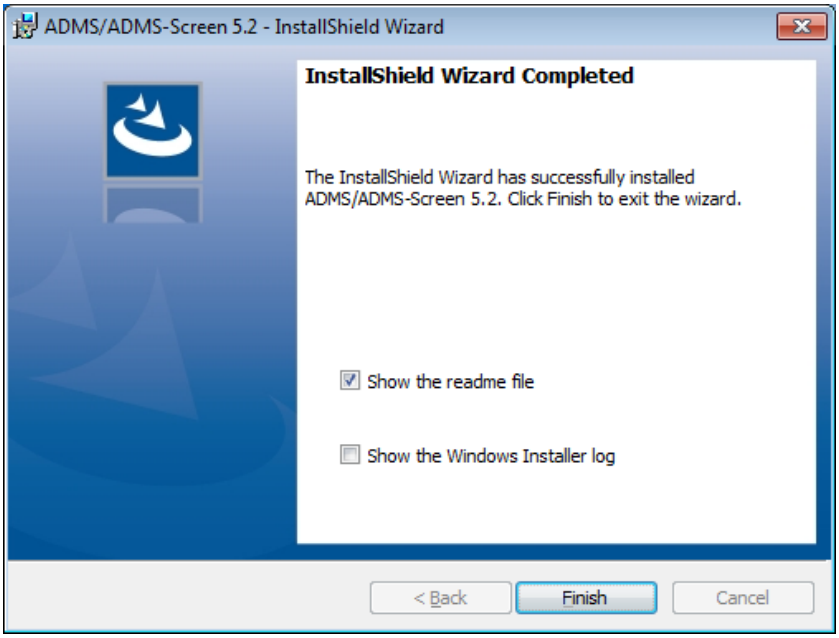


Figure 2.5 – The ADMS 5 **InstallShield Wizard Completed** screen.

Step 7 Click **Finish** to complete the installation. The installation procedure automatically puts a shortcut to ADMS 5 on your Windows desktop. If the **Show the readme file** box is checked the document *What's New in ADMS 5?* will be opened automatically once you click on **Finish**.

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 8 To install the ADMS 5 licence, copy the file *ADMS.lic* to the *<install_path>* directory.

*Launching ADMS 5 and checking the licence details (through **Help, Licence Details**) will give the location of the licence currently being used. Although the model will run when the licence file is in the Windows WINNT directory, it is recommended that the location be the* <install_path> *directory.*

Step 9 Restart your computer: you are now ready to use the model.

Step 10 The first time you start the model you will be presented with a choice of models, as shown in **Figure 2.6** , ensure ADMS is selected and click **OK**.

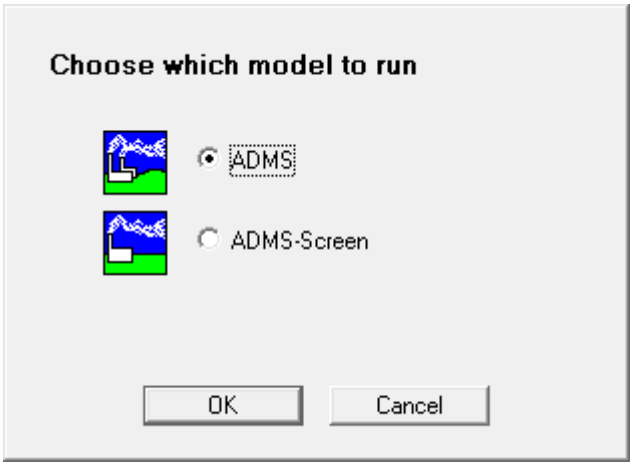


Figure 2.6 − The model choice screen.

2.2.3 Installing the ADMS GIS extension

If you will be using ADMS 5 in conjunction with the GIS packages ArcGIS or MapInfo, you should install the extension now. The installations for the extensions will have been copied to the *<install_path>\Support* directory. Further instructions for installing these links can be found in their respective User Guides in the *<install_path>\Documents* directory.

2.2.4 Setting up ADMS 5 in Run Manager

Run Manager is a tool to help users manage their model runs and make the most of their CERC licence(s). Please contact CERC if you would like to purchase a Run Manager licence. Instructions for setting up ADMS 5 as a model in Run Manager are included here for convenience.

Adding ADMS 5 to the available models in Run Manager can be done using the **Models** screen. To view the **Models** screen in Run Manager, choose **Options > Global > Models** from the main menus, or use the keyboard shortcut (**Ctrl+M**).

To add a new model, click **<New Model>** at the end of the list of **Available Models** and complete the bottom half of the screen. **Figure 2.7** shows the standard settings for ADMS 5.2.

The standard required *.dll* files (*sx32w.dll* and *netcdf.dll*) are automatically included after selecting the **Executable** using the top .. button. The ADMS 5 licence file (*ADMS.lic*) and three *.dat* files (*Decay.dat*, *GDatbase.dat* and *Halflife.dat*) must also be added to the list of required files. To add these files click the middle .. button and browse to the files, the *.dat* files can be found in the ADMS 5 installation directory. To remove a file from the list, select the file in the **Required Files** list and press the **Delete** key on the keyboard.

The `/e2` and `/remote` **Command Line Parameters** are required to enable Run Manager to monitor the progress of an ADMS 5 run, these parameters are added by default. To add additional parameters, click the lowest .. button and type the parameter. The parameters should be added individually, and can be added with or without the leading slash. To remove a parameter from the list, select the parameter in the list and press the **Delete** key on the keyboard.

The settings below will enable Run Manager to execute typical ADMS 5 model runs. Advanced model users may wish to consult the model documentation for additional files and command line parameters for very specialised runs.

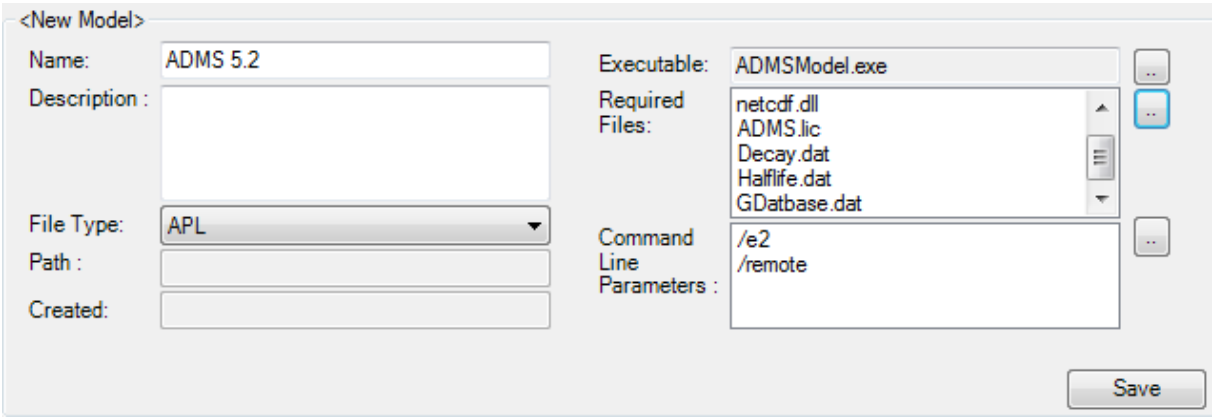


Figure 2.7 – Run Manager Model settings for ADMS 5.

Press **Save** to make the new model available for new runs. This can be checked using the **Add Runs** screen.

When the `/remote` command line flag is used, as is standard when the model is run through Run Manager, the model will create an additional output file with the *.OK* extension, to indicate that the run has successfully completed.

2.3 Getting around the interface

2.3.1 Mouse buttons

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

2.3.2 Keyboard access

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

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

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

Table 2.1 − Keystrokes to enable you to move through the ADMS 5 interface.

2.4 Main menu options

The menu bar has five headings: **File**, **Run!**, **Results**, **Utilities** and **Help**. All menu headings except **Run!** have drop-down lists of options (see for example the **File** menu options shown in **Figure 2.8**). **Table 2.2** gives the list of options and roles of each menu item.

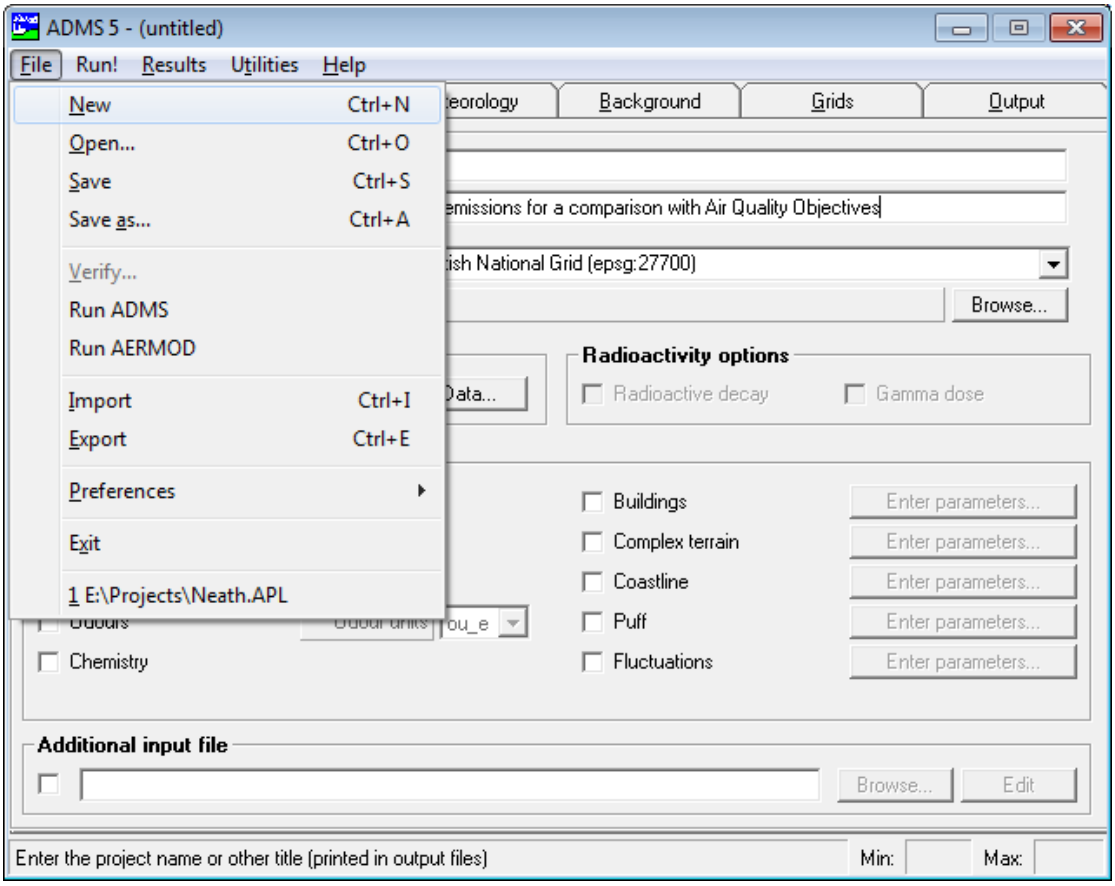


Figure 2.8 − The **File** menu from the menu bar in ADMS 5.

Menu			Role	Ref.
File	New		Reset the parameters in the *.apl* file to their default values	
	Open…		Open a previously saved parameter file	
	Save		Save the current parameters under the current file name	
	Save As…		Save the current parameters with a user-specified file name	
	Verify…		Run verification checks on the *.apl* file	2.5.5
	Run ADMS		Check the validity of the input data and run the ADMS 5 code using the current parameter file	2.5.6
	Run AERMOD		Run AERMOD using the current parameter file	2.5.7
	Import		Import sources, pollutants and buildings from a set of comma-separated input files	5.2
	Export		Export sources, pollutants and buildings to a set of comma-separated input files	5.3
	Preferences	Percentiles method	Force the model to use one of the available percentiles methods (normally, the model chooses the method)	
	Preferences	APL verification	Change the settings that affect when APL verification should run and how the results of APL verification should be presented	2.5.5
	Preferences	Model execution	Choose options for the run time window state and exit mode	2.5.6
	Preferences	Viewing options	Choose the application for viewing numerical results files in ADMS 5 (the default application is Microsoft Excel)	6.5
	Preferences	Coordinate system	Specify the coordinate system to be used by default	3.1.2
	Preferences	Start as	Choose the product mode to run	
	Exit		Quit ADMS 5	
	1, ...		Show the names of the *.apl* files most recently opened in ADMS 5 (click on the required file name to open the selected file in ADMS 5)	
Run!			Check the validity of the input data and run the ADMS 5 code using the current *.apl* file	2.5.6
Results	Plot in Surfer…		Launch the 2D plotting facility (if Surfer is installed)	6.3
	Line Plot		Launch the line plotting facility	6.4
	Numerical…		Open the numerical output of the current *.apl* file in the preferred viewing software	6.5
	Process comprehensive output		Launch the ADMS Comprehensive Output File Processor	#
	Reports from model run		Show the verification report and/or other information output files for the current *.apl* file	6.6

Table 2.2 − Options and roles of menu items. The last column (Ref.) indicates the section of the user guide where the item is further described. # Please refer to the *ADMS Comprehensive Output File Processor User Guide* for more details.

Menu		Role	Ref.
Results	**Open results folder**	Open the folder containing the *.apl* file in an Explorer window	
	Automate Surfer contour plots	Launch an Excel template containing macros for the automation of contour plotting in Surfer (if Surfer and Excel are installed)	*
	Make Slideshow	Launch an Excel template containing macros for creating a slideshow (if Excel is installed)	*
Utilities	**Open Mapper**	Launch the ADMS Mapper	7.1
	Visualise Input in Surfer…	Select features in the current *.apl* file, such as location of sources, for visualisation in a Surfer plot (if Surfer is installed)	7.2
	Convert met. data	Convert US format met. data to ADMS format	7.3
	Create terrain file	Prepare a terrain file from digital terrain data	7.4
	Download terrain data	Link to a website from which SRTM data can be downloaded	7
	Make/Run batch File…	Create and run batch files	7.5
	View a wind rose	Launch the Wind Rose viewer	7.6
	Create import templates	Create templates for the import facility	5.1.7
	Start Excel	Launch Excel (if Excel is installed)	
	Start Surfer	Launch Surfer plotting package (if Surfer is installed)	
Help	**User Guide**	Open the User Guide in a PDF viewer	
	What's New	Open the *What's New* in a PDF viewer	
	Email CERC	Auto-addresses a new email to the ADMS 5 helpdesk in the user's default email client	
	CERC website	Opens the CERC homepage in the user's default internet browser	
	Licence details	Show licensee and licence number of the model	
	About model	Show model version number along with contact information for CERC and the ADMS 5 Helpdesk	

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

2.5 Creating a model file and running the model

To generate results using ADMS 5, there are several steps to complete:

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

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

2.5.1 Creating a model file

When the ADMS 5 model interface is loaded or when you select the **New** command from the **File** menu, a new model file, or scenario, is created and default values are loaded into the screens for you to edit.

To open an existing model file for editing or running, choose **Open**... from the **File** menu. By default, ADMS 5 will display only files with the *.apl* extension.

---

*Files prepared in previous versions of ADMS (version 3.0 and above) are converted into the current ADMS 5 format as they are loaded into the interface. Simply open the file you wish to convert, by selecting **Open...** from the **File** menu, and follow the instructions given on screen.*

2.5.2 Entering information

Changing values in the input screens

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

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

The helpline

This is a single line of text that appears at the bottom of the active screen. The information in the helpline changes when different controls on the screen are selected.

It gives a brief description of the selected control's function. Where you are prompted for a numerical value, the helpline will give the minimum and maximum values allowed.

For example, when the **Latitude (°)** box in the **Meteorology** screen is selected, the help bar will display the line:

Approximate latitude of the source site (°), Min: -90, Max: 90

Data validity and integrity checking

As you enter data, the ADMS 5 model interface performs checks to ensure that all user-entered data are consistent with the model's logic and that minimum and maximum values are satisfied.

2.5.3 Saving input data to a model file

When you are ready to run the model, choose **Save** from the **File** menu. If the current scenario has not been saved before, you will be prompted to choose a directory and file name. ADMS 5 model files are always saved with the extension *.apl*.

2.5.4 Additional model input files

There are a number of additional model input files that may be required by the model in order to run. These are most often comma-delimited text files in a particular format. The most common example of a file used by the model is the *.met* file, which contains the meteorological data required by the ADMS met pre-processor. Other model files include the *.bgd* file that contains hourly values of background concentrations, the *.ter* and *.ruf* files that contain variable terrain height and surface roughness values respectively, and the *.asp* file that contains a list of additional output points.

Most commonly, the user is invited to **Browse**... to locate these additional model input files associated with a model run, and the absolute paths of the files are automatically written to the ADMS 5 model *.apl* file.

If you use Excel to save the additional model input files in comma-delimited format, it is important to open them in a text editor such as Notepad, remove any trailing commas, and re-save them, prior to use by the model.

2.5.5 Verifying the model file

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

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

Additionally, the ADMS 5 interface can be set so that the *.apl* file is verified each time it is saved and also before the model is run. To alter these settings select **File, Preferences, APL verification**. This will bring up the **Data file verification preferences** screen, shown in **Figure 2.9**.

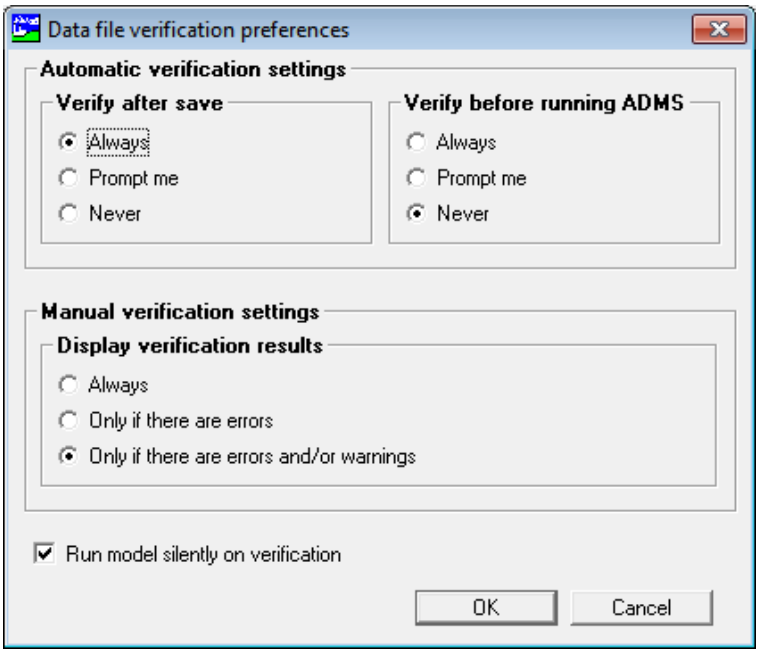


Figure 2.9 − The **Data file verification preferences** screen.

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

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

2.5.6 Running ADMS 5

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

Alternatively, you can run the model with a batch file. Such a file allows you to run several files consecutively without opening and running each *.apl* separately. Batch files can be created with the **Make/Run Batch File** utility (see Section 7.5).

While the model is running, information is displayed in a progress window (see **Figure 2.10**) to show the status of the model run in terms of the number of lines of meteorological data completed. When all the calculations have successfully completed, the progress window displays "`***RUN SUCCESSFULLY COMPLETED***`".

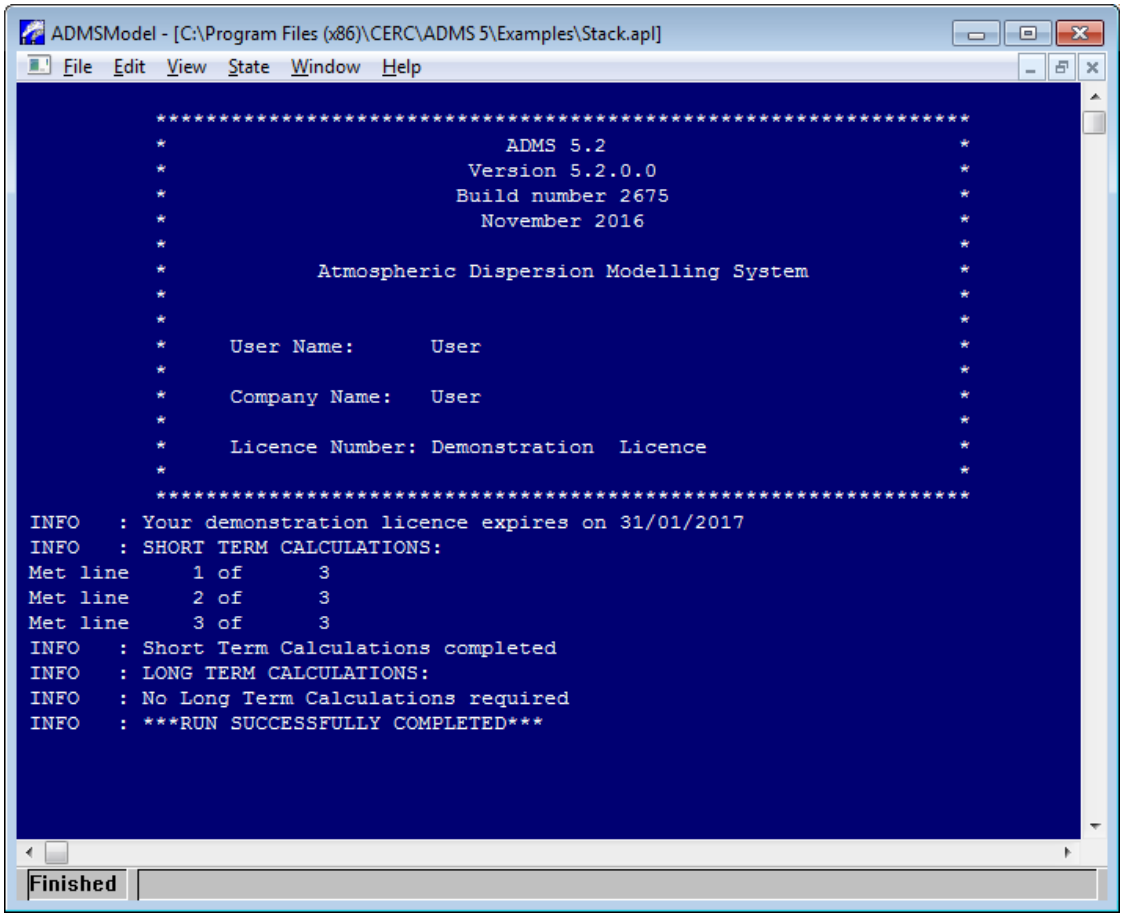


Figure 2.10 − Progress window of an ADMS 5 run.

Runtime preferences

The user can edit the runtime options from the **File, Preferences, Model execution** menu. This will bring up the **Runtime Preferences** screen shown in **Figure 2.11**, 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 whether 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.

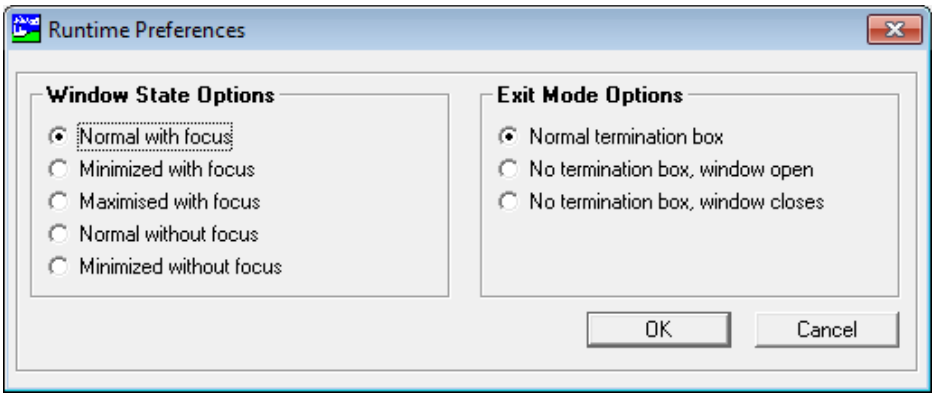


Figure 2.11 – Runtime Preferences screen.

2.5.7 Running AERMOD

To run the current scenario in the AERMOD dispersion model, save the scenario as an *.apl* file and then select **Run AERMOD** from the **File** menu. This will bring up the **Choose met data for AERMOD run** screen (**Figure 2.12**). Select:

- **Use ADMS met. processor** to use the meteorological data entered on the **Meteorology** screen of the ADMS 5 interface;
- **Use AERMET** to use the meteorological data entered on the **Meteorology** screen of the ADMS 5 interface, along with an upper air file, with the AERMET meteorological processor;
- **Define AERMOD met. files**: to use AERMOD surface and profile files instead of the information entered on the **Meteorology screen** of the ADMS 5 interface.

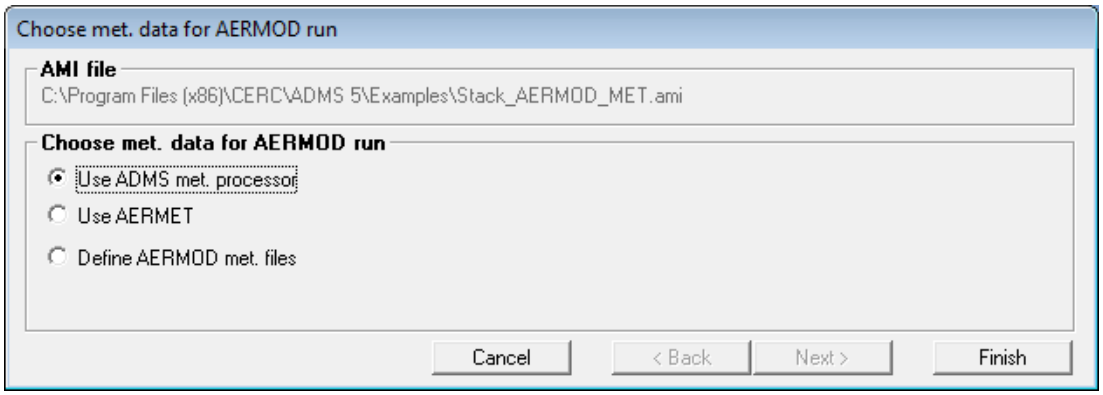


Figure 2.12 – The **Choose met. data for AERMOD run** screen.

Use AERMET

After selecting **Use AERMET** and clicking **Next >** the **Choose met. data for AERMOD run** screen will show the **Use AERMET** form (**Figure 2.13**). Enter the following information:

- **Longitude**: the longitude of the site.
- **Upper air file**: a file containing the upper air soundings required for input into AERMET. The upper air file will be used in AERMET directly and so must be

in a format which can be read by AERMET.

- **Upper air file type**: the type of the upper air file, choose from **FSL**, **6201FB** or **6201VB**.
- **Upper air station ID**: the station ID of the upper air file.
- **Midday Bowen ratio**: how the midday Bowen ratio will be determined. Select either **Calculate**, for the midday Bowen ratio to be calculated from the modified Priestley-Taylor parameter entered into the ADMS 5 interface, or **Define**, to specify a value.

Once the information has been entered click **Finish** to run AERMOD. Click on **< Back** to change the choice for how the meteorological data will be entered.

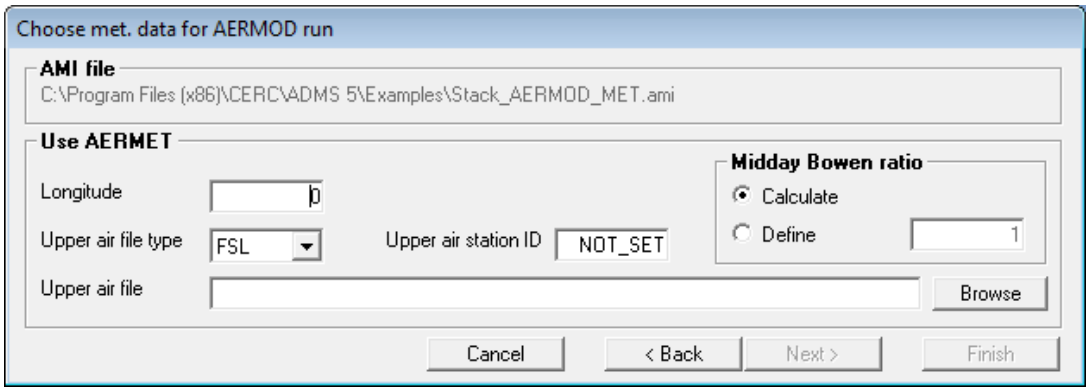


Figure 2.13 – The **Choose met. data for AERMOD run** screen for entering the data required to run AERMET.

Define AERMOD met. files

After selecting **Define AERMOD met. files** and clicking **Next >**, the **Choose met. data for AERMOD run** screen will show the **Define AERMOD met. files** form (**Figure 2.14**). Enter the file paths of the AERMOD **Surface file (SFC)** and **Profile file (PFL)** and click on **Finish**. Click on **< Back** to change the choice for how the meteorological data should be entered.

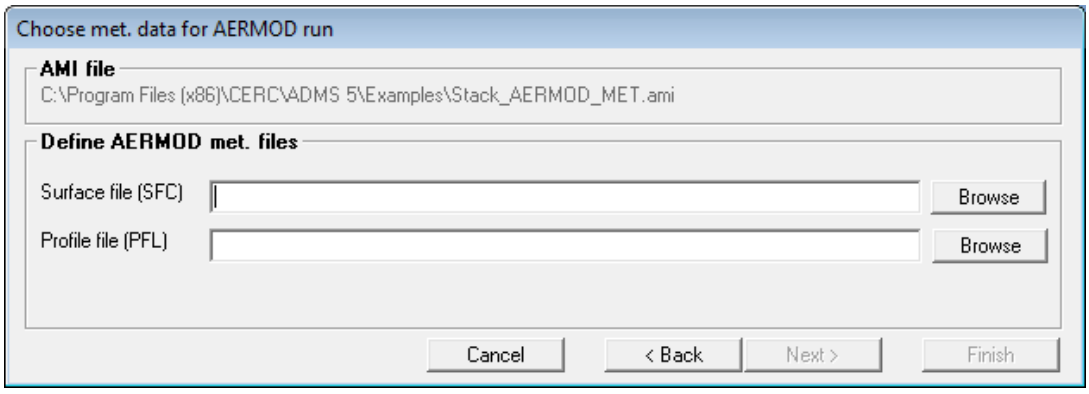


Figure 2.14 – The **Choose met. data for AERMOD run** screen for entering the AERMOD format met files.

The restrictions on which options are available when running with AERMOD are given in Appendix A.4 and descriptions of the output files produced are given in Section 6.2.

2.5.8 Displaying model output

Please refer to Section 6 for details about displaying output from ADMS 5 model runs.

SECTION 3 Model Input

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

There are six basic input screens associated with an ADMS 5 model run, as shown in **Figure 3.1**. Data must be entered into each of these basic screens for every model run. These are:

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

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

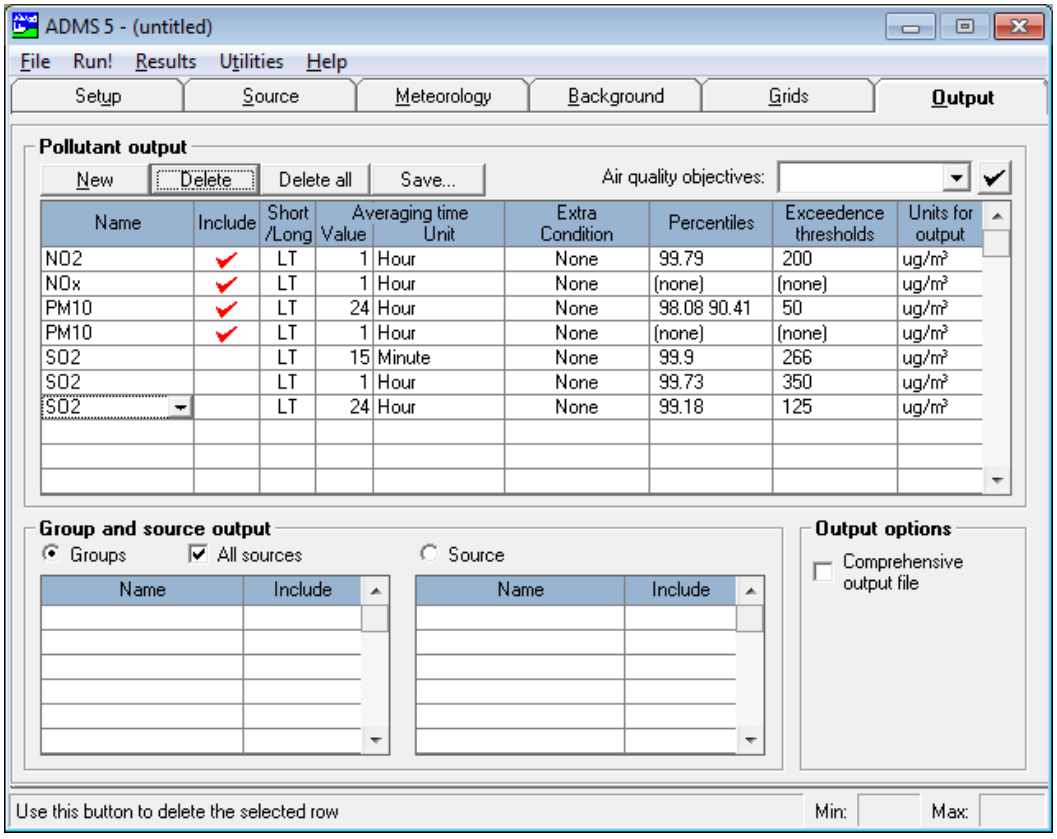


Figure 3.1 − The ADMS 5 interface.

3.1 Setup screen

The **Setup** screen is shown in **Figure 3.2**, and is the screen that appears when ADMS 5 is first opened. The general site details, palette choice and modelling options are entered here.

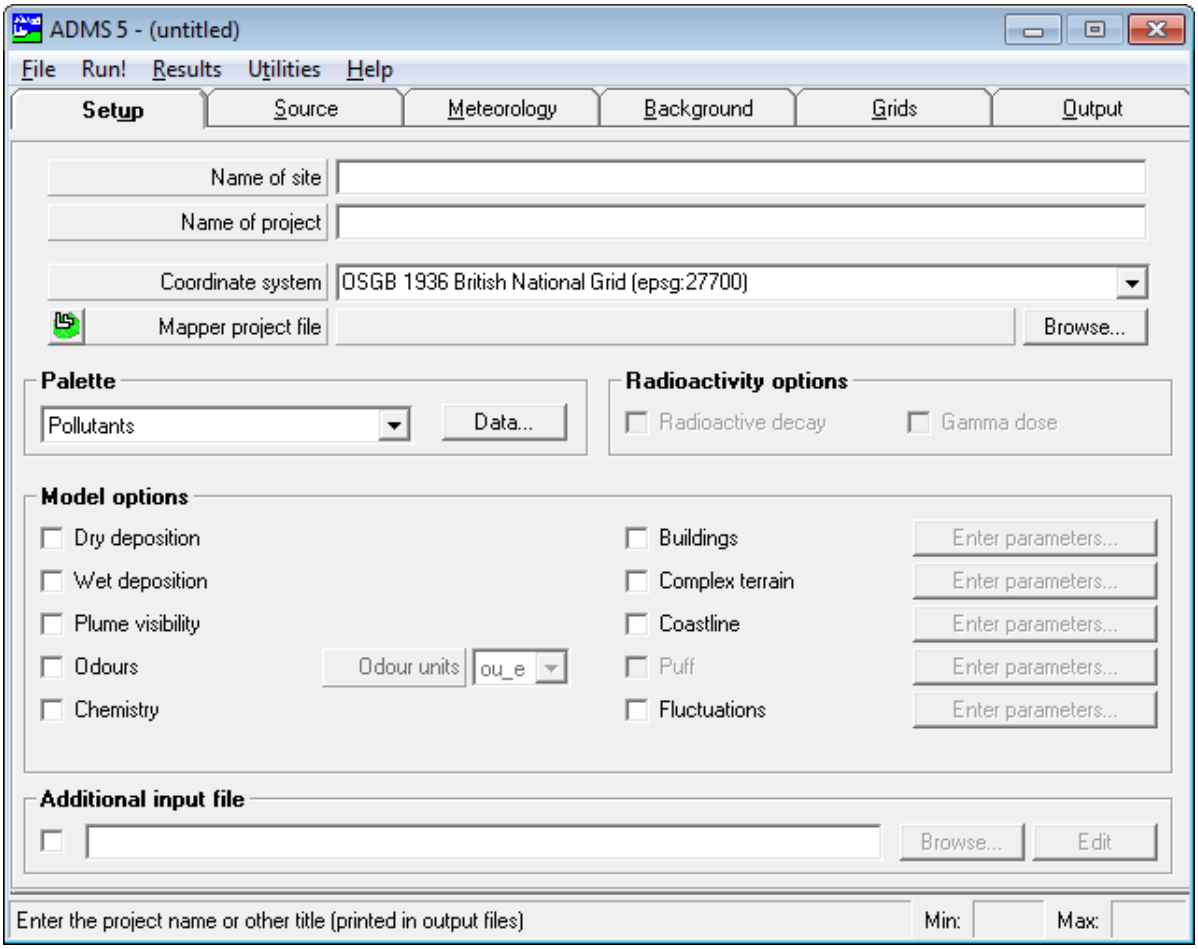


Figure 3.2 − The **Setup** screen.

3.1.1 Name of site, Name of project

Up to 80 characters may be entered for each of these, or the boxes may be left empty. Text entered here will appear in the output "report" file, so it is recommended that meaningful entries are included each time.

3.1.2 Coordinate system

Spatial data entered into ADMS 5 should be in a Cartesian coordinate system, measured in metres. The coordinate system may either be a standard coordinate system, e.g. British National Grid, or site-specific, e.g. centred on the source. The **Coordinate system** should be selected from the list, if a site-specific coordinate system is being used then select **Unspecified regular Cartesian**. The coordinate system is used by the ADMS Mapper and for some export options a recognised coordinate system must be used, for more information refer to the *ADMS Mapper User Guide*.

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

3.1.3 Mapper project file

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

3.1.4 Palette

The **Palette** box provides a choice of palettes. Two palettes are available, **Pollutants** or **Isotopes**, and the required palette should be selected from the list.

Some model options are only available with a particular palette.

The **Data**... button brings up the **Palette of Pollutants** screen which is shown in **Figure 3.3**. The **Palette of Pollutants** screen can also be accessed by clicking on **Pollutants**... in the **Emissions** screen, described in Section 3.2.4.

The purpose of the palette of pollutants is to define properties of pollutants for a given problem. Up to 30 pollutants may be defined: 12 pollutants are pre-defined, namely the gases NO_X, NO_2, NO, O_3, VOC (volatile organic compound), SO_2, CO, benzene, butadiene and HCl (hydrogen chloride), together with the generic particulates PM_{10} and $PM_{2.5}$. NO_X, NO_2, NO, O_3 and VOC may not be deleted from the palette.

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

The **Reset** button restores the palette to its default contents (including the removal of user-defined pollutants).

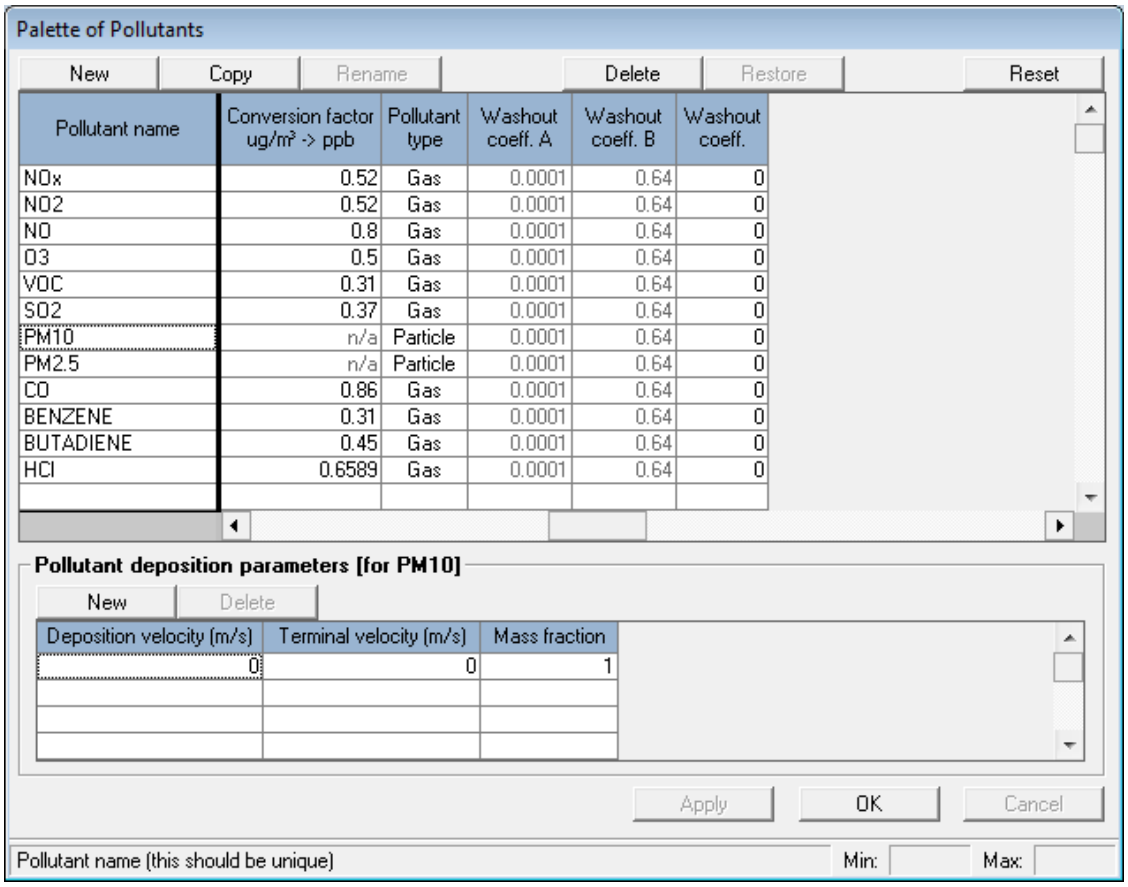


Figure 3.3 − Palette of pollutants in ADMS 5, showing default values for the pre-set pollutants, including the dry deposition parameters for PM_{10}.

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

1. **Pollutant name**

   Each pollutant must have a unique name before the information may be saved. The pollutant name must not contain commas.

2. **Conversion factor (ug/m³ → ppb)**
   For gaseous pollutants, the user should specify a factor to convert output concentrations from $\mu g/m^3$ to parts per billion by volume (ppb). The conversion factor from $\mu g/m^3$ to ppb is 24.06/M (derived from the Ideal Gas Equation), where M is the molecular mass of the pollutant in grams. Default values:

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

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

   Double-click in the **Pollutant type** column to change between **Gas** and **Particle**. The number of columns in the lower table changes according to the different options in the upper table (see 5).

4. **Wet deposition parameters**

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

precipitation). Wet deposition is described further in Section 4.3.

5. **Dry deposition parameters**

   The lower table in the palette gives parameters that are related to dry deposition processes. As different pollutants are selected in the upper table the pollutant name in the title of the pollutant deposition parameters box changes. Different variables must be entered for different pollutant types: deposition velocity or gas type for gaseous emissions; and deposition and terminal velocities or particle size, density and mass fraction for particulate emissions. Dry deposition parameters are described in detail in Section 4.2.

3.1.5 Radioactivity options

The box presents a choice of modelling options which require the isotopes palette. These options are fully described in Section 4.4. Permitted combinations of model options are defined in Appendix A. The **Isotopes** palette must be selected to allow the **Radioactivity options** to be selected.

3.1.6 Model options

The box presents a choice of modelling options to be used in the ADMS 5 run. These options are fully described in Section 4. Permitted combinations of model options are defined in Appendix A.

If no model options are selected, then ADMS 5 will calculate concentrations for a continuous (plume) release in flat terrain.

3.1.7 Additional input file (.aai)

There are a number of additional model options that are accessed via an additional input file (*.aai*). These options are fully described in Section 4. Permitted combinations of model options are defined in Appendix A.

The additional input files used by ADMS 5.2 are version 3.

To use an additional input file check the box in the **Additional input file** section and then enter the path to the file or click on **Browse**... to browse to the file. The **Edit** button launches the **Additional Input file editor** (shown in **Figure 3.4**) for the editing of existing *.aai* files or creation of new ones.

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

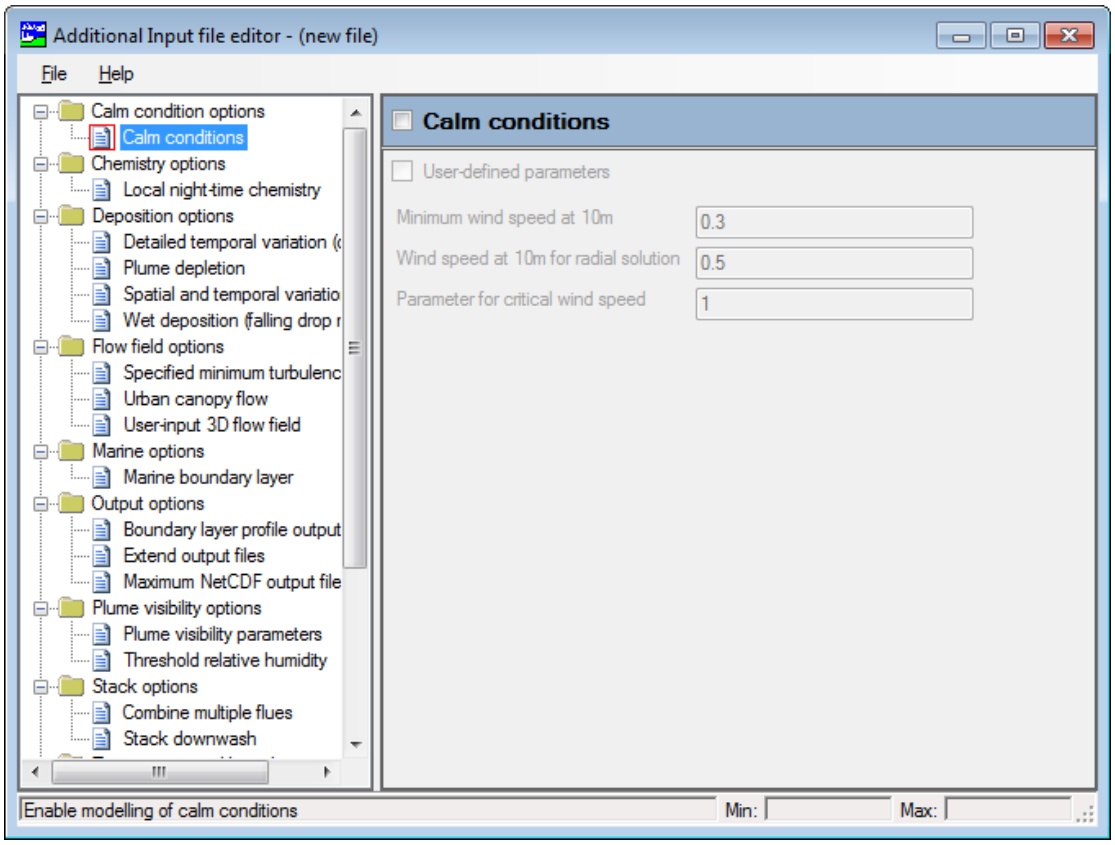


Figure 3.4 – **Additional Input file editor** with no options selected.

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

Clicking on an option in the left hand side takes you to the screen for that option; check the box at the top of the screen to enable the option. Most of the options then require further details regarding that option to be entered, these are described in more detail in the corresponding model option sections in Section 4. Each of the sections in the additional input file are identified by keywords, which are indicated alongside the model option descriptions.

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

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

3.2 Source screen

The **Source** screen is shown in **Figure 3.5** as it appears for a new model run. All source properties are entered here: type of source, dimensions, location, pollutants, etc.

3.2.1 Sources and groups

The **Source** screen allows the user to add sources and to create groups of sources. Data may be entered for up to 300 sources depending on the source type, and up to 20 user-defined groups of sources may be created. Different source types may be modelled in the same calculation.

Sources

To enter data for individual sources, select **Sources**, and create or delete sources using the **New** and **Delete** buttons. Clicking on **New** will enter a new line of data in the table with default values, which may then be edited to meet the requirements of the problem. The type of source to be added can be selected from the list below the **New** button. The **Delete all** button brings up the **Delete sources by type** screen, shown in **Figure 3.6**, which allows all sources of the chosen type(s) to be deleted.

*A source can be duplicated by right-clicking on the name of the source and selecting **Copy** from the menu which appears.*

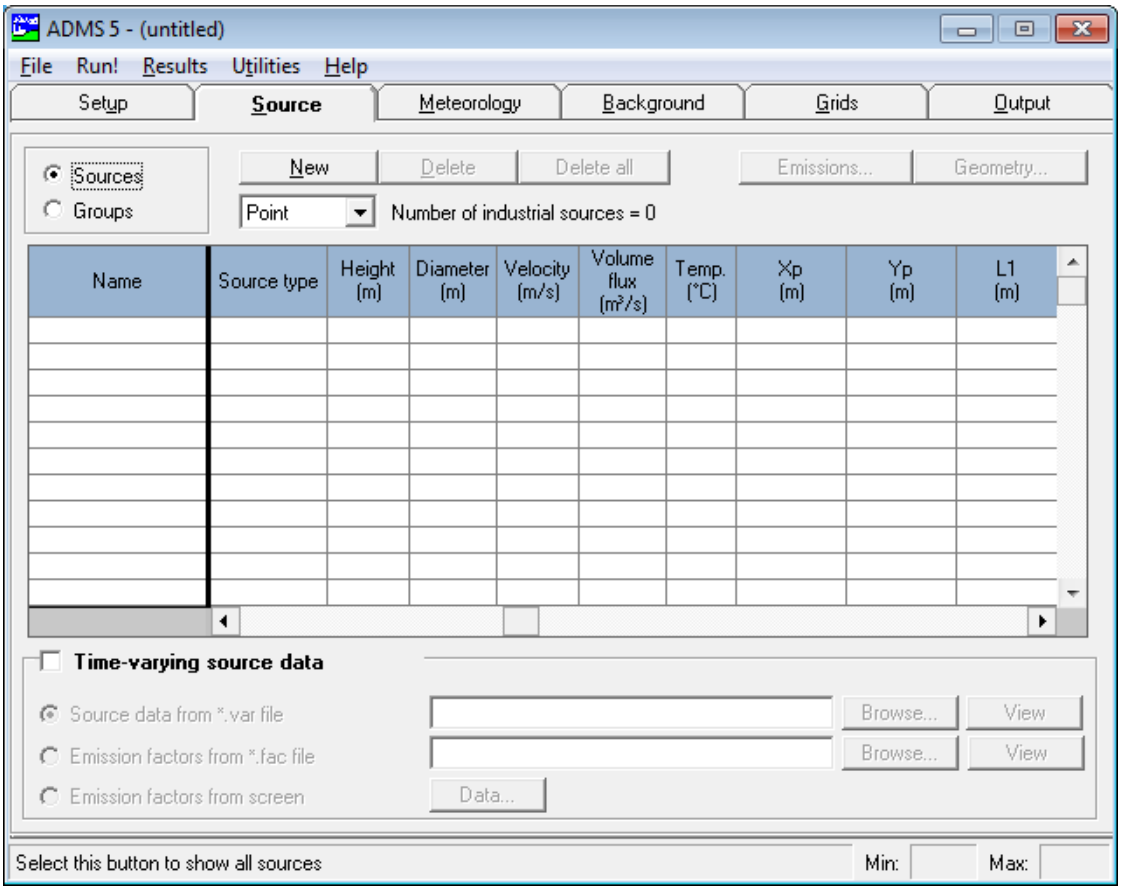


Figure 3.5 − The **Source** screen.

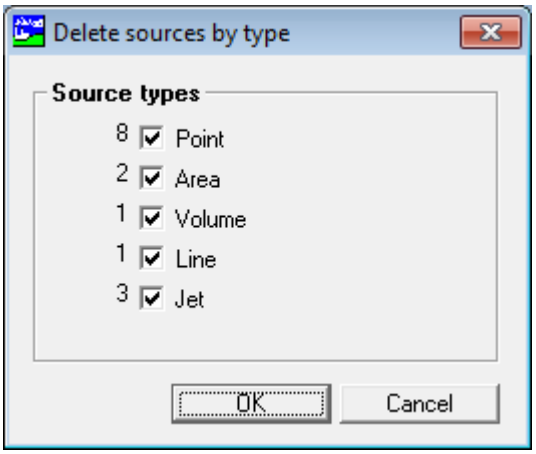


Figure 3.6 − The **Delete sources by type** screen.

Groups

Groups of sources may be created by selecting **Groups** on the **Source** screen, shown in **Figure 3.7**. The **New** and **Delete** buttons are used to add and remove existing groups, respectively. The **Delete all** button removes all existing groups. The **Rename** button can be used to rename an existing group and the **Copy** button to duplicate an existing group. The **Group/Source** button will add a set of groups, one for each source. This option is only available if the number of sources is small enough that the groups created will remain within the limit of 20 user-defined groups.

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

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

*Right-clicking on a source name in the **Sources** screen allows the group membership of that source to be viewed and modified from the **Groups** section of the menu. Groups of which the source is a member are shown with a tick next to the name, clicking on the group name toggles the group membership for that source.*

In addition to the 20 user-defined groups it is also possible to select for output a group containing all of the sources, refer to Section 3.6.2 for details.

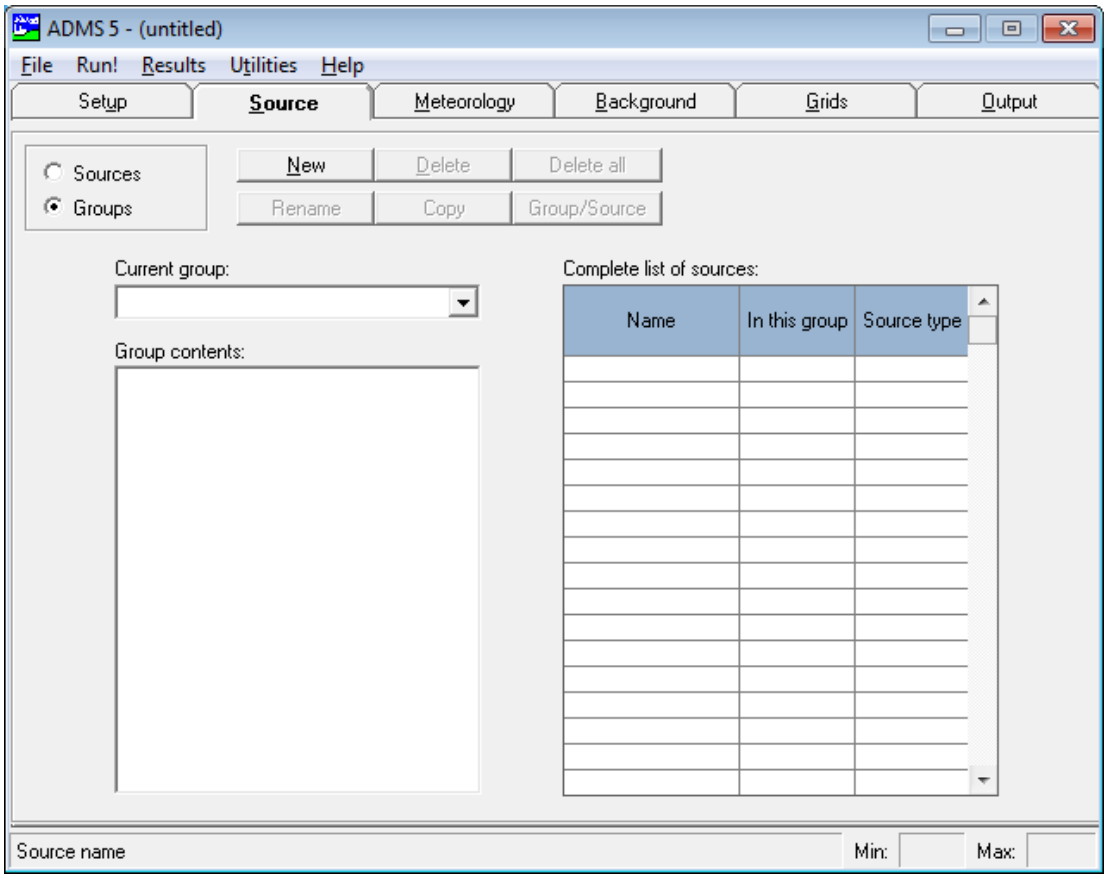


Figure 3.7 – Source screen with **Groups** option selected.

3.2.2 Source data

Data for an individual source should be entered in the table shown in **Figure 3.5**. A total of up to 300 sources are allowed. The list of parameters that may be entered for each source is given hereafter. Some parameters should be entered by hand, others chosen from a list and others modified by clicking on the displayed value.

1. **Name**: it is advised to give a meaningful source name for the study, for example "incinerator". By default, a name such as Source001 is used. The source name must not contain commas.

2. **Cp (J/°C/kg)**: specific heat capacity of the source material.

   Minimum = 1 J/kg/°C
   Maximum = 10^5 J/kg/°C
   Default = 1012 J/kg/°C (typical value for air)

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

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

4. **Density (kg/m³)**: density of the release material.

   Minimum = 0.01 kg/m^3
   Maximum = 2 kg/m^3
   Default = 1.225 kg/m^3 (typical value for air)

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

   Default = **T**

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

   Default = **Actual**

7. **Efflux**: efflux type to be specified.

   Exit V. exit velocity
   Vol. volumetric flow rate
   Fm, Fb. momentum flux and heat release rate (for example for fires)
   Mass mass flow rate

   Default = **Exit V.**

8. **Source type**: type of the source.

   Point point source
   Area area source
   Volume volume source

Line line source
Jet jet source

The default source type depends on the choice in the menu below the **New** button.

9. **Height (m)**: height of source above the ground. For a volume source, it is the mid-height of the volume above the ground.

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

10. **Diameter (m)**: internal diameter of the source.

    Minimum > 0 m
    Maximum = 100 m
    Default = 1 m

11. **Velocity (m/s)**: velocity of release at source exit (in m/s if **Actual** is selected in 6 or in Nm/s if **NTP** is selected where N means normal pressure and temperature).

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

12. **Volume flux (m^3/s)**: volume flow rate of the release (in m^3/s if **Actual** is selected in 6 or in Nm^3/s if **NTP** is selected).

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

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

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

14. **Xp (m)**, **Yp (m)**: X and Y coordinates of the centre of a point or jet source.

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

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

---

15. **L1 (m)**: width of a line source or vertical dimension of a volume source.

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

16. **Fm (m4/s2)**: momentum flux of emission as defined in the HMIP D1 document (HMIP, 1993). **Fm** should always be specified at the release conditions, i.e. **Actual** temperature and pressure. Refer to Section 9.22 for more information.

    Minimum = 1 m^4/s^2
    Maximum = 10^6 m^4/s^2
    Default = 1 m^4/s^2

17. **Fb (MW)**: heat release rate of emission as defined in the HMIP D1 document (HMIP, 1993). **Fb** should always be specified at the release conditions, i.e. **Actual** temperature and pressure. Refer to Section 9.22 for more information.

    Minimum = 0.0001 MW
    Maximum = 10^4 MW
    Default = 1 MW

18. **Mass flux (kg/s)**: mass flux of the emission.

    Minimum = 0.1 kg/s
    Maximum = 10^5 kg/s
    Default = 1 kg/s

19. **Angle 1 (°), Angle 2 (°)**: angles of the jet release (required for jet sources only). Angle 1 is the elevation angle, measured upwards from the horizontal, and angle 2 is the horizontal direction, measured anticlockwise from a line running from west to east (see Section 9.5).

    Minimum = 0°
    Maximum = 90° (angle 1), 360° (angle 2)
    Default = 0°

Three additional columns appear in the source table only if specific options are selected:

20. **Use VAR file**: shows whether the source is included (tick) in the time-varying emission file or not included (no tick). This is only shown if **Time-varying source data** is selected and a *.var* file is used (see Section 4.1).

    *Right-clicking in the **Use VAR file** column allows for the manipulation of the **Use VAR file** setting for multiple sources.*

21. **Main building**: if buildings are being modelled, then any of the buildings defined can be selected as the main building for each source. The default **Main building** is the building ticked on the **Buildings** screen. For further details, please refer to Section 4.9. This option is only available for point sources.

22. **Mass H2O (kg/kg)**: initial mixing ratio of the plume, i.e. mass of water per unit mass of dry release. This is only shown if **Plume visibility** is selected in the **Setup** screen.

    Minimum = 0 kg/kg
    Maximum = 9 999 999 kg/kg
    Default = 0 kg/kg

The source name (1), efflux type (7), source type (8), and source height (9) should always be entered. **Table 3.1** summarises the required source dimension/orientation data for each source type. The source geometry parameters 10, 14, 15 and 19 should be defined according to the source type. Efflux parameters 11 to 13 and 16 to 18 should be specified as summarised in **Table 3.2**. The main building (21) is only required if buildings are included in the model run.

Default values (those for air) would normally be used for parameters **Cp** (2) and **Mol. Mass** (3) unless a very pure gas is being emitted. If **Cp** and **Mol. Mass** are being altered it must be ensured that these two parameters are set to consistent values.

Source type	Height	Diameter	Xp	Yp	L1	Angle 1	Angle 2
P	✓	✓	✓	✓	×	×	×
A	✓	×	×	×	×	×	×
V	✓	×	×	×	✓	×	×
L	✓	×	×	×	✓	×	×
J	✓	✓	✓	✓	×	✓	✓

Table 3.1 − Source dimensions that must be specified for each source type.

Efflux type	Actual or NTP	Velocity	Volume flux	T or RHO	Fm	Fb	Mass Flux
Exit V.	✓	✓	×	✓	×	×	×
Vol.	✓	×	✓	✓	×	×	×
Fm, Fb	×	×	×	×	✓	✓	×
Mass	✓	×	×	✓	×	×	✓

Table 3.2 − Efflux parameters that must be specified for each efflux type.

3.2.3 Source geometry

The source type of an individual source may be selected as point, area, volume, line or jet. For each source type, the position and size are defined differently. Recall that the source height (Zs) has already been defined in Section 3.2.2 (list element 9).

The units of the emissions will also depend on the source type.

The geometry of a **Point** or **Jet** source should be given as follows.

Point source

A point source is a release at the specified height, located at (**Xp**,**Yp**). It is assumed to be horizontal and circular in cross-section, with diameter as specified by the user. Sources of large diameter should be modelled as area sources. This is because large point sources are subject to large stack downwash, which may not be physically representative of the source in question.

Jet source

A jet source is a release whose exit velocity has horizontal components as well as a vertical component (positive or zero), located at (**Xp**,**Yp**). It is specified in terms of its height, diameter, exit velocity and the angles made by the release with the vertical (**Angle 1**) and with the horizontal (**Angle 2**). More details of jet source geometry can be found in Section 9.5.

For an **Area**, **Volume** or **Line** source, the **Geometry**... button becomes available to allow the definition of the source vertices (see **Figure 3.8**). Adjacent vertices for such sources should be at least 1 m apart. The geometry parameters for volume and line sources are shown in **Figure 3.9**.

Area source

An area source is a release at the specified height over a horizontal *convex* polygon with between 3 and 50 vertices, e.g. emissions from a sewage tank. The X and Y coordinates of the vertices of the area source must be specified by the user sequentially around the source. **Figure 3.8** (left) shows the geometry screen for an area source. To add a new vertex click on the **New** button and edit the X and Y coordinates as appropriate. To remove a vertex select it and click the **Delete** button. Vertices should be at least 1 m apart. The **< Back** and **Next >** buttons can be used to access and edit the geometry of all area, volume and line sources.

Volume source

A volume source is a release from an area source with vertical extent but no plume rise, e.g. fugitive emissions around a building. The vertical extent of the source is **L1** (the height over which the source is initially well-mixed) and the source height (Zs) is the height of the *middle* of **L1** (see **Figure 3.9**). The source must be wholly above ground level. The horizontal cross-section of the volume is defined as for an area source, i.e. it must be a *convex* polygon with between 3 and 50 vertices, defined by the X and Y coordinates of the vertices specified sequentially around the polygon. Note that as no plume rise is modelled, the efflux parameters are automatically given the value 0. **Figure 3.8** (middle) shows the geometry screen for the horizontal area of a volume source. To add a new vertex click on the **New** button and edit the X and Y coordinates as appropriate. To remove a vertex, select it and press the **Delete** button. Vertices should be at least 1 m apart. The **< Back** and **Next >** buttons can be used to access and edit the geometry of all area, volume and line sources.

Line source

A line source is a release emitted uniformly along a straight line joining two points both at the source height. The line is defined by the X and Y coordinates of the centre points of its ends, and its width L1, distributed equally about its centreline (see **Figure 3.9**). The default values for the end points are shown in **Figure 3.8** (right). The vertices should be at least 1 m apart. The **< Back** and **Next >** buttons can be used to access and edit the geometry of all area, volume and line sources.

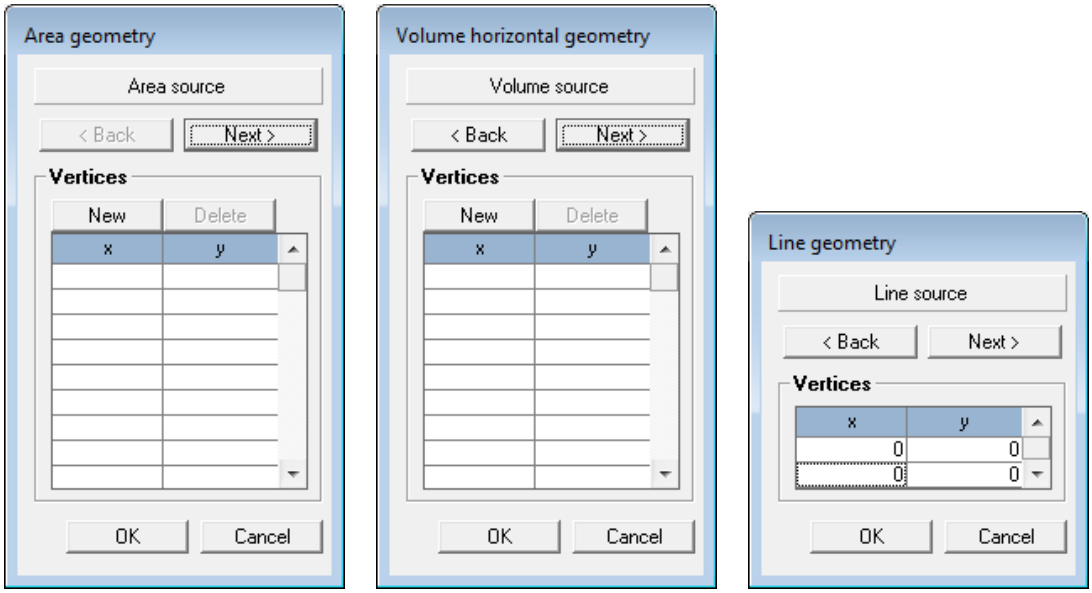


Figure 3.8 − Screen for the definition of the source geometry: area source (left), volume source (middle), line source (right).

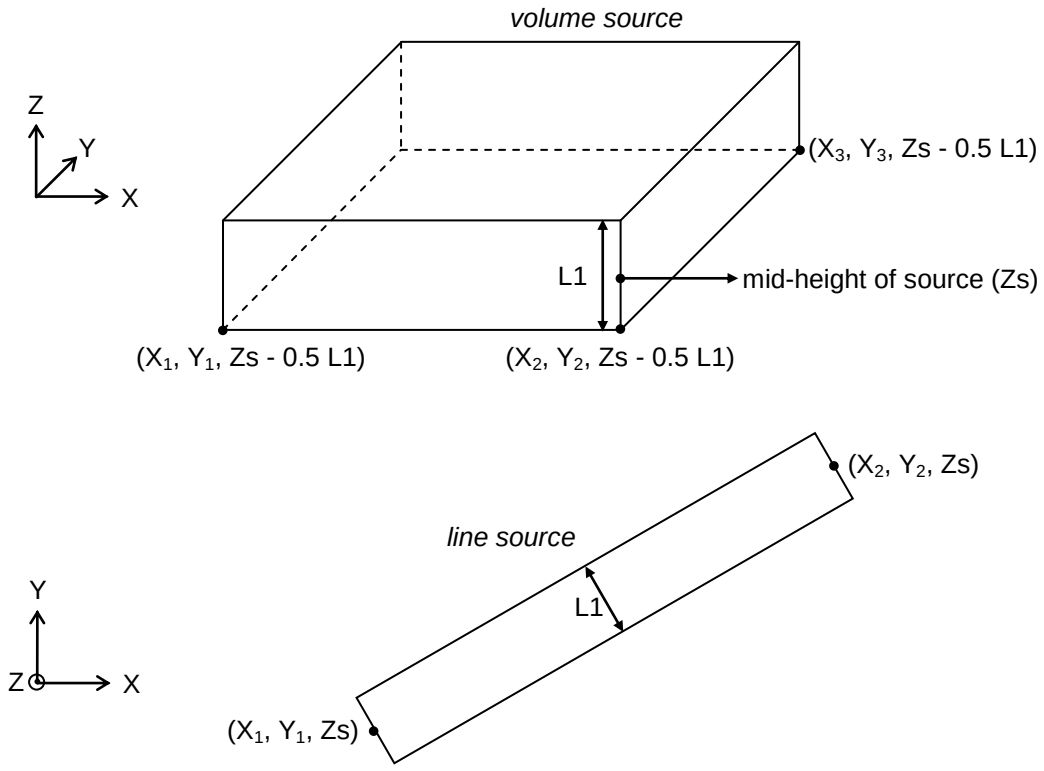


Figure 3.9 – Definition of geometry parameters for a volume source (top) and a line source (bottom). X_1, Y_1, etc., refer to the vertex coordinates entered, while Zs is the source height.

3.2.4 Source emissions

To enter emissions data for individual sources, select **Enter source data** on the **Source** screen, then click on the **Emissions**... button. The **Emissions** screen is then displayed (see **Figure 3.10)**. Up to 10 pollutants may be emitted by each source; these emissions are defined using the **New** and **Delete** buttons. Properties of pollutants are defined in

ADMS 5 by means of the **Palette of Pollutants** screen, which is accessed by selecting **Pollutants**... from the **Emissions** screen or the **Data**... button in the **Palette** section of the **Setup** screen.

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

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

*Before running the model, emissions data **must** be entered for **all** the sources – no default data are assumed.*

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

g/s	for point or jet sources,
$g/m^2/s$	for area sources,
$g/m^3/s$	for volume sources,
g/m/s	for line sources.

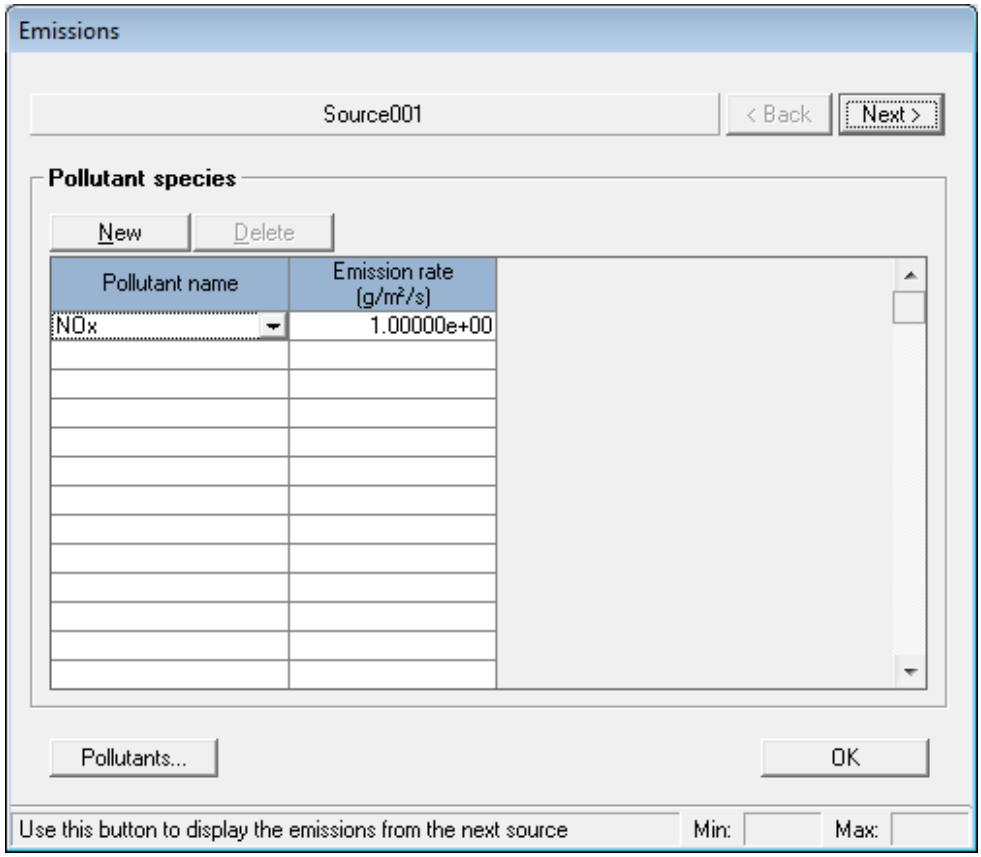


Figure 3.10 – **Emissions** screen for specifying pollutant emission rates.

If radioactive isotopes are being emitted the units will be Bq/s for point sources, $Bq/m^2/s$ for area sources, etc. Refer to Section 4.4 for more information about the radioactivity model option.

If the **Odours** option, has been selected on the **Setup** screen emissions must be given in ou_e/s for point sources, ou_e/m/s for line sources, ou_e/m^2/s for area sources and ou_e/m^3/s for volume sources. Refer to Section 4.7 for more information about the **Odours** option.

3.2.5 Time-varying emissions

Unless otherwise specified, the emission rate is assumed constant. However, it is also possible in ADMS 5 to model concentrations from emissions that vary with time. To select this option, check the **Time-varying source data** box on the **Source** screen (see **Figure 3.5**). Data to describe the time variation may then be entered directly in the interface or from a file. The following options are available:

- **Source data from *.var file**: enter time-varying values of source parameters, by hour or statistical period, using a *.var* file.
- **Emission factors from *.fac file**: enter diurnal and monthly profiles of emission factors along with wind dependent operation using a *.fac* file.
- **Emission factors from screen**: enter a diurnal profile of emission factors to be applied to all sources.

Full details of entering time-varying emissions can be found in Section 4.1.

3.3 Meteorology screen

The **Meteorology** screen shown in **Figure 3.11** allows details about the site and the meteorological conditions to be entered.

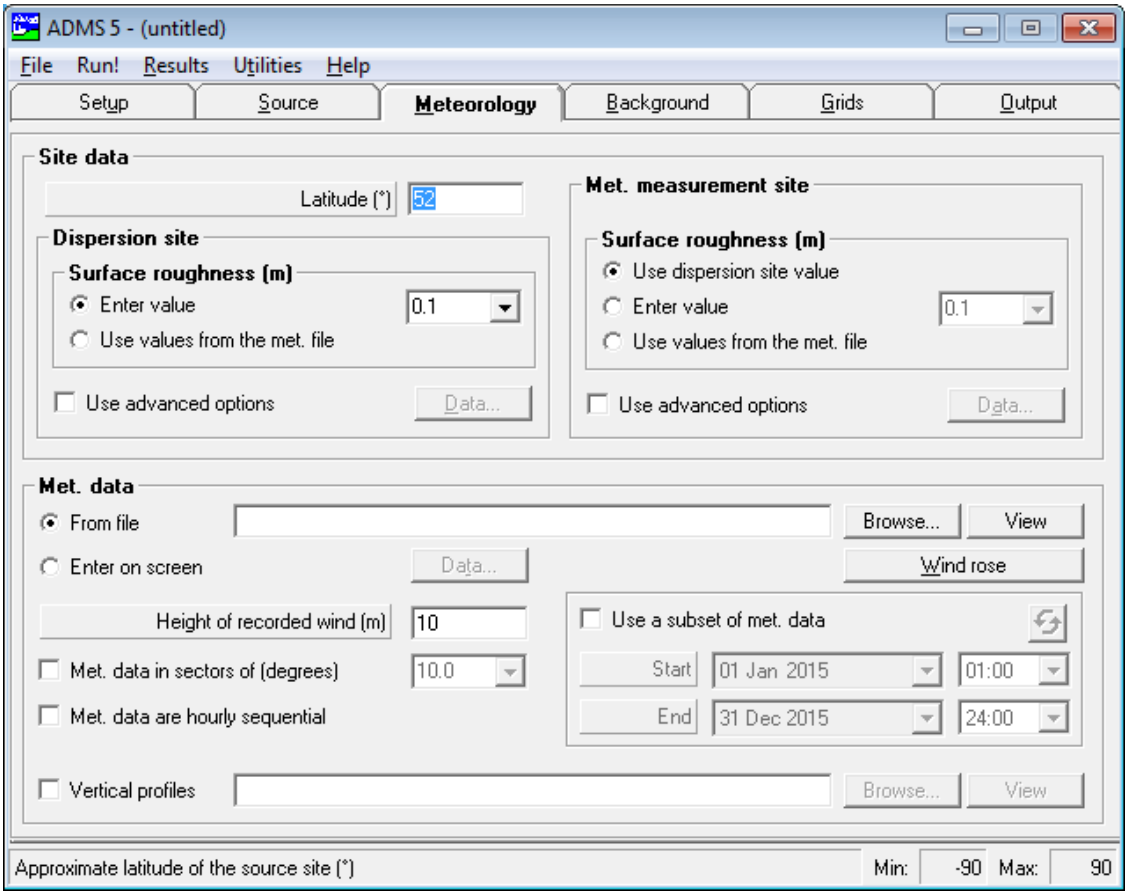


Figure 3.11 − The **Meteorology** screen in the ADMS 5 interface.

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

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

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

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

Examples of each of these can be found in the *<install path>\Data* directory. Some of the files in this directory such as *oneday.met* and *stat.met* are for use in the worked examples and do not contain meaningful data to be used in actual calculations.

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

- The Julian day number, time of day (local solar time) and cloud cover (TDAY, THOUR and CL);

or

- Sensible surface heat flux (FTHETA0);

or

- Reciprocal of Monin-Obukhov length (RECIPLMO).

Additional meteorological parameters can be added to the meteorological data file, and should be included if known. Some model options require further parameters to be entered in this input file. Full details of the parameters that can be included can be found in Section 9.

3.3.1 Site data

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

On other screens Cartesian coordinates of sources and receptors are required. It is assumed that the positive X and Y axes point east and north, respectively.

In the **Dispersion site** box, enter the **Surface roughness** at the dispersion site (in metres) based on land use. Either enter a value directly in the **Enter value** box, subject to the model limits (minimum = 10^{-7} m, maximum = 10 m), or alternatively use the drop-down box to select a value based on the land use − **Table 3.3** shows the land use types and corresponding values of surface roughness. The default value of surface roughness is 0.1 m, corresponding to a region of root crops. To enter hourly varying surface roughness values for the dispersion site via the meteorological data file, select the **Use values from the met. file** option and add the surface roughness values to the file using the variable name 'Z0 (D)'.

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

Table 3.3 − The surface roughness values for the different land uses available on the **Meteorology** screen.

In general, the surface roughness length selected for the dispersion site is assumed to apply throughout the domain. However, ADMS 5 also has advanced features that make it possible to define a *distribution* of surface roughness over the domain (refer to Section 4.10).

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

For both the dispersion site and met. measurement site, **Use advanced options** can be selected to allow the user to alter the values of surface albedo, Priestley-Taylor parameter, minimum Monin-Obukhov length and precipitation differences between the dispersion site and meteorological measurement site. Details of how to use these options are given in Section 4.15.

3.3.2 Met. data

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

- **From file**: meteorological data are to be read from a file (the name of the file should be entered by the user). Clicking **Browse**... displays the directory structure and lets the user select prepared meteorological data files, which, by convention, have the extension *.met*. The **View** button allows the current *.met* file to be viewed. Further information on entering meteorological data using a *.met* file can be found in Sections 3.3.3 and 9.1.
- **Enter on screen**: meteorological data are to be entered on screen. Click on the **Data**... button to enter the data; full details of this option can be found in Section 3.3.4.
- **Wind rose**: shows the wind rose for the meteorological data file selected, including the effects of wind sector size and subset start and end.
- **Height of recorded wind (m)**: height of the wind measurements (default = 10 m).
- **Met. data in sectors of (degrees)**: if the wind measurements are in sectors, check the box and enter the sector size, either by selecting one of the options from the drop-down list or typing directly into the box (refer to Section 9.1 for further details).
- **Met. data are hourly sequential**: check the box if the meteorological data are hourly sequential or are to be interpolated to become hourly sequential (refer to Section 9.1). If a series of unrelated meteorological data are entered, for instance to examine what happens during specific meteorological conditions, this box should not be checked.
- **Use a subset of met. data**: check this box to only use a subset of the meteorological data; the dates and times of the first and last met. lines should

be entered. The start and end dates can be automatically set to those of the meteorological data file by clicking the reset button, .

- **Vertical profiles**: enter user-defined vertical profile data for wind speed, turbulence parameters, temperature and specific humidity using a *.prf* file. For details of this option refer to Section 4.16.

3.3.3 Entering meteorological data from a file

If the meteorological data are to be entered from a file, then on the **Meteorology** screen first select **From file**, then click on **Browse**... and search to find the file. The selected file can then be viewed using the **View** button. Meteorological data files are comma-separated files with a *.met* extension, by convention. The format of the meteorological data file should be as follows.

The first line should contain the keyword "VARIABLES:". The second line should contain the number of parameters to be entered in the *.met* file. Then follows a list of the parameters by name (one per line) for which data are entered in the *.met* file (details of the names to be used for each parameter can be found in Section 9.1). The data section starts with the keyword "DATA:" followed by the actual data. For each line of meteorological data, the parameters should be entered in a row in the order that they are listed in the "VARIABLES:" section. Comments can be added before the "VARIABLES:" and between the "VARIABLES:" and "DATA:" sections, no blank lines should appear in the file except as part of the comments. Further details about the format of the *.met* file can also be found by looking at *METDEMO.met* which can be found in the <*install path*>*Data* directory.

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

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

Using one of the prepared example files

There are a number of example *.met* files provided with ADMS 5. They are installed in the <*install_path*>*Data* directory of your computer. A particularly useful file is *R91A-G.met*, which contains seven lines of meteorological data that correspond approximately to stability classes A-G. This file can be viewed in an editor such as Notepad or WordPad and is shown in **Figure 3.12**.

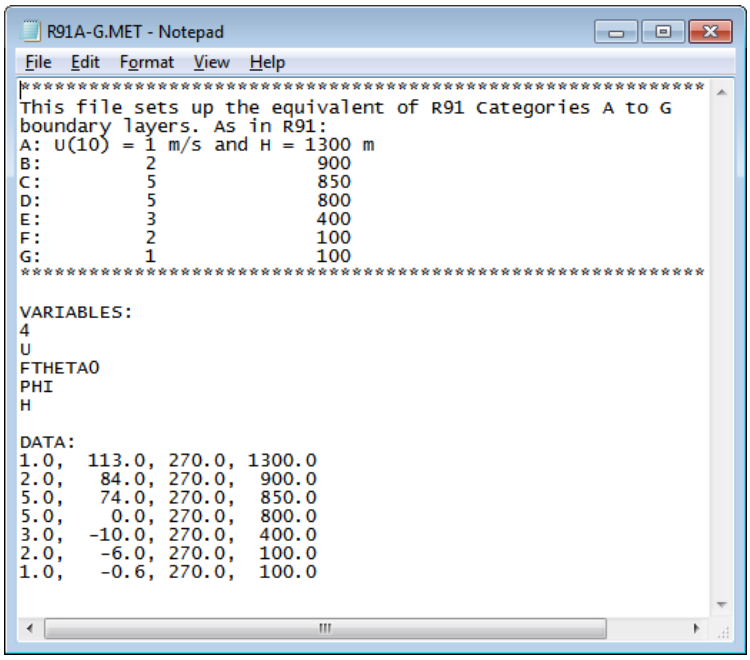


Figure 3.12 − The *.met* file *R91A-G.met*.

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

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

Hourly sequential meteorological data

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

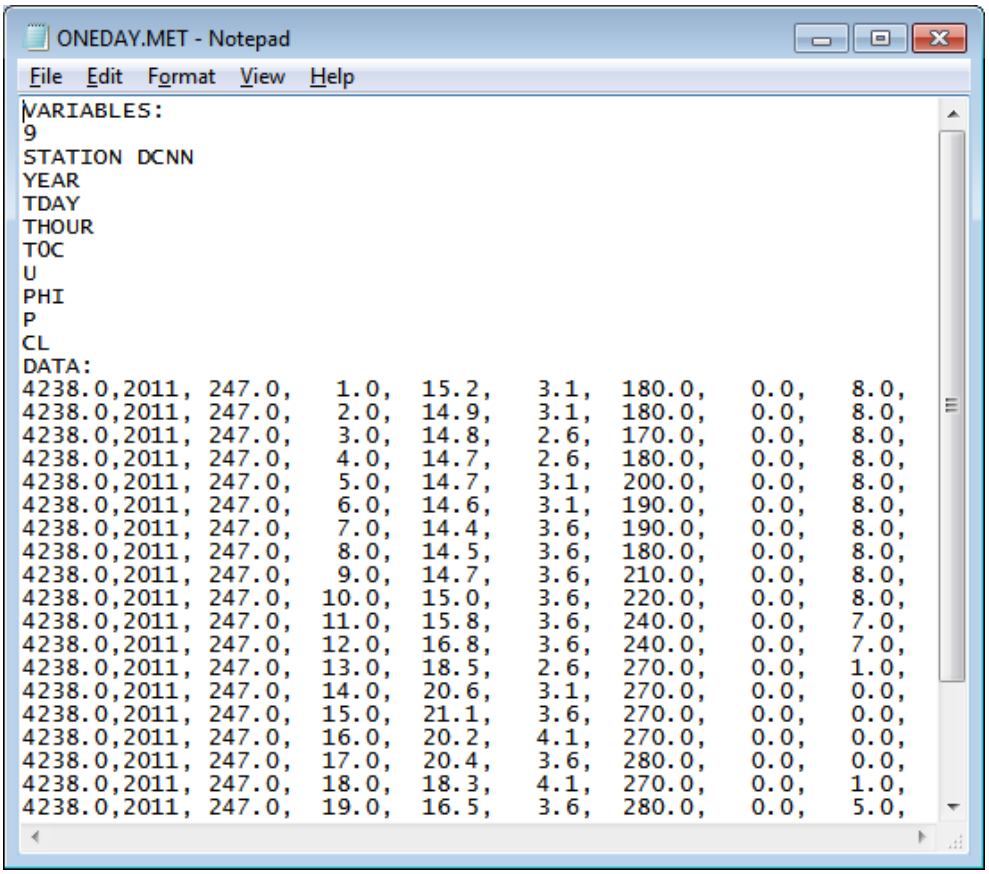


Figure 3.13 − Example of hourly sequential meteorological data file.

An example of the beginning of an hourly sequential file is shown in **Figure 3.13**. If this type of file is being used then the **Met. data are hourly sequential** box should be checked on the **Meteorology** screen. Wind direction measurements (`PHI`) are often reported to the nearest 10 degrees, i.e. the wind data are in 10 degree sectors. If this is the case, the **Met. data in sectors of (degrees)** option should be selected and the sector size entered.

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

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

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

Statistical meteorological data

Statistical data files normally include meteorological data for a period of ten years. They can be obtained from a supplier, e.g. the UK Met Office. They can be viewed in a text editor, and an example showing the beginning of a statistical file from the UK Met Office is given in **Figure 3.14**.

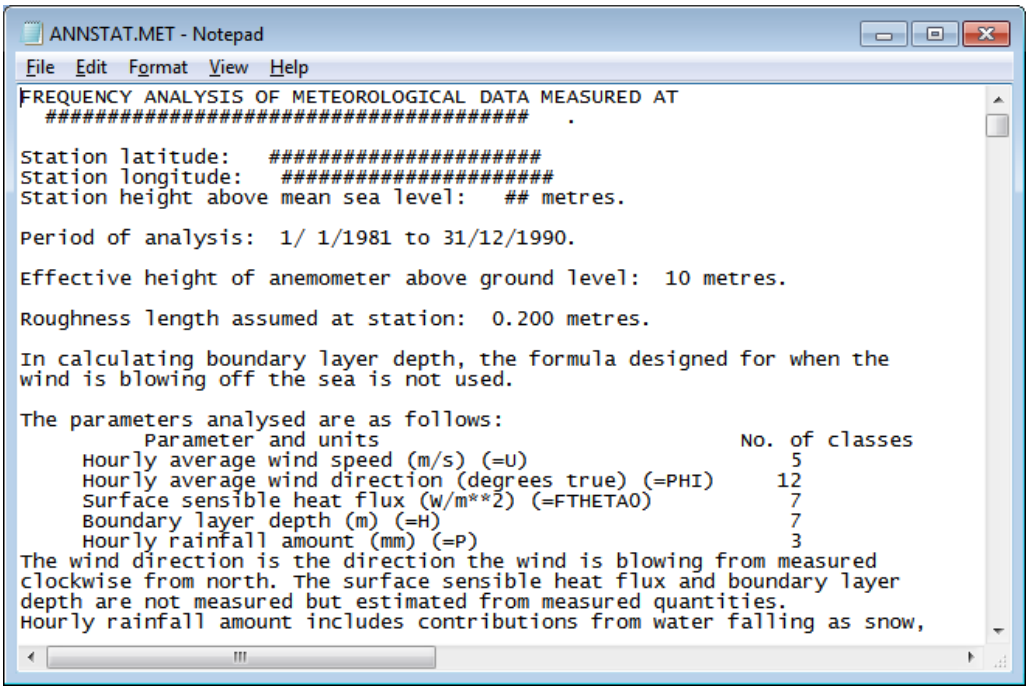


Figure 3.14 − Example of statistical meteorological data file.

The aim of using statistically analysed data is to take account of the variation in meteorological conditions from year to year. Each hourly measurement for that period is recorded and put into a bin or class representing a range of different values for different meteorological parameters. For example, in the file represented in **Figure 3.14**, the data were categorized into five classes of wind speed (`U`), twelve classes of wind direction (`PHI`) (i.e. a wind sector size of 30 degrees), seven classes of surface sensible heat flux (`FTHETA0`), seven classes of boundary layer depth (`H`), and three classes of precipitation (`P`).

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

Users should ensure they read the information at the beginning of a statistical meteorological data file in particular the 'Effective height of anemometer above ground'. For the majority of files it is 10 m but occasionally it is at some other height and the correct value should be entered into the ***Meteorology*** *screen of ADMS 5.*

Statistical data has a number of advantages over sequential data.

- There are about 2000 lines of data to process in ADMS 5 with a statistical meteorological data file rather than 8760 in a one year sequential file; run times are therefore significantly shorter for statistical than sequential data.
- Statistical data is representative of the meteorological conditions which occurred over several years (usually ten); year-to-year variations in conditions are therefore accounted for. This is not true for sequential data.

A disadvantage of using statistical meteorological data is that because the data are categorised into bins with pre-defined upper and lower bounds, it is the mean for each bin rather than the actual hourly value of a parameter that is processed by ADMS 5. Thus extreme conditions are smoothed out. For example, there may be a few hours in a particular year (e.g. 1995) when the surface heat flux is extremely high, thus generating a lot of convective mixing and bringing an elevated plume down to the ground very quickly. This would normally result in high maximum ground-level concentrations near the source. Using sequential data, the data file would contain the actual high surface heat flux value for that particular hour, thus representing actual conditions. Using statistical data, the value of surface heat flux used would be the mean value of the appropriate class. In this case, this mean value would be less than the true hourly value and would thus underestimate the amount of convective turbulence hence underestimating ground level concentrations for that hour.

In general, high percentiles obtained with sequential data are larger than those obtained with statistical data. Sequential data must be used when using background data from files. Sequential data are also required for generating results for comparison with regulatory standards defined in terms of rolling or running means as statistical data are inappropriate for such calculations. Sequential data are also required for modelling with averaging times greater than 1 hour.

3.3.4 Entering meteorological data on screen

If the meteorological data are to be entered on screen, first select the **Enter on screen** option on the **Meteorology** screen, then click on the **Data**... button. This will bring up the **Meteorological Data** screen shown in **Figure 3.15**.

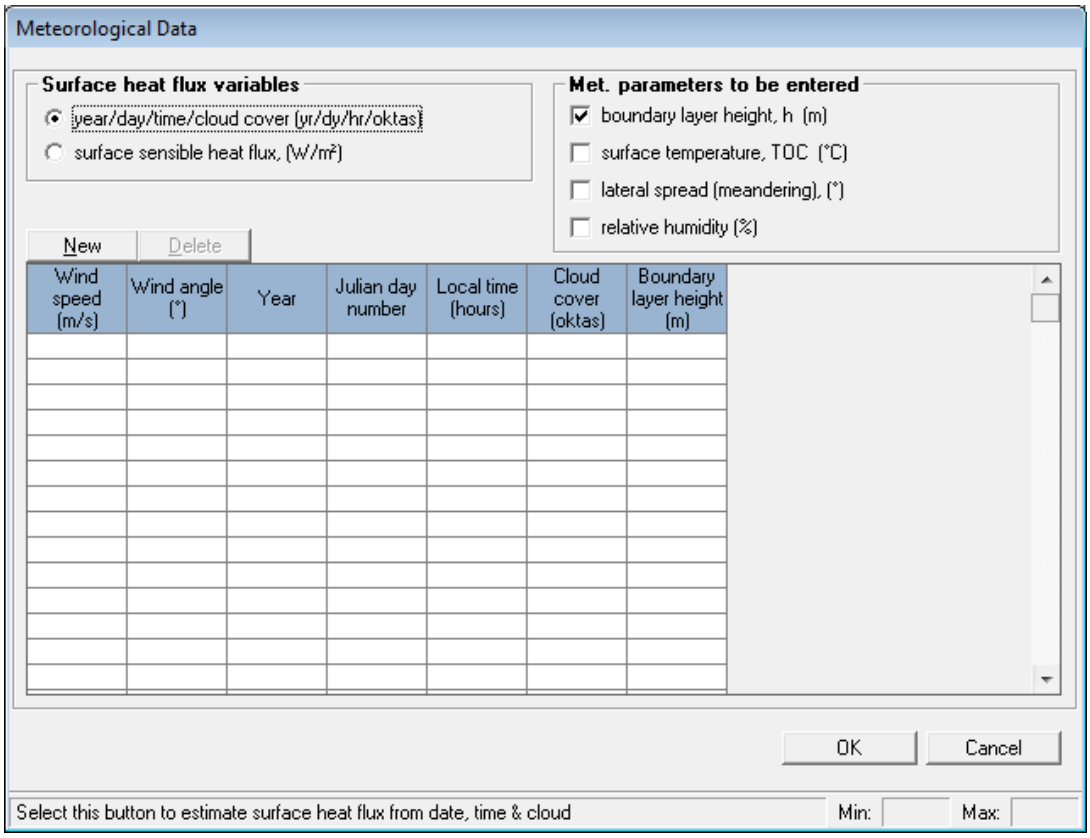


Figure 3.15 − The **Meteorological Data** screen used for entering meteorological data on screen.

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

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

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

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

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

The columns of data are as follows:

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

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

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

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

- **Year**: year

  Minimum = 1900
  Maximum = 2100
  Default = Current year

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

  Minimum = 1
  Maximum = 366
  Default = 90

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

  Minimum = 0
  Maximum = 24
  Default = 8

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

  Minimum = 0
  Maximum = 8
  Default = 0

- **Surface sensible heat flux (W/m^2)**: surface sensible heat flux

  Minimum = -100 W/m^2
  Maximum = 1000 W/m^2
  Default = 0 W/m^2

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

  Minimum = 40 m
  Maximum = 3000 m
  Default = 800 m

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

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

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

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

- **Relative humidity (%)**: relative humidity

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

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

3.3.5 Interpolation of meteorological data

In order to model averaging times greater than one hour, hourly sequential meteorological data are required. If hourly sequential data are not available, but a chronological sequence of meteorological data are available then, providing certain constraints are met, the data will be interpolated to become hourly sequential. To enable this option enter your meteorological data as described in sections 3.3.3 or 3.3.4 and check the **Met. Data are hourly sequential** box.

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

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

---

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

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

3.4 Background screen

The **Background** screen shown in **Figure 3.16** allows the user to enter background data for any of the pollutants entered in the **Palette of Pollutants**.

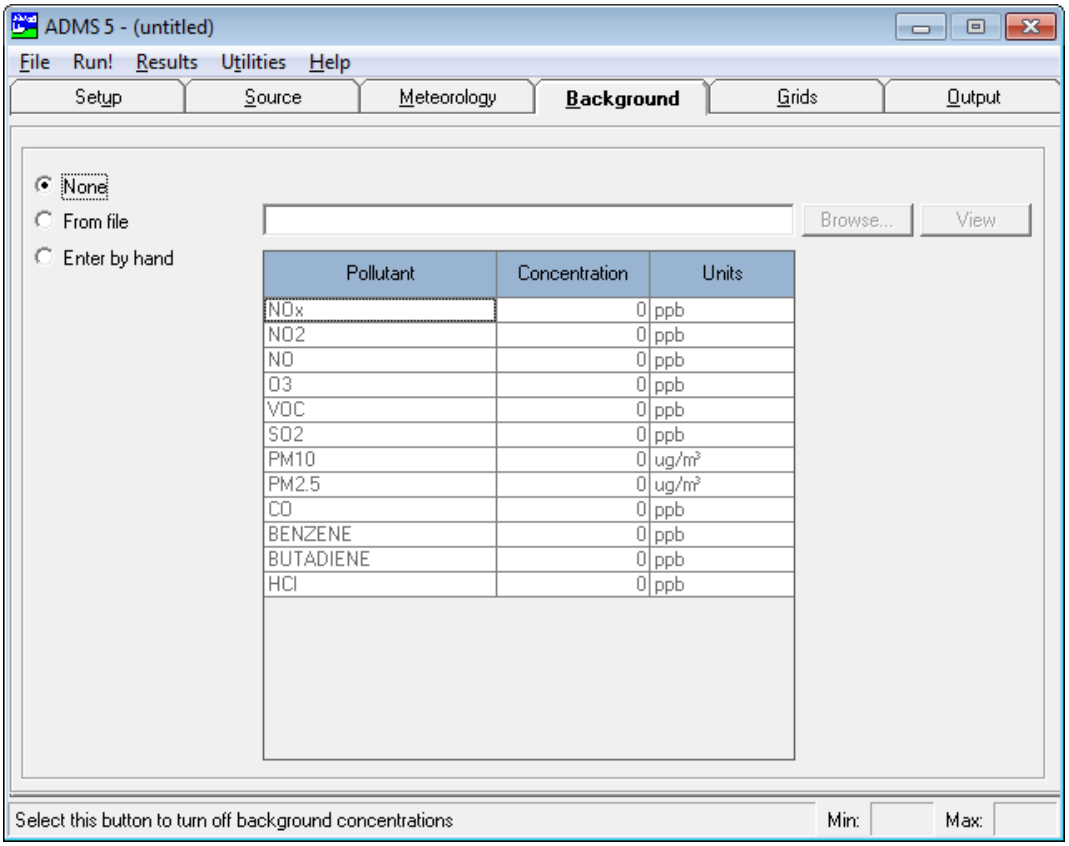


Figure 3.16 – The **Background** screen.

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

For certain model options background concentrations cannot be entered. In these cases the background screen just contains a message explaining that background concentrations cannot be entered with the current model options.

*Background data cannot be entered if the **Isotopes** palette is being used or if the **Fluctuations** or **Time-integrated Puffs** model options are being used.*

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

If background concentrations are permitted, then the **Background** screen presents three options:

- **None**: no background concentrations.
- **From file**: hourly varying background data from a prepared background file (refer to Section 3.4.1 for more details).
- **Enter by hand**: constant background values entered directly on screen (refer to Section 3.4.2 for more details).

3.4.1 Entering background data from a file

To specify the file containing the background data, select the **From file** option and click on **Browse**.... This allows you to browse through the directory structure to find the appropriate background file (*.bgd* extension by convention). The **View** button allows the currently selected background file to be viewed. The format of the background file is as follows.

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

The next line states the number of pollutants for which the file contains background data. The names of the pollutants are then listed (one per line) and should match the pollutant definitions in the **Palette of Pollutants**. Any pollutants which are not listed in the background file are assumed to have zero background concentrations. Data for unrecognised pollutants will be ignored.

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

If odour units are modelled then the background values should be in appropriate units.

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

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

Background pollutant data must be given as hourly sequential values and must be supplied for each hour to be modelled. An example of a background file is given in **Figure 3.17**.

If NO_X data are included in the background file then NO_2 data must also be included.

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

If any data are missing on the first line of the background file or if there are more than 24 consecutive lines of missing data for any pollutant, then the model will stop with an error message.

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

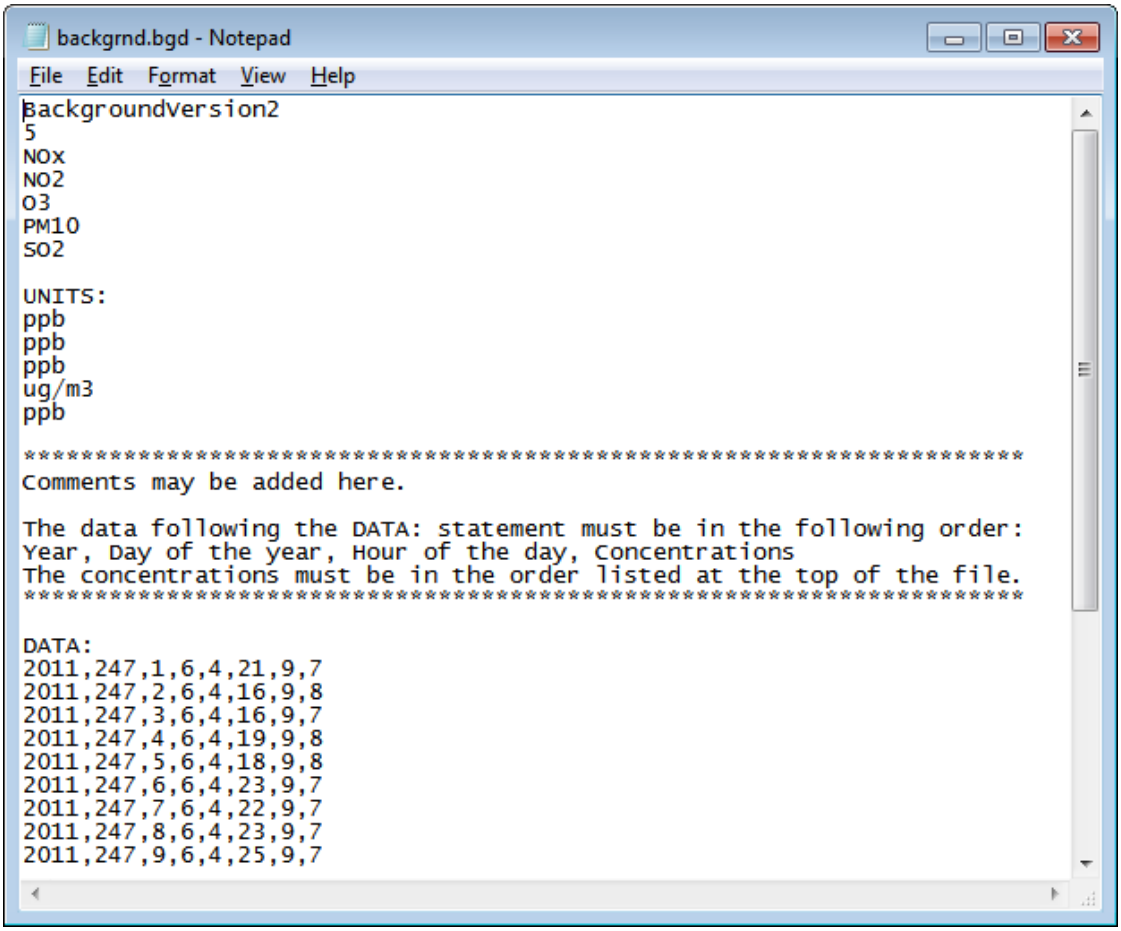


Figure 3.17 − An example showing the format of the background file.

3.4.2 Entering background data by hand

To enter constant values of background data directly on screen, first select the **Enter by hand** option. The table contains all of the pollutants in the **Palette of Pollutants** and their background values, along with the units of these background values. The following should be noted:

- The background value and units of NO cannot be edited; instead the value is calculated from the background values of NO_X and NO_2 that have been entered.
- Background concentrations of particulate pollutants can only be specified in mass units.

3.5 Grids screen

The **Grids** screen shown in **Figure 3.18** allows the user to define output points at which concentration values will be output by the model. The coordinate system may be Cartesian or Polar. Output points may be defined as a grid, individually (specified points) or both. Grids of output points may be defined with regular or variable spacing. Units are metres (or degrees for angles).

Cartesian coordinate system

A Cartesian coordinate system refers to a point by a pair of X and Y coordinates. Usually, the X direction points eastwards, and the Y direction northwards.

Polar coordinate system

A polar coordinate system refers to a point by a pair of coordinates: the radius (R) and the angle (Theta). The radius is the distance between the origin of the coordinate system and the point. The angle is the angle made between the radius and a reference (the east). The angle increases in an anti-clockwise direction. A 0° angle corresponds to the X axis (the radius points to the east), a 90° angle corresponds to the Y axis (the radius points to the north), etc.

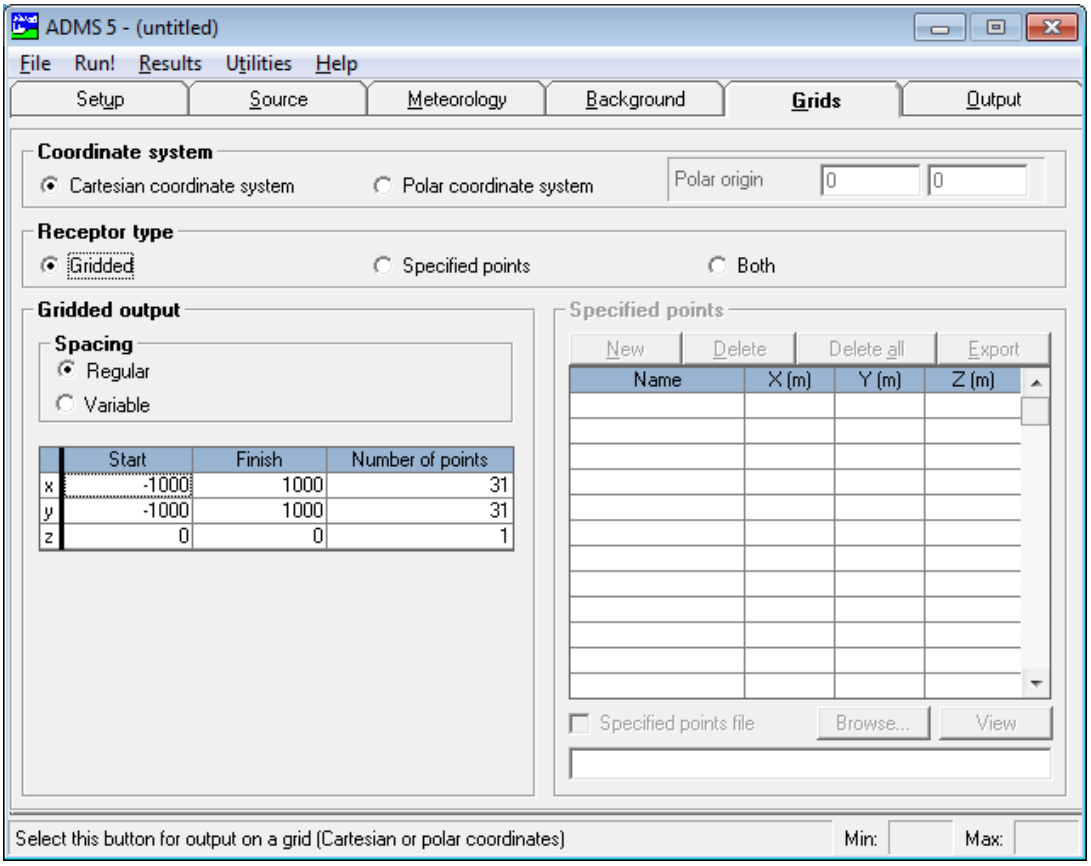


Figure 3.18 − The **Grids** screen for selecting an output grid and/or specified output points.

3.5.1 Gridded output

Cartesian grids

Regular Cartesian grids are specified by the **Start** and **Finish** X, Y and Z coordinates, representing the minimum and maximum values in each direction, together with the **Number of points** along the X, Y and Z axes. The maximum resolution in the X and Y directions are 2001 points and the maximum number in the Z direction is 501. The grid spacing is automatically calculated once the number of points in each direction has been set.

Care should be taken with high-resolution output grids since the model run time and memory requirements partly depend on the number of output locations.

Variable Cartesian grids are specified by X, Y and Z coordinates entered in the **X**, **Y** and **Z** boxes as shown in **Figure 3.19**. To enter **X**, **Y** and **Z** values, click in the small text box in that column, type the value then press the **SPACE** bar. To remove points, select them in the larger, upper text box, and press the **DELETE** key. Results will be calculated on a grid of points at the specified **X**, **Y** and **Z** values. For example, if three **X** values, four **Y** values and one **Z** value are specified, results will be calculated on a grid of 12 points. The maximum size of the variable grid in any dimension is 100 points.

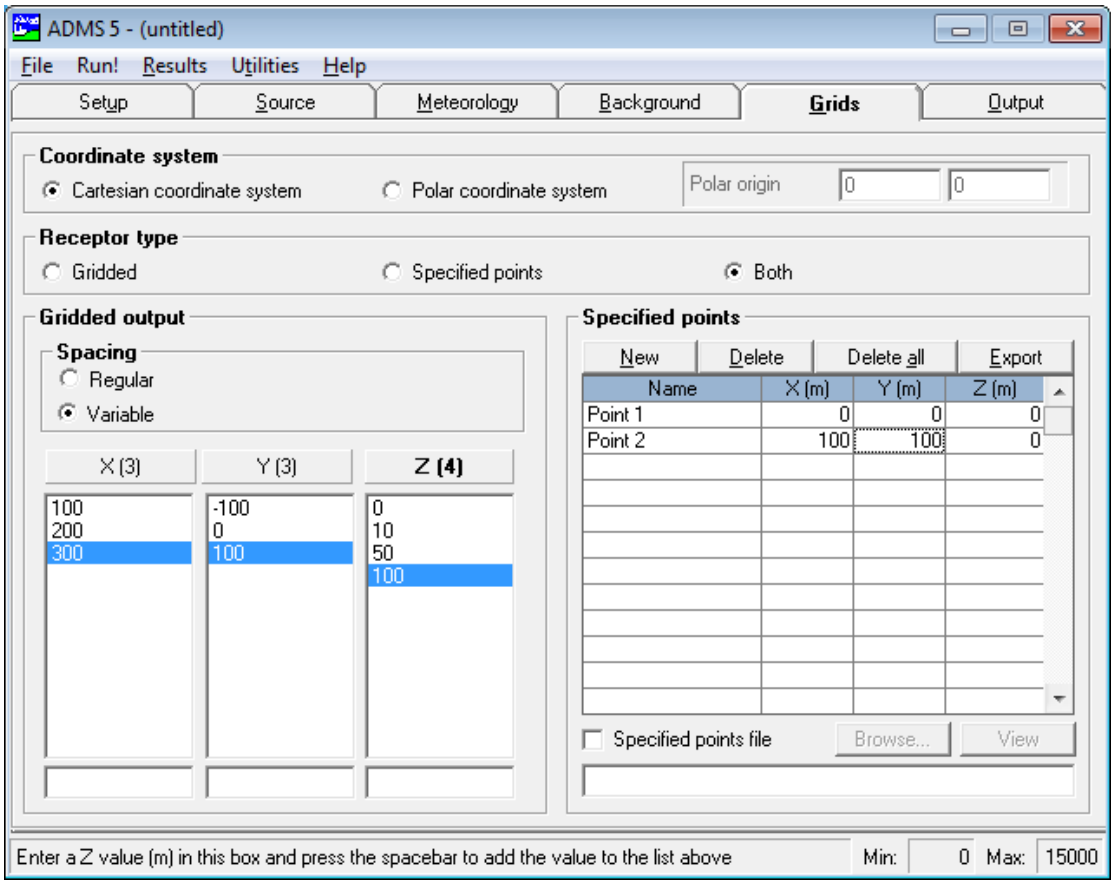


Figure 3.19 − The **Grids** screen when a variable Cartesian grid is specified. The numbers in brackets in the column headers indicate the number of points in each column box.

Polar grids

Regular polar grids are specified by the location of the origin of the polar grid (X and Y coordinates), plus the **Start** and **Finish** values of the radius, angle and height, together with the number of points in the radial direction, the number of angles into which the circumference is divided and the number of heights (up to 2001 for the angle and radial directions and 501 for the heights), as shown in **Figure 3.20**. The spacings are automatically calculated once the number of points has been set.

Variable polar grids are specified by R (radius), Theta (angle) and Z (height) values entered in the **R**, **Theta** and **Z** boxes as shown in **Figure 3.21**, in an analogous way to Cartesian grids.

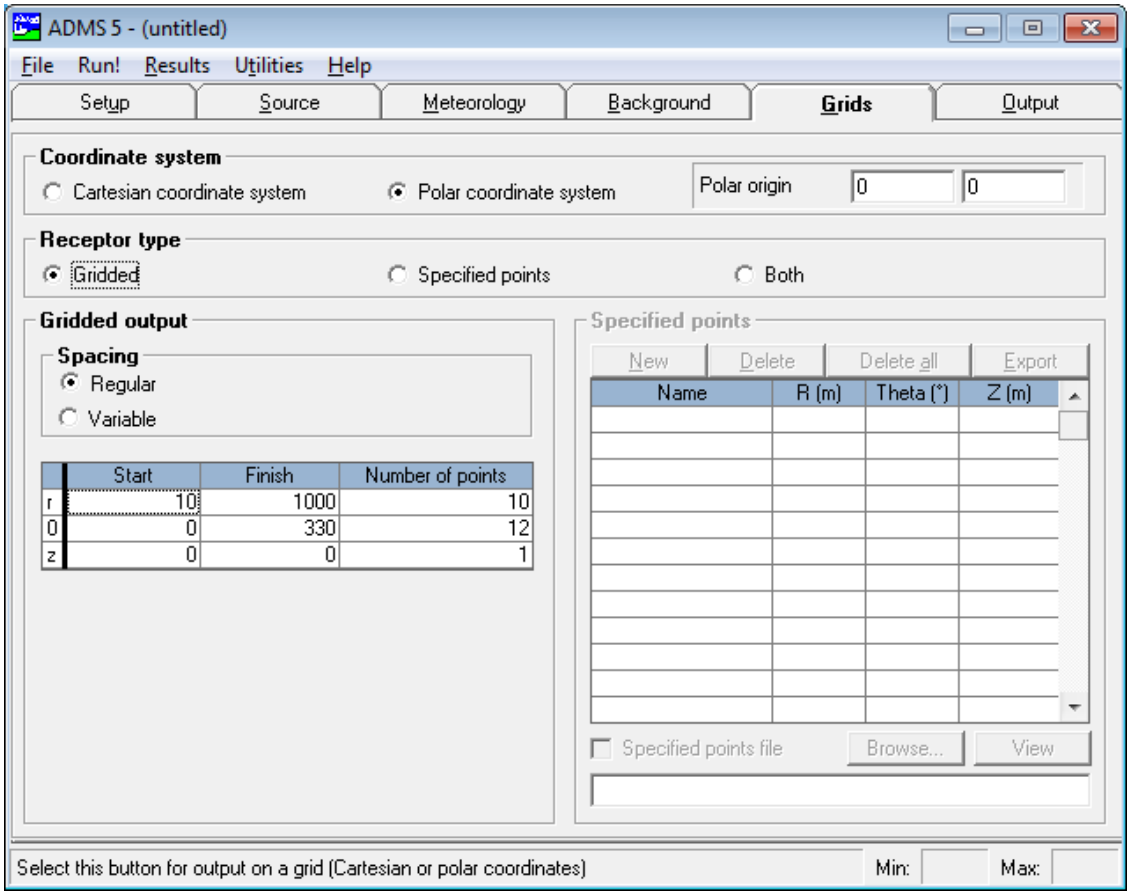


Figure 3.20 − The **Grids** screen when a regular polar grid is specified.

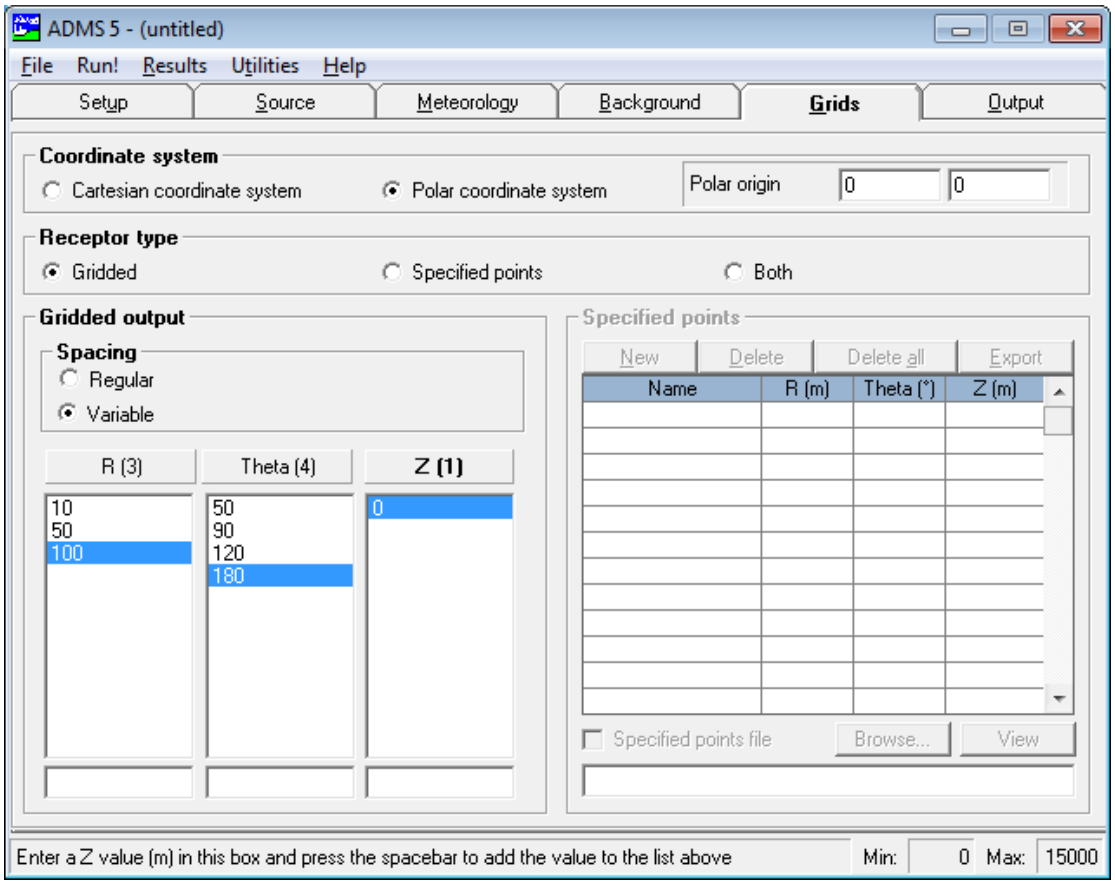


Figure 3.21 − The **Grids** screen when a variable polar grid is specified.

3.5.2 Specified points output

Specified points may be added to or deleted from the table on the right-hand side of the **Grids** screen by clicking on the **New** and **Delete** buttons, respectively, if the **Specified points** or **Both** option is selected for **Receptor type**. The maximum number of points which can be defined in the interface is 100. Points may be given a name by the user (e.g. "point 1", "School") and have to be specified with **X, Y** and **Z** coordinates (or **R**, **Theta** and **Z** coordinates). The names given to the specified points must not contain commas.

As well as entering up to 100 specified points in the interface, you can enter extra specified points using an additional specified points file, which is a comma-separated text file with the extension *.asp* that contains a line of data for every extra specified point required. Each line contains the point name, the X (or R) coordinate, the Y (or Theta) coordinate, and Z coordinate, in that order. The location of the *.asp* file is entered in the interface on the bottom right hand corner of the **Grids** screen (as shown in **Figure 3.22** below).

An example *.asp* file is supplied with the model. There is no limit on the number of points which may be included in the *.asp* file. The specified points entered into the interface can be exported to an *.asp* file by clicking on the **Export** button.

If a very large number of output points are included the model run time and memory requirements may be substantial.

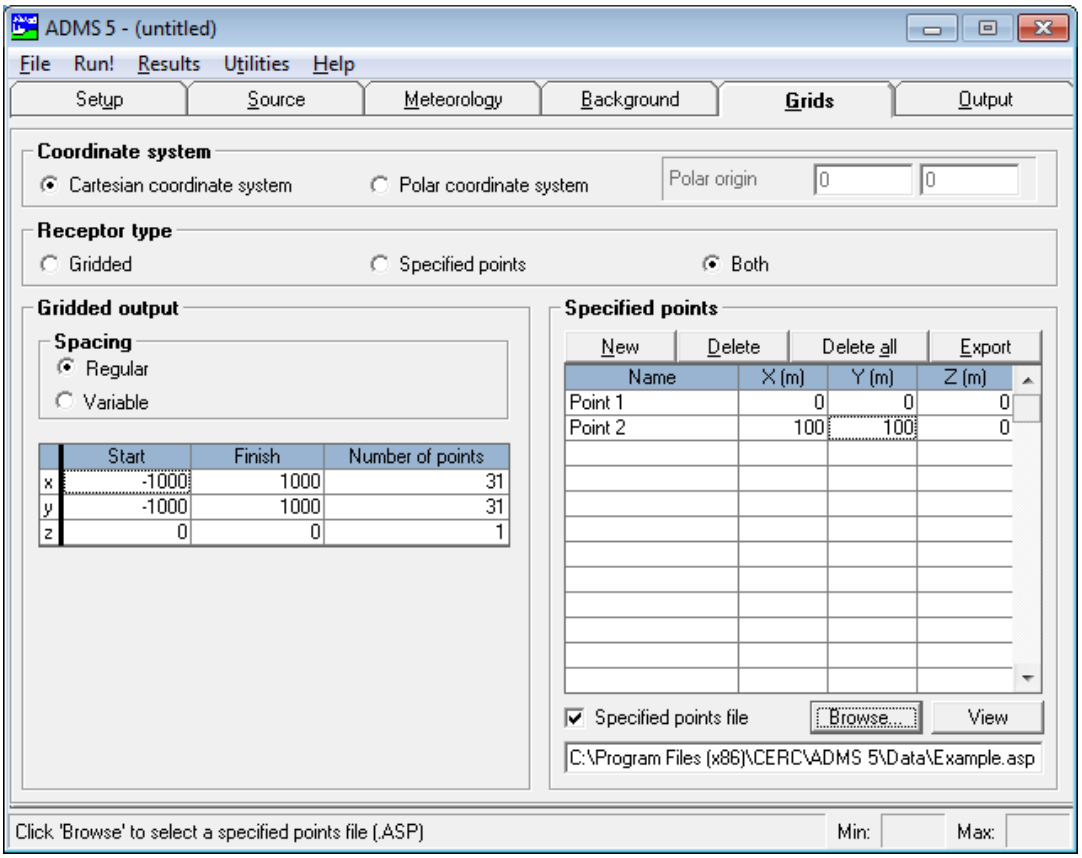


Figure 3.22 − The **Grids** screen showing specified points entered via the interface and the *.asp* file.

3.6 Output screen

The pollutants and sources for which output is to be calculated, together with the type of output required, are selected in the **Output** screen, shown in **Figure 3.23**.

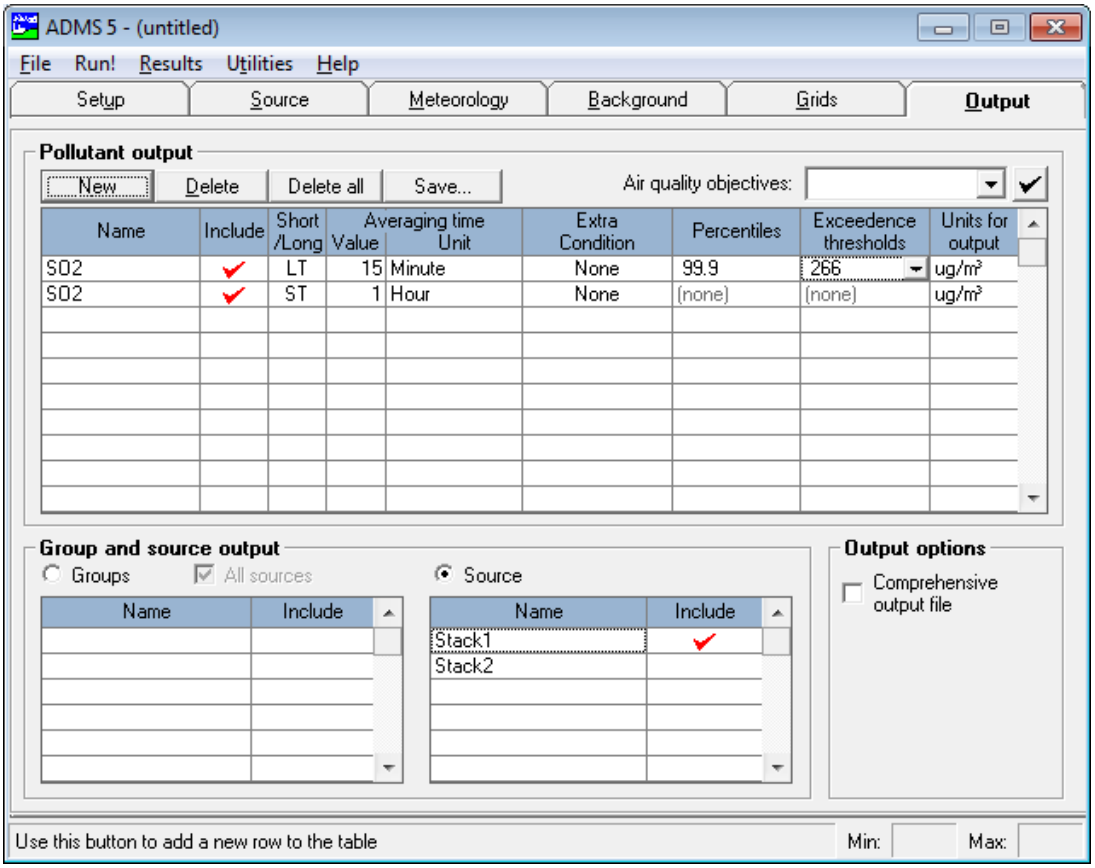


Figure 3.23 − The **Output** screen shown for the case of a short-term calculation for a single source.

3.6.1 Pollutant output

Pollutants may be added to or deleted from the upper table by clicking on the **New** and **Delete** buttons. For each pollutant, parameters to be specified are described below. The user **must** select the pollutant name (1), indicate whether the pollutant will be included in the output (2), whether short- or long-term calculations are required (3), and select the averaging time (4) and the output units (8). The other parameters (5, 6, 7) are optional.

1. **Name**

   The name of the pollutants for which concentrations are to be calculated can be selected from the list. The list contains all pollutants from the **Palette of Pollutants**. Two additional pollutants (NO and O_3) are also available if **Chemistry** is modelled (refer to Section 4.8 for details).

   Default = **NOx**

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

2. **Include**

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

Default = included

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

3. **Short/Long**

For each output pollutant, the type of average (short-term or long-term) to be calculated must be specified. Click in this cell to toggle between **ST** (short-term) and **LT** (long-term). Following these first three columns, various other options are available in the rest of the table depending on whether **ST** or **LT** is selected, as illustrated in **Figure 3.23**.

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

If long-term averages are selected, then the concentration output files (e.g. *filename.glt* for gridded output and *filename.plt* for specified points) contain *a single set of concentration data*, averaged over all the lines of meteorological data.

Default = **ST**

4. **Averaging time (Value, Unit)**

Select the appropriate averaging time over which concentrations should be averaged for each pollutant. Note that ADMS 5 allows the user to select different averaging times for different pollutants or the same pollutant *in the same run* (e.g. 15 minutes for SO_2 and 1 hour for NO_X). A drop-down list presents commonly-used averaging times, and may be accessed by clicking on the **Value** cell and then pressing the **ALT** and ↓ keys.

See Appendix C for the averaging times relevant to air quality guidelines (e.g. NAQS objectives) and Section 3.6.3 for a convenient way to enter these.

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

*It is **essential** to use hourly sequential meteorological data (and select **Met. Data are hourly sequential** on the **Meteorology** screen) in order to be able to select an averaging time greater than 1 hour.*

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

5. **Extra condition**

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

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

Default = **None**

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

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

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

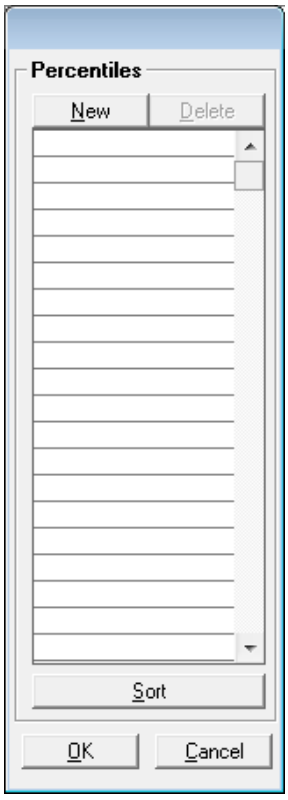


Figure 3.24 − The **Percentiles** screen in the **Output** screen.

Percentile values are entered via a table (as shown in **Figure 3.24**) which is made available when the user clicks on the **Edit as Table** option from the list in the **Percentiles** value cell. Click on the **New** and **Delete** buttons to add/remove values, respectively. The resulting list of percentiles will then be used for the given pollutant. Click on the **Sort** button to arrange the values in descending order for convenience.

7. **Exceedences** (long-term average only)

   If long-term averages are being calculated, then up to 20 exceedence values may be entered. For each pollutant, the exceedence values are entered in the same units as displayed in the **Units for output** cell. The value of number of exceedences per annum will then be calculated by the model at each output point for each exceedence value entered. This is a useful way of looking at the results from the model because many air quality standards are stated in terms of number of exceedences per year (refer to Section 3.6.3 and Appendix C for more details).

   As for percentiles, different exceedence values may be entered for different pollutants. Exceedence values are entered via a table, which is made available when the user clicks on the **Edit as Table** option from the list in the exceedence value cell. Click on the **New** and **Delete** buttons to add/remove values, respectively. The resulting list of exceedences will then be used for the given pollutant. Click on the **Sort** button to arrange the values in descending order for convenience.

8. **Units for output**

For each output pollutant, the units may be selected from the list available by clicking on this cell. Unless radioactive isotopes are modelled, or the odours option is selected, the units available are:

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

Default = $\mu g/m^3$

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

For radioactive isotopes, output is given in Bq/m^3.

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

3.6.2 Group and source output

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

To obtain output for a specific source, select the **Source** option above the lower right-hand table on the **Output** screen (shown in **Figure 3.23**). The table will show in the **Name** column a list of all individual sources available for the current model run. **Only one** of these sources may be selected (click in the **Include** cell next to the required source or enter **Y** in that cell). Choosing the single source option generates files containing model parameters on the plume centreline, e.g. σ_y, σ_z, that it may be useful to plot using the **Line Plot** utility (refer to Section 6.4 for further details).

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

A group may be defined to contain only one source, so that output may be generated for several individual sources if required. If more than 20 user-defined groups are to be modelled, more than one model run will be necessary.

3.6.3 Comprehensive output file

Selecting long-term output means that the output files contain a single set of output data, for example, the average or maximum over all of the lines of meteorological data. By selecting the **Comprehensive output file** option, an additional file containing output data for each met line, for all the pollutants selected for long-term output, is also created. This file is created in netCDF format and a full description of the file is given in Section 6.1.7.

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

A Comprehensive Output File Processor is supplied with ADMS 5 and can be launched from the ***Results****,* ***Process comprehensive output*** *menu. A full User Guide for the Comprehensive Output File Processor can be found in the* <install dir>\Documents *directory.*

The netCDF files produced by this option can be very large. By default the run will stop if a netCDF file of greater than 20 Gb is required, based on the output options selected. The file size limit can be altered using the *.aai* file. This is done by creating an *.aai* file or editing an existing *.aai* file (refer to section 3.1.6), enabling the **Maximum NetCDF output file size** (keyword NETCDFOUTPUT) and entering the **Maximum file size** in Mb.

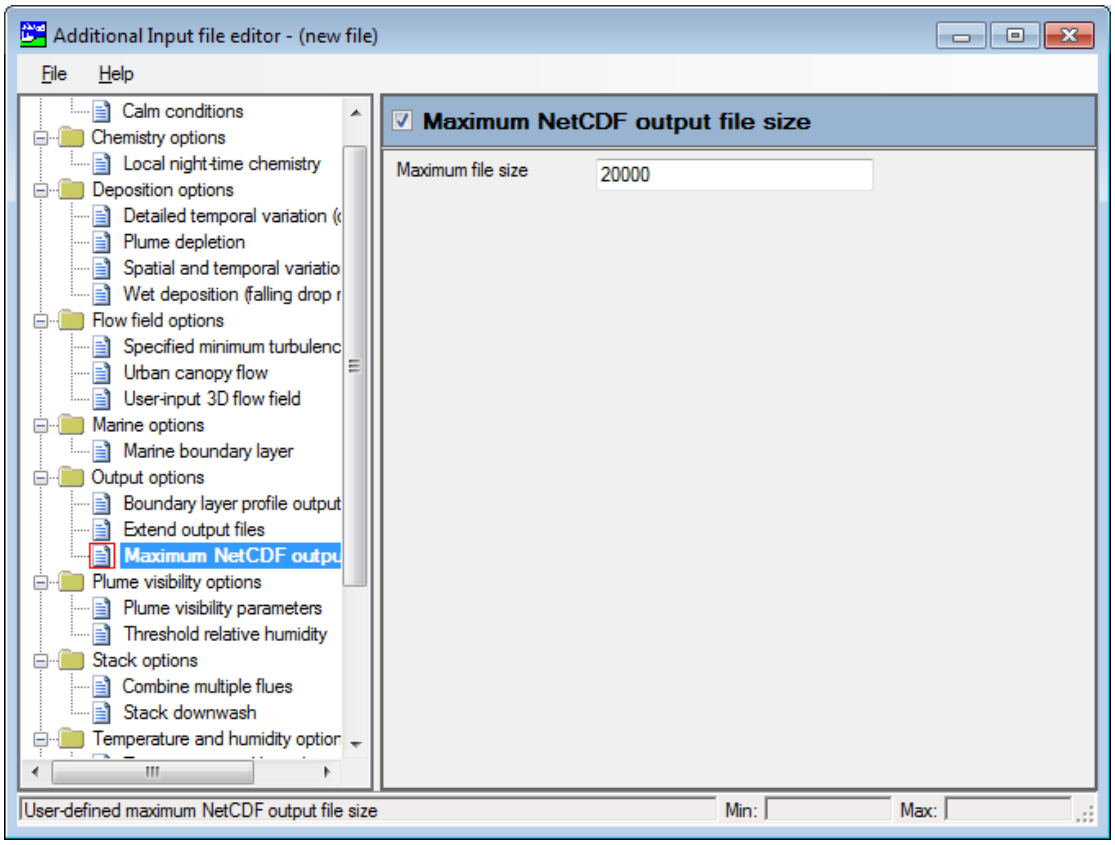


Figure 3.25 − The **Maximum NetCDF output file size** screen in the **Additional Input file editor**.

3.6.4 Air quality objectives

Many users are interested in comparing model results with air quality objectives. An option is available to facilitate this for some standard sets of objectives. Click on the **Air quality objectives** list on the **Output** screen, select a set of objectives and click the tick (see **Figure 3.26**). This action will populate the output table with the relevant air quality objective values (see **Figure 3.27**). When using this tool, the user is advised to check that the objectives are up-to-date.

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

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

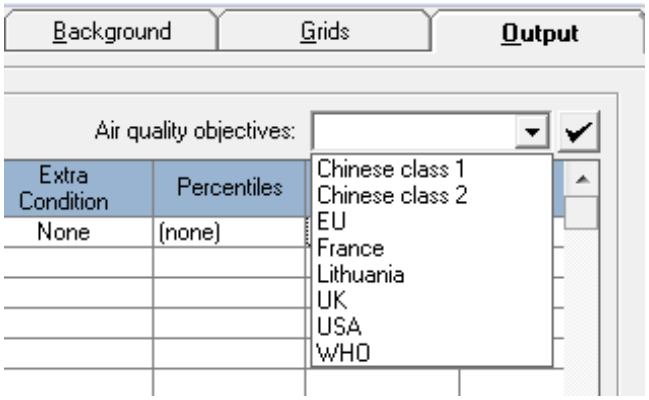


Figure 3.26 The **Air quality objectives** list

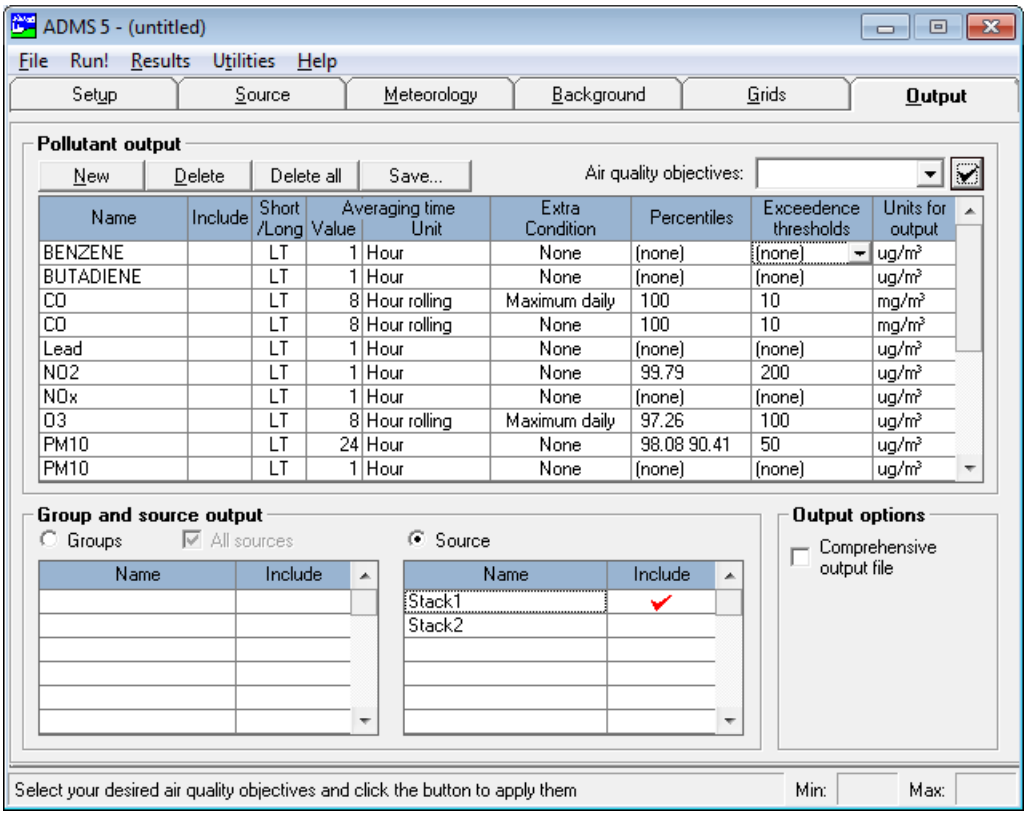


Figure 3.27 − The **Output** screen showing the effect of selecting **UK** from the **Air quality objectives** list and then clicking the tick.

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

Limitations

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

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

Saving air quality objective files

It is also possible to save your own set of **Air quality objectives**. Firstly, the required outputs should be entered into the **Pollutant output** table. Selecting **Save**... then brings up the **Save pollutant outputs to file** screen, as shown in **Figure 3.28**. In this screen enter:

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

then click **OK**.

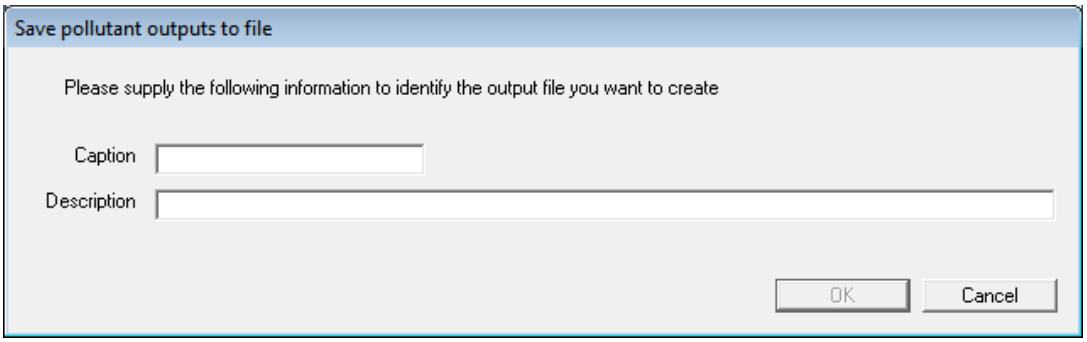


Figure 3.28 − The **Save pollutant outputs to file** screen.

SECTION 4 Additional Model Options

This section describes additional model options available in ADMS 5.

Options such as deposition, radioactive decay, plume visibility, odours, chemistry, buildings, complex terrain, coastline, puffs, and fluctuations are available in the **Model options** section of the **Setup** screen, as illustrated in **Figure 4.1**. Options related to the meteorological data are available in the **Meteorology** screen. Time-varying source data options are available in the **Source** screen. Other options are available by means of the *.aai* file, refer to Section 3.1.6 for details of how to create an *.aai* file.

Some of these model options require only basic model input, some require further information to be entered in the ADMS 5 interface and some require information to be entered both in the ADMS 5 interface and in an *.aai* file.

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

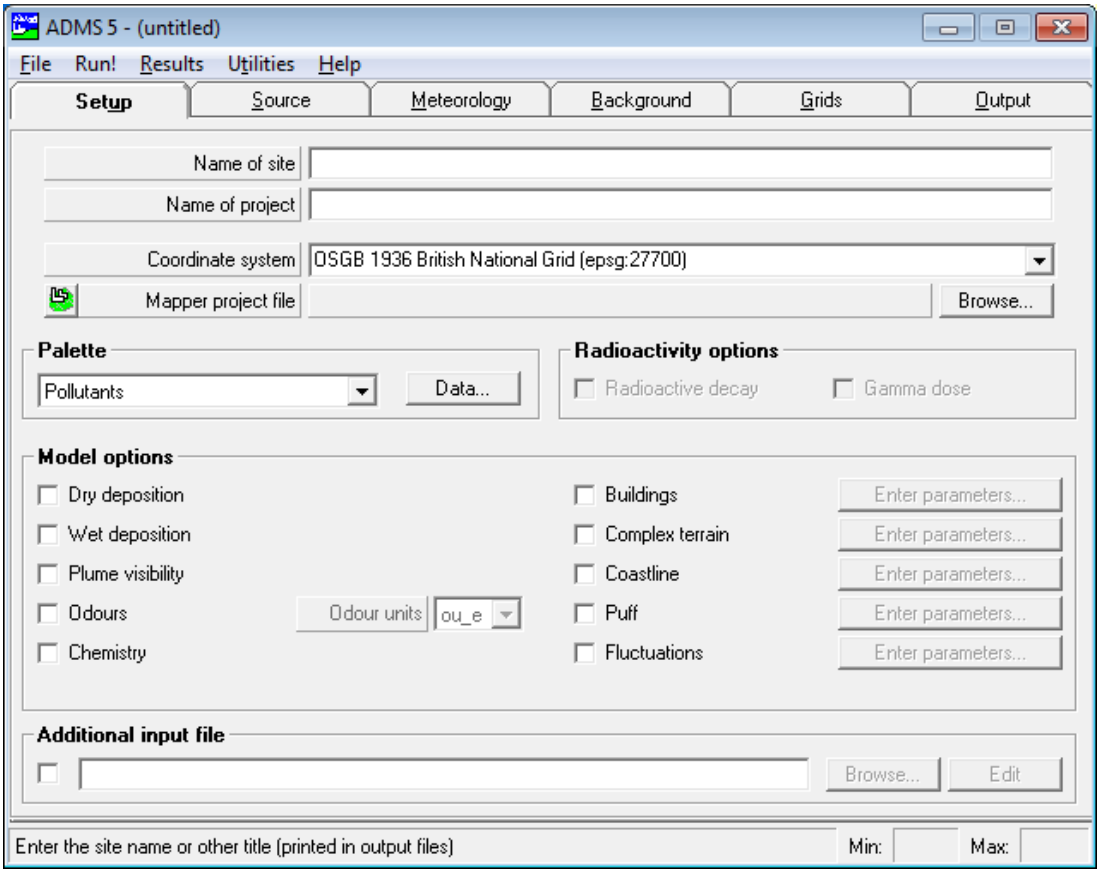


Figure 4.1 − The ADMS 5 **Setup** screen.

4.1 Time-varying source data

ADMS 5 includes the option to model concentrations from emissions that vary with time. To do this the user should tick the **Time-varying source data** box in the **Source** screen (as shown in **Figure 4.2**). Data to describe the time variation may then be entered directly in the interface or from a file. The following options are available:

- **Source data from *.var file**: Enter time variation of source parameters by hour or statistical period using a *.var* file.
- **Emission factors from *.fac file**: Enter diurnal and/or monthly profiles of emission factors along with wind dependent operation using a *.fac* file.
- **Emission factors from screen**: Enter a diurnal profile of emission factors on screen to be applied to all sources.

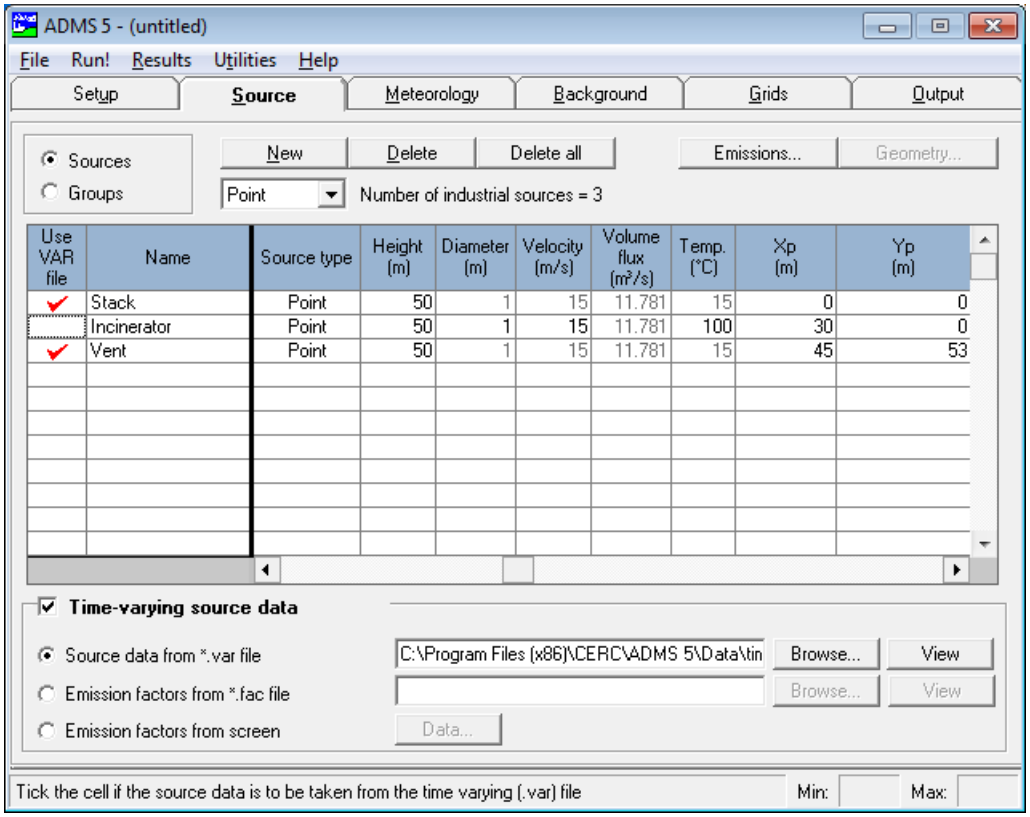


Figure 4.2 − The ADMS 5 **Source** screen with the **Time-varying source data** and **Source data from *.var file** options selected.

4.1.1 Source data from a .var file

A *.var* file can be used to enter source parameters and emission rates for sources so that they vary either by hour or by statistical period. A range of parameters can be entered depending on the nature of the time-varying emissions, but the following data must be entered:

1. Time information for each line of meteorological data. This is either **year**, **day**, **hour** for sequential data or **season**, **period** for statistical data. These parameters must be given prior to the other time-varying parameters.
2. A combination of momentum and buoyancy parameters, unless the source is a volume source. This can be one of:

   * **volume flow rate**, **mass flow rate**, or **exit velocity**

   together with one of:

   * **exit temperature**, **density**, or **ambient**;

   or it can be both:

   * **momentum flux** (F_M) and **buoyancy flux** (F_B).

   More details on these options are given below.

3. When a point or jet source is being modelled and a *.var* file is being used for that source, then the file must include the source **diameter**. Likewise if plume visibility is being modelled for a source that is included in the *.var* file then the **initial mixing ratio of water** must also be specified in the file.
4. Pollutant **emission rates**.

Other optional parameters which may be varied are:

5. Source **location** (**easting** and **northing**) (for point and jet sources only).
6. Source **height**
7. Source **width** (for line sources only).
8. Source **depth** (for volume sources only).
9. **Angle 1**, **Angle 2** (for jet sources only).

To enter source data using a *.var* file select **Source data from *.var file** and use the **Browse**... button to locate the *.var* file. The **View** button can be used to view the *.var* file once it has been selected. For sources whose parameters are to be entered using the *.var* file, create a source of the correct name on the **Source** screen of the ADMS 5 interface, then place a tick in the **Use VAR file** column for that source by clicking in the appropriate cell. Fill in the non-varying source parameters e.g. source location and/or geometry. The columns for the data that will be unused by the model, because they must be in the *.var* file, are greyed out (as shown in **Figure 4.2**). The optional time-varying data are not greyed out, but any value given in the interface will be replaced with the values in the *.var* file.

Either statistical or hourly sequential meteorological data may be used with the *.var* file option. However, if statistical data files are used then the frequency of occurrence of each meteorological condition for the hours of the day and the month of the year must be included in the meteorological file, i.e. it is not sufficient to have only the overall frequency. Two example files, *timevary.var* and *statvary.var*, are provided with the model and show the format that such a file should follow when sequential and statistical meteorological data, respectively, are being used.

The format for each of the file types is described in more detail below.

File format

.var files are comma-separated text files, which means that they can be readily imported and edited in a spreadsheet package, such as Microsoft Excel. The data are entered in separate rows and columns, as described below.

The first two lines in the *.var* file specify formatting parameters. The first line is the version parameter, which is currently:

```
VarFileVersion2
```

The second line differs depending on whether the data are sequential or statistical. The format of this line for a sequential file is:

```
NSVARY, NSTATORSEQ, NMET
```

or, for a statistical file:

```
NSVARY, NSTATORSEQ, NSEAS, NDAY
```

where:

`NSVARY` is the number of time varying sources.

`NSTATORSEQ = 0` or `1` according to whether the meteorological data are sequential or statistical, respectively. For statistical data, to allow for the full effect of emissions varying with time, the meteorological data file must include the full frequency information, that is frequencies of meteorological conditions for months of the year (`MONTHS n1 TO n2`) as well as hours of the day (`HOURS n3 TO n4`).

`NMET` is the number of hourly observations in the meteorological data modelled

`NSEAS` is the number of seasons of the year for which frequency data are specified in the meteorological data file, i.e. the number of sections of `MONTHS n1 TO n2`. The seasons must cover all months throughout the year but need not be of equal duration.

`NDAY` is the number of periods of the day for which frequency data are specified in the meteorological data file, i.e. the number of sections into which `HOURS n3 TO n4` are divided. The periods must cover all hours throughout the day but need not be of equal duration.

The next two lines in the file are source-specific headers, which are repeated at the top of the section for each source included in the *.var* file. In the first of these lines is the source name, which must match the name in the *.apl* file.

The next line contains the header information identifying which time-varying variables are being specified for the source. For sequential data the first three variables must be `YEAR, DAY, HOUR`. For statistical data the first two variables must be `SEASON, PERIOD`. Then the other time-varying variables can be specified in any order, using a specific naming convention as shown in **Table 4.1**.

The time-varying data are then entered from the fifth line onwards for either: the number of hourly meteorological lines (NMET) for a sequential file, or the total number of periods (NSEAS x NDAY) for a statistical file.

The source-specific header information (source name and then time-varying variables) for the next source is then added in the next two lines immediately after the data for the previous source, followed by the time-varying data. These are repeated for every source included in the *.var* file.

An example sequential *.var* file is shown in **Figure 4.3** which specifies exit velocity and temperature for Stack 1 and volume flow rate and density for Stack 2. For clarity the example only uses 3 met lines, but in a standard sequential run there would be data for each hour of the year for each source.

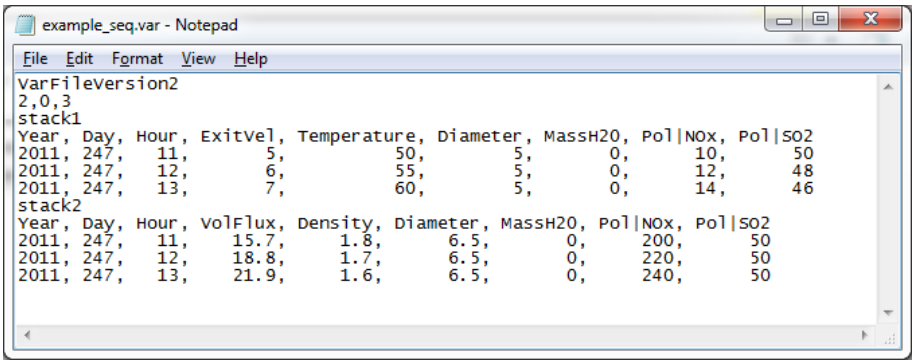
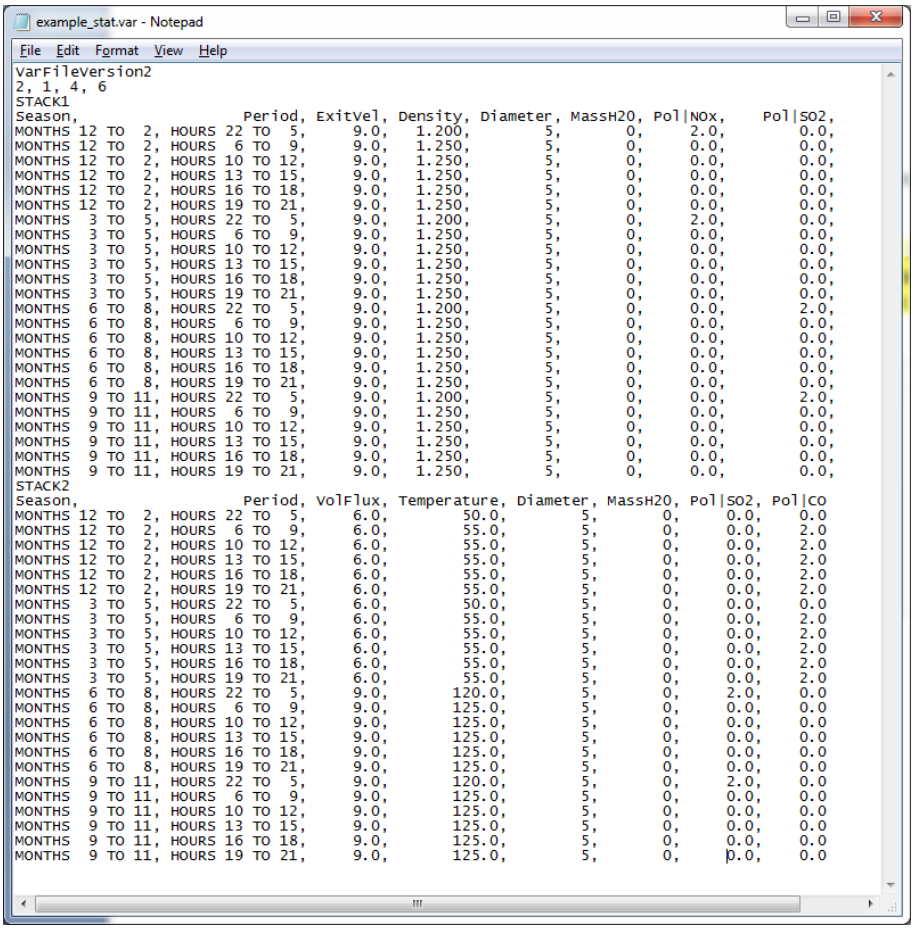


Figure 4.3 − Example of a time varying data file for hourly sequential meteorological data.

Figure 4.4 shows an example statistical *.var* file which specifies exit velocity and density for Stack 1 and volume flow rate and temperature for Stack 2.

ADMS 5.2 supports the use of version 1 .var *files for backwards compatibility. For information on the format of version 1* .var *files, please refer to the User Guide for ADMS 5.1 or older (available from CERC on request).*



example_stat.var - Notepad

File Edit Format View Help

```
VarFileVersion2
2, 1, 4, 6
STACK1
Season,                    Period, ExitVel, Density, Diameter, MassH2O, Pol|NOx,    Pol|SO2,
MONTHS 12 TO  2, HOURS 22 TO  5,      9.0,   1.200,         5,        0,     2.0,        0.0,
MONTHS 12 TO  2, HOURS  6 TO  9,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS 12 TO  2, HOURS 10 TO 12,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS 12 TO  2, HOURS 13 TO 15,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS 12 TO  2, HOURS 16 TO 18,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS 12 TO  2, HOURS 19 TO 21,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  3 TO  5, HOURS 22 TO  5,      9.0,   1.200,         5,        0,     2.0,        0.0,
MONTHS  3 TO  5, HOURS  6 TO  9,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  3 TO  5, HOURS 10 TO 12,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  3 TO  5, HOURS 13 TO 15,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  3 TO  5, HOURS 16 TO 18,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  3 TO  5, HOURS 19 TO 21,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  6 TO  8, HOURS 22 TO  5,      9.0,   1.200,         5,        0,     0.0,        2.0,
MONTHS  6 TO  8, HOURS  6 TO  9,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  6 TO  8, HOURS 10 TO 12,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  6 TO  8, HOURS 13 TO 15,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  6 TO  8, HOURS 16 TO 18,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  6 TO  8, HOURS 19 TO 21,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  9 TO 11, HOURS 22 TO  5,      9.0,   1.200,         5,        0,     0.0,        2.0,
MONTHS  9 TO 11, HOURS  6 TO  9,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  9 TO 11, HOURS 10 TO 12,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  9 TO 11, HOURS 13 TO 15,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  9 TO 11, HOURS 16 TO 18,      9.0,   1.250,         5,        0,     0.0,        0.0,
MONTHS  9 TO 11, HOURS 19 TO 21,      9.0,   1.250,         5,        0,     0.0,        0.0,
STACK2
Season,                    Period, VolFlux, Temperature, Diameter, MassH2O, Pol|SO2, Pol|CO
MONTHS 12 TO  2, HOURS 22 TO  5,      6.0,        50.0,         5,        0,     0.0,    0.0
MONTHS 12 TO  2, HOURS  6 TO  9,      6.0,        55.0,         5,        0,     0.0,    2.0
MONTHS 12 TO  2, HOURS 10 TO 12,      6.0,        55.0,         5,        0,     0.0,    2.0
MONTHS 12 TO  2, HOURS 13 TO 15,      6.0,        55.0,         5,        0,     0.0,    2.0
MONTHS 12 TO  2, HOURS 16 TO 18,      6.0,        55.0,         5,        0,     0.0,    2.0
MONTHS 12 TO  2, HOURS 19 TO 21,      6.0,        55.0,         5,        0,     0.0,    2.0
MONTHS  3 TO  5, HOURS 22 TO  5,      6.0,        50.0,         5,        0,     0.0,    0.0
MONTHS  3 TO  5, HOURS  6 TO  9,      6.0,        55.0,         5,        0,     0.0,    2.0
MONTHS  3 TO  5, HOURS 10 TO 12,      6.0,        55.0,         5,        0,     0.0,    2.0
MONTHS  3 TO  5, HOURS 13 TO 15,      6.0,        55.0,         5,        0,     0.0,    2.0
MONTHS  3 TO  5, HOURS 16 TO 18,      6.0,        55.0,         5,        0,     0.0,    2.0
MONTHS  3 TO  5, HOURS 19 TO 21,      6.0,        55.0,         5,        0,     0.0,    2.0
MONTHS  6 TO  8, HOURS 22 TO  5,      9.0,       120.0,         5,        0,     2.0,    0.0
MONTHS  6 TO  8, HOURS  6 TO  9,      9.0,       125.0,         5,        0,     0.0,    0.0
MONTHS  6 TO  8, HOURS 10 TO 12,      9.0,       125.0,         5,        0,     0.0,    0.0
MONTHS  6 TO  8, HOURS 13 TO 15,      9.0,       125.0,         5,        0,     0.0,    0.0
MONTHS  6 TO  8, HOURS 16 TO 18,      9.0,       125.0,         5,        0,     0.0,    0.0
MONTHS  6 TO  8, HOURS 19 TO 21,      9.0,       125.0,         5,        0,     0.0,    0.0
MONTHS  9 TO 11, HOURS 22 TO  5,      9.0,       120.0,         5,        0,     2.0,    0.0
MONTHS  9 TO 11, HOURS  6 TO  9,      9.0,       125.0,         5,        0,     0.0,    0.0
MONTHS  9 TO 11, HOURS 10 TO 12,      9.0,       125.0,         5,        0,     0.0,    0.0
MONTHS  9 TO 11, HOURS 13 TO 15,      9.0,       125.0,         5,        0,     0.0,    0.0
MONTHS  9 TO 11, HOURS 16 TO 18,      9.0,       125.0,         5,        0,     0.0,    0.0
MONTHS  9 TO 11, HOURS 19 TO 21,      9.0,       125.0,         5,        0,     0.0,    0.0
```

Figure 4.4 − Example of time varying data file for statistical meteorological data.

Variable Name	Variable & Units	Comments
ExitVel	Exit velocity (m/s)	Cannot be specified with VolFlux, MassFlux or Fm and Fb. Cannot be specified for volume sources.
VolFlux	Volume flow rate (m^3/s)	Cannot be specified with ExitVel, Mass Flux or Fm and Fb. Cannot be specified for volume sources.
MassFlux	Mass flow rate (kg/s)	Cannot be specified with ExitVel, VolFlux or Fm and Fb. Cannot be specified for volume sources.
Temperature	Exit temperature (°C)	Cannot be specified with Density or Ambient or Fm and Fb. Cannot be specified for volume sources.
Density	Exit density (kg/m^3)	Cannot be specified with Temperature or Ambient or Fm and Fb. Cannot be specified for volume sources.
Ambient	Exit temperature relative to ambient temperature (°C)	Cannot be specified with Temperature or Density or Fm and Fb. This is the temperature relative to ambient conditions so 0°C indicates ambient; 5°C indicates ambient+5°C and so on. Cannot be specified for volume sources.
Fm	Momentum flux (m^4/s^2)	Both Fm and Fb must be included if either is to be specified. Cannot be specified with any other momentum/buoyancy efflux option. Cannot be specified for volume sources.
Fb	Buoyancy flux (MW)	
Diameter	Source diameter (m)	Mandatory for point and jet sources.
MassH20	Initial mixing ratio of water (kg/kg)	Mandatory when plume visibility is modelled for point sources.
Xp	Source location, easting (m)	Optional. Can only be specified for point or jet sources. Both Xp and Yp must be included to specify a time-varying position.
Yp	Source location, northing (m)	
Height	Source height (m)	Optional.
Width	Source width (m)	Optional. Can only be specified for line sources.
Depth	Source depth (m)	Optional. Can only be specified for volume sources.

Table 4.1 – Time-varying variables that can be specified in a *.var* file.

Variable Name	Variable & Units	Comments
`Angle1`	Angle of elevation for jet source (degrees from horizontal)	Optional. Can only be specified for jet sources. Both angles must be included to specify a time-varying angle of release
`Angle2`	Angle in horizontal for jet source (degrees from west-east axis, measured anti-clockwise)	
`Pol|<Pol name>`	Pollutant emission rate. Repeated for each pollutant. Units are (g/s), (g/m/s), ($g/m^2/s$) or ($g/m^3/s$) depending on source type. Or equivalent for radioactivity or odour options.	Each pollutant emission rate is specified. Pollutants are named with the prefix `Pol|` followed by the standard pollutant name, as given in the pollutant palette in the interface, e.g. for NO_X specify `Pol|NOx`.

Table 4.1 – Time-varying variables that can be specified in a *.var* file. *(continued)*

4.1.2 Emission factors from a .fac file

ADMS 5 can take account of diurnal and seasonal changes in emissions for each source, and can model sources that are only in operation for a specific range of wind directions. This is done by setting up an extra input file, *.fac*, which contains the emission factors and data linking each source to the appropriate set(s) of emission factors. The format of the *.fac* file is described below.

The full functionality of this option is as follows:

- Hourly factors can be defined for weekdays, Saturdays and Sundays ('3-day diurnal profiles') or for each day of the week ('7-day diurnal profiles'). A different set of factors can be applied to each source.
- Monthly factors can be defined for each month ('monthly profiles'). A different set of factors can be applied to each source.
- Each source can be defined to be in operation only for a specific range of wind directions.

To use a *.fac* file in a modelling run, select **Emission factors from *.fac file** on the **Source** screen in the ADMS 5 interface.

.fac file format

An example time-varying emission factors file is shown in **Figure 4.5**. The file contains two 3-day diurnal profiles, one 7-day diurnal profile and one monthly profile, followed by data for three point sources, point1, point2 and point3. Note that the data in the file are factors, i.e. the emission rates entered on the **Emissions** screen of the interface will be multiplied by these values. The data in the file are as follows.

1. **FacVersion3** the version number for the file

   Note that ADMS 5.2 will also accept .fac *files in FacVersion2 format, but future versions may not.*

2. **3-day Diurnal profiles**

   Line 1: Number of 3-day diurnal profiles included in the file
   Line 2: Name of the 3-day diurnal profile, up to 20 characters long
   Line 3: The 24 hourly factors for weekdays, in order, from hour 1 to hour 24
   Line 4: The 24 hourly factors for Saturdays, in order, from hour 1 to hour 24
   Line 5: The 24 hourly factors for Sundays, in order, from hour 1 to hour 24

   Line 1 must always be included, even if no 3-day diurnal profiles are included in the file. Lines 2-5 are repeated for each 3-day diurnal profile.

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

3. **7-day Diurnal profiles**

   Line 1: Number of 7-day diurnal profiles included in the file
   Line 2: Name of the 7-day diurnal profile, up to 20 characters long

Line 3: The 24 hourly factors for Mondays, in order, from hour 1 to hour 24
Line 4: The 24 hourly factors for Tuesdays, in order, from hour 1 to hour 24
Line 5: The 24 hourly factors for Wednesdays, in order, from hour 1 to hour 24
Line 6: The 24 hourly factors for Thursdays, in order, from hour 1 to hour 24
Line 7: The 24 hourly factors for Fridays, in order, from hour 1 to hour 24
Line 8: The 24 hourly factors for Saturdays, in order, from hour 1 to hour 24
Line 9: The 24 hourly factors for Sundays, in order, from hour 1 to hour 24

Line 1 must always be included, even if no 7-day diurnal profiles are included in the file. Lines 2-5 are repeated for each 7-day diurnal profile.

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

4. Monthly profiles

Line 1: Number of monthly profiles included in the file
Line 2: Name of the monthly profile, up to 20 characters long
Line 3: The 12 monthly factors for the 12 months, in order, from January to December

Line 1 must always be included, even if no monthly profiles are included. Lines 2-3 are repeated for each monthly profile.

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

5. Source-specific data

Line 1: **`SrcName, HourlyFlag, MonthlyFlag, WindFlag`**
Line 2: Name of the diurnal profile to be applied to this source (only included for sources for which **`HourlyFlag`** = 1)
Line 3: Name of the monthly profile to be applied to this source (only included for sources for which **`MonthlyFlag`** = 1)
Line 4: **`PhiStart`**, **`PhiEnd`** (only included for sources with **`WindFlag`** = 1)

where

`SrcName` is the source name.

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

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

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

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

`PhiEnd` is the end of the range. Values are in degrees moving clockwise from

North.

Lines 1, and 2-4 if appropriate, are repeated for each source.

In the example shown in **Figure 4.5**, the emissions from point1 vary with the hour using the 7-day diurnal profile, and with the month. The emissions from point2 vary with the hour, using the 3-day diurnal profile profile1, and the stack is only operational if the wind direction is greater than or equal to 350° or less than or equal to 10°. The emissions from point3 vary with the hour, using the 3-day diurnal profile profile2.

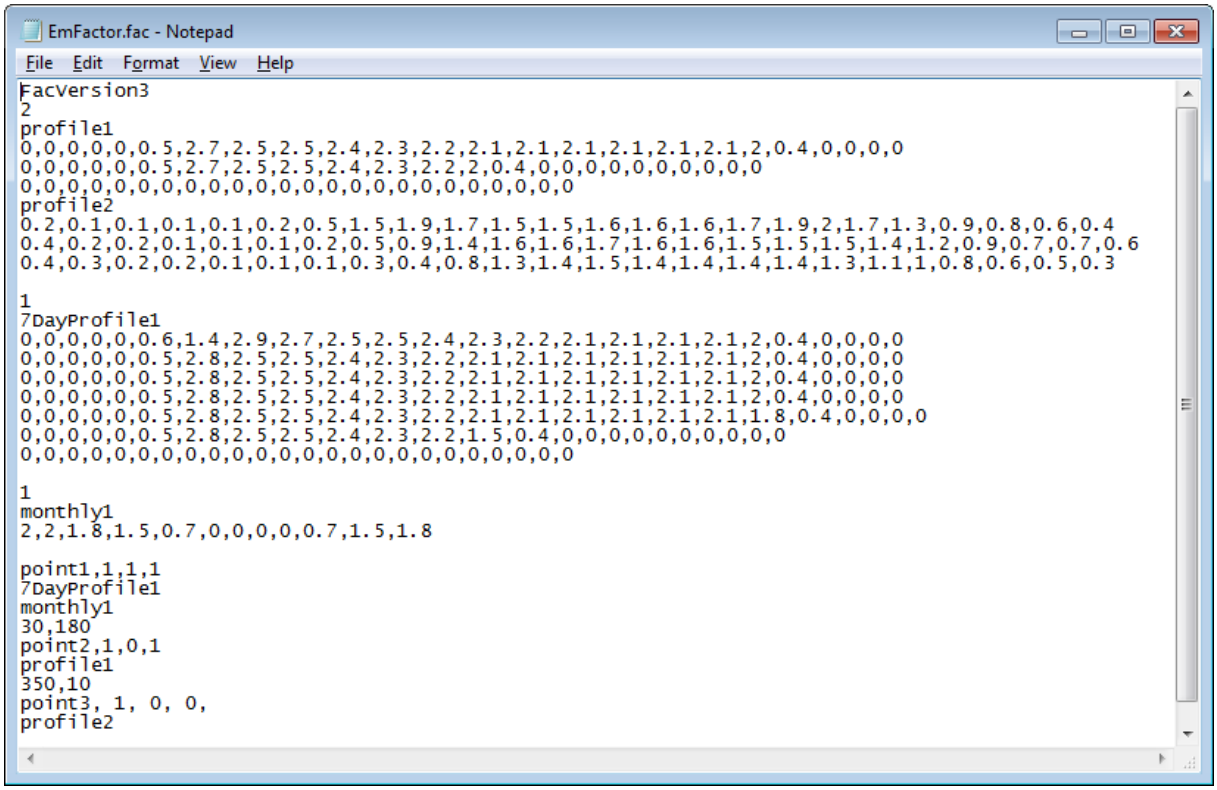


Figure 4.5 − Example time-varying emission factors file (*.fac*).

Important points to note:

- None of the text in the file is case-sensitive.
- Source and profile names can contain spaces.
- The file must be in comma-separated format, and source and profile names must not contain commas.
- There should be no blank lines, except for the lines between the 3-day diurnal profile data and the 7-day diurnal profile data, between the 7-day diurnal profile data and the monthly profile data, and between the monthly profile data and the source-specific data.
- Each source can use either a 3-day diurnal profile, with separate hourly factors for weekdays, Saturdays and Sundays, or a 7-day diurnal profile, with separate hourly factors for each day of the week, but not both. Both 3-day and 7-day diurnal profiles can be included in the same *.fac* file, for use by different sources.

- Any wind direction sector limits are applied to the wind direction following the pre-processing of the meteorological data, i.e. the wind direction given in the *.mop* file rather than the input *.met* file.
- Any sources omitted from the *.fac* file are assumed to have constant emissions.

4.1.3 Emission factors from screen

To enter a set of emission factors to be used by all sources, first select the **Emission factors from screen** option and then click on the **Data**... button in the lower section of the **Source** screen to display the screen shown in **Figure 4.6**.

Factors may be entered for each hour in a 24-hour period, with different factors according to whether it is a weekday, Saturday or Sunday. The release rate given in the **Emissions** screen for each source will be multiplied by the appropriate factor for each hour.

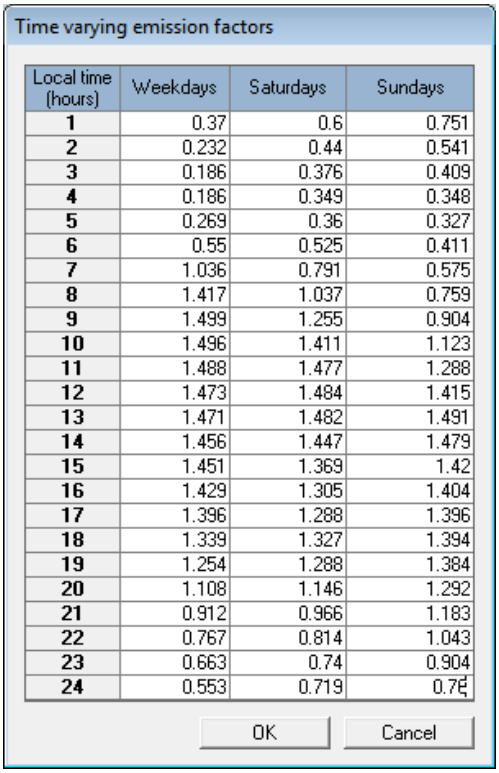


Figure 4.6 − Time varying emission factors screen.

4.1.4 Daylight saving time

Times entered into ADMS 5 should be entered as hour ending values in local solar time and should be consistent between all input files including the time-varying emissions factors. In many countries daylight saving time is used during the summer to shift the times forward by one hour. This changes the relationship between local solar time and clock time during the daylight saving time periods.

For input files where the year, day and hour are input, including the *.var* file, the input should all remain in local solar time. For emissions factors entered on screen or using

a *.fac* file it may be required to include effects of the one hour shift during daylight saving time.

Daylight saving time is included in the model by creating a new *.aai* file, or editing the existing *.aai* file (refer to Section 3.1.6) and enabling the **Daylight saving time** option (keyword ADJUSTFACTORS_DST), **Figure 4.7**. Into the table enter the **START** and **END** times of the daylight saving periods, using the **New** button. The times must be entered in the format `YYYY, MM, DD, HH`, eg. for 2 am on 27th March 2016 enter `2016, 03, 27, 02`. The start and end periods must be entered in sequential order. During daylight saving time the emission factor for one hour ahead of local solar time will be used for the source.

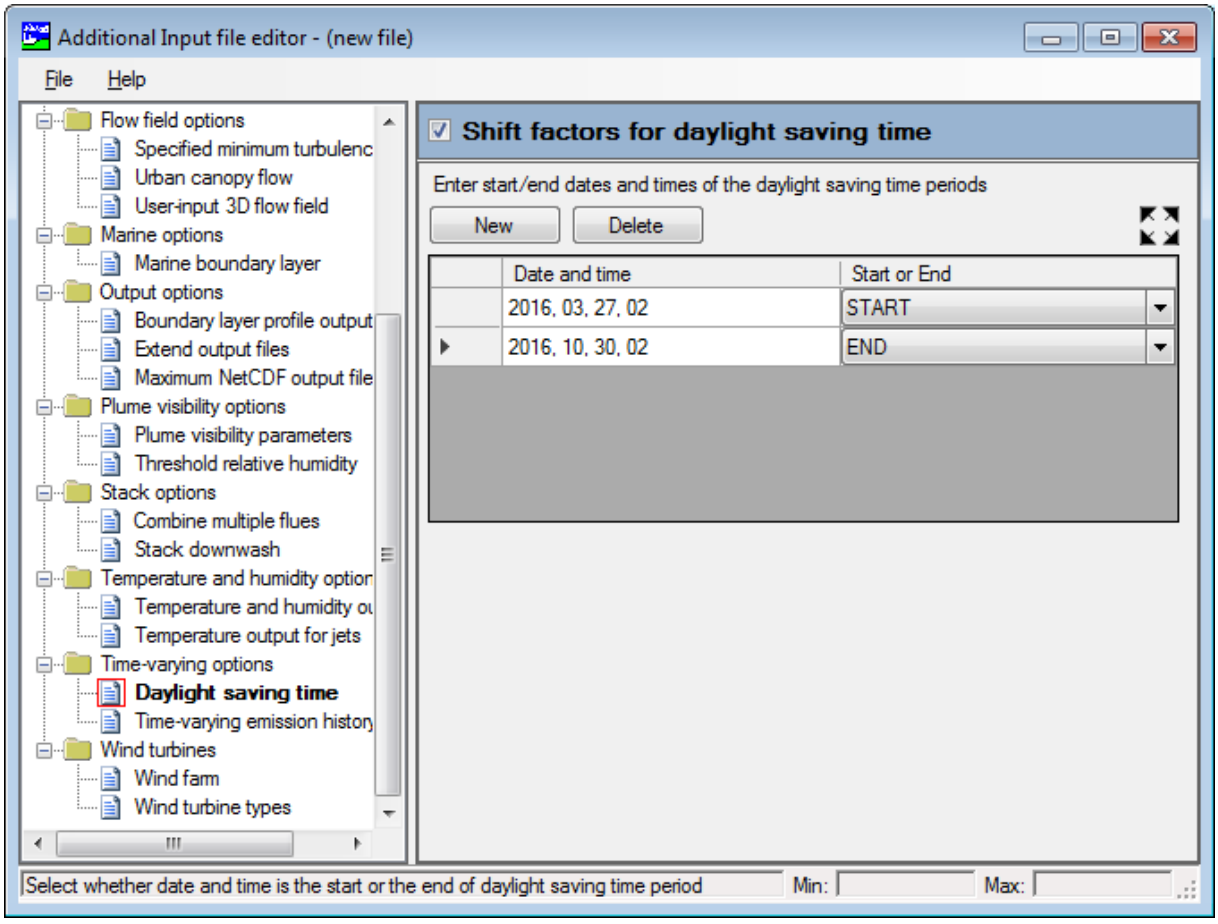


Figure 4.7 – **Daylight saving time** option in the **Additional Input file editor** showing the daylight saving time period for 2016 in the UK.

4.1.5 Time-varying emission history

By default ADMS 5 models each source assuming that the source parameters do not change for that meteorological data line. If the effects of sources are being examined over a large distance then this assumption becomes less valid. For certain source types the emissions are likely to be the quantity which varies the fastest with time. ADMS 5 includes a time-varying emission history option which allows for the effect of the temporal variation of the emission to be taken into account for each meteorological data line.

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

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

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

To enable the time-varying emission history option create a new *.aai* file, or edit the existing *.aai* file (refer to Section 3.1.6) and enable the **Time-varying emission history** option (keyword TIMEVARYQ), as shown in **Figure 4.8**. There are no further parameters required for this option.

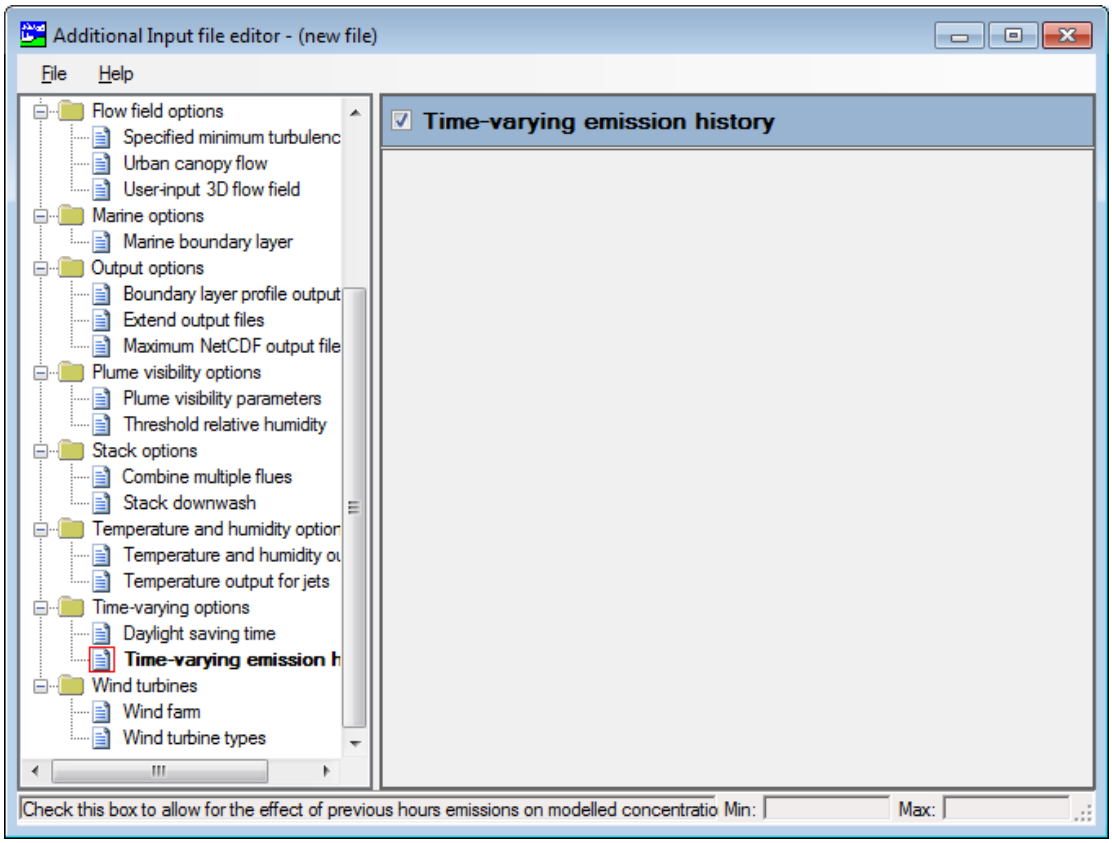


Figure 4.8 – Time-varying emission history option in the **Additional Input file editor**.

The following options cannot be used with the time-varying emission history option:

- time-varying source data from a *.var* file;
- the puff module;
- the plume visibility module;
- temperature and humidity output;
- gamma dose output;
- sources which are combined into a single stack; and
- the wind turbine module.

4.2 Dry deposition

Dry deposition occurs when material is lost from the plume at the surface of the ground. This changes the airborne concentration of pollutant compared with the case of no deposition, by depleting the plume concentration with increasing distance downstream from the source, and adjusting the vertical profile of concentration through removal of material at the surface.

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

$$F = v_d\, C(x, y, 0) \tag{4.1}$$

where F is the rate of deposition per unit area per unit time, v_d is the deposition velocity, and C is the predicted airborne concentration at the position $(x,y,0)$.

The deposition velocity contains a diffusive part, which is commonly referred to as the deposition velocity itself, and the terminal velocity of a particle, which is due to gravitational settling in the plume. The reciprocal of the diffusive part is called the resistance. This resistance is calculated as a sum of aerodynamic, sub-layer and surface resistances.

Both the deposition velocity (diffusive part) and the terminal velocity may be input directly in the interface. If the values are unknown, they can be estimated by the model, based on gas type for gaseous substances or particle size and particle density for particulate substances. For gaseous substances, if the deposition velocity (diffusive part) is not known, the value of the surface resistance used in the calculation of the deposition velocity (diffusive part) can either be input in the interface or determined by the model from the gas type. For both the deposition velocity and the surface resistance, the values entered can be constant, or vary temporally (seasonally or hourly) and/or spatially.

Further details of the dry deposition calculations are given in Section 9.

*Note that for the particular case of modelling **dry deposition of ammonia**, the current guidance given by the UK Environment Agency recommends modelling **without plume depletion being taken into account**. This methodology has been implemented as a model option in ADMS 5; please refer to Section 4.2.3 for further details.*

4.2.1 Modelling dry deposition

To model dry deposition effects with ADMS 5, pollutant deposition parameters must be specified in the **Palette of Pollutants**.

The palette can be entered by clicking on the **Data**... button in the **Palette** section of the **Setup** screen or by clicking **Pollutants**... on the **Emissions** screen, which is accessed from the **Source** screen.

If temporally (seasonally or hourly) or spatially varying values of the deposition velocity, or user input values of the surface resistance, are required, then the *.aai* file should be used (refer to Section 4.2.2 for more details about how to do this).

It is only necessary to enter deposition parameters if deposition is being modelled, otherwise the model will not use the deposition parameters in the table so the user may retain the default values.

The **Palette of Pollutants** is shown in **Figure 4.9** for the case of a particulate pollutant ('NewPol') for which both deposition and terminal velocities are unknown.

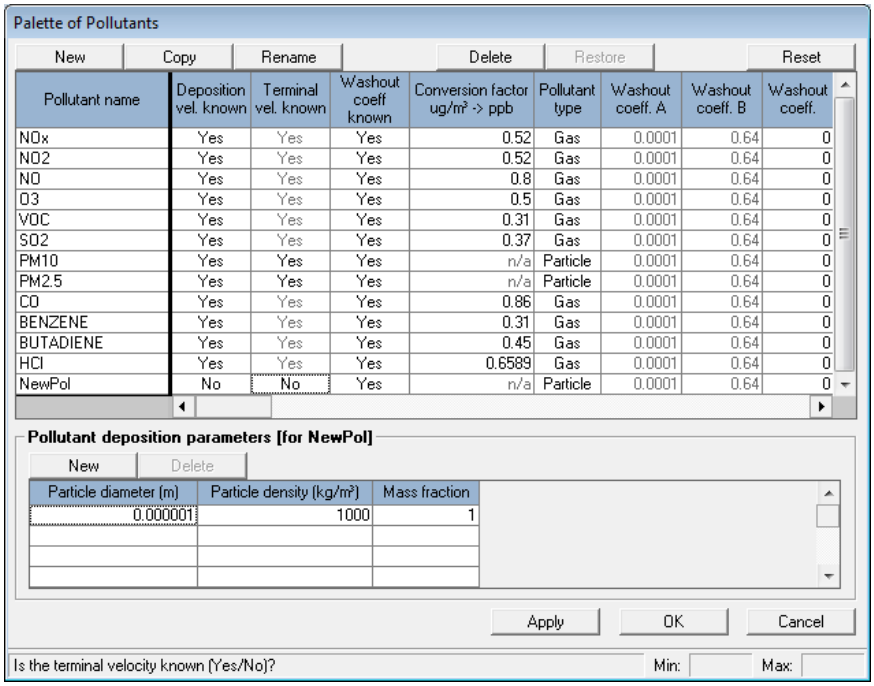


Figure 4.9 − The **Palette of Pollutants** screen showing additional deposition parameters for a particulate pollutant.

The deposition parameters that must be defined depend on whether the pollutant type has been specified as **Gas** or **Particle**.

Each parameter is defined below and **Table 4.2** summarises the input options available for each pollutant type.

1. **Deposition vel. known** (required for both gases and particles)

   Select **Yes** or **No** depending on whether the deposition velocity is known.

   Default = **Yes** (for all pollutants)

2. **Terminal vel. known** (required for particles only)

   Select **Yes** or **No** depending on whether the terminal velocity is known.

   Default = **Yes** (for all particles)

3. **Deposition velocity (m/s)**: rate at which material is deposited to the ground due to diffusion

   For both **Gas** and **Particle** pollutants, if a constant value is to be specified, this should be done in the lower table of **Figure 4.9** (the column **Deposition velocity (m/s)** will appear in the lower table if **Yes** was chosen in the upper table for

Deposition vel. known). Alternatively, if temporally (seasonally or hourly) or spatially varying values are to be used these should be entered using the *.aai* file.

Minimum = 0 m/s
Maximum = 10 m/s
Default = 0 m/s

4. **Nature of gas** (gases only)

If the deposition velocity of a gas is unknown (**Deposition vel. known** = **No**), then the gas nature should be specified (**Figure 4.10**) with one of the following:

Reactive: gas expected to undergo significant chemical reaction with the surface (for example HCl)

Non-Reactive: gas not expected to undergo significant chemical reaction with the surface (for example CO_2, CH_3I, O_3, SO_2, NO_2)

Inert: noble gas (for example He, Ne, Ar, Kr, Xe, Rn)

ADMS 5 uses the above information to calculate the surface resistance. Alternatively, values of the surface resistance may be entered using an *.aai* file.

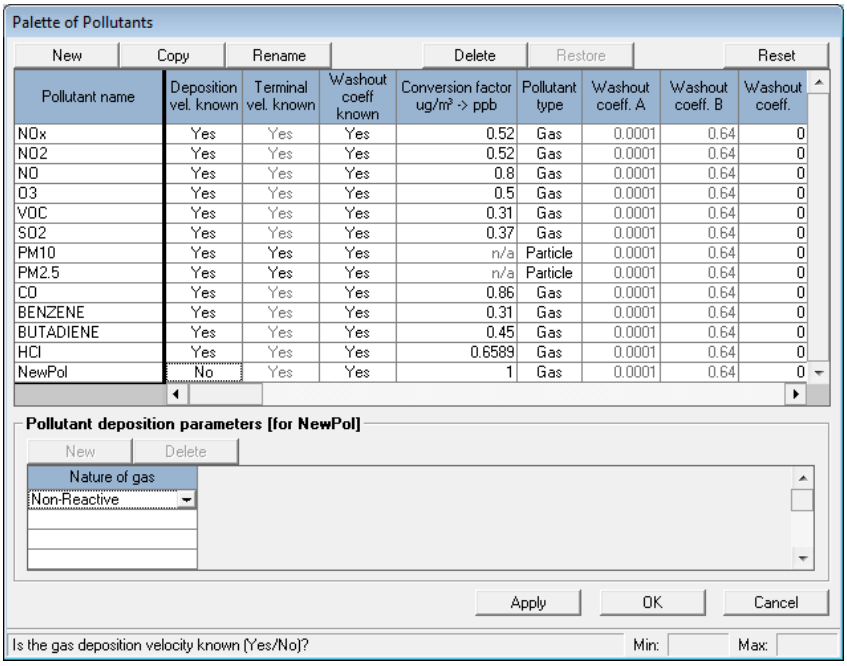


Figure 4.10 − The **Palette of Pollutants** screen showing additional deposition parameters for a gaseous pollutant.

5. **Terminal velocity (m/s)** (particles only)

This represents the velocity at which material is deposited to the ground due to gravitational settling of particles.

Minimum = 0 m/s
Maximum = 10 m/s
Default = 0 m/s (for both pre-defined and user-defined particles)

6. **Particle diameter (m)** (particles only)

   More than one particle diameter may be defined for each particulate pollutant.

   Minimum = 10^{-9} m (= 0.001 μm)
   Maximum = 0.01 m
   Default = 0.00001 m (= 10 μm) for PM_{10}, 0.0000025 m (= 2.5 μm) for $PM_{2.5}$, 0.000001 m (= 1 μm) for user-defined particles

   This is used in the estimation of deposition velocity and/or terminal velocity, for particulate emissions.

7. **Particle density (kg/m³)** (particles only)

   Density of material of which particles are composed. More than one particle density may be defined for each particulate pollutant.

   Minimum = 1 kg/m^3
   Maximum = 1000000 kg/m^3
   Default = 1000 kg/m^3

   This is used in the estimation of terminal velocity for particulate emissions.

8. **Mass fraction** (particles only)

   Mass fraction of the particles with specified densities/diameters (must add up to 1 ± 0.05).

   Minimum = 10^{-10}
   Maximum = 1
   Default = 1

   This is always used for particle emissions and allows the user to model particulate mixtures. As long as the total mass fraction sums to 1, up to 10 different particle size/density combinations can be specified by the user.

The greater the number of particle size/density combinations, the longer the run time required for the calculation.

Pollutant type	Dep. vel. known	Term. vel. known	Dep. velocity	Nature of gas	Term. velocity	Particle diameter	Particle density	Mass fraction
Gas	Yes	-	✓	-	-	-	-	-
Gas	No	-	-	✓	-	-	-	-
Particle	Yes	Yes	✓	-	✓	-	-	✓
Particle	No	Yes	-	-	✓	✓	-	✓
Particle	Yes	No	✓	-	-	✓	✓	✓
Particle	No	No	-	-	-	✓	✓	✓

Table 4.2 − Dry deposition parameters that must be specified for each pollutant type.

4.2.2 Additional input options

It is often the case that the dry deposition parameters change with time, for example due to the diurnal opening and closing of leaf stomata. In addition, deposition parameters may vary spatially due to land use changes over the model domain. ADMS 5 is able to model both temporally and spatially varying deposition parameters. Full details are given in the sections below.

Section 4.2.2.1 describes how hourly or seasonally varying values of the deposition velocity or surface resistance can be modelled; Section 4.2.2.2 gives details of how spatially varying values of dry deposition parameters can be modelled. Sometimes it is useful to be able to model both the hourly and spatial variation of these parameters at the same time, for example when modelling deposition of ammonia; this final model option is described in Section 4.2.2.3.

4.2.2.1 Using temporally varying dry deposition parameters

For both gaseous and particulate pollutants hourly or seasonally varying values of the deposition velocity can be entered. If these are to be entered then the deposition velocity should be set to known in the **Palette of Pollutants**. If seasonally varying values are to be entered then these values should be entered directly into the *.aai* file. If hourly varying values are to be entered these should be entered into a *.vdp* file and the path of this entered into the *.aai* file. See below for definitions of seasons and the format of a *.vdp* file.

For gaseous pollutants, if the deposition velocity is not known then constant, seasonally or hourly varying values of the surface resistance (one of the components used to calculate the deposition velocity) can be entered. If constant or seasonally varying values are known then these are entered directly into the *.aai* file, alternatively if hourly varying values are known then these are entered into a *.rsp* file, the path of which is entered into the *.aai* file. See below for the definition of seasons and the format of the *.rsp* file.

If seasonally or hourly varying values of either the deposition velocity or surface resistance are used then the meteorological data must contain year, day and hour values.

Additional input file

Temporally varying values of the dry deposition parameters are entered using an additional input file. To use these options, create an new additional input file or edit the existing additional input file (see Section 3.1.6) and enable the **Detailed temporal variation (dry deposition)** section (keyword DRYDEP, **Figure 4.11**).

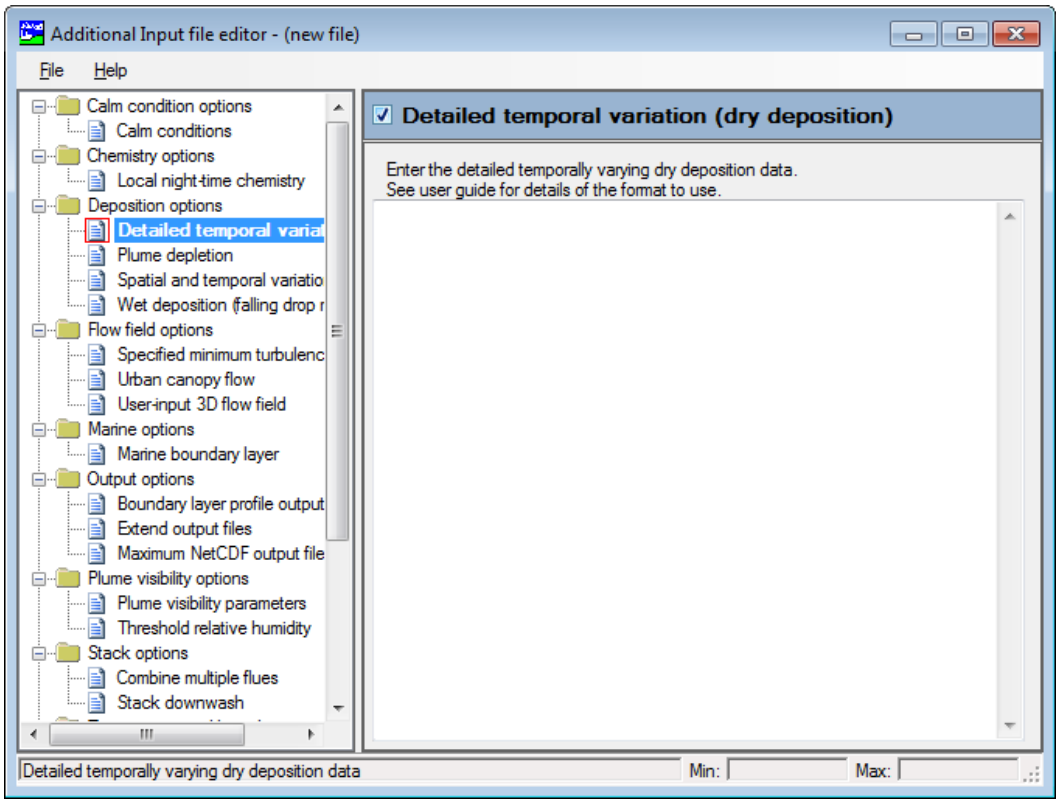


Figure 4.11 – The initial **Detailed temporal variation (dry deposition)** screen of the **Additional Input file editor**.

This section should then be filled in as follows:

- **`NPols`**: number of pollutants for which dry deposition data are listed here.

Then, for each pollutant:

- **`PolName`**: pollutant name.
- **`VdKnownFlag`**: `Y` if deposition velocity known, `N` if unknown.

If `VdKnownFlag` = `Y`:

- **`VdType`**: `0` if deposition velocity is constant, `1` if eight seasonal/diurnal values are specified, or `2` if hourly sequential values of deposition velocity are to be read from a separate file.

  If `VdType` = `1`, enter:

  `Vd_D_Spr`, **`Vd_D_Sum`**, **`Vd_D_Aut`**, **`Vd_D_Win`**: daytime deposition velocity values,
  `Vd_N_Spr`, **`Vd_N_Sum`**, **`Vd_N_Aut`**, **`Vd_N_Win`**: night time deposition velocity values.

  If `VdType` = `2`, enter:

  `PATH = "VdFile"`: where *VdFile* is the full file path of the hourly deposition velocity data file (*.vdp*).

If `VdKnownFlag` = `N`:

- **`RsType`**: `0` if surface resistance constant, `1` if eight seasonal/diurnal values are specified, `2` if hourly sequential values of surface resistance are to be read from a separate file, or `3` if surface resistance is to be inferred from the gas type.

  If `RsType` = `0`, enter:

  `Rs`: constant surface resistance value.

  If `RsType` = `1`, enter:

  `Rs_D_Spr`, **`Rs_D_Sum`**, **`Rs_D_Aut`**, **`Rs_D_Win`**: daytime surface resistance values,
  `Rs_N_Spr`, **`Rs_N_Sum`**, **`Rs_N_Aut`**, **`Rs_N_Win`**: night time surface resistance values.

  If `RsType` = `2`, enter:

  `PATH = "RsFile"`: where *RsFile* is the full file path of the hourly surface resistance data file (*.rsp*).

  If `RsType` = `3`, no additional information is required for this pollutant.

Figure 4.12 shows an example of the **Additional Input file editor** containing detailed dry deposition data for the following four pollutants:

- Pol1 (deposition velocity unknown, constant surface resistance = 750),
- CO (deposition velocity unknown, surface resistance to be inferred from the gas type),
- NO_X (deposition velocity known, 8 seasonal/diurnal deposition velocity values are specified), and
- SO_2 (deposition velocity known, hourly sequential values of deposition velocity to be read from a *.vdp* file).

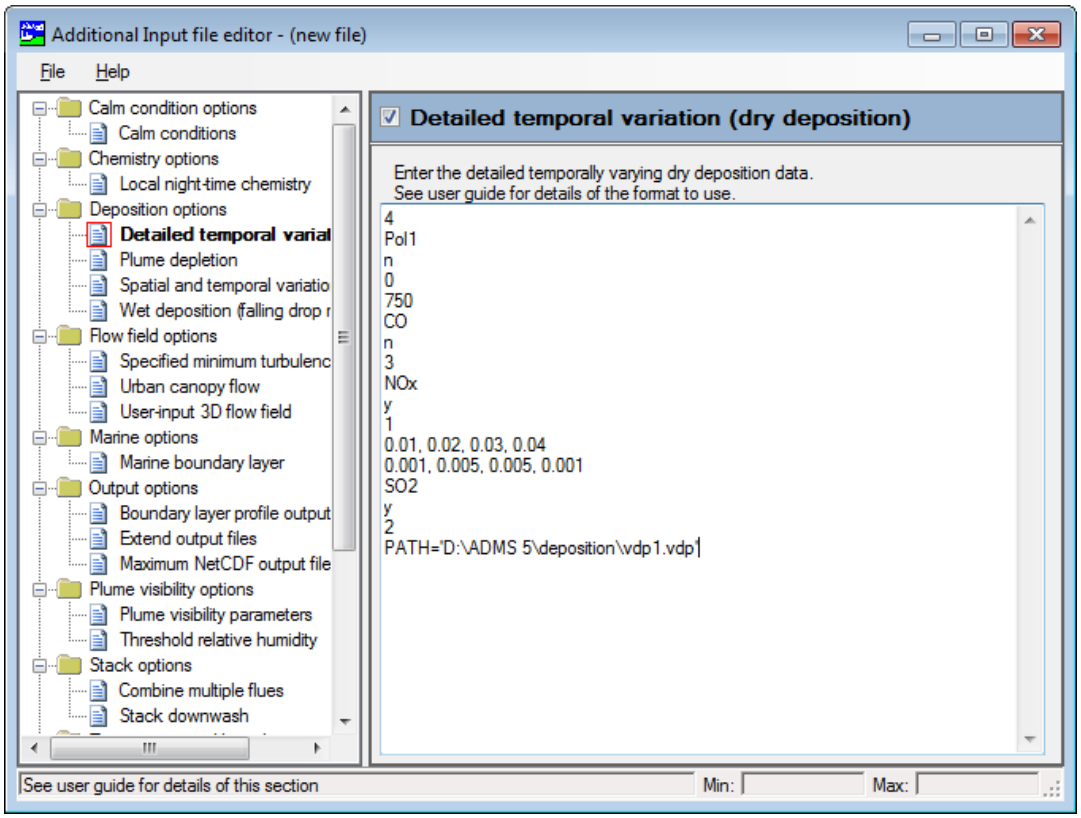


Figure 4.12 – The **Detailed temporal variation (dry deposition)** screen of the **Additional Input file editor** with sample data entered.

Seasons

Seasonally-varying values of either the deposition velocity or the surface resistance can be entered. Separate day and night values are used for each of the four seasons. The seasons are defined using the UK Met Office definition:

Spring = March to May
Summer = June to August
Autumn = September to November
Winter = December to February

Hourly-varying file format

Hourly-varying values of the deposition velocity or surface resistance are entered using a *.vdp* or *.rsp* file respectively. These files have the same format, which is outlined below.

The first line contains the number of pollutants for which data are listed in the file; after that each of the pollutants for which data are entered are listed in turn. Then any comments may be added. The data are then entered: first there is a line containing the key word "`DATA:`", then for each line of meteorological data, the data are entered in the following format:

```
Year, Day, Hour, Value(Pol 1), Value(Pol 2), …
```

where `Value(Pol 1)` is the value of either the deposition velocity or surface resistance for pollutant 1, etc., entered in m/s for deposition velocity or s/m for surface

resistance. An example *.vdp* file with data for three pollutants is shown in **Figure 4.13**.

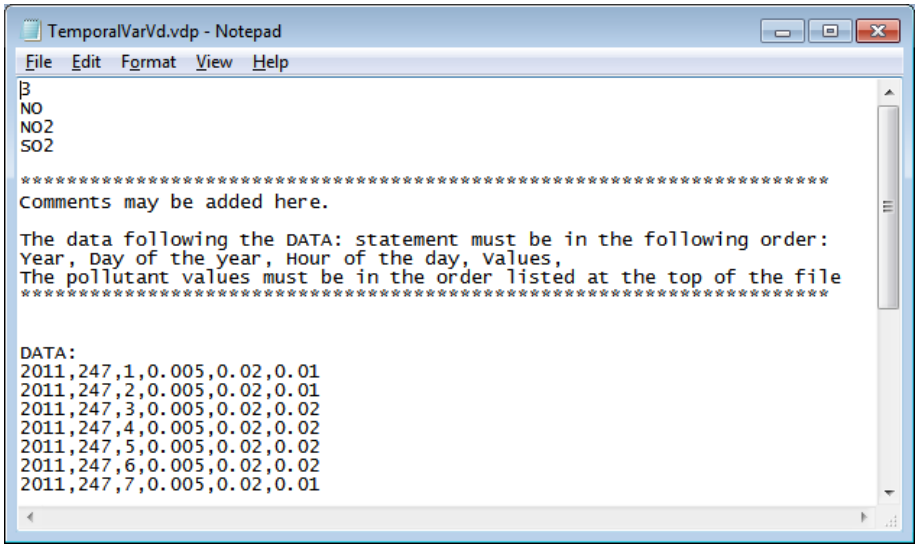


Figure 4.13 − An example of a *.vdp* file.

4.2.2.2 Using spatially varying dry deposition parameters

For both gaseous and particulate pollutants, spatially varying values of the deposition velocity can be used by the model. If these are to be entered for a particular pollutant, then the deposition velocity should be set to known in the **Palette of Pollutants**. For gaseous pollutants, if the deposition velocity is unknown, but the spatial variation of the surface resistance parameter is known, then this information can be used by the model to calculate the deposition velocity. In this case, the deposition velocity should be set to unknown in the **Palette of Pollutants**.

One set of deposition velocity and surface resistance values that vary over the model domain can be entered into a *.din* file and the path of this entered into the *.aai* file. The values specified will be used for all pollutants listed in the relevant section in the *.aai* file. Details of how to set up the additional input file and the format of a *.din* file are given below.

Additional input file

Spatially varying values of the deposition parameters are entered using an additional input file. To use these options, create an new additional input file or edit the existing additional input file (see Section 3.1.6) and enable the **Spatial and temporal variation of deposition parameters** section (keyword VARDEPOSITION, **Figure 4.14**).

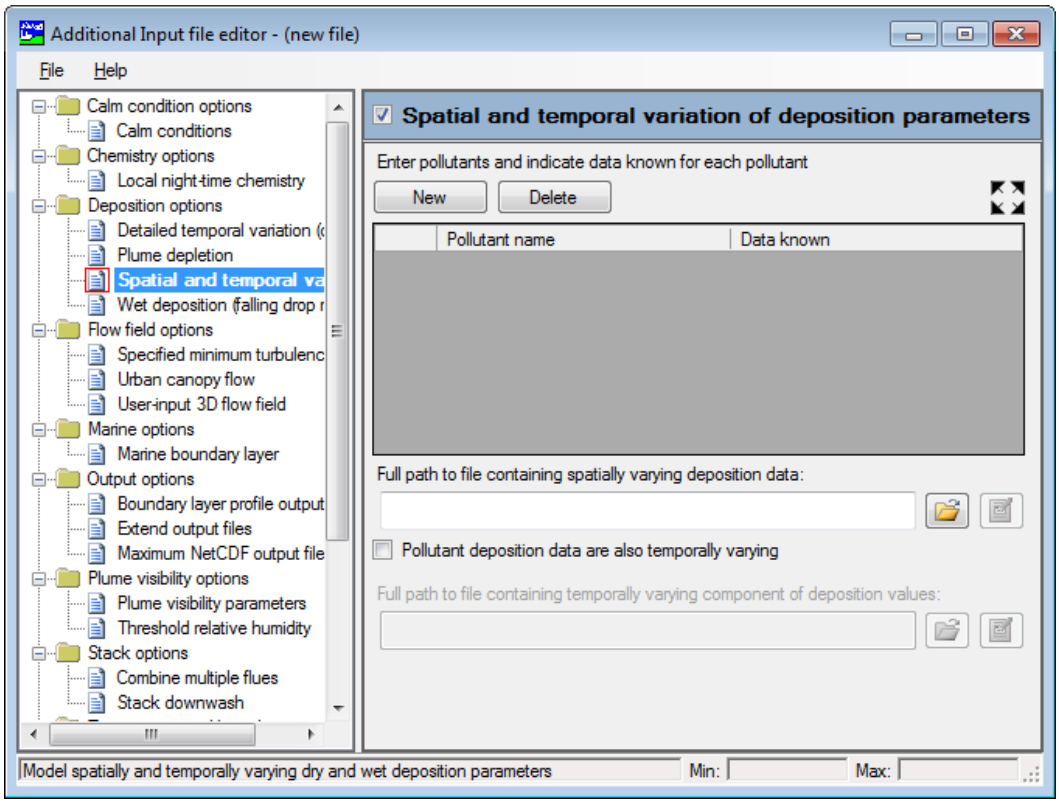


Figure 4.14 – The **Spatial and temporal variation of deposition parameters** screen of the **Additional Input file editor**.

Each of the pollutants for which spatially varying deposition are to be entered should be entered into the table with the choice of dry deposition data type as follows:

- **Deposition velocity** if **Deposition vel. known** is set to **Yes** in the **Palette of Pollutants**
- **Surface resistance** if **Deposition vel. known** is set to **No** in the **Palette of Pollutants**

The path to the *.din* file must also be entered; the format of the *.din* file is described below. The button can be used to view the file in the viewer selected from the **File\Preferences** menu. For details on specifying that the deposition data are also temporally varying see Section 4.2.2.3.

Spatially-varying file format

Spatially-varying values of the deposition velocity or surface resistance are entered using a *.din* file. Note that this file also contains the associated wet deposition parameter values. Further details of how to model the spatial variation of wet deposition parameters is given in Section 4.3.2.

The first line contains the file version number; for ADMS 5 this is `VarDepVERSION1`. The data are then entered in the following format:

```
N, X, Y, Vd, Rs, Lambda, A, B
```

where:

- `N` is an incrementing counter for each line,

- `X` is the X coordinate of the data point,
- `Y` is the Y coordinate of the data point,
- `Vd` is the deposition velocity at this data point,
- `Rs` is the surface resistance at this data point,
- `Lambda` is the washout coefficient at this data point (refer to Section 4.3 for details),
- `A` is the washout coefficient A at this data point (refer to Section 4.3 for details), and
- `B` is the washout coefficient B at this data point (refer to Section 4.3 for details).

Note that data for all of the parameters must be entered even if some of the parameters are not to be used. The data values for the unused columns can be set to 0.

Data point coordinates should be in metres, in the same coordinate system as all other input coordinate data (e.g. source locations, output grid etc).

An example *.din* file is shown in **Figure 4.15**.

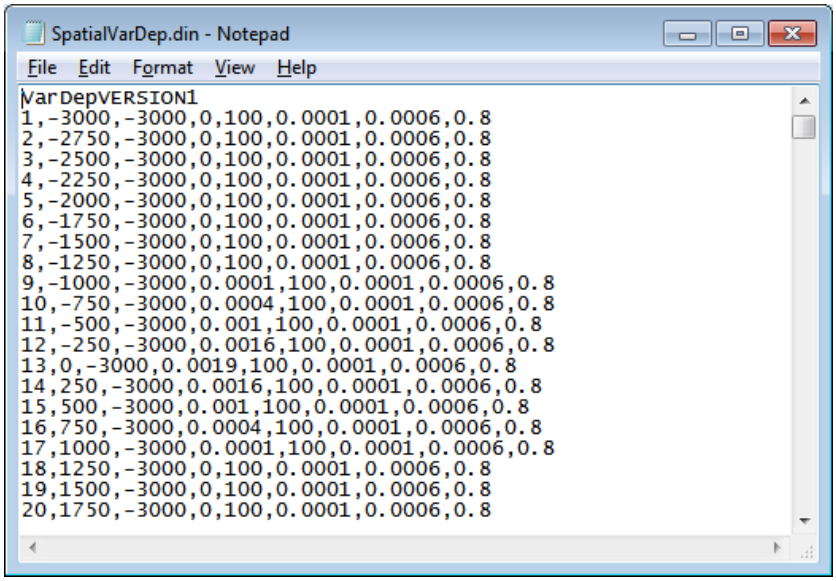


Figure 4.15 − An example of a *.din* file.

4.2.2.3 Using a combination of hourly and spatially varying dry deposition velocities

The hourly and spatially varying deposition model options can be combined. However, this option is restricted to modelling the variation of deposition velocity only and to one pollutant per model run.

In order to model a combination of hourly and spatially varying dry deposition velocities, a *.din* file should be set up representing the required spatial variation of deposition velocities, as described in Section 4.2.2.2 above. The hourly variation of the deposition velocity is introduced by a scaling factor entered into a *.dnf* file and the path of this entered into the *.aai* file. The *.aai* file should be set up as outlined in

Section 4.2.2.2 but with the **Pollutant deposition data are also temporally varying** option checked and the path of the *.dnf* file entered into the appropriate section. Details of the format of a *.dnf* file are given below.

If this option of modelling the hourly variation of deposition velocity is used then the meteorological data must contain values of the year, day and hour.

Hourly scaling factor file format

Hourly-varying scaling factors to be applied to the spatially varying values of the deposition velocity in the *.din* file are entered using a *.dnf* file. The file format is outlined below.

The first line contains the file version number; for ADMS 5 this is `VarDepFacVersion1`. Then any comments may be added. The data are then entered: firstly there is a line containing the key word "`DATA:`", then for each line of meteorological data, the data are entered in the following format:

```
Year, Day, Hour, Factor
```

where `Factor` is the scaling factor by which the deposition velocity specified in the *.din* file should be multiplied. An example *.dnf* file is shown in **Figure 4.16**.

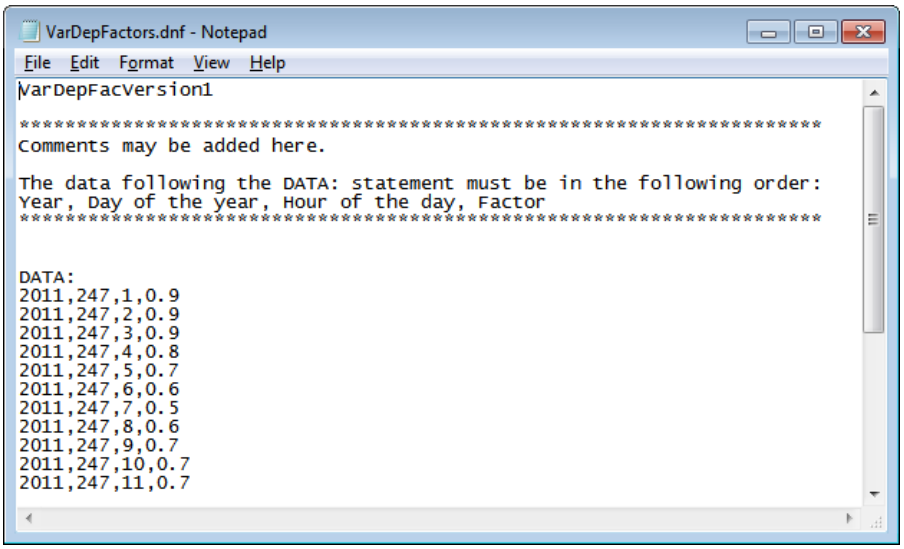


Figure 4.16 − An example of a *.dnf* file.

4.2.3 Modelling without plume depletion

It is possible to switch off the modelling of plume depletion due to dry deposition by use of an *.aai* file. When this option is enabled, the modelled concentrations are identical to those that would be predicted if the dry deposition option were not selected. The values of dry deposition calculated by the model are equal to the product of the dry deposition velocity and the ground level concentration.

To switch off the plume depletion for dry deposition, create an *.aai* file or edit the existing *.aai* file, see Section 3.1.6, enable the **Plume depletion** section (keyword SWITCHOFFDEPLETION) and check the **Switch off plume depletion for DRY deposition** option.

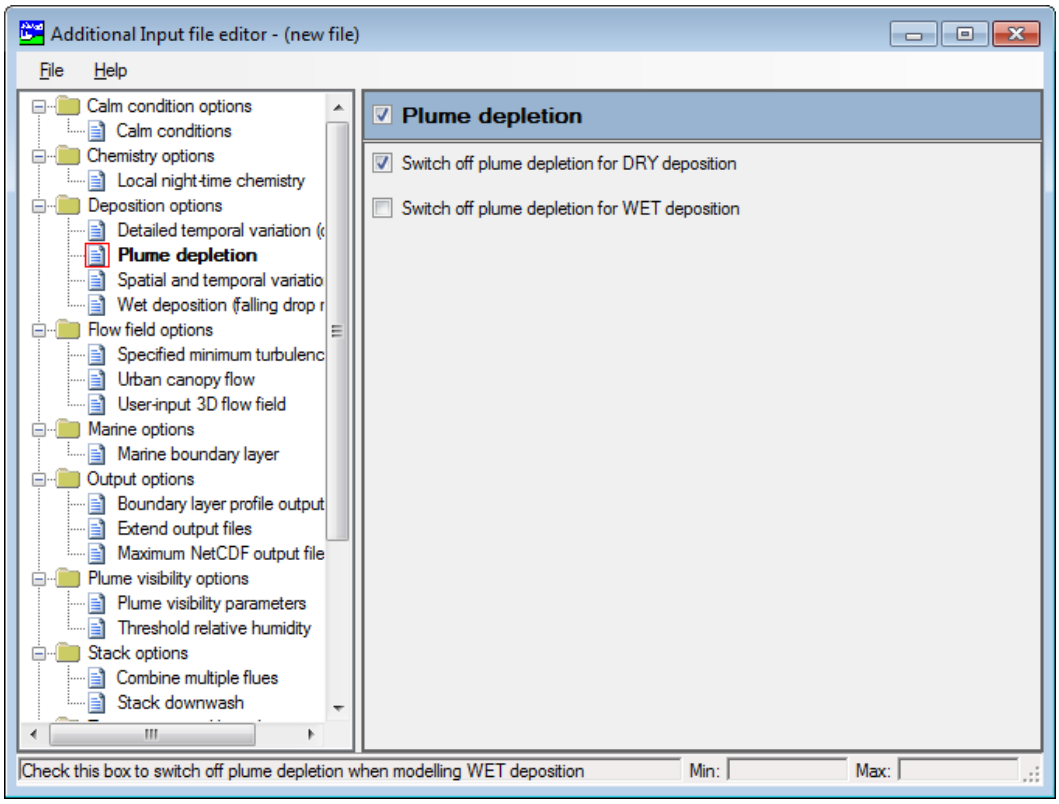


Figure 4.17 – **Additional Input file editor** screen for switching off plume depletion for dry deposition.

4.2.4 Output

Dry deposition results are output to *.gst* and *.pst* files for short-term runs and *.glt* and *.plt* files for long-term runs. The output units are mass units/m^2/s, e.g. μg/m^2/s for a plume or time-dependent puff. These are units of deposition flux or rate. For a time-integrated puff, the output units are mass units/m^2, e.g. μg/m^2, which is the total deposition (deposits) per unit area.

Dry deposition results do not include background concentrations, except for NO_X when modelling chemistry.

To view contours of the dry deposition, select **Results, Contour Plot in Surfer** from the menu at the top of the interface. Select **Short term** or **Long term** as required, click on the appropriate file name and select a **Dataset to plot** from the list. Further details on viewing output may be found in Section 6. If both wet and dry deposition have been calculated then total deposition results are also output. Total deposition is the sum of dry and wet deposition rates.

The values of dry deposition velocity and terminal velocity entered by the user and/or calculated by the model for each pollutant are output to the *.dep* file, for the first 24 lines of meteorological data.

4.2.5 Guidance on dry deposition parameters

Dry deposition velocities depend on the nature of the surface. **Table 4.3** contains suggested deposition velocities for short vegetation (such as grass) for NO, NO_2, SO_2, and NH_3.

Parameter	NO_2	NO	SO_2	NH_3
deposition velocity (m/s)	0.0015	0.00015	0.012	0.02

Table 4.3 − Suggested deposition velocities for short vegetation.

A conservative estimate for both concentrations and deposition rates of NO_x can be obtained by setting the gas property to Non-reactive and switching off plume depletion as described in Section 4.2.3.

Also note that for the particular case of modelling dry deposition of ammonia, the current guidance given by the UK Environment Agency recommends modelling without plume depletion being taken into account.

4.3 Wet deposition

Wet deposition occurs when there is precipitation (rain, sleet, snow, etc) and material is washed out of the plume. The type of wet deposition that can be modelled in ADMS 5 depends on the source and pollutant type:

- Wet deposition of all pollutants from all source types can be modelled using a washout coefficient.
- Wet deposition of SO_2 and CO_2 from all source types calculated using a washout coefficient is pH-limited.
- Wet deposition of SO_2 and HCl from point sources can be modelled using the 'falling drop' method.

The washout coefficient method is simple, and pollutants are treated independently. The pH-limited method goes some way to representing the effect of the reduced uptake of acidic pollutants when the concentration of these pollutants becomes high in the raindrops. The 'falling drop' method is more complex. The wet deposition of HCl and SO_2 are not independent, and the method models the kinetic and thermodynamic processes that occur at the surface of, and within, the raindrop; this method consequently requires significant computational time. Only point sources can be modelled with the 'falling drop' method.

The wet deposition model options are summarised in **Table 4.4**, and a brief description of each is given below.

Method	Algorithms implemented in code							
	Point sources				Non-point sources			
	SO_2	HCl	CO_2	Other	SO_2	HCl	CO_2	Other
Wet deposition	pH-limited WC	WC	pH-limited WC	WC	pH-limited WC	WC	pH-limited WC	WC
Wet deposition with falling drop method	FD	FD						

Table 4.4 − Summary of wet deposition model options in ADMS 5 for the different source types and pollutants (WC: washout coefficient, FD: falling drop).

ADMS has the option to model wet deposition without including the effect of plume depletion, selected through the *.aai* file. Further details are given in Section 4.3.4.

Washout Coefficient

Wet deposition is modelled through a washout coefficient Λ such that the mass of material captured by the precipitation and therefore removed from the plume is ΛC per unit volume per unit time, where C is the local airborne concentration. Λ is dependent on the nature of the pollutant, the rainfall rate and droplet size.

The washout coefficient Λ can be taken to have a constant value, in which case the wet deposition is independent of the rainfall rate. In fact, even if there is zero rainfall, or if rainfall rate is not a parameter in the meteorological file, a constant washout coefficient can be specified and the wet deposition can be calculated. This is equivalent to assuming a constant drizzle.

Alternatively, the washout coefficient Λ can depend on the rainfall rate P, and the model will assume a relationship between Λ and P of the form

$$\Lambda = AP^B \tag{4.2}$$

where A and B are constants entered in the interface (see **Figure 4.18**). If Λ is specified in this way then the rainfall rate in mm/hr must be a variable in the meteorological data.

It is possible to model the spatial variation of washout coefficients Λ, or A and B over the model domain in order to represent changes in precipitation rates. Please refer to Section 4.3.2 for details of how to do this.

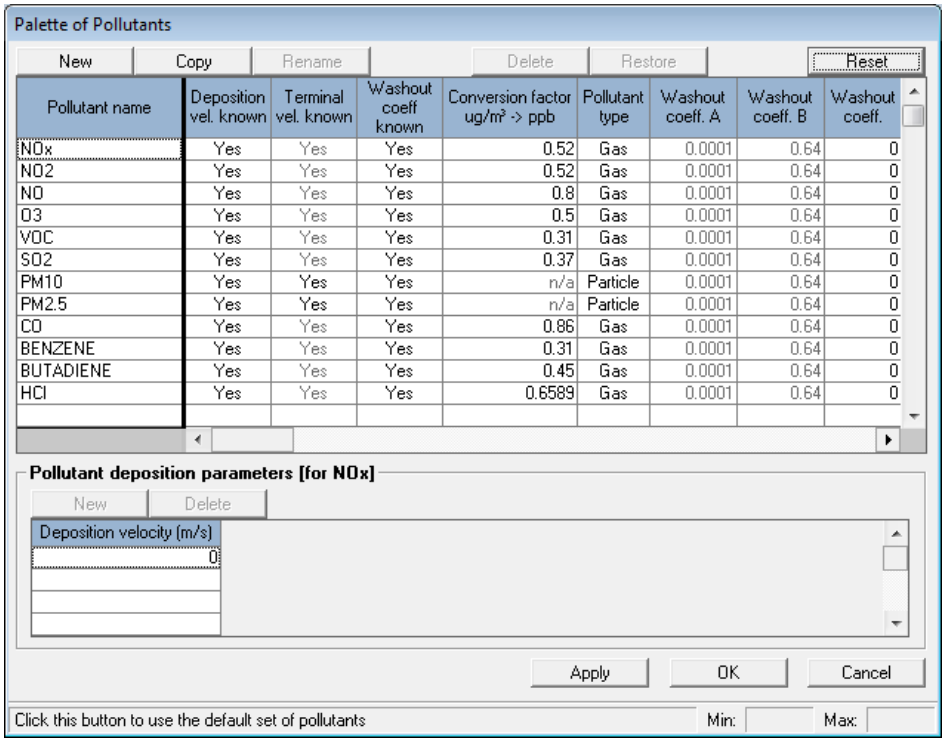


Figure 4.18 − The **Palette of Pollutants** screen showing parameters required to model wet deposition.

pH-limited washout coefficient

For acidic species, SO_2 and CO_2, a pH-limited washout coefficient is calculated, i.e. the amount of SO_2 or CO_2 dissolved in the precipitation may be limited due to the high acidity resulting from the SO_2 or CO_2 already dissolved.

'Falling drop' method

Using a washout coefficient to model wet deposition implicitly assumes that the uptake of pollutants is irreversible. However, after passing through the plume centreline, raindrops often fall through some relatively 'clean' air before reaching the ground. If the pollutant in question undergoes out-gassing (i.e. where the concentration gradient between the drop and the air causes the pollutant to pass back into gaseous form), the pollutant concentration within the drop at the ground will be reduced relative to the maximum concentration that was achieved at, or in fact often just below, the plume centreline. The gas of most interest that undergoes significant out-gassing before the drop reaches the ground is SO_2.

The 'falling drop' method incorporates both the out-gassing of pollutants, and the limiting of the uptake of acidic pollutants (such as SO_2) due to the acidity of the raindrops.

For further technical information on wet deposition see Section 9.11.

4.3.1 Modelling wet deposition using a washout coefficient

To model wet deposition effects using a washout coefficient, the appropriate pollutant deposition parameters must be specified in the **Palette of Pollutants**. The palette can be entered via the **Emissions** screen (launched from the **Source** screen) or by clicking on the **Data**... button in the **Palette** section of the **Setup** screen of the ADMS 5 interface.

The **Palette of Pollutants** screen is shown in **Figure 4.18**. Note the scroll bar at the bottom of the upper table, which allows the user to view other pollutant parameters.

It is only necessary to enter deposition parameters if deposition is being modelled, otherwise the user may retain the default values.

Default values apply for all pollutants including user-defined pollutants.

1. **Washout coeff known**

   The user should select **Yes** or **No** depending on whether or not the *washout coefficient* Λ is known. If **Yes** then wet deposition is calculated using a constant washout coefficient that is independent of the rainfall rate. If **No** then it is calculated using parameters *A* and *B* in equation (4.2).

   Default = **Yes**

2. **Washout coeff. A**

   This parameter is required if the washout coefficient is not known (see 1) and represents *A* in equation (4.2). If this parameter is used then wet deposition is taken to be a function of the rainfall rate.

   Minimum = 0
   Maximum = 100
   Default = 0.0001

3. **Washout coeff. B**

   This parameter is required if the washout coefficient is not known (see 1) and represents *B* in equation (4.2). If this parameter is used then wet deposition is taken to be a function of the rainfall rate.

   Minimum = 0
   Maximum = 100
   Default = 0.64

The default values of A *and* B *may be used unless the user has specific data, e.g. from experiments, for the particular pollutant.*

---

4. **Washout coeff. (/s)**

   If the washout coefficient is known then wet deposition is independent of the rainfall rate.

   Minimum = 0 s^{-1}
   Maximum = 1 s^{-1}
   Default = 0 s^{-1}

The washout coefficient values are not used for NO as the wet deposition of NO is always assumed to be 0.

4.3.2 Additional input options

Wet deposition parameters may vary spatially due to changes in precipitation rates across the model domain. ADMS 5 is able to model this variation, with details given below.

For both gaseous and particulate pollutants, spatially varying values of the washout coefficients Λ, or A and B can be used by the model. If these are to be entered for a particular pollutant, then the **Washout coeff known** option should be set up as appropriate in the **Palette of Pollutants.**

One set of washout coefficients Λ, A and B that vary over the model domain can be entered into a *.din* file, and the path of this entered into the *.aai* file. The values specified will be used for all pollutants listed in the relevant section in the *.aai* file. Details for setting up the *.aai* file and of the format of a *.din* file are given in Section 4.2.2.2.

4.3.3 Modelling wet deposition using the 'falling drop' method

Wet deposition of emissions of SO_2 and HCl from point sources can be modelled in ADMS 5 using the 'falling drop' method by use of an *.aai* file.

To use the 'falling drop' method for SO_2 and HCl, create a new *.aai* file or edit the existing *.aai* file, see Section 3.1.6, so the **Wet deposition (falling drop method)** option (keyword FALLINGDROP) is enabled. The **Initial pH for droplets above plume** should be entered and the **Wet deposition method for HCl and SO2** set to **Use 'falling drop'**, see **Figure 4.19** for an example with an initial pH of 5.6.

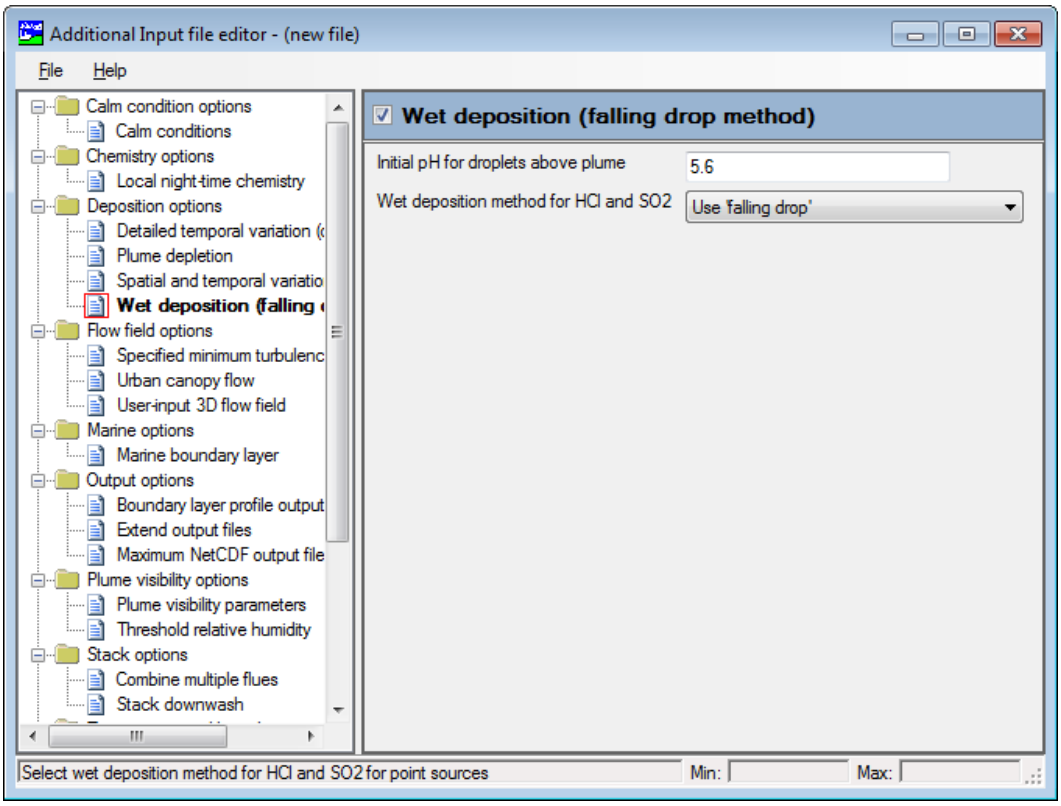


Figure 4.19 – Additional Input file editor screen for enabling the 'falling drop' wet deposition method.

Note that:

- The values of the washout coefficients specified in the pollutant palette are ignored for pollutants that are modelled using the 'falling drop' method.
- The 'falling drop' method is computationally intensive compared to the washout coefficient method.
- If calm conditions are being modelled and the falling drop method is used to calculate wet deposition, then the wet deposition flux is calculated as the weighted average of wet deposition flux from the radial and Gaussian plume components separately.

4.3.4 Modelling without plume depletion

It is possible to switch off the modelling of plume depletion due to wet deposition by use of an *.aai* file. When this option is enabled, the modelled concentrations are identical to those that would be predicted if the wet deposition option were not selected.

To switch off the plume depletion for wet deposition, create a new *.aai* file or edit the existing *.aai* file, see Section 3.1.6, to enable the **Plume depletion** section (keyword SWITCHOFFDEPLETION) and check the **Switch off plume depletion for WET deposition** option.

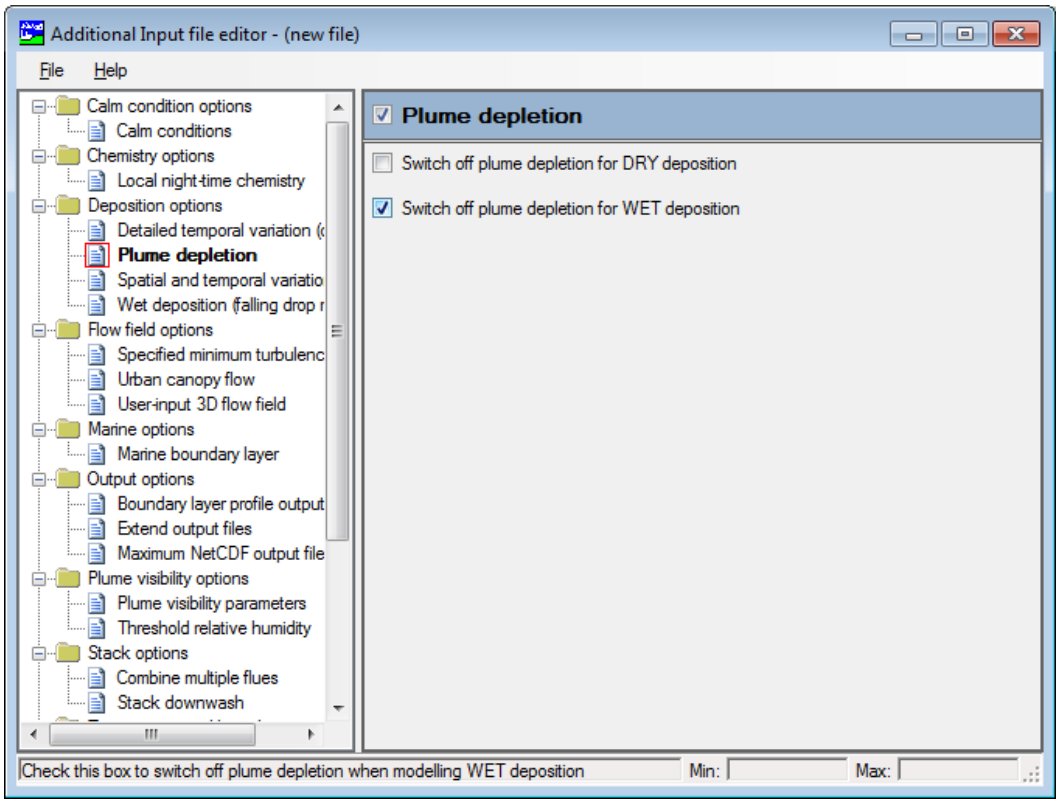


Figure 4.20 – **Additional Input file editor** screen for switching off plume depletion for wet deposition.

4.3.5 Output

Wet deposition rates are output to *.gst* and *.pst* files for short-term runs and *.glt* and *.plt* files for long-term runs. The output units are mass units/m^2/s as this is a deposition flux or rate. To view contours of the wet deposition, select **Results, Contour Plot in Surfer** from the menu at the top of the interface. Select **Short term** or **Long term** as required, click on the appropriate file name and select a **Dataset to plot**. Further details about viewing output may be found in Section 6. Note that dry deposition (if selected) and total deposition results are also available in the same way. Total deposition is the sum of dry and wet deposition rates.

The wet deposition washout coefficients entered by the user and/or calculated by the model for each pollutant are output to the *.dep* file for the first 24 lines of meteorological data.

4.3.6 Guidance on wet deposition parameters

Table 4.5 gives some suggested values for wet deposition parameters for NO_2, NO, NH_3 and HCl. Note that NO is very insoluble and the wet deposition rate will be 0 even if different values for the wet deposition parameters are entered. The deposition rate of NO_2 is also small.

Parameter	NO_2	NO	NH_3	HCl
A	0.0001	0	0.005	0.0003
B	0.64	n/a	0.64	0.66

Table 4.5 − Deposition parameters for NO_2, NO and NH_3.

For further information on wet deposition parameters, users are advised to consult the reports of the Review Group on Acid Rain (RGAR). The following advice on wet deposition parameters for SO_2 and sulphate (SO_4^{2-}) is based on RGAR reports (CLAG, 1994).

For SO_2, if P < 103 mm/year, $\Lambda = 1.0 \times 10^{-6}(P/500)$

if P > 103 mm/year, $\Lambda = 1.0 \times 10^{-6}\left[2 + (0.336/0.84)(P - 1000)/500\right]$

For SO_4^{2-}, if P < 103mm/year, $\Lambda = 1.3 \times 10^{-5}(P/500)$

if P > 103mm/year, $\Lambda = 1.3 \times 10^{-5}\left[2 + (0.336/0.84)(P - 1000)/500\right]$

where P is the rainfall rate in mm/year, and Λ is the constant value of the washout coefficient, independent of hourly rainfall rates.

4.4 Radioactive emissions

The main pathways by which the public may be exposed to radiation as a result of atmospheric discharges are external exposure, inhalation, contact with contaminated materials or consumption of contaminated foods. Deposition of radioactive substances onto soil and vegetation can result in contamination of food products. Regulatory limits on radioactive emissions are specified in terms of the radioactive dose received by members of the public, or as maximum permitted levels in particular types of food. Radioactive dose is calculated from air concentrations and deposition to the ground.

Air concentrations and deposition rates may be calculated for emissions of radioactive isotopes using ADMS 5. Isotopes may be modelled with or without the effects of radioactive decay (Sections 4.4.1 and 4.4.2 respectively). In addition, the gamma dose may be calculated, as described in Section 4.4.3. There is also a utility to view the list of isotopes that can be modelled, their half-lives and decay products.

4.4.1 Modelling isotopes without radioactive decay

In ADMS 5 the user can model emissions of isotopes without including the effects of radioactive decay. This may be useful for comparison purposes, or to enable the modelling of isotopes using advanced options unavailable with radioactive decay.

1. On the **Setup** screen select **Isotopes** from the **Palette** type options.
2. Make sure the **Radioactive decay** model option is NOT selected.
3. Go to the **Palette of Radioactive Pollutants (Isotopes)** screen by clicking on the **Data**... button. Isotopes may now be added to the palette using the **New** button and either selected from the list or entered manually.
4. Specify emission rates for the isotopes in Bq/s in the **Emissions** screen.
5. Add isotopes for which output should be calculated to the **Output** screen and run the model. This will run the model to give the same output parameters as when non-radioactive pollutants are emitted but the units will be in terms of Bq instead of g.

4.4.2 Modelling isotopes with radioactive decay

To model releases of radioactive isotopes, including the effects of radioactive decay, select **Radioactive decay** in the **Setup** screen. The **Radioactive decay** option is only available when **Isotopes** is selected for the **Palette** type. Model runs may be set up in a similar way to that described for the basic modelling studies in Section 3, but with the following differences.

1. The **Palette of Pollutants** is replaced by the **Palette of Radioactive Pollutants (Isotopes)** (**Figure 4.21**). This is accessed by clicking on the **Data**... button in the **Palette** section of the **Setup** screen, or by clicking on the **Emissions**... button and then **Pollutants**... on the **Source** screen. The default **Palette of Radioactive Pollutants (Isotopes)** screen contains a single isotope (Sr-90) in the pollutants table.

Both gaseous and particulate radioactive pollutants may be defined.

Up to 30 isotopes in total may be added to the palette by clicking **New** at the top of the table (**Figure 4.21**). The required isotope(s) may be selected from the drop-down list, which contains isotopes typically emitted in the UK. If the isotope you want to model is not in the drop down list but is in the list of isotopes which can be emitted, edit the name of the isotope by hand. For further details about the **Palette** screen see Section 3.1.4.

All isotopes that can be emitted are contained in the file <install_path>\Isotopes.pdf. *Isotopes are listed in order of atomic mass and the file contains the half-life in seconds and the daughter products for each isotope. For each daughter, the figure in brackets corresponds to the fraction of the original isotope that decays to the given daughter. To view the full list of isotopes, their half-lives and daughter isotopes, click on* ***Isotopes*** *from the* ***Palette of Radioactive Pollutants (Isotopes)*** *screen. The file* Isotopes.pdf *is opened in a PDF viewer. This file cannot be modified by the user.*

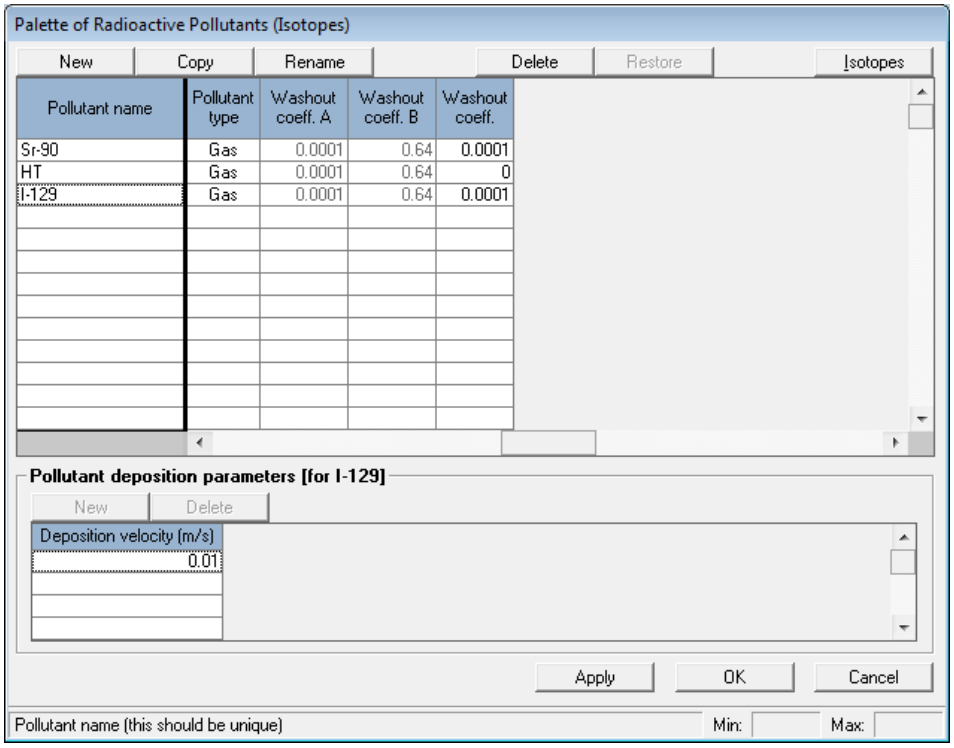


Figure 4.21 − The **Palette of Radioactive Pollutants (Isotopes)** screen.

The appropriate deposition characteristics should also be entered in the palette for each isotope if deposition is to be modelled. The deposition parameters present when the isotopes are first added are default values; the appropriate value should be entered if known. Please refer to Sections 4.2 and 4.3 for details of the deposition parameters.

For radioactive releases there are many possibilities for the choice of entries in the **Palette of Radioactive Pollutants**. If the user is likely to want to model the same set of isotopes (or a sub-set of these) frequently, then they may wish to set up a base *.apl* file (e.g. *factory1.apl*) with a suitable short list of isotopes in

the **Palette of Radioactive Pollutants** (along with their deposition characteristics). This file can then be used as a template file for other modelling runs.

2. The **Emissions** screen is replaced by the **Radioactive Emissions (Isotopes)** screen (**Figure 4.22**). Emissions of isotopes are entered in the same way as for non-radioactive emissions.

   Up to ten isotopes may be emitted from each source. These are added to the table or removed from it by clicking **New** or **Delete**, respectively.

   To select another isotope from the palette, click on the cell containing the pollutant name and on the down arrow that appears to the right of the cell. The isotopes in the list appear in the order of definition in the **Palette of Radioactive Pollutants (Isotopes)**; move around the list by typing a letter, which highlights a member of the list with that initial letter.

*Isotopes must be entered in the **Palette of Radioactive Pollutants (Isotopes)** screen before they can be selected in the **Radioactive Emissions (Isotopes)** screen.*

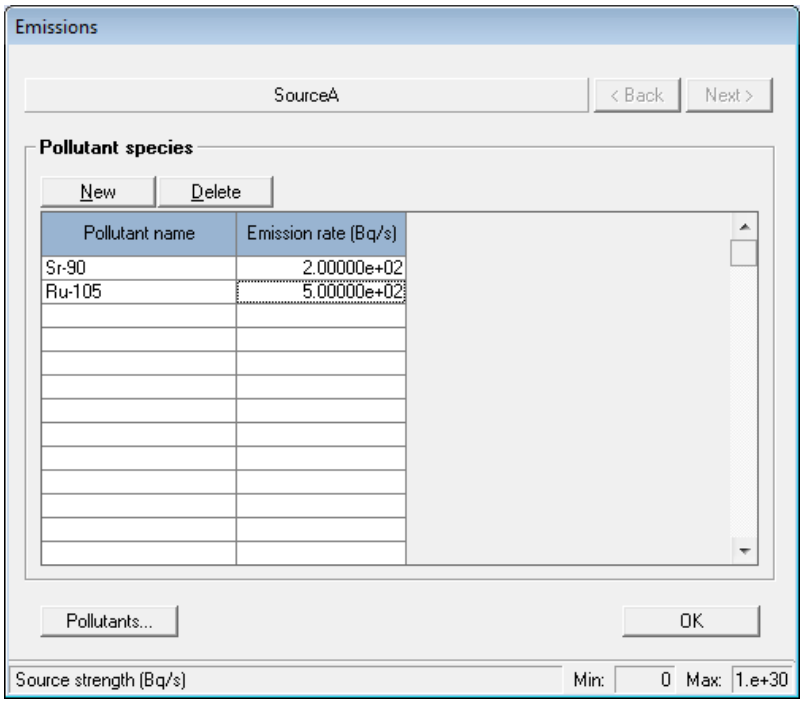


Figure 4.22 − The **Radioactive Emissions (Isotopes)** screen.

3. Output options should be chosen in the **Output** screen (**Figure 4.23**), as described for non-radioactive emissions in Section 3.6. If isotopes are being emitted, the drop-down list of pollutant names in the **Pollutant output** table will only contain isotopes from the **Palette of Radioactive Pollutants (Isotopes)**; the output units are Bq/m^3.

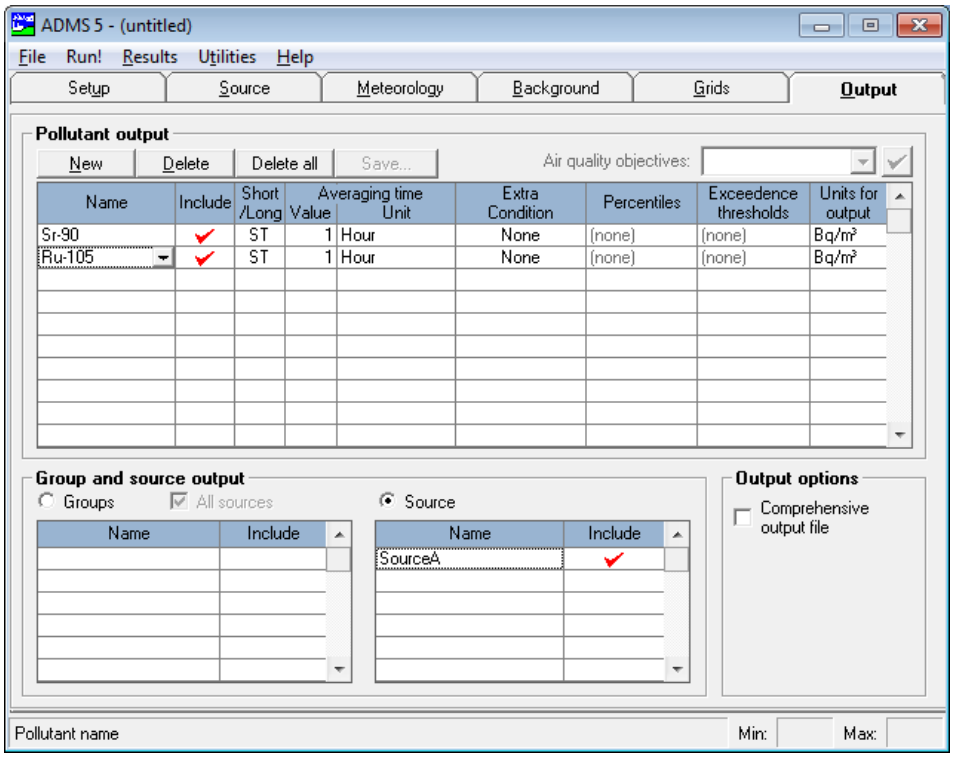


Figure 4.23 − The **Output** screen when modelling radioactive isotopes.

4.4.3 Gamma dose

The gamma dose option allows the user to calculate the dose of gamma radiation at a point within a cloud of a gamma-emitting isotope (e.g. Ar-41). In addition, the gamma dose due to deposited material is also calculated, when dry deposition is modelled.

Gamma dose may be calculated at all output points provided they are at ground level. However, it is important to be aware that gamma dose calculations take significantly longer than, say, radioactive decay calculations. Therefore is it advisable, at least in the early stages of a study, to restrict the number of gamma dose output points considered.

To calculate gamma dose, make sure the **Radioactive decay** option has been selected in the **Setup** screen and check the **Gamma dose** box. If this option is greyed out (and therefore not available), go to the **Output** screen and select the **Source** option, since gamma dose can only be calculated for one source at a time.

Emissions data should be input as described in Section 4.4.2 (2) above. Output should also be entered as in Section 4.4.2. Gamma dose will be calculated at all output points in units of Sieverts per second (Sv/s).

*To calculate gamma dose, **Single source** output must be selected and the **Radioactive decay** option selected – if groups have been selected for output, it is not possible to select **Gamma dose**. Gamma dose output points are restricted to ground level, i.e. z = 0.*

4.5 Plume visibility

It is possible in ADMS 5 to calculate whether a plume is visible due to the presence of condensed water. This is relevant for emissions with high moisture content. Using this option also calculates the effects of the presence of moisture on plume rise.

To model plume visibility, go to the **Setup** screen and click on **Plume visibility**. The following additional parameters must then be input:

1. **Mass H2O (kg/kg)**: initial mixing ratio of the plume

   In the **Source** screen, there is an additional column in the source table for **Mass H2O (kg/kg)**, shown in **Figure 4.24**. The initial mixing ratio of the plume should be specified here, that is, the mass of water per unit mass of dry release.

   Minimum = 0 kg/kg
   Maximum = 9,999,999 kg/kg
   Default = 0 kg/kg

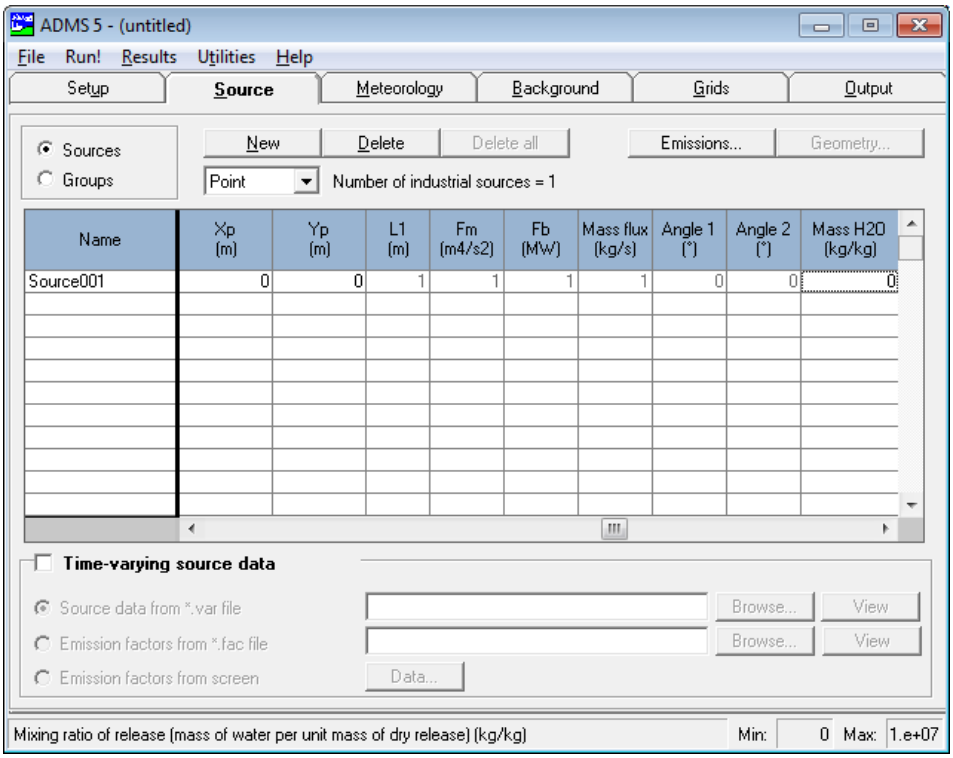


Figure 4.24 − The **Source** screen showing the extra column of data required to model plume visibility.

In addition, meteorological data for humidity and temperature must be specified for plume visibility to be modelled. **Figure 4.25** shows an example of the **Meteorology** screen for entering meteorological data by hand. It shows the two additional columns in which data must be entered when plume visibility is to be modelled.

2. **Surface temperature T0C (°C)**: the near surface temperature of the air, in degrees Celsius

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

3. **Relative humidity (%)**: relative humidity of the atmosphere

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

   If meteorological data from a file are used, then the meteorological file must include relative humidity (RHUM (%)) or specific humidity (S HUMIDITY (kg/kg)), together with surface temperature (TOC (°C)).

TOC is normally provided as a standard parameter in meteorological files supplied by the UK Met Office but users may need to request the inclusion of relative or specific humidity when ordering data.

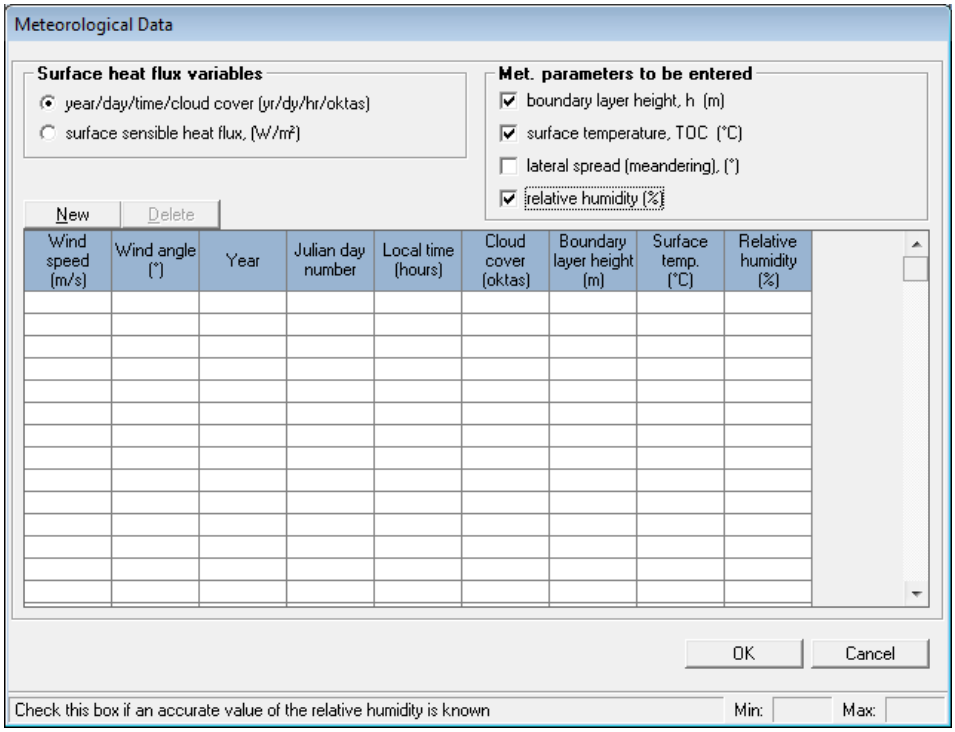


Figure 4.25 − The **Enter on screen** option in the **Meteorology** screen when modelling plume visibility.

Note that plume visibility is only considered within the extent of the output region covered by the output grid and specified points.

4.5.1 Threshold for ambient cloud

When calculating the plume visibility, if the ambient humidity at the plume centreline is above 98% then the plume is deemed not to be visible as it is considered indistinguishable from cloud, see Section 9.14 for further details.

To specify a different threshold relative humidity, create a new *.aai* file or edit the existing *.aai* file, see Section 3.1.6, to enable the **Threshold relative humidity** section (keyword RELATIVEHUMIDITYFORCLOUDS), **Figure 4.26**. Then enter the **Threshold for ambient cloud**.

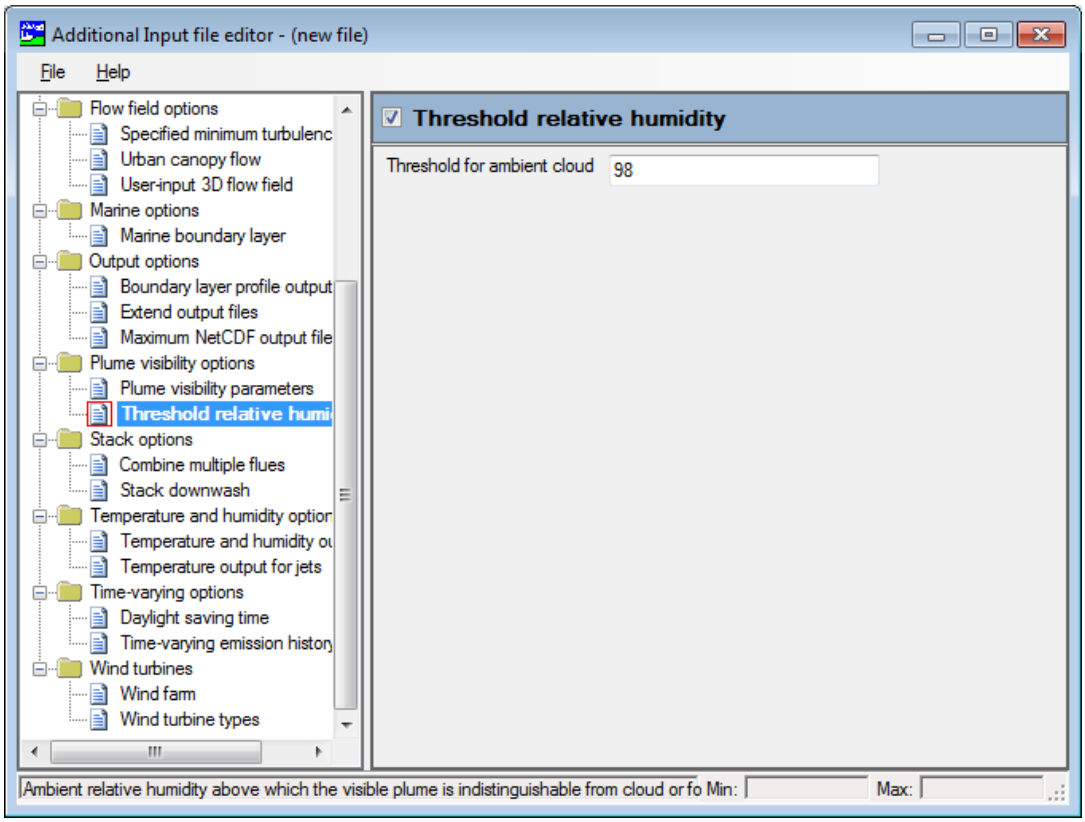


Figure 4.26 – Additional Input file editor screen for setting the threshold relative humidity

4.5.2 Plume visibility parameters

The method for determining whether a plume is visible is outlined in Section 9.14. As part of those calculations three parameters are used:

- liquid water threshold,
- number of droplets in the plume,
- opacity factor.

The values of these parameters can be altered using an *.aai* file. To alter these parameters, create a new *.aai* file or edit the existing *.aai* file, see Section 3.1.6, to enable the **Plume visibility parameters** section (keyword PLUMEVISPARAMS), shown in **Figure 4.27**. The **Opacity factor**, **Number of liquid droplets** and **Critical liquid water threshold** should be entered.

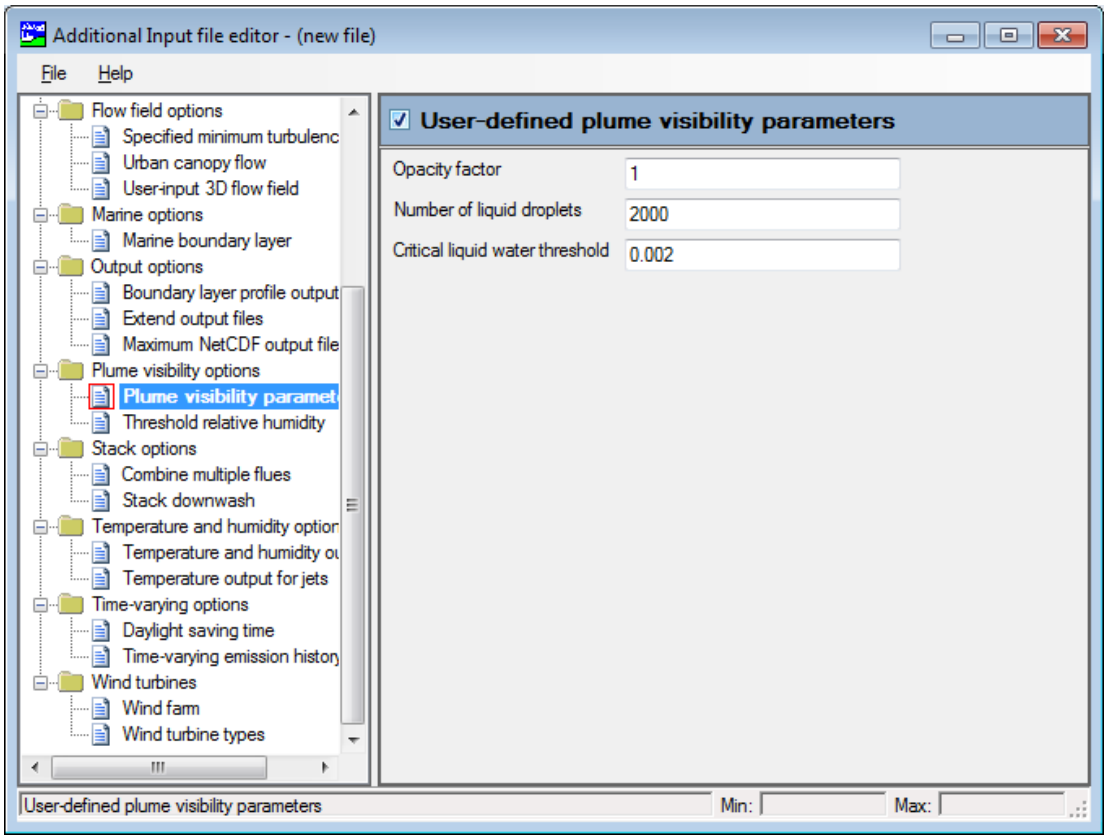


Figure 4.27 – **Additional Input file editor** screen for altering the plume visibility calculation parameters.

4.5.3 Restrictions

The following restrictions apply when using the **Plume visibility** option.

- Calculations are restricted to point sources.
- Calculations are restricted to sources with plume rise i.e. non-zero release velocity.
- If buildings are modelled in the run, they do not affect model predictions of plume visibility or plume grounding. However, additionally modelling plume visibility when modelling buildings may affect predicted model concentrations, and temperature and humidity output; this is because the water content of the plume is accounted for in the plume rise calculations when plume visibility is modelled.
- Meteorological data must include surface temperature and either relative or specific humidity.
- In addition to the threshold for ambient cloud, which is applied at the plume centreline, plumes are not reported as visible if the ambient relative humidity is 100% at ground level as conditions are considered to be foggy.

4.6 Temperature and humidity output

ADMS 5 is able to calculate the plume-affected temperature, specific humidity and/or relative humidity at each output point. The number of exceedences of user-input thresholds of these parameters can also be output, as well as simultaneous exceedences of pairs of threshold values of temperature and specific humidity and/or temperature and relative humidity.

This option may be of particular interest to users interested in the dispersion of live micro-organisms such as Legionella. These micro-organisms are sensitive to the conditions within the plume; for example, they may only survive if the temperature and/or relative humidity are above or below certain threshold values.

Temperature and humidity output is restricted to runs modelling a single point source, refer to Section 4.6.1 for details of setting up an *.aai* file to enable this option. However, short-term temperature output can also be obtained for a single downstream pointing jet source, refer to Section 4.6.2 for details of setting up this option. Details of the output produced by this option are given in Section 4.6.3 and restrictions on the use of this option are given in Section 4.6.4. Technical details about this option are given in Section 9.14**.**

4.6.1 Temperature and humidity output for point sources

To enable the temperature and humidity output for a single point source, an *.aai* file should be created or the existing *.aai* file edited, as described in Section 3.1.6, and the **Temperature and humidity output** option (keyword TEMPHUM) enabled (**Figure 4.28**).

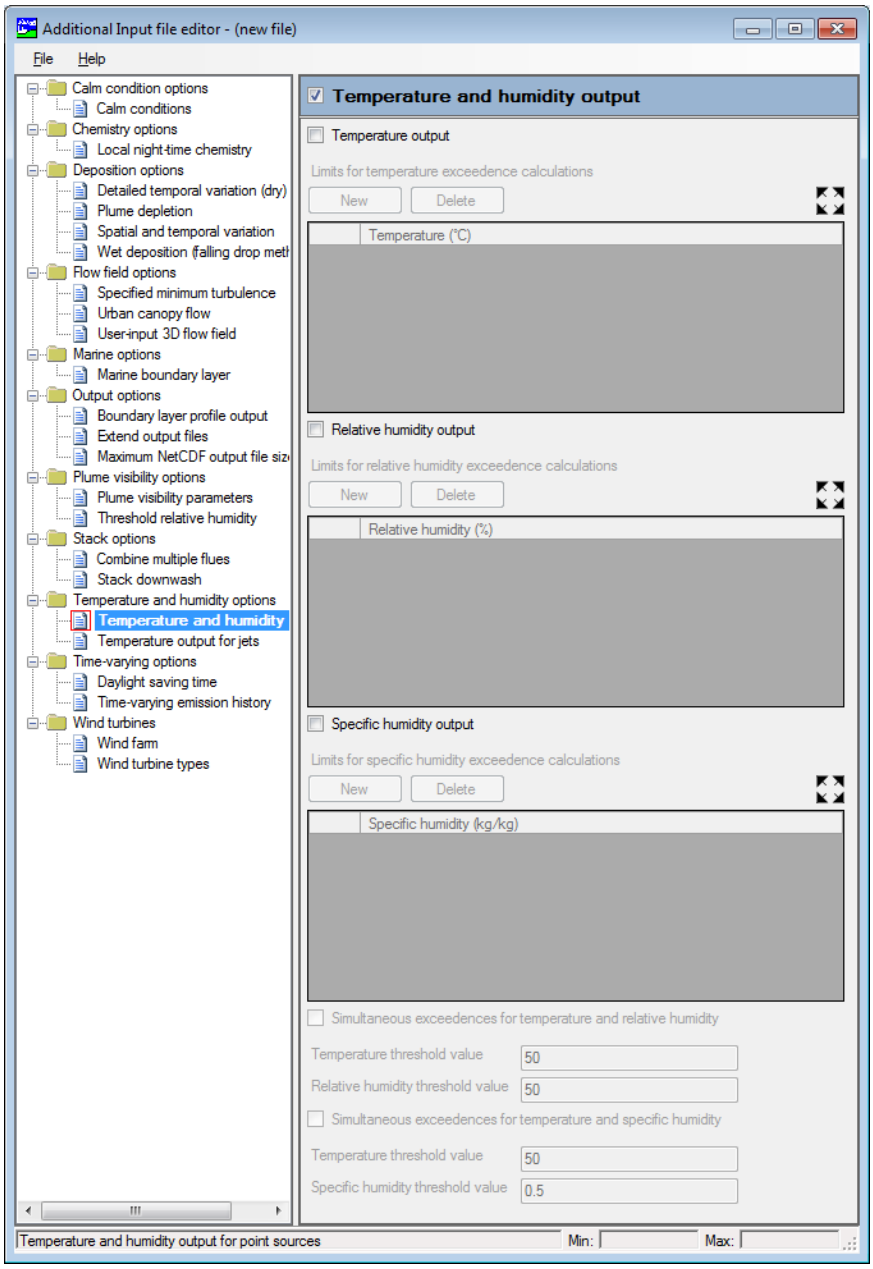


Figure 4.28 − The **Temperature and humidity output** option in the **Additional Input file editor**.

Choices of output parameters are then made as follows:

- **Temperature output**: If temperature output is required, a number of temperature thresholds to use in the calculation of exceedences may be specified in the table.
- **Relative humidity output**: If relative humidity output is required, a number of relative humidity values to use in the calculation of exceedences may be specified in the table.
- **Specific humidity output**: If specific humidity output is required, a number of specific humidity thresholds to use in the calculation of exceedences may be specified in the table.
- **Simultaneous exceedences for temperature and relative humidity**: If both temperature

and relative humidity output are selected then output of simultaneous exceedences may be requested and a pair of thresholds specified.

- **Simultaneous exceedences for temperature and specific humidity**: If both temperature and specific humidity output are selected then output of simultaneous exceedences may be requested and a pair of thresholds specified.

4.6.2 Temperature output for downstream jet sources

To enable temperature output for a single downstream pointing jet source for short-term calculations, an *.aai* file should be created or the existing *.aai* file edited, see Section 3.1.6 and the **Temperature output for jets** option (keyword JETTEMP) enabled (**Figure 4.29**). No further information is required for this section

If the jet is pointing upstream, i.e. the wind aligned component of the release velocity is negative, no temperature output will be given for that meteorological data line.

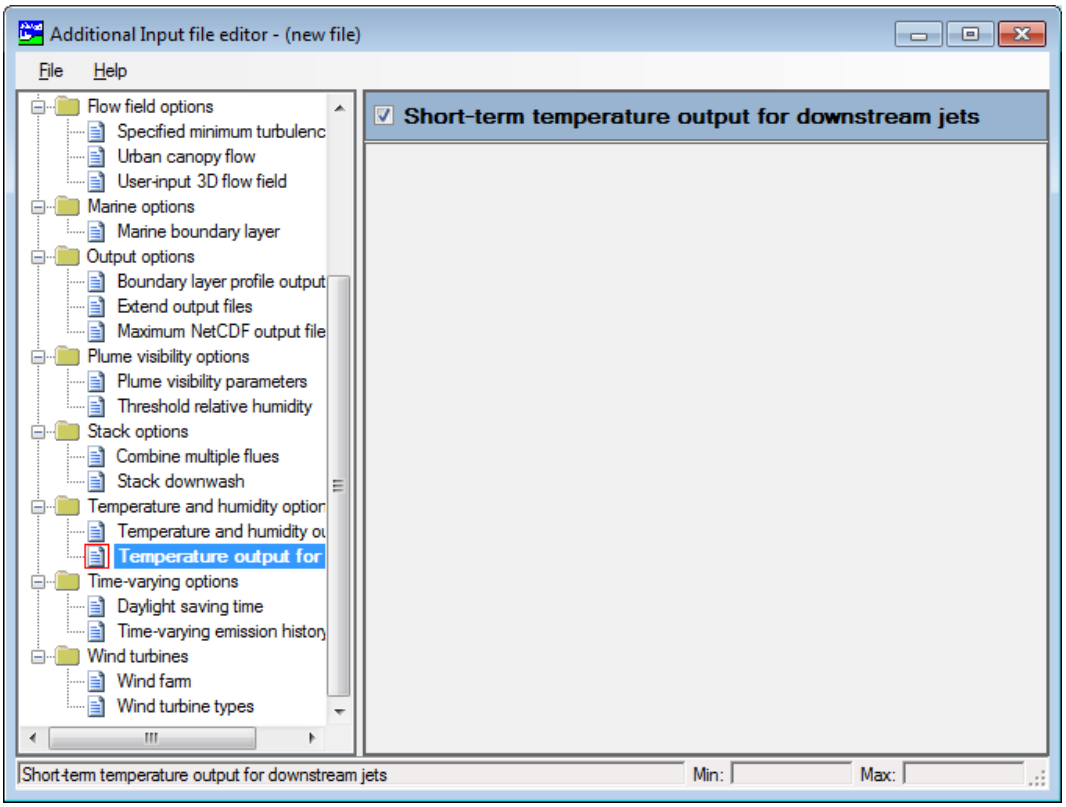


Figure 4.29 − The **Temperature output for jets** option in the **Additional Input file editor**.

4.6.3 Output

When the temperature and humidity output option is used, a number of new temperature and humidity outputs are given depending on the selections made. Details of these outputs are given below. Temperature output is given in units of °C, relative humidity in % and specific humidity in kg/kg.

Short-term

The temperature, relative humidity and/or specific humidity at each output point are listed in the *.gst* and/or *.pst* files. These new columns of data are labelled 'Temp'. 'RHum' and 'SHum' respectively. Contour plots of gridded output of these parameters can be created in the same way as concentration contours, using the **2-D Output Plotter** (refer to Section 6.3).

The temperature, relative humidity and/or specific humidity on the plume centreline, and under the plume centreline at the ground, are given in the *.cen* file. The six new columns are labelled 'Tmp0', 'TmpZP' (temperature at the ground and at the centreline), 'RH0', 'RHZP' (relative humidity at the ground and at the centreline) and 'SH0', 'SHZP' (specific humidity at the ground and at the centreline). Note that the *.cen* file is only created for single point source runs with gridded output.

Long-term

The long-term average temperature, relative humidity and/or specific humidity at each output point are listed in the *.glt* and/or *.plt* files. These new columns of data are labelled 'LTTmp', 'LTRH' and 'LTSH'. Contour plots of gridded output of these parameters can be created in the same way as concentration contours, using the **2-D Output Plotter** (refer to Section 6.3).

The numbers of exceedences of the threshold values of temperature, relative humidity and/or specific humidity are listed in the *.glt* and/or *.plt* files. Again, contour plots of the data can be created. The columns of data are labelled as follows:

- EXT: Number of exceedences of threshold value of temperature.
- EXRH: Number of exceedences of threshold value of relative humidity.
- EXSH: Number of exceedences of threshold value of specific humidity.
- EXTRH: Number of exceedences of the simultaneous temperature and relative humidity thresholds.
- EXTSH: Number of exceedences of the simultaneous temperature and specific humidity thresholds are.

Note that the exceedence output is adjusted to a per-annum value (as for the standard concentration exceedence output); the output is always the expected number of exceedences per year, irrespective of the length of the period covered by the meteorological data.

4.6.4 Restrictions

- Only one pollutant may be selected for output, and if the pollutant modelled is a particulate, then it may only have one particle size.
- The source must have non-zero efflux. i.e. there must be some plume rise.
- To output specific humidity and/or relative humidity, the **Plume visibility** option must also be used.
- This option may not be used in conjunction with the **Radioactive decay**, **Gamma dose** or **Coastline** options.

4.7 Odours

Odours are typically measured in 'odour units'. The **Odours** option in ADMS 5 uses the odour unit (ou_E) defined in the CEN standard (EN 13725:2003). One ou_E is the mass of pollutant that, when evaporated into 1 m^3 of odourless gas at standard conditions, is at the detection limit.

4.7.1 Input data

When the **Odours** option is selected on the **Setup** screen the emission rate and output units are changed to be based on ou_E, refer to Sections 3.2.4 and 3.6.1 for more details.

4.7.2 Restrictions

This option may not be used in combination with either the **Radioactivity** (and hence **Gamma dose**), **Chemistry** or **Puff** options.

4.8 Chemistry

ADMS 5 has the ability to calculate the chemical reactions in the atmosphere between nitric oxide (NO), nitrogen dioxide (NO_2) and ozone (O_3) and the resultant concentrations of each pollutant in the plume. In order to include the effects of chemical reactions, the user should go to the **Setup** screen and select **Chemistry** from the model options. The following sections describe the model options which also need to be taken into consideration if chemistry is selected.

4.8.1 Background concentrations

If chemistry is being modelled then background concentrations of NO_X, NO_2 and O_3 must be entered. Details of how background concentrations are entered are given in Section 3.4. During the day, the background values are adjusted to ensure that they are in equilibrium. Further details are given in Section 9.15. During the night no adjustments are made to the background concentrations, so if the background concentrations are not in equilibrium it may be appropriate to use the **Local night-time chemistry option**, described in Section 4.8.6

4.8.2 Pollutant parameters

If chemistry is modelled with dry deposition then the user must input deposition velocities for NO_2 **and** NO, or set these parameters to unknown to allow the model to make its own estimate (see Section 4.2.1).

If user-entered values of the $\mu g/m^3$ to ppb conversion factors are entered for NO and NO_2 then these should be consistent, i.e. calculated using the same ambient conditions.

4.8.3 Meteorology

The reaction rates used in the chemistry calculations depend on the temperature and the solar radiation. Hence if the **Chemistry** option is selected, the meteorological data must include temperature, and either solar radiation or year, day, hour and cloud cover (from which the model is able to calculate solar radiation).

4.8.4 Averaging times

If the **Chemistry** option is selected then the **Averaging times** entered on the **Output** screen for all of the pollutants involved in the chemistry calculations (NO_X, NO_2, NO, O_3) must either all be the same, or all be greater than or equal to one hour.

4.8.5 Emissions data

ADMS 5 requires all NO_X emissions to be 'NO_X as NO_2'. Please refer to Appendix B for more details.

In order to implement the chemistry option, the model requires sufficient information to determine how much of the total NO_X emitted is in the form of NO_2. The remainder

is assumed to be NO. For sources emitting NO_X, the total NO_X emissions must always be specified in the **Emissions** screen. The emissions of NO_2 may then be specified, either by explicitly entering an NO_2 emission rate in the Emissions table in g/s, or by specifying the percentage of the NO_X emissions that are in the form of NO_2. This second option is entered in the **% of NOx emission that is NO2** box (**Figure 4.30**), subject to these restrictions:

Minimum = 0%
Maximum=100%
Default = 5%

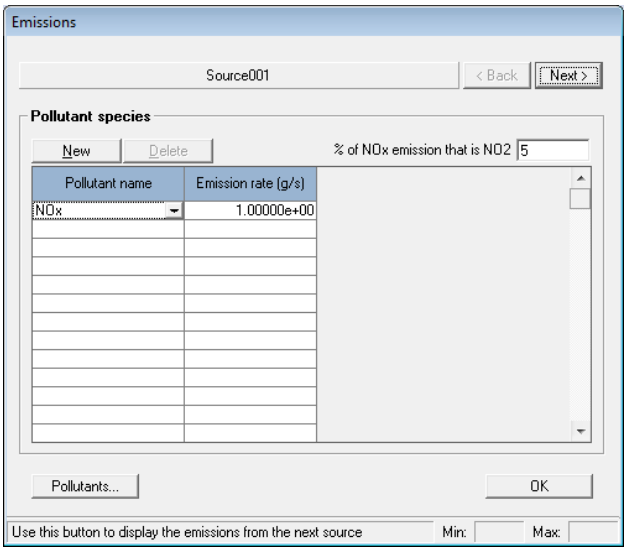


Figure 4.30 − Specifying the percentage of NO_X emission that is NO_2 in the **Emissions** screen.

*If NO_2 is the only pollutant species specified the user will see the message shown in **Figure 4.31** when attempting to leave the **Emissions** screen.*

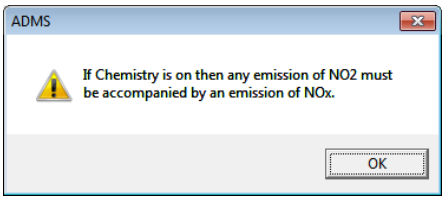


Figure 4.31 − Error message if NO_X is not specified.

*The emissions of total NO_X must be greater than or equal to the NO_2 emissions. If not then the message in **Figure 4.32** will be shown when attempting to save the* .apl *file.*

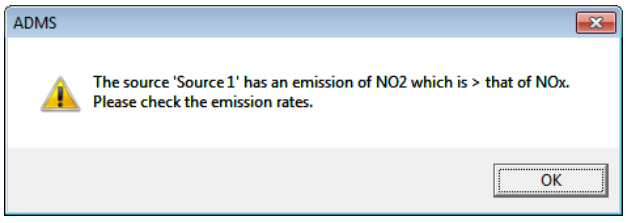


Figure 4.32 − Error message if NO_2 > NO_X.

Using a .var file with chemistry

If a *.var* file is being used to specify the emission rates for some of the sources, then for any source for which the emission rate of NO_x is given in the *.var* file, the emission rate of NO_2 must also be given in the *.var* file.

4.8.6 Local night-time chemistry

During the day the background concentrations of species involved in chemistry are adjusted to ensure they are in equilibrium. This is not possible during the night as the chemistry is more complicated. A local night-time chemistry scheme is available, which allows for the effects of chemistry on the emitted NO_X and NO_2 to be taken into account without spuriously affecting the background values. Technical details of this scheme are given in Section 9.15.

To use the local night-time chemistry scheme, the **Chemistry** option should be enabled in the **Setup** screen of the interface, and an *.aai* file should be created or the existing *.aai* file edited, see Section 3.1.6 so the **Local night-time chemistry** section (keyword NIGHTCHEM) is enabled, **Figure 4.33**. No further parameters are required for this section.

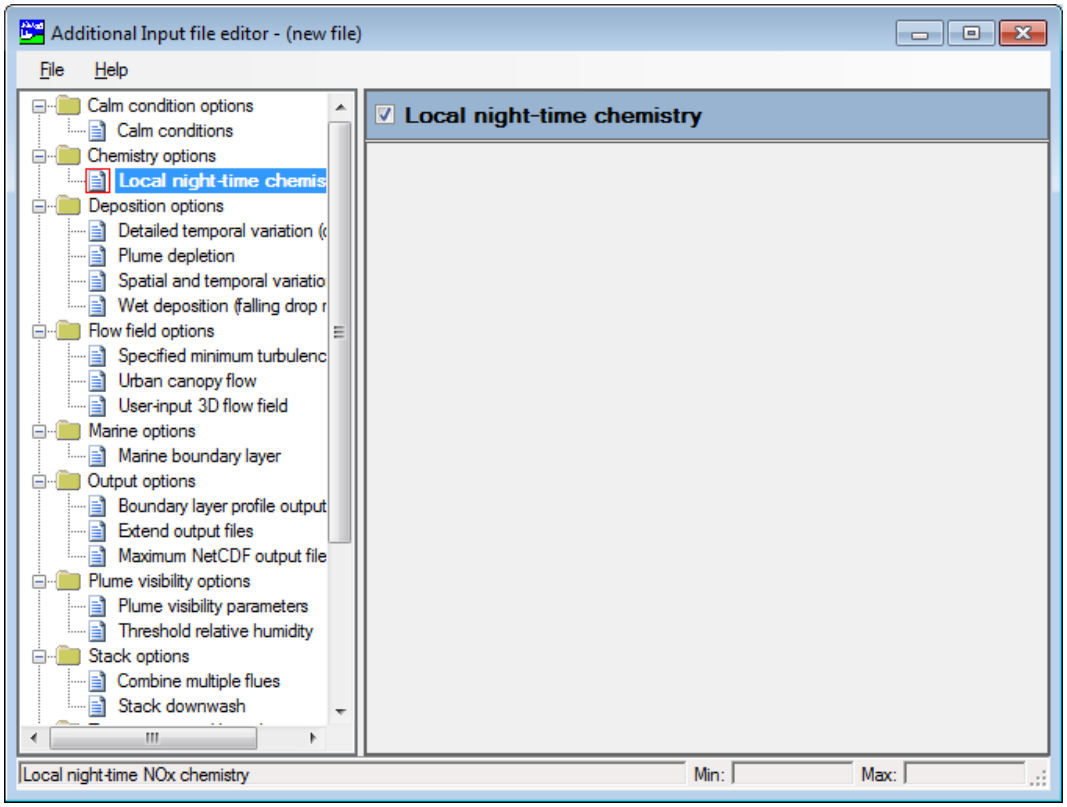


Figure 4.33 – **Local night-time chemistry** scheme in the **Additional Input file editor**.

4.9 Buildings

4.9.1 Building effects

Chimney stacks often protrude from the roof of a building or may be attached to the side of a building.

Buildings can have a significant effect on the dispersion of pollutants from sources and can increase the maximum predicted ground level concentrations. The main effect of a building is to entrain pollutants into the cavity region in the immediate leeward side of the building, bringing them rapidly down to ground level. As a consequence, concentrations near the building are increased but further away concentrations are decreased.

The buildings module in ADMS 5 allows the user to enter up to 25 buildings in any one run. However, this does not mean that this many buildings *should* be included in a given run and consideration must be given as to which buildings to include.

For each source, the program combines the individual buildings into a single effective building for each wind direction in the meteorological file, the height of which is the height of the building specified as the main building for that source. This is described in more detail in Section 9.16.

4.9.2 Defining buildings

Select the **Buildings** button on the **Setup** screen and click **Enter parameters**... to go to the **Buildings** screen shown in **Figure 4.34**.

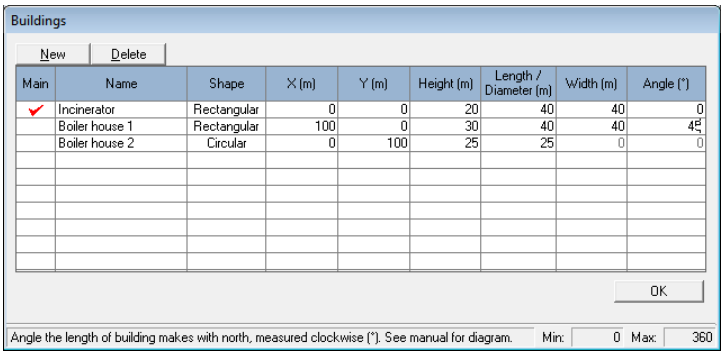


Figure 4.34 − The **Buildings** screen.

The user may add or remove buildings in the table using the **New** and **Delete** buttons, respectively. At least one building must be specified, with others up to a maximum of 25, each defined by the following parameters.

1. **Main**: The user must define one 'main building' by placing a tick in this column. This is likely to be the building that has the most significant effect on dispersion. To change the main building, double click in the **Main** column or type **Y** or **N** to display or remove the tick.

Default = a tick appears next to the first building created.

Note that the main building defined in this screen is the default main building. Source-specific main buildings are defined on the **Source** screen, refer to Section 4.9.3 for further details.

2. **Name**: The model will use a default name when a new building is added (e.g. Building001). The user is advised to change this to something more meaningful (e.g. boiler house). Building names must not contain commas.
3. **Shape**: shape of the building, either **Rectangular** or **Circular**. To change the shape of the building click on the cell.

   Default = **Rectangular**.

4. **X (m)**: X coordinate of the centre of the building
   Y (m): Y coordinate of the centre of the building

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

   The large maximum value allows the user to input UK National Grid coordinates or worldwide UTM (Universal Transverse Mercator) coordinates. Note that the building position should be specified in the same coordinates as the source position and output grid.

5. **Height (m)**: height of the building.

   Minimum = 0.001 m
   Maximum = 500 m
   Default = 10 m

6. **Length (m)**: length of a rectangular building or diameter of a circular building. For a rectangular building this is simply one horizontal dimension.

   Minimum = 0.001 m
   Maximum = 1 000 m
   Default = 10 m

7. **Width (m)**: width of a rectangular building, not necessarily smaller then the length. This parameter is not used for a circular building.

   Minimum = 0.001 m
   Maximum = 1 000 m
   Default = 10 m

8. **Angle (°)**: the angle between north and the previously defined **Length**, measured clockwise from north. This is not used for a circular building. Refer to Section Section 9.16 for a diagram and more information.

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

To duplicate an existing building right-click on the name of the building and select ***Copy****.*

4.9.3 Defining the 'main building' for each source

The user may specify a 'main building' for each source, i.e. indicate which building is likely to have the most effect on dispersion from that source. **Figure 4.35** shows the **Source** screen when **Buildings** are selected in the **Setup** screen. In the **Main building** column for each source the user may select a building from the list that they would like to define as the **Main building** for that source. The list includes all the named buildings defined in the **Buildings** screen, as well as **"(Main)"** and **"(None)"**.

If **"(Main)"** is chosen then the building defined as the **Main building** in the **Buildings** screen will be the main building for this source. This is the default selection.

If **"(None)"** is selected then building effects will not be modelled for this source. This might be useful, for example, where the user knows that a source lies outside the region of influence of all buildings and wishes to keep run-times to a minimum.

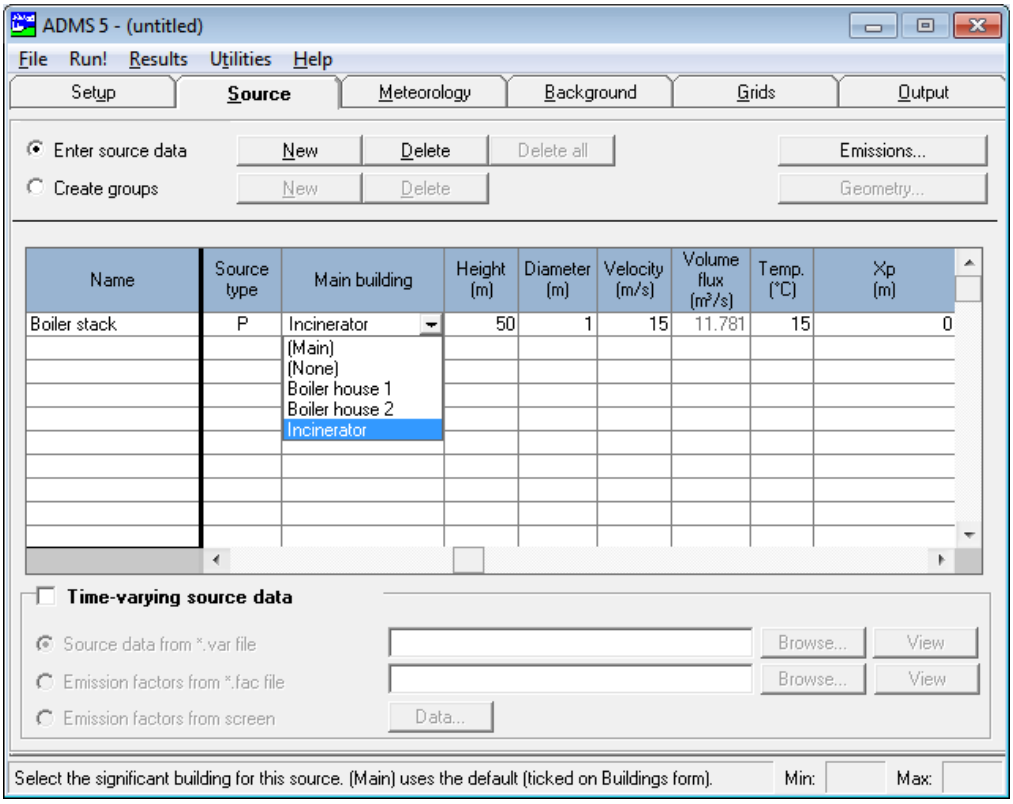


Figure 4.35 − The **Source** screen when a building is present.

4.9.4 Guidance

Wherever possible, a complicated site with a large number of buildings should be simplified. The user should consider the building or buildings to which the sources are attached or to which they are nearest.

A few hints and tips for modelling buildings are listed below.

- Buildings such that the building height, H, is less than a fraction ($1/\alpha$) of the source height are ignored when setting up the effective building, so need not be entered. Here $\alpha = 1 + 2*min(1,W/H)$ with W the crosswind width of the building.
- Where there is a cluster of similar buildings on a site such as a row of warehouses, the user may decide to enter them as a single building.
- If there are a large number of buildings on a large site, the user should consider whether to include those that are nearest to/attached to the sources and/or those that will have the greatest effect on dispersion (tallest/largest), or consider a higher surface roughness, which can be entered in the **Meteorology** screen, as a means of representing the buildings in a complex site.
- The effect of buildings on dispersion can only be modelled for point sources. Area, line or volume sources may be included in the modelling run, but the effect of the buildings on dispersion from these sources will not be modelled. In order to model building effects for a line, area or volume source, the source should be modelled as a number of point sources.
- Sources inside buildings cannot be modelled, though sources on the sides or roof of a building can be modelled.

4.10 Complex terrain

For many industrial sites located in areas of complex terrain, the effect of the terrain height and/or the spatial variation of surface roughness on plume dispersion should be considered.

The effect of complex terrain is modelled by changing the plume trajectory and dispersion to account for disturbances in the air flow due to the terrain. This may increase or decrease the concentrations calculated.

It is not always necessary to include the effects of surrounding terrain in a modelling calculation. Usually terrain height effects are only included if the gradient exceeds 1:10; gradients of up to 1:3 can be modelled. The influence of the terrain height will vary with the source height and position and the local meteorology.

4.10.1 Terrain height and surface roughness files

To run the **Complex terrain** option, additional data file(s) are required that contain the spatially varying terrain height and/or surface roughness data; these are called the *.ter* and *.ruf* files respectively. The file(s) should consist of a series of lines with

```
N, X, Y, Z
```

entered on each line, where `N` is an incrementing counter for each line, `X` is the X coordinate of the data point, `Y` is the Y coordinate of the data point, and `Z` is:

- the height of the terrain (in metres) at that data point for the *.ter* file (see example in **Figure 4.36**), and
- the surface roughness value (in metres) at that data point for the *.ruf* file.

The .ter/.ruf *files should be comma-separated and the maximum number of data points that may be included in each file is 66,000.*

The data points do not need to be regularly spaced, but the area described should be rectangular. Note that the coordinate system used should be the same as that in which the sources, output points, etc., are defined. An example *terrain.ter* file describing a bell-shaped hill is supplied in *<install_path>\Data* and shown in **Figure 4.36**. This file may be edited to create new *.ter* files using a spreadsheet program or text editor. Section 7.4 provides further details of how to create new terrain files using the **ADMS Terrain Converter** utility and Section 5.6 of the ADMS Mapper User Guide provides details on using the ADMS Mapper to create terrain files. An example *ROUGHNES.RUF* file is also supplied in the *<install_path>\Data*.

An easy way to create a roughness file is to edit the Z values of the terrain file being used so that the same domain, with the same X and Y coordinates, is used for both files. This can be carried out using a spreadsheet package such as Excel.

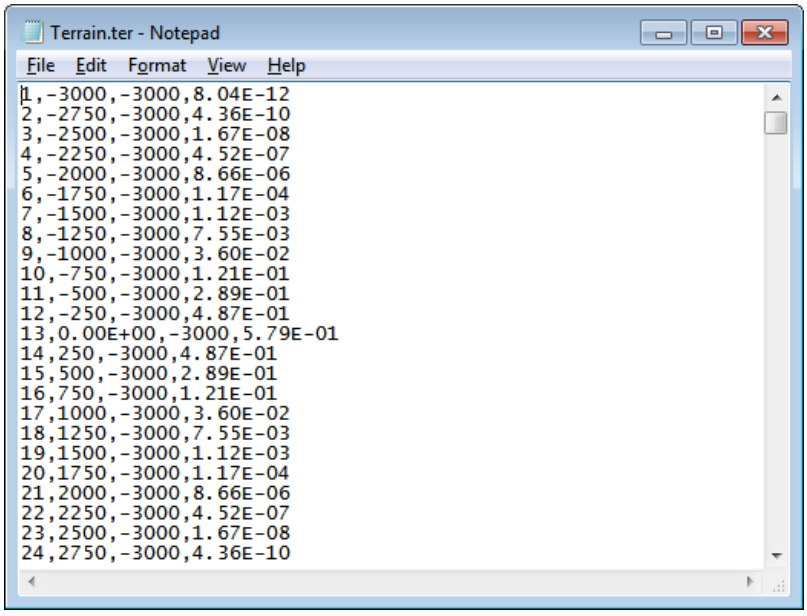


Figure 4.36 − The start of the terrain file *terrain.ter*.

The surface roughness file must cover the same area as the terrain file.

The values in the terrain height file should not all be equal.

4.10.2 Modelling the effects of complex terrain

To model the effects of complex terrain, go to the **Setup** screen. Select **Complex terrain** from the model options and click on **Enter parameters**... to go to the **Complex terrain** screen shown in **Figure 4.37**.

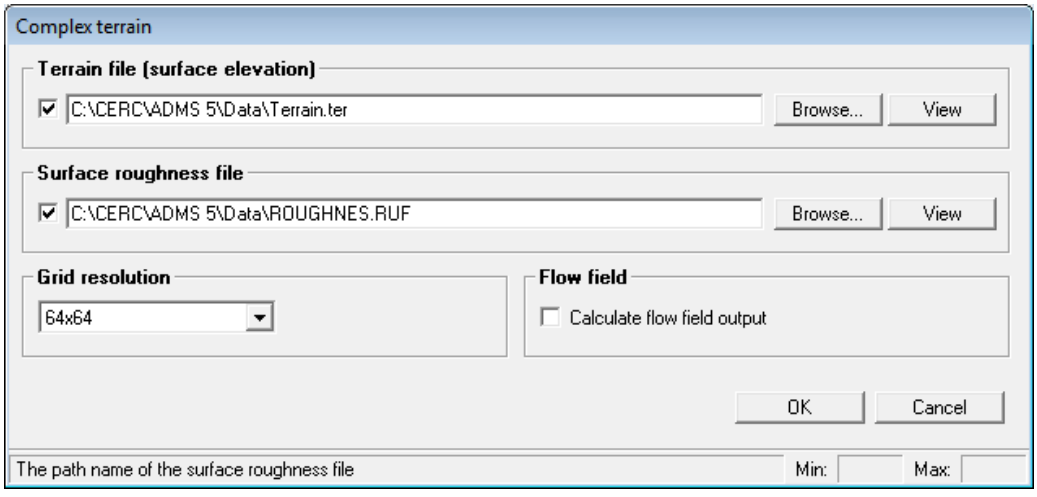


Figure 4.37 − The **Complex terrain** screen.

1. **Terrain file (surface elevation)**: Check the box if this option is required.

   In the text box, type the full path name of the required terrain height file, or use the **Browse**... button to find existing *.ter* files.

2. **Surface roughness file**: Check the box if this option is required.

In the text box, type the full path name of the required surface roughness file, or use the **Browse**... button to find existing *.ruf* files.

3. **Grid resolution**: the following choices of internal calculation grid sizes are offered:

   * **256 x 256** - this option should only be used if the complex terrain input data are highly resolved, and cover large or complicated areas of terrain; run times may be prohibitive.
   * **128 x 128**
   * **64 x 64** – this is the default
   * **32 x 32**
   * **16 x 16** - for testing only

   For most calculations the 32 x 32 or 64 x 64 grids are sufficient.

   Note that the grid resolution here refers to the internal grid used to calculate the flow field and is different to that in the **Grids** screen, which refers to the grid of points on which concentrations are output. The terrain grid extent must always be bigger than the output grid.

4. **Flow field**: Check the box if flow field output is required. The flow field output consists of the velocity and turbulence components and is given at the output locations entered in the **Grids** screen. For more detail on the flow field output refer to Sections 6.1.13 to 6.1.18 and Section 6.3.2.

4.10.3 Restrictions

The following restrictions apply when using the **Complex terrain** option.

The geographical extent covered by the terrain data must be larger than the modelling region, where the modelling region comprises all of the sources, buildings, specified output points and the output grid. Specifically:

- For runs with a wind sector size of less than or equal to 15° the terrain region must be at least 10% larger than the modelling region.
- For runs with a wind sector size of greater than 15° the terrain region must be at least 50% larger than the modelling region.
- Additionally the terrain region must be at least 500 m larger than the modelling region in each direction.

If the terrain requirements are not met then the model will stop with a message giving the minimum region that must be covered by the terrain to run with the current sources, buildings, output points and model options.

4.11 User-input 3D flow field data

In ADMS 5, users can input their own 3D flow fields to be used in place of the standard flat terrain or hills flow fields. For example, users can enter flow fields output from a mesoscale meteorological model or a CFD program.

In order to enter a user-input 3D flow field file, select the **Complex terrain** option in the interface and browse to select any ADMS 5 terrain file, for example the standard example data file *terrain.ter* in the *Data* sub-directory of the ADMS 5 installation directory.

This terrain file will not be used by the model, but it is necessary to select a terrain file in order to save the .apl *file.*

Details of the format of the flow field data that can be input into the ADMS 5 model are given in Section 4.11.1 below; Section 4.11.2 outlines how these data can be included in a model run using an *.aai* file. Section 4.11.3 gives some guidance on the use of user-input 3D flow field data with ADMS 5, and Section 4.11.4 summarises some restrictions of this option.

4.11.1 Format of the user-input 3D flow field file

When entering user-input 3D flow fields it is necessary to enter values for three velocity components and additionally either the turbulent kinetic energy or three turbulence components.

The tab- or comma-separated flow field data should be in a text file with the following format (there is no header row):

```
NX NY NZ
X Y Z U V W TKE
…
```

or

```
NX NY NZ
X Y Z U V W SigU SigV SigW
…
```

On the first row, `NX` is the number of grid points in the X horizontal direction, `NY` is the number of grid points in the Y horizontal direction, and `NZ` is the number of grid points in the vertical direction.

On the second and subsequent rows, `X`, `Y`, and `Z` are the coordinates of the grid point in metres, using the same coordinate system as the rest of the modelling input. `Z` should be measured in metres from the ground. `U`, `V`, and `W` are the X, Y and Z components of the wind velocity (m/s). `TKE` is the turbulent kinetic energy (m^2/s^2). `SigU`, `SigV`, and `SigW` are the X, Y and Z components of the turbulence velocity (m/s).

The data must be ordered as follows: data for each (x,y) location must be grouped together, with corresponding vertical heights being distinct, and increasing in magnitude.

The first few lines of an example file using TKE are shown in **Figure 4.38**.

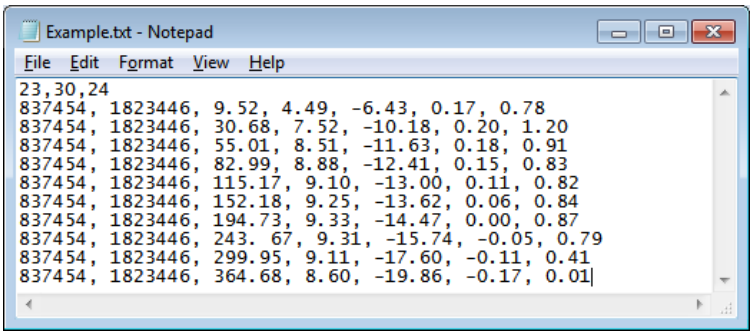


Figure 4.38 − Example user-input 3D flow field file using TKE.

4.11.2 Additional input file

To use user-input 3D flow field data an *.aai* file should be created or the existing *.aai* file edited, see 3.1.6, so the **User-input 3D flow field** option (keyword MESO) is enabled (**Figure 4.39**).

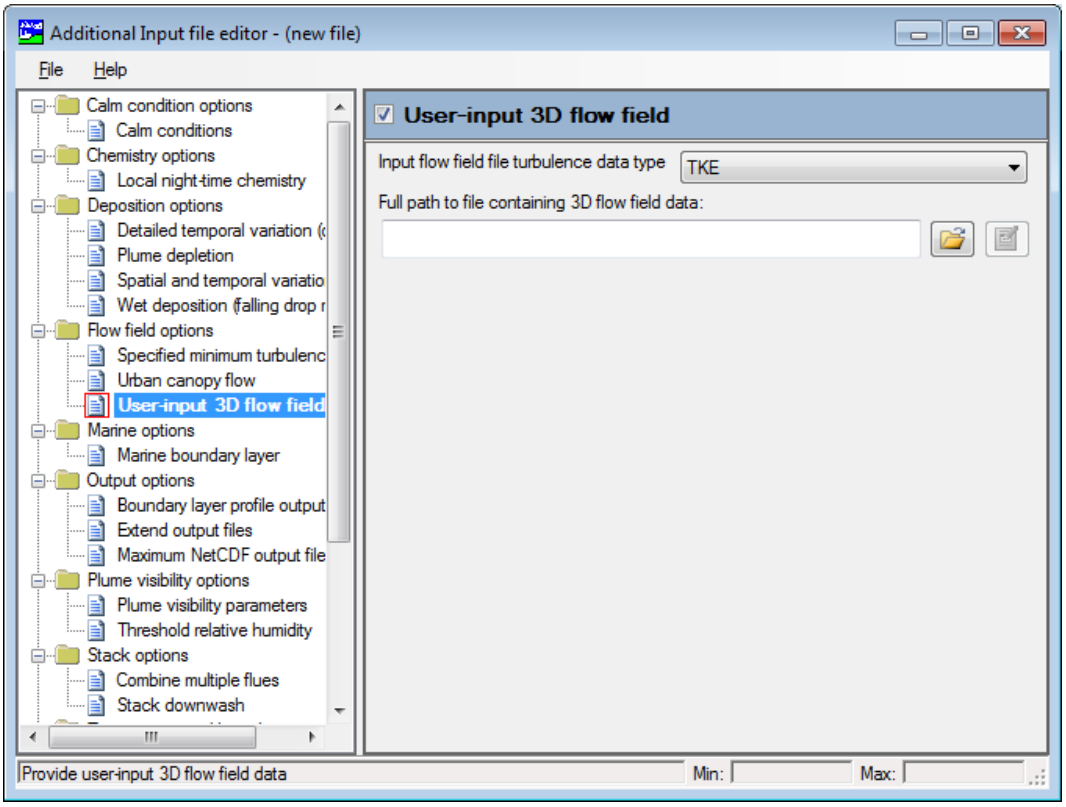


Figure 4.39 – **User-input 3D flow field** option enabled in the **Additional Input file editor**.

The type of turbulence data being entered in the file should be selected, either:

- **TKE** if turbulent kinetic energies are being entered or
- **Turbulent velocities** if turbulent velocities are being entered.

The path to the file containing the 3D flow field data must also be entered. Use the button to browse to the file and the button to view the file in the default

application, as selected by the **File\Preferences...\Viewing** menu option in the **Additional input file editor**.

4.11.3 Guidance

All input values are used without modification in the model apart from TKE, which is adjusted to account for the fact that mesoscale numerical models have a tendency to predict TKE values that are substantially lower than those typically used in ADMS 5. Further details of this adjustment are given in Section 9.18.

Users should note that:

- In addition to the 3D flow field, the user must provide a standard ADMS 5 *.met* file containing the meteorological conditions upstream of the domain covered by the 3D flow field.
- The user-input 3D flow field data should include data for at least one height below 50 m and at least three heights below 100 m. Higher vertical resolution in the region of the source height will improve modelling results.
- No flow field output can be produced if the 'User-input 3D flow field data' option is selected.

4.11.4 Restrictions

There are a number of restrictions with the user-input 3D flow field option:

- User-input 3D flow field data can only be used for short-term runs with one line of meteorological data.
- Using the notation introduced above, the maximum number of distinct (x,y) points, NX×NY, is 1000. At each (x,y) point, up to 50 vertical points (NZ) are allowed.
- The model assumes that the flow field data are defined on a rectangular grid, although this grid does not have to be north-south / east-west aligned.
- The geographical extent covered by the flow field data must be larger than the modelling region, where the modelling region comprises of all of the sources, buildings, specified output points and the output grid. Specifically, the flow field data must cover a region at least 10% larger than the modelling region. Additionally the region covered by the flow field data must be at least 500 m larger than the modelling region in each direction.
- If the flow field data does not cover a sufficiently large area then the model will stop with a message giving the minimum region that must be covered by the flow field data to run with the current sources, buildings, output points and model options.

4.12 Coastline

The effect of a coastline will generally only be significant for emissions from elevated point sources that are within a few kilometres (up to a maximum of 5 km) of the coast. The coastline module is only invoked when:

- the wind is blowing onshore (from the sea to the land),
- the land is warmer than the sea,
- the meteorological conditions over the land are unstable (convective).

When the **Coastline** option is selected, these conditions are checked automatically by the model for each line of meteorological data used in the calculation. If they are not satisfied then the coastline module is disabled for that meteorological data line. In addition to the usual meteorological data variables, the meteorological file used when modelling coastline effects must contain either:

> DELTAT: the difference in temperature (in degrees) between the sea surface temperature and the near surface temperature over land (e.g. as in *coast.met*); **or**
>
> both TSEA (the sea surface temperature) and T0C (the surface temperature over land).

*It is not possible to enter meteorological data by hand when modelling coastal effects because the necessary sea temperature data above are not included in the table (see **Figure 4.25**).*

To model coastal effects select the **Coastline** option on the **Setup** screen and click **Enter parameters**... to go to the **Coastline** screen shown in **Figure 4.40**. The coastline position and orientation (sense) is specified by means of the coordinates of two points lying on the coastline and one point on the land. The model will assume that the coastline is a straight line between the two points on the coastline. This is illustrated in **Figure 4.41**.

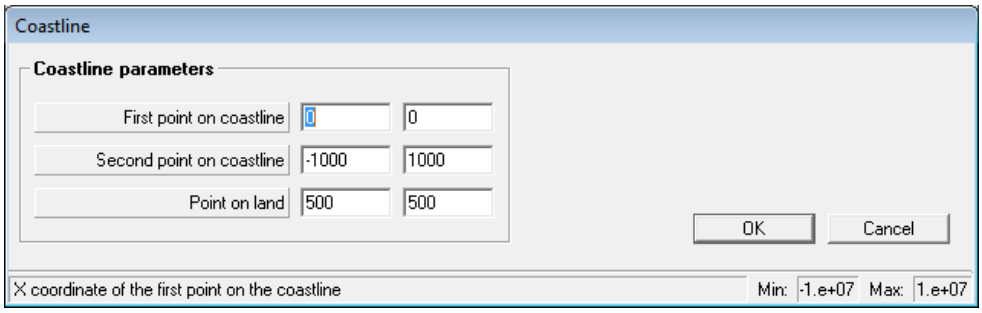


Figure 4.40 − The **Coastline** screen.

The following parameters should then be defined.

1. **First point on the coastline**: X and Y coordinates of the first point on the coastline

   Minimum = -9,999,999 m
   Maximum = 9,999,999 m
   Default = (0,0)

2. **Second point on the coastline**: X and Y coordinates of the second point on the coastline

   Minimum = -9,999,999 m
   Maximum = 9,999,999 m
   Default = (-1000,1000)

3. **Point on land**: X and Y coordinates of a point on land (X_L, Y_L in **Figure 4.41**)

   Minimum = -9,999,999 m
   Maximum = 9,999,999 m
   Default = (500,500)

The coordinate system used to define the coastline should be the same as that used for the rest of the model input file.

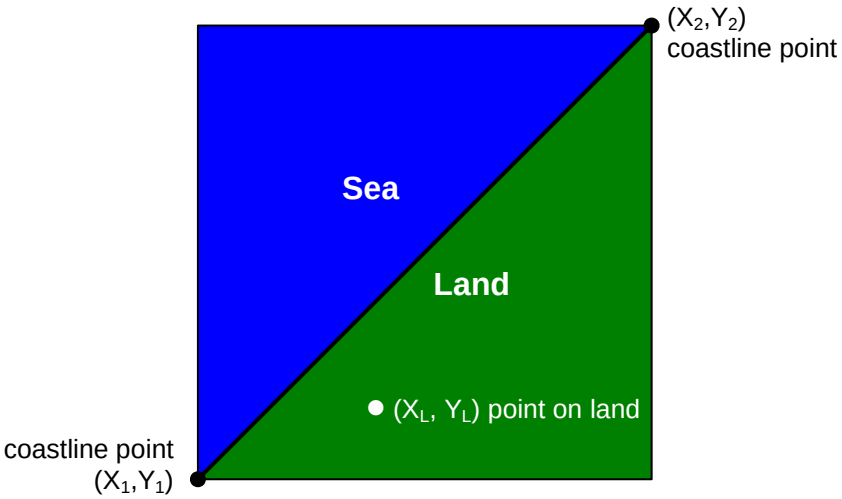


Figure 4.41 − An example of the coastline coordinates to be specified.

4.13 Puff

The **Puff** option in ADMS 5 can be used when a substance is released over a specified short period of time, usually less than 1 hour but possibly up to 24 hours. Situations when this could occur might be blowdowns, accidental discharges or explosions.

The model allows the user to calculate two types of output. These are: time dependent concentrations, i.e. a "snapshot" of the concentrations at various distances for a particular time after the release started; and time-integrated concentrations or dose. The dose can be thought of as a measure of the total quantity of pollutant to which a person would be exposed to by remaining at a particular location as the puff passes that point, from first to last exposure.

The model assumes that meteorological conditions remain constant throughout the whole duration of the release, even if the release lasts more than 1 hour. Separate output is given for each meteorological condition modelled.

To model a puff release, check the **Puff** box on the **Setup** screen.

---

*Only short-term output is available for puffs. Only one source can be modelled so single source output should be selected on the **Output** screen.*

Release details

The user should first visit the **Source** screen and click on the **Emissions**... button to go to the **Emissions** screen. This is shown in **Figure 4.42**.

Note that the column headings have changed from those that appear when modelling a continuous release.

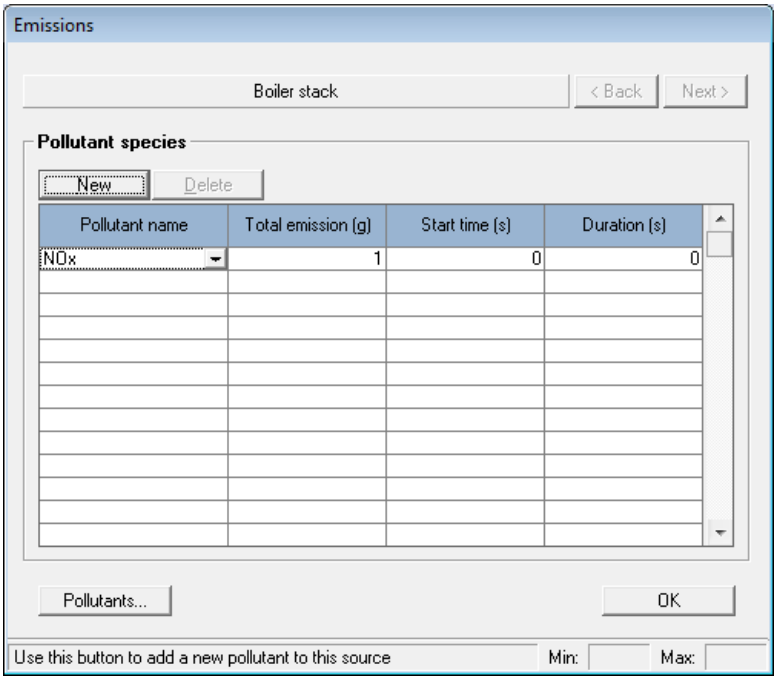


Figure 4.42 − The **Emissions** screen when modelling a puff release.

The following parameters are required to define the release:

1. **Total emission (g)**: total quantity of pollutant released

   Minimum = 10^{-10} g
   Maximum = 10^{30} g
   Default = 1 g

2. **Start time (s)**: start time of release

   Minimum = 0 s
   Maximum = 86 400 s (= 24 hours)
   Default = 0 s

3. **Duration (s)**: duration of release

   Minimum = 0 s
   Maximum = 86 400 s (=24 hours)
   Default = 0 s

Different pollutants can be released at different start times and for different durations.

Output details

Now return to the **Setup** screen and click **Enter parameters**... to go to the **Puff** screen shown in **Figure 4.43**.

Select one of the following:

- **Time-integrated (dose)**: to calculate the total quantity of material to which a person would be exposed at a particular location as the puff passes that point i.e. from first to last exposure. If this option is selected, no further parameters are defined. Results are calculated in units of g.s/m^3 (or mg.s/m^3, etc.).
- **Time dependent**: to calculate a series of snapshots of the dispersing pollutant, as defined by the following parameters:

  1. **Start time for output (s)**

     Minimum = 0 s
     Maximum = 82 800 s (=23 hours)
     Default = 100 s

  2. **Timestep (s)**

     Minimum = 1 s
     Maximum = 10 800 s (=3 hours)
     Default = 100 s

  3. **Number of timesteps**

     Minimum = 1
     Maximum = 50
     Default = 10

For example, the default parameter values would calculate 10 snapshots of concentration at 100-second intervals starting from 100 s (i.e. at 100 s, 200 s, 300 s, etc. up to 1000 s). Results are calculated in units of g/m^3 (or mg/m^3, etc.).

*The averaging (sampling) time in the **Output** screen should be set to equal to the duration of the release as given on the **Emissions** screen. The averaging time is used to determine the horizontal (across-wind) spread of the puff.*

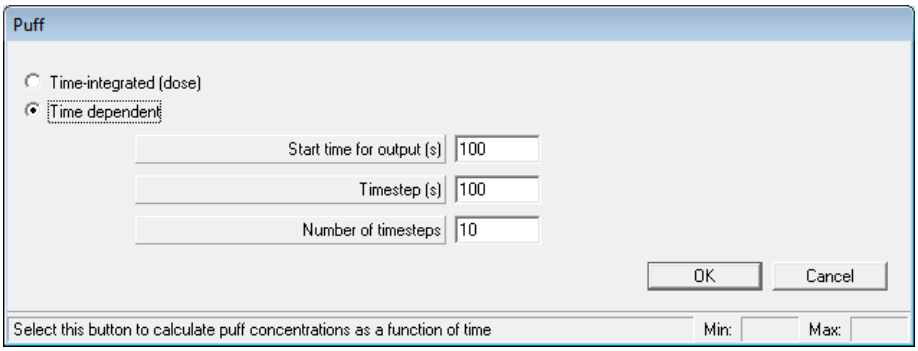


Figure 4.43 − The **Puff** screen, used to specify the output required.

4.14 Fluctuations

The **Fluctuations** option is particularly useful for modelling odour problems and making comparisons between predicted concentrations and air quality standards for averaging times of less than 1 hour.

For applications such as dispersion modelling, a period of one hour is taken as the time scale over which the meteorological conditions are roughly constant. Time scales less than an hour are then considered "small" or "short". Changes in meteorological conditions generally occur on time scales greater than an hour while turbulent time scales are generally less than an hour.

The **Fluctuations** option allows the user to take account of the variations in concentration caused by the "short" time scale turbulence in the lower atmosphere. If either the observation or the release period is less than an hour, most changes are due to boundary-layer turbulence and constant meteorology is assumed, except for changes in mean wind direction. Changes on a time scale longer than 1 hour are said to be due to changing meteorology.

Note that in ADMS 5 the effect of fluctuations over short time scales may be included in long-term average calculations. This allows the user to compare modelled long-term 15-minute average concentrations of SO_2 with the National Air Quality Strategy (NAQS) objective of the 99.9th percentile of 15-minute average concentrations for SO_2. The model can also be used to calculate the number of exceedences of particular concentrations in a year.

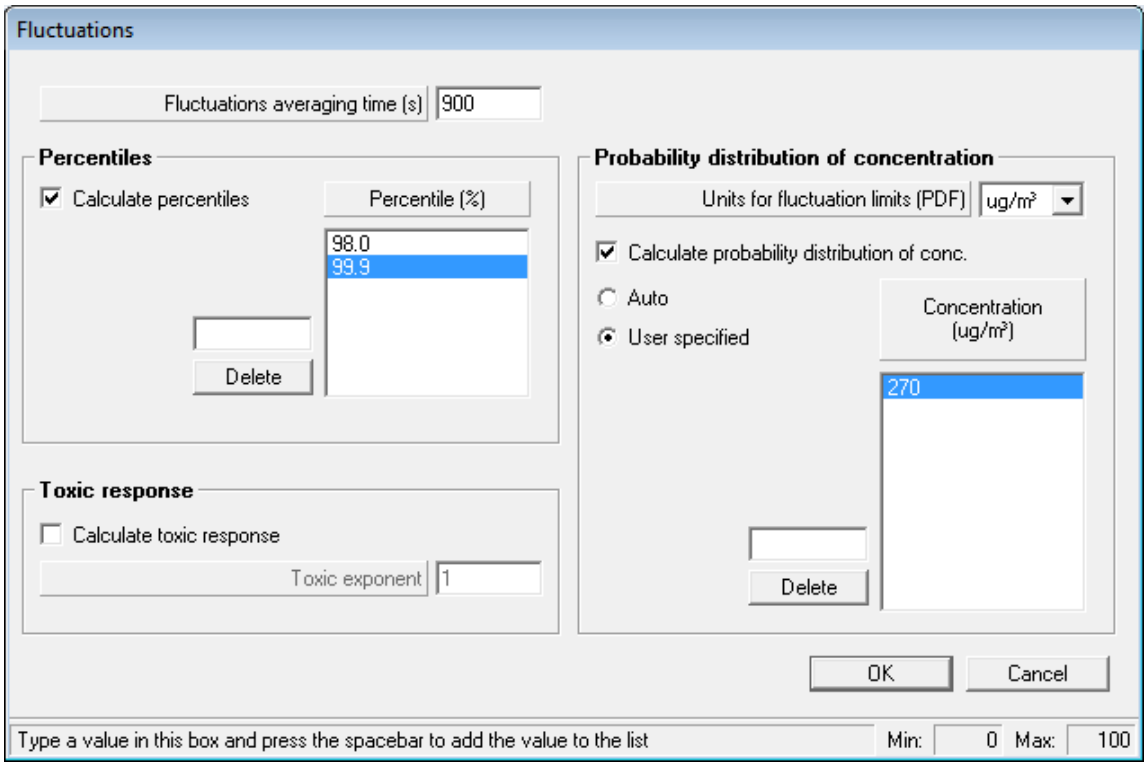


Figure 4.44 − The **Fluctuations** screen.

To model the effects of fluctuations, go to the **Setup** screen, select **Fluctuations** from the model options and click on **Enter parameters**... to go to the **Fluctuations** input screen shown in **Figure 4.44**. The available parameters are described below.

1. **Fluctuations averaging time (s)**: the observation time in seconds

   Minimum = 0 s
   Maximum = 3600 s

   The user does not need to enter a value here if a release is discrete (puff). For a time dependent puff the relevant averaging time is 0 seconds and for a time-integrated puff it tends to infinity.

2. **Percentiles**: Click on the **Calculate percentiles** check box and add the percentile(s) to be calculated to the **Percentile (%)** list box, by typing the number in the small box and pressing the space bar. To remove a number from the list highlight the number and click on the **Delete** button.

   Minimum = 0
   Maximum = 100

   *The **Percentiles** option here is only available for short-term average runs i.e. not for annual averages. To calculate, say, the long-term 99.9th percentile of 15-minute means without the effect of fluctuations, specify this in the **Output** screen of ADMS 5.*

3. **Units for fluctuations limits (PDF)**: select the units for probability distribution output from the list available by clicking on this cell and then on the down arrow that appears in the right of the cell. The units available are ng/m^3, µg/m^3, mg/m^3, g/m^3, ppb (parts per billion by volume), ppm (parts per million by volume). If odours are also being modelled then the units will be fixed as ou_e/m^3.

   Default = ug/m^3

4. **Probability distribution of concentration**: this information is used to calculate the probability distribution function (PDF) of concentrations, i.e. the probability of certain concentrations being exceeded. Click on the **Calculate probability distribution of conc.** check box and select the **Auto** option to use program-selected concentration values or the **User specified** option to use specific values of concentration input by the user. The program-selected values will vary for each output point and will cover several decades of concentration (orders of magnitude). To add values to the **Concentration (units)** list, type the concentration in the small box and press the space bar to enter. To remove a number from the list, highlight the number and click on the **Delete** button. The units of concentration will be shown as specified in **Units for fluctuation limits (PDF)** above.

   Minimum = 10^{-20} units
   Maximum = 10^{20} units

5. **Toxic response**: click on the **Calculate toxic response** check box if you want the model to calculate the ensemble mean of concentration to a chosen power. Then type a value in the **Toxic exponent** box.

   Minimum = 0
   Maximum = 10

Default = 1

The toxic response calculation is not available if the release is a time-integrated puff because the calculation uses the instantaneous puff concentrations and not the time-integrated ones.

4.14.1 Restrictions

The following restrictions apply when using the **Fluctuations** option.

- The maximum number of point sources that can be modelled is 50 and the maximum number of line, area and volume sources that can additionally be modelled is 20.
- Fluctuations output is given for the first pollutant listed in the **Output** screen only.
- If the **Groups** option is chosen, fluctuations output is given for the first group only.
- The averaging time on the **Output** screen should be set to whatever is appropriate for the meteorological data, e.g. 1 h.

4.15 Advanced meteorological parameters

Advanced meteorological data can be entered for the dispersion site and/or the meteorological measurement site. To enter advanced meteorological data go to the **Meteorology** screen, select the **Use advanced options** box in either of the **Dispersion site** and **Met. measurement site** boxes, then click on **Data**....

The **Advanced dispersion site data** screen is shown in **Figure 4.45**; the **Advanced met. site data** screen is shown in **Figure 4.46**.

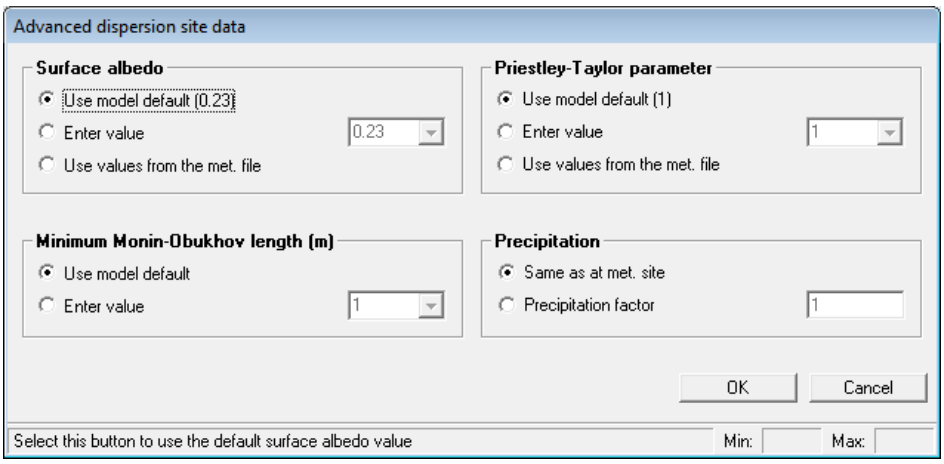


Figure 4.45 − The **Advanced dispersion site data** screen

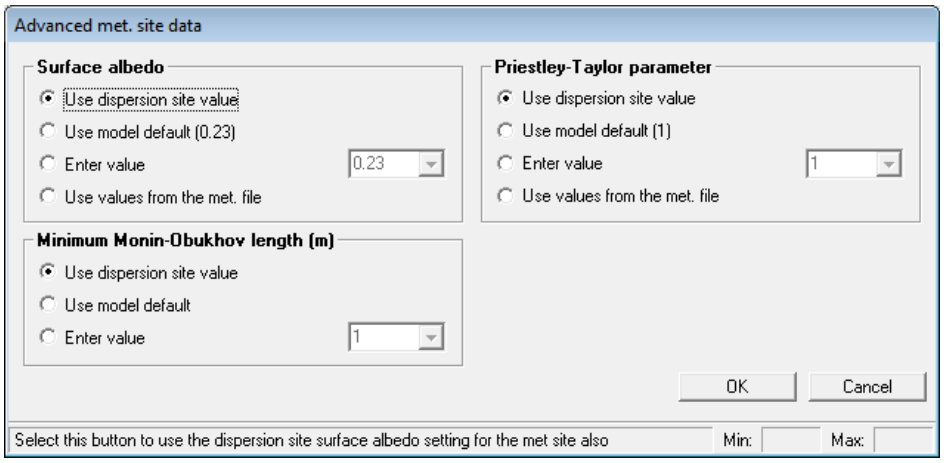


Figure 4.46 − The **Advanced met. site data** screen

The parameters used by ADMS 5 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 5 includes the option to specify values of these parameters more suitable for the site being modelled.

*Note that if the **Use advanced options** box has not been ticked for the **Met. measurement site**, then each of the parameters will take the dispersion site value.*

The option to alter these parameters at either the dispersion or meteorological measurement site is described further in Sections 4.15.1 to 4.15.3. 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 4.15.4.

4.15.1 Surface albedo

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

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

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

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

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

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

4.15.2 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.

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

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

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

the meteorological file.

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

4.15.3 Minimum Monin-Obukhov length

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

It is useful to note that the minimum value of the Monin-Obukhov length is used by the model to determine a minimum value for the turbulence parameters, σ_{min}, in the following way:

$$\sigma_{min} = \begin{cases} 0.01 & L_{MOmin} \le 10 \\ 0.01 + 0.0095(L_{MOmin} - 10) & 10 < L_{MOmin} < 30 \\ 0.2 & L_{MOmin} \ge 30 \end{cases} \quad (4.3)$$

In addition if complex terrain is modelled a further minimum of 0.1 m/s is applied. See Section 4.22 for further details regarding the minimum value for the turbulence parameters including a way to specify this minimum value.

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.

Enter value: Enter a value, either by typing directly into the box, or by selecting from a list using the drop down menu:

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

At the meteorological measurement site there is an additional option of **Use value at dispersion site**; select this option to ensure the minimum Monin-Obukhov lengths at the

dispersion site and the meteorological measurement site are the same. This is the default option for the meteorological measurement site.

4.15.4 Precipitation

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

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

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

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

4.16 User-input vertical profiles of meteorological data

User-defined vertical profiles of wind speed, turbulence, temperature and specific humidity can be entered. These data are entered using a *.prf* file.

To use this option first select the **Vertical profiles** option on the **Meteorology** screen, then **Browse**... to select the *.prf* file required.

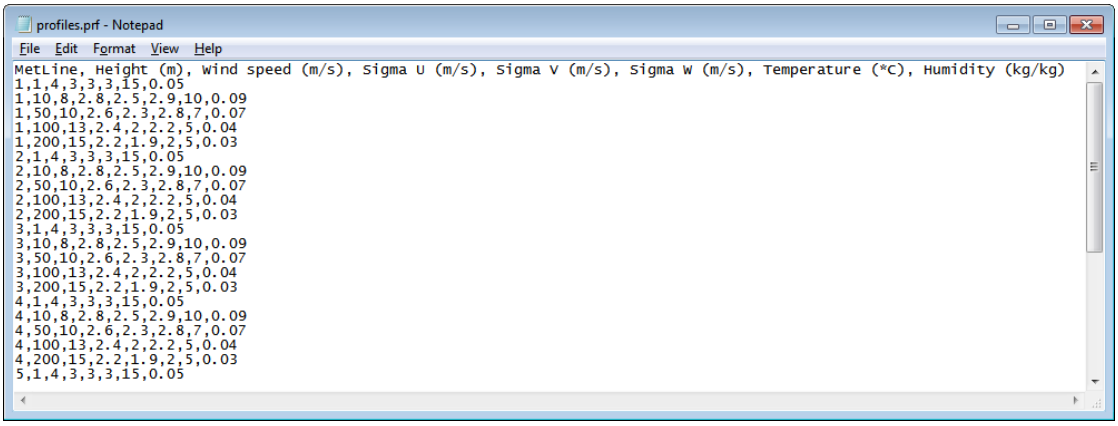


Figure 4.47 − An example *.prf* file.

The *.prf* file should be comma-separated. An example *.prf* file is shown in **Figure 4.47** to illustrate the format of the file. 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 in the meteorological file. The lines should be in order with the heights in ascending order for each meteorological data line. 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 line number corresponding to this line of data

   If the option to use a meteorological data subset is selected then the .prf *file should contain only data for that subset with meteorological line numbering relative to the start of the subset.*

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 in the meteorological file.

   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`**: the specific humidity in kg/kg

   Minimum = 0 kg/kg
   Maximum = 0.1 kg/kg

4.17 Boundary layer profile output

ADMS 5 calculates a vertical profile of the boundary layer properties, and these properties may be output to a *.pro* file at a range of heights using an *.aai* file option. Section 6.1.9 describes the format of the *.pro* file and Section 9.2 together with Technical Specification document P09/01 (CERC, 2015) give more details regarding the calculation of these variables.

To output a *.pro* file, first create an *.aai* file or edit the existing *.aai* file, refer to Section 3.1.6, so the **Boundary layer profile output** option (keyword BLPROFILE) is enabled (Figure 4.48).

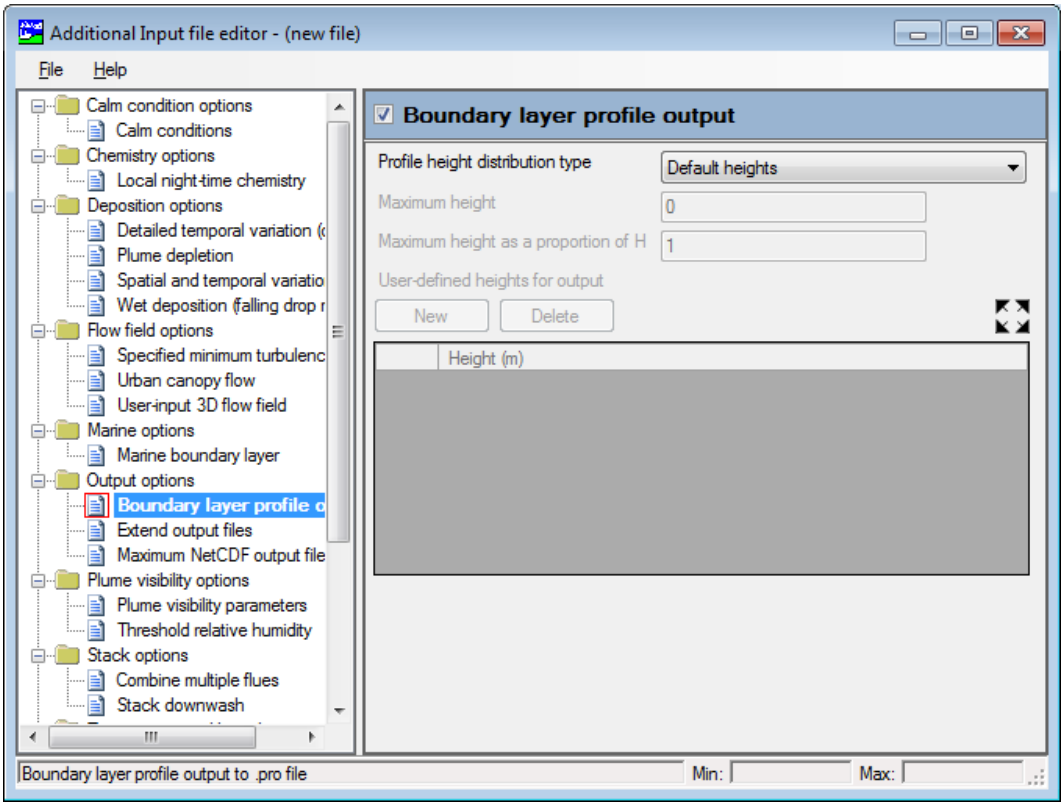


Figure 4.48 – The **Boundary layer profile output** section of the **Additional Input file editor**.

Select the type of profile distribution to use from the list:

- **Default heights**: A geometric series of 100 heights up to the boundary layer height will be used.
- **Define absolute max height**: A geometric series of 100 heights up to the specified fixed height will be used.
- **Define max height relative to H**: A geometric series of 100 heights up to the specified proportion of the boundary layer will be used.
- **Define all heights**: The heights defined in the table will be used. Up to 200 heights may be defined.

4.18 Calm conditions

ADMS 5 includes an option to model 'calm' meteorological data (meteorological data with wind speed at 10 m, *U*, less than 0.75 m/s); in standard ADMS 5 runs, such meteorological data are not modelled.

To model calm conditions, an *.aai* file should be created or the existing *.aai* file edited, refer to Section 3.1.6, and the **Calm conditions** option (keyword CALMS) enabled (**Figure 4.49**).

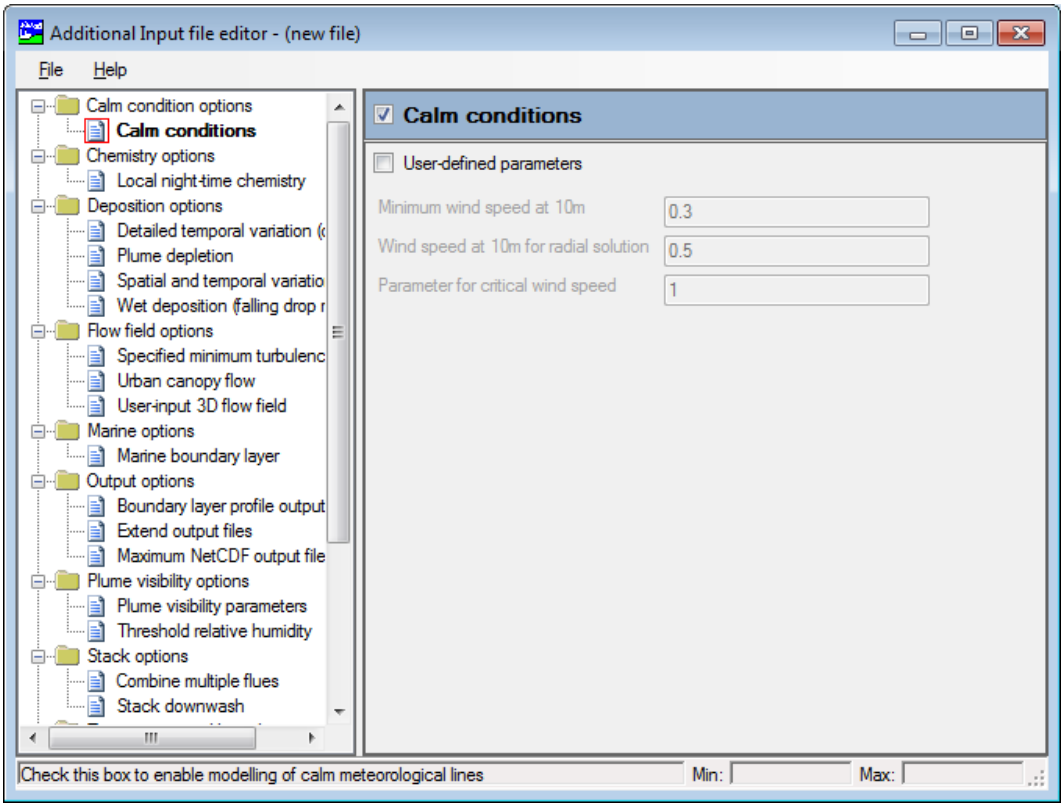


Figure 4.49 – **Calm conditions** option enabled in the **Additional Input file editor**, using the default values for the parameters.

When modelling calm conditions, three parameters are required to define the behaviour of the model. User-defined values can be set for these parameters by checking the **User-defined parameters** box and entering the following:

- **Minimum wind speed at 10m**

  The minimum wind speed at 10 m (MinU10) is the lowest value of *U* at which the model will calculate concentrations. The default value of MinU10 is 0.3 m/s. If U is lower than MinU10 (which includes meteorological data with $U = 0$ m/s), then it will be increased to MinU10 (with friction velocity u_* and surface heat flux $F_{\theta 0}$ adjusted accordingly). Section 4.18.1 gives some guidance on the most appropriate value of MinU10.

- **Wind speed at 10m for radial solution**

  The wind speed at 10 m for the radial solution ($U_{G\,\min}$) is the lower limit of the

validity of the normal Gaussian type plume. The default value of $U_{G\min}$ is 0.5 m/s. For meteorological data with U less than $U_{G\min}$, the model calculates the radial solution only.

- **Parameter for critical wind speed**

  The parameter for critical wind speed (U_{calm}) is a parameter value used in the calculation of the critical value of U, U_{crit}, above which the model calculates the Gaussian solution only. The default value of U_{calm} is 1.0 m/s.

Technical details regarding the calm conditions option are given in Section 9.3.

4.18.1 Notes on selecting the appropriate value of MinU10

Users should choose a value for MinU10 on the basis of the distribution of wind speed data in the meteorological data used. A good 'rule of thumb' is to choose a value that is mid-way between 0 m/s and the next lowest wind speed recorded in the meteorological data.

For example, in some old meteorological datasets, many meteorological data lines may have U = 0 m/s and many may have U = 1 m/s but there may be no wind speeds recorded between 0 m/s and 1 m/s. This would suggest that the instrument recording the wind speed data was not sensitive below 1 m/s and that '0 m/s' actually means '< 1 m/s'. In this case, using the default value of MinU10 = 0.3 m/s would mean that all meteorological data with U = 0 m/s would be modelled with U = 0.3 m/s, which is likely to be too low. A better value in this case would be MinU10 = 0.5 m/s.

However, newer data recorded with more sensitive instruments may have a lowest non-zero recording of 0.5 m/s, in which case the default value of MinU10 = 0.3 m/s would be appropriate.

4.18.2 Restrictions

ADMS 5 cannot currently model calm meteorological data lines if one or more of the following model options are selected, in which case meteorological data lines with wind speed at 10 m less than 0.75 m/s will not be modelled:

- **Puff**;
- **Fluctuations**;
- **Radioactive decay**;
- **Coastline**; and
- **Wind turbines**.

If the complex terrain module is used in conjunction with the calm conditions option, then for meteorological data lines with wind speed at 10 m greater than 0.75 m/s the complex terrain module will be used. For meteorological data lines with wind speed at 10 m less than 0.75 m/s a flat terrain calm conditions solution will be used.

4.19 Marine boundary layer

ADMS 5 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 emissions from stacks on oil extraction platforms. To use this option, all sources and receptors should be over the sea; it is not suitable for coastal modelling. Technical details about this option are given in Section 9.9.

Points to note about this scheme:

1. ADMS 5 will not use the surface roughness value at the dispersion site specified in the interface, as the surface roughness over the sea will be calculated from the wind speed. However, if a dispersion site surface roughness is entered in the meteorological data (*.met*) file this will be used.
2. It is recommended that meteorological data from a marine site be used. However, if marine *wind* data are unavailable, wind data from a site on land can be used. The surface roughness at the met. site should either be entered on the **Meteorology** screen or entered via the meteorological data file. The surface roughness at the dispersion site (i.e. over the sea) will still be calculated from the wind data. Note that it is still important that the temperature data (and humidity data if used) are from a marine site.

*If the marine boundary layer option is used and the surface roughness at the met. site is input either on the **Meteorology** screen or via the meteorological data file, it will be assumed that the met. site is on land and the dispersion site is at sea. These options should **not** be used if the reverse is true, i.e. met. site at sea and dispersion site on land.*

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:

Meteorological data requirements

- Wind speed and direction

  Specify the height of the wind measurements above mean sea level on the **Meteorology** screen.

- Temperature (ºC)

  Users are strongly recommended to include temperature data in runs using the marine boundary layer option. However, if no temperature data are available, then users should select the option to assume that the marine boundary layer is neutral (refer to the section concerning the additional input file below).

- Sea surface temperature (ºC)

Additional input file

To use the marine boundary layer option, create a new *.aai* file or edit the existing *.aai* file, refer to Section 3.1.6, so the **Marine boundary layer** (keyword MARINE) option is enabled (**Figure 4.50**).

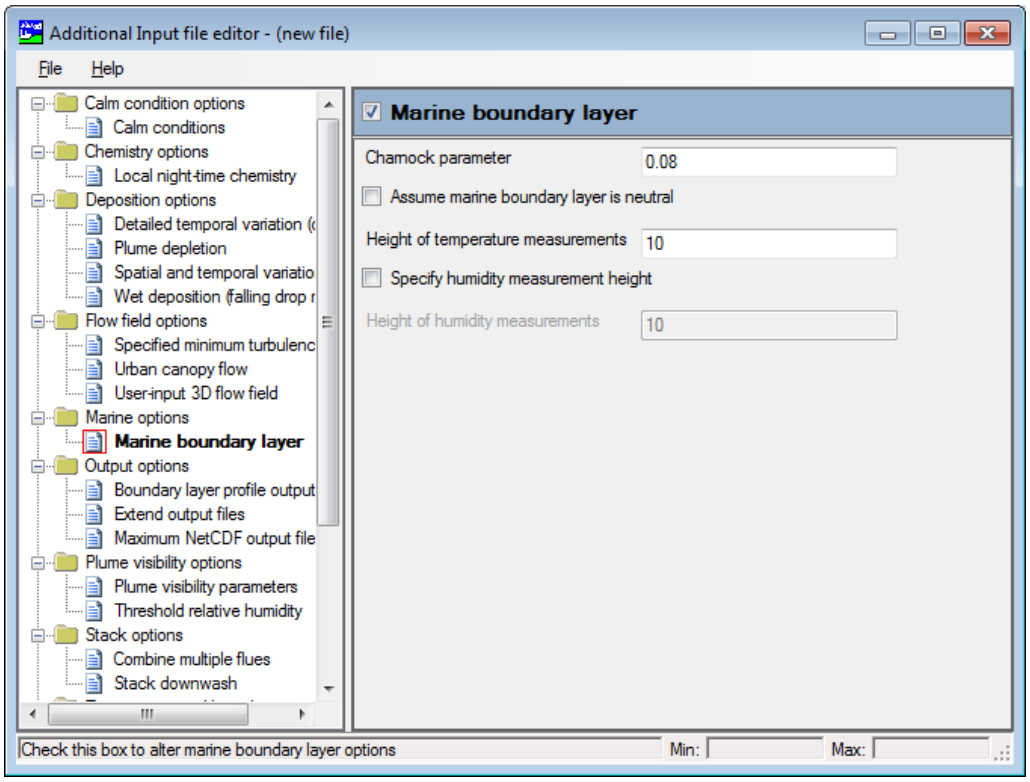


Figure 4.50 – **Marine boundary layer** option enabled in the **Additional Input file editor**.

For the marine boundary layer enter the following information:

- **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 5 for regions not altogether remote from land (e.g. North Sea).

- **Assume marine boundary layer is neutral**, this option should be checked if no temperature measurements are available
- **Height of temperature measurements** (in metres), if temperature data are included in run
- **Specify humidity measurement height**, if humidity data are included in the run the measurement height for the humidity measurements needs to be specified. Check this option and specify the height of the humidity measurements (in metres) in **Height of humidity measurements.**

4.20 Combine flues into a single stack

If there are emissions from multiple flues within the same stack, the plumes will in effect act as a single plume with combined source characteristics. When modelling such situations the data for the individual flues can be entered as individual point sources, and the model set to calculate the combined source parameters and model all the flues in the stack together as a single source. Full details of how the combined source parameters are calculated from the individual sources are given in Section 9.6.

Additional input file

To combine multiple flues into a single stack, a new *.aai* file should be created or the existing *.aai* file edited, refer to Section 3.1.6, so the **Combine multiple flues into a single stack** option (keyword COMBINEFLUES) is enabled (**Figure 4.51**).

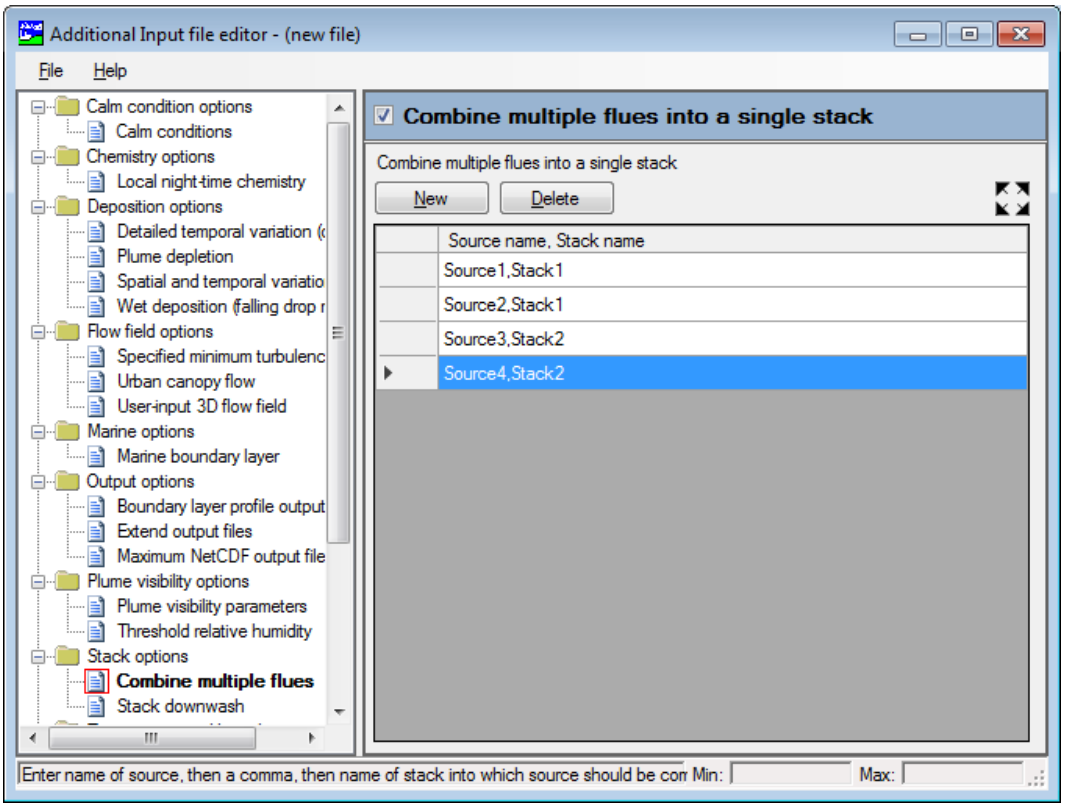


Figure 4.51 – Combine multiple flues into a single stack option enabled in the **Additional Input file editor**.

Into the table enter pairs of 'Source, Stack' where Source is the source name and Stack is the stack name. **Figure 4.51** shows an example where at least four sources have been entered into the interface 'Source1', 'Source2', 'Source3' and 'Source4'. Source1 and Source2 are to be combined as 'Stack1'; Source3 and Source4 are to be combined as 'Stack2'. There may be other sources in the interface; these will be modelled in the usual way.

Restrictions

The following restrictions apply to the sources which can be combined using this option:

- Only point sources can be combined.
- The sources in a particular stack must be of the same height.
- The sources in a particular stack should be located very close to each other. A warning is issued if the sources are more than 20 m apart and the model will stop if they are more than 50 m apart.
- There are no restrictions on the time-variation of the sources within a stack; none, some or all of the sources in an individual stack may have time-varying emissions and/or source parameters.
- If a group is selected for output, then if one source from a particular stack is in that group, all of the sources in that stack must be in that group.
- Single source output cannot be selected for a source that is within a stack.
- If buildings are being modelled, then all of the sources in a particular stack must have the same main building selected.
- The total number of sources that can be entered is the same regardless of whether any of the sources are to be combined into a stack.

4.21 Switch off stack downwash

For point sources, by default ADMS 5 incorporates the reduction in the mean height of the plume just downwind of the source caused by the flow and pressure fields around the stack, refer to Section 9.4 for more details. In some cases this stack-induced downwash may not be appropriate, it is therefore possible to turn off the stack-induced downwash for selected sources.

To turn off the stack-induced downwash for selected sources, create a new *.aai* file or edit the existing *.aai* file, as described in Section 3.1.6, so the **Stack downwash** option (keyword NOSTACKDOWNWASH) is enabled (shown in **Figure 4.52**). Enter the names of the sources for which stack downwash should not be modelled into the table.

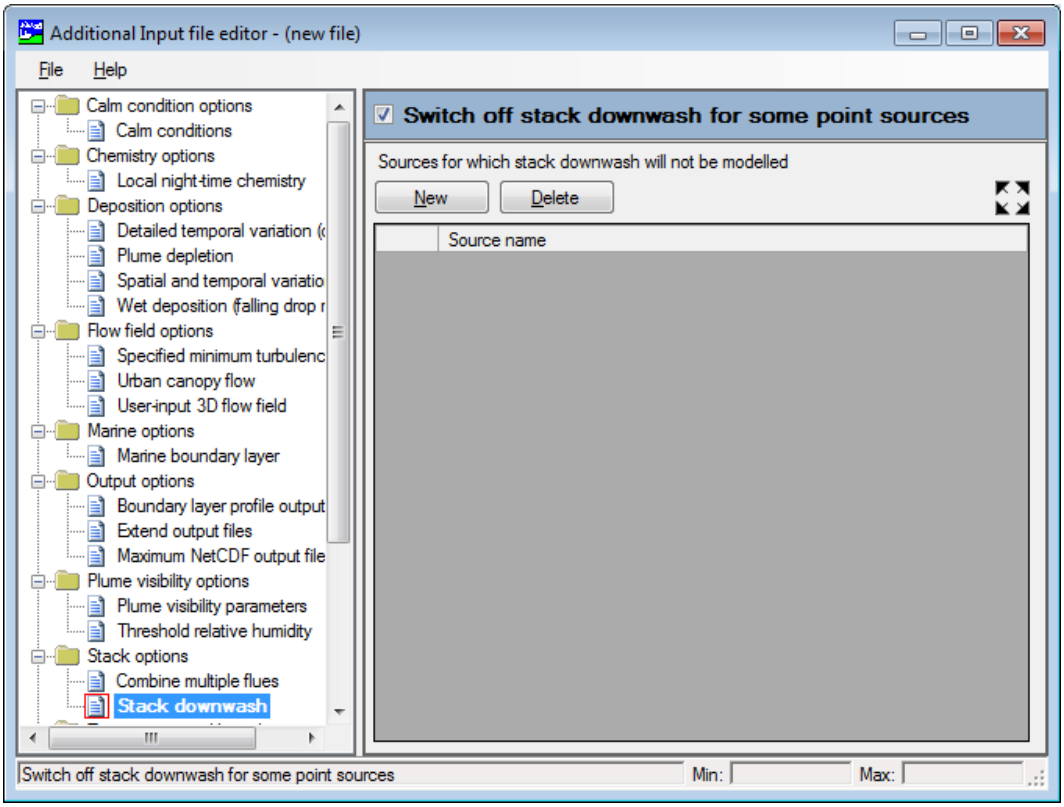


Figure 4.52 – The **Stack downwash** option, which is used to disable stack downwash for certain sources, in the **Additional Input file editor**.

4.22 User-defined minimum turbulence

As part of the dispersion calculations, a minimum turbulence value is applied. This minimum turbulence value varies between 0.01 and 0.2 m/s depending on the minimum value of the Monin-Obukhov length specified in the **Advanced meteorological** screen, as described in Section 4.15. Additionally, a further minimum value of 0.1 m/s is applied if variable terrain height is being modelled.

An option exists to specify the minimum value of the turbulence parameters. First create a new *.aai* file or edit the existing *.aai* file, as described in Section 3.1.6, so the **Specified minimum turbulence** option (keyword DEFINEMINTURBULENCE) is enabled (**Figure 4.53**). Then enter the minimum value to be used for the turbulence parameters in m/s.

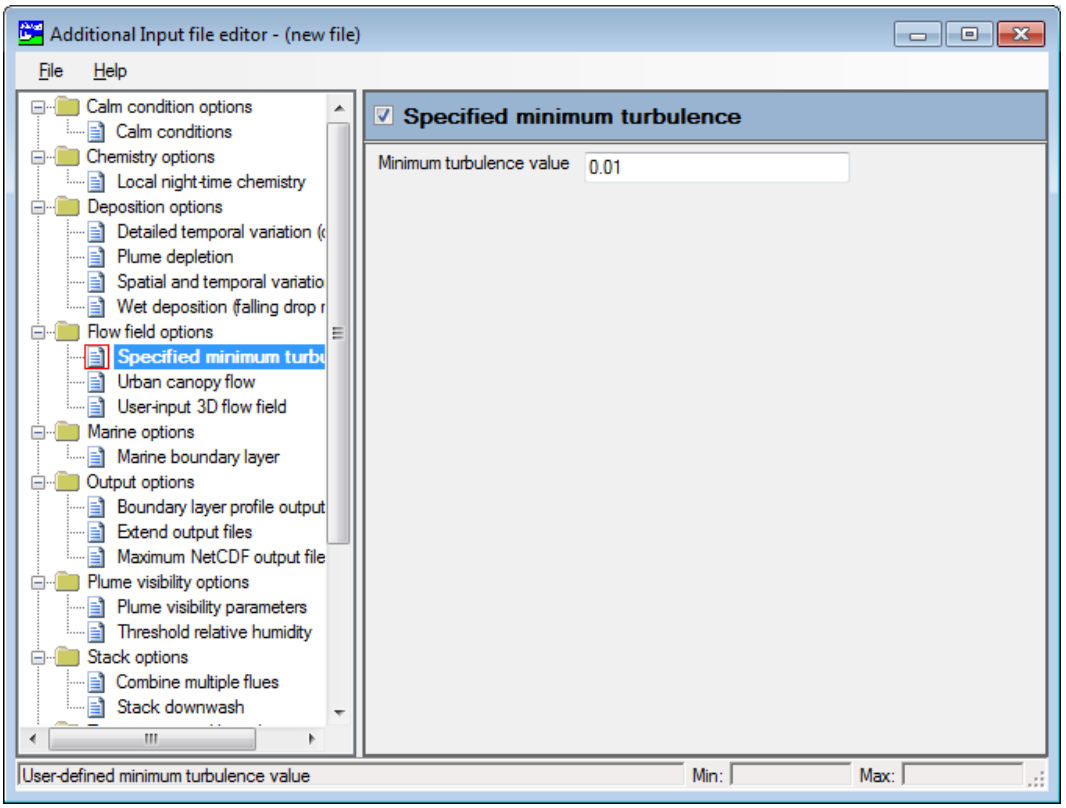


Figure 4.53 – The **Specified minimum turbulence** section enabled in the **Additional Input file editor**.

4.23 Wind turbine effects

ADMS 5 includes the ability to model the effects of wind turbines on plume dispersion. The model calculates changes in the flow field due to the wind turbines and this modified flow field affects the plume dispersion. If several turbines are modelled then the cumulative effect of each turbine on the flow field is taken into account. Full details on the scientific methodology used can be found in the Technical Summary in Section 9.23.

To use the wind turbine effects option, create a new *.aai* file or edit the existing *.aai* file, as described in Section 3.1.6, so the **Wind turbine types** (keyword WINDTURBINETYPES) and **Wind farm** (keyword WINDFARM) options are enabled (see **Figure 4.54** and **Figure 4.55**).

*Both the **Wind turbine types** and **Wind farm** sections of the* .aai *file must be included when modelling wind turbine effects.*

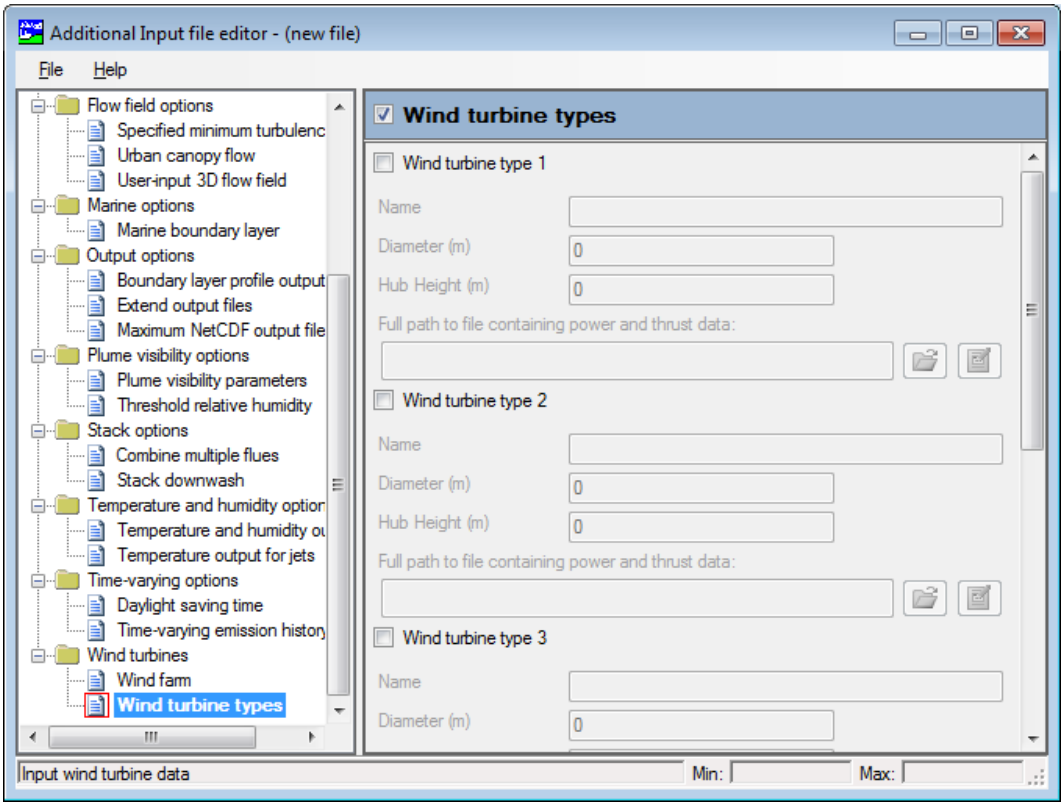


Figure 4.54 – The **Wind turbine types** section which is required when modelling the effects of wind turbines on dispersion.

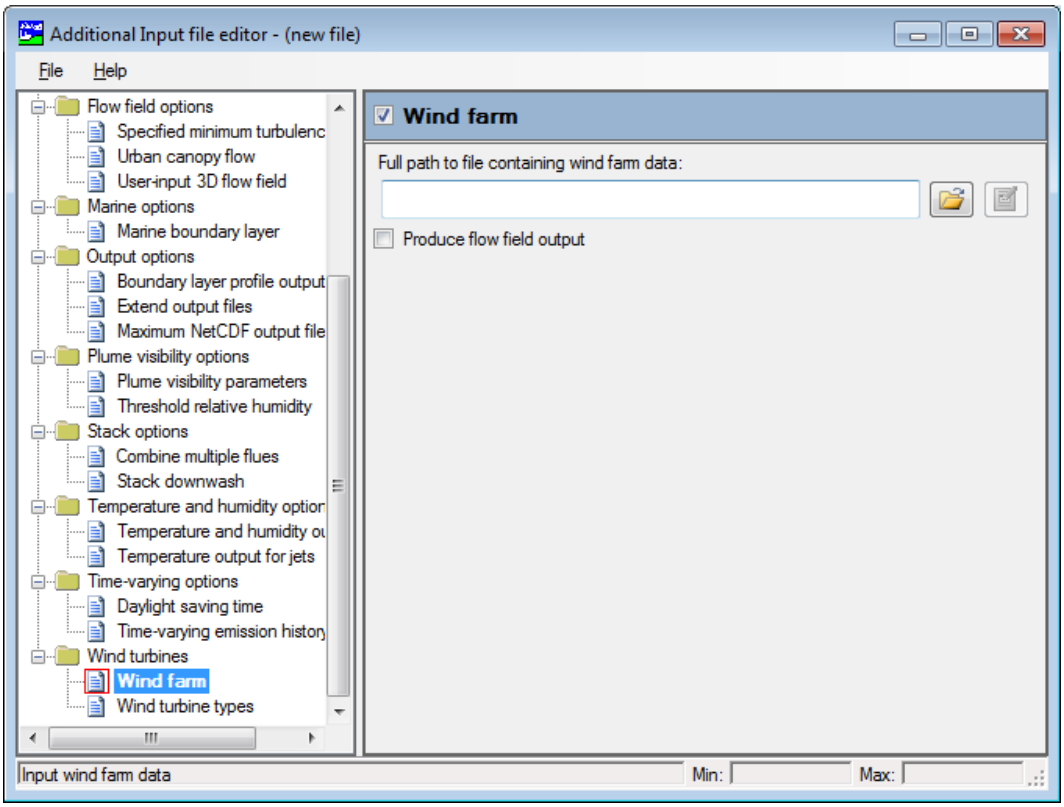


Figure 4.55 – The **Wind farm** option which is also required when modelling the effect of wind turbines on dispersion.

For the **Wind turbine types** section first click on the check box next to **Wind turbine type 1** so that information can be entered. Then specify the following information:

- **Name**: this is the name of the make, or type, of turbine. This name is referenced in the wind farm data file, described in Section 4.23.2.

- **Diameter (m)**: this is the diameter of the turbine rotor in metres

- **Hub Height (m)**: this is the hub height of the turbine in metres. The hub height is measured from ground level to the turbine hub.

- **File path to file containing power and thrust data**: enter the file path to the wind turbine data (*.wtd*) file. This file is used to input values of the turbine power and turbine thrust as functions of wind speed for the different makes or types of wind turbines that are included in the model. A full description of the file is given in Section 4.23.1.

These steps may be repeated so that information for up to 5 types of turbines can be included in the model.

For the **Wind farm** section enter the following information:

- **File path to file containing wind farm data**: enter the file path to the wind farm data (*.wfd*) file. This is used to specify the locations of the turbines within the wind farm and the type of turbine at each location. A full description of the file is given in Section 4.23.2.

- **Produce flow field output**: check this box if you wish to obtain flow field output files, refer to Sections 6.1.13 to 6.1.18 and Section 6.3.2 for details.

4.23.1 Wind turbine data file

The wind turbine data (*.wtd*) file is used to provide the model with the power and thrust coefficients for a particular type of turbine as functions of wind speed. This data is usually available from the turbine manufacturer. The *.wtd* file must be created by the user, and this can be done in a text editor, such as Notepad, or in a spreadsheet package such as Excel. An example file illustrating the required format is shown in **Figure 4.56**.

Each line of data in the file includes:

- `U_HUB`: the wind speed at the turbine hub height in metres per second
- `CT`: the thrust coefficient for the turbine at the corresponding wind speed

To create the file, follow these steps:

- Add the version string `WTDFileVersion2` as the first line in the file
- Create the "`VARIABLES:`" section

  This is used to list the variables that are included in the file.

  * On the second line type `VARIABLES:`
  * On the third line type `2`

    This indicates that there are values for two variables in the file.
  * On lines four to five list the variable names `U_HUB`, `CT`

    These can appear in any order, but the data in the "`DATA:`" section must match this order.
- Create the "`DATA:`" section.
  * On a new line type `DATA:`
  * After the "`DATA:`" keyword enter a line of data giving the wind speed and the corresponding thrust coefficient at that wind speed. The order of these variables must match the order specified in the "`VARIABLES:`" section.
  * Add an additional line of data for each wind speed.

A *.wtd* file must be provided for each type of turbine defined in the model. Note that the *.wtd* file defines the range of wind speeds for which the wind turbine can be operational. If the hub height wind speed for a given met line is outside this range then

the wind turbine will not be operational and therefore the wind turbine will be considered to have no effect on dispersion.

An example .wtd *file is included in the* \Data *folder in the install directory. This file may be edited to create new* .wtd *files.*

If you have created your .wtd *files with a spreadsheet package, it is always advisable to check the file format in a text editor afterwards to see if it is correct. In particular, trailing commas may need to be removed from the "*`VARIABLES:`*" section.*

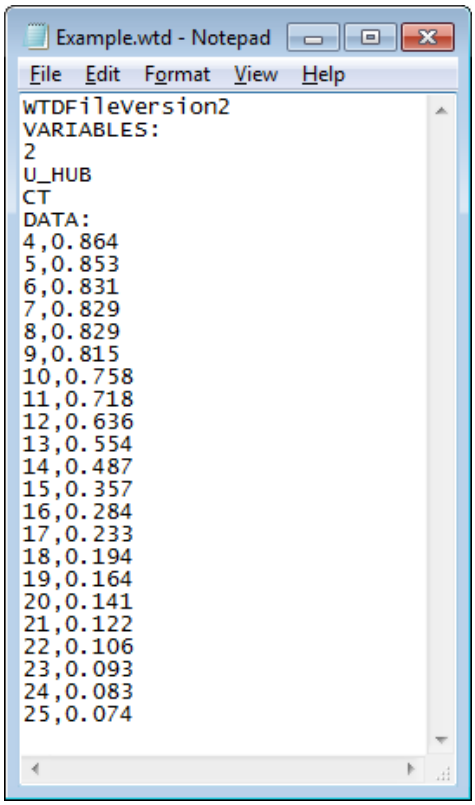


Figure 4.56 – Example *.wtd* file

4.23.2 Wind farm data file

The wind farm data (*.wfd*) file is used to input the locations of each turbine and the make, or type, of turbine at each location. The file can be created in a text editor, such as Notepad, and then saved with the extension *.wfd*; or it could be created in a spreadsheet package such as Excel. If the latter is done the file must be saved in comma-separated format (*.csv*) but the file extension must be changed to *.wfd*. An example file is shown in **Figure 4.57**.

Each line of data in the wind farm data file includes:

- `TURBINE NAME`: the name of the turbine at the location, e.g. "Turbine 1".
- `TURBINE TYPE`: the name of the make or type of turbine at the location. This

must match the name of one of the types specified in the **Wind turbine types** section in the advanced file editor.

- X: the easting of the turbine location in metres.
- Y: the northing of the turbine location in metres.

The coordinate system used when specifying the turbine locations must be the same as that used in the rest of the .apl *file.*

To create the file, follow these steps:

- Create the "VARIABLES:" section

  This is used to list the variables that are included in the file.

  * On the first line type VARIABLES:
  * On the second line type 4

    This indicates that there are values for four variables in the file.
  * On lines three to six list the variable names TURBINE NAME, TURBINE TYPE, X, Y

    These can appear in any order, but the data in the "DATA:" section must match this order.

- Create the "DATA:" section.
  * On a new line type DATA:
  * After the "DATA:" keyword enter a line of data that gives the turbine name, turbine type, easting coordinate and northing coordinate for each turbine in the wind farm. The variables must appear in the order specified in the "VARIABLES:" section.
  * Add an additional line of data for each turbine.

An example .wfd *file is included in the* \Data *folder in the install directory. This file may be edited to create new* .wfd *files.*

If you have created your .wfd *files with a spreadsheet package, it is always advisable to check the file format in a text editor afterwards to see if it is correct. In particular, trailing commas may need to be removed from the "*VARIABLES:*" section.*

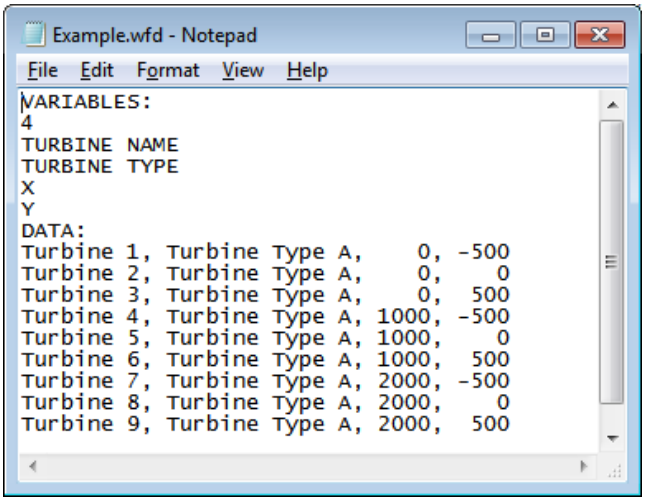


Figure 4.57 – Example *.wfd* file.

4.23.3 Wind turbine output

If the option to produce flow field output has been selected in the **Wind farm** section of the **Additional Input file editor** then additional output files will be produced giving the flow field and turbulence field. These files are described in Sections 6.1.13 to 6.1.18. Otherwise there are no additional output files when modelling wind turbines, but the output concentrations will take into account the effects of the turbines.

4.23.4 Restrictions

The **Puff**, **Coastline**, **Gamma dose**, **Calm conditions**, **Urban canopy** and **User-input 3D flow field** model options cannot be used in conjunction with the wind turbines option.

Jet sources cannot be modelled with wind turbines.

If the meteorological wind direction data is in sectors then the sector size must be less than 15°.

4.24 Urban canopy flow

The Urban canopy flow option in ADMS 5 calculates the changes in the vertical profiles of velocity and turbulence caused by the presence of buildings in an urban area. It is generally accepted that as wind approaches a built-up urban area, the profile is displaced vertically by a height related to the mean height of the buildings, while the flow within the building canopy is slowed by the buildings (Belcher *et al.* 2013).

To use the urban canopy flow field in a run, first create a *.aai* file or edit the existing *.aai* file (as described in Section 3.1.6) so that the **Urban canopy flow** option (keyword URBANCANOPYFLOW) is enabled, as shown in **Figure 4.58**.

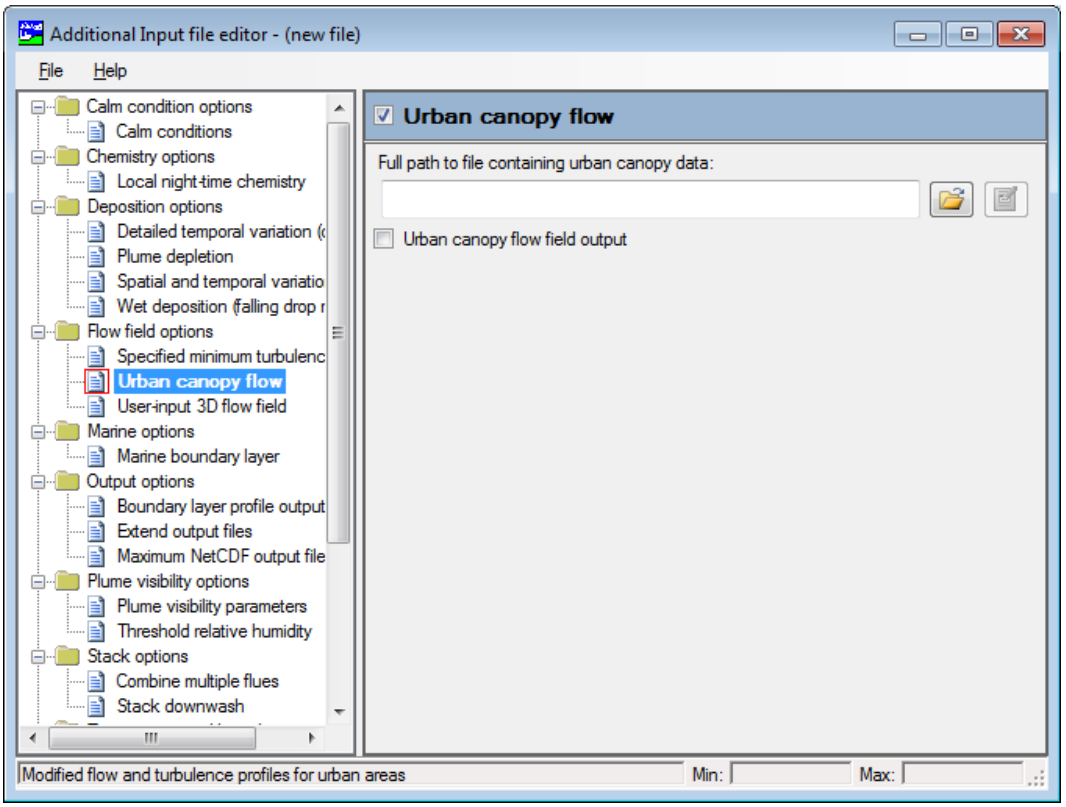


Figure 4.58 – The **Urban canopy flow** section enabled in the **Additional Input file editor**.

The user must specify:

- **Full path to file containing urban canopy data**: Use the button to browse to an input file containing gridded urban canopy data, please refer to Section 4.24.1 for details of the input file format and contents. Use the button to view the file in the default viewer selected in the **File\Preferences...\Viewing** menu option of the **Additional input file editor**;
- **Urban canopy flow field output**: check this box if you wish to obtain flow field output files, refer to Sections 6.1.13 to 6.1.18 and Section 6.3.2 for details.

4.24.1 Urban canopy flow input data file format

The structure of the input data file for the urban canopy flow option is similar to the ADMS meteorological file format, with a list of variable names followed by a table of data. At the beginning of the data file, an additional section specifies the grid layout of the urban canopy data. An example data file is shown in **Figure 4.59**.

CERC have developed an ArcGIS tool to create urban canopy data files from a buildings data file, with a choice of grid layout and lambdaF wind directions. Please refer to the Urban Canopy Tool User Guide, which can be found in the *Support\ArcGIS Python Tools\Urban Canopy* directory of your ADMS 5 installation, for more details.

The input variables used to determine the urban canopy flow parameters in each cell of a regular x, y urban canopy grid are as follows:

- `H` average building height within the cell
- `G` average street canyon width within the cell
- `LambdaP` ratio of plan area occupied by buildings to total plan area within the cell (must be between 0 and 1)
- `LambdaF phi1 to phi2` ratio of total frontal area of buildings perpendicular to a wind direction to total plan area within the cell (may be greater than 1), for a range of wind directions `phi1` to `phi2` in degrees

There must be one line of data for each urban canopy grid cell. If there are no buildings within a particular urban canopy grid cell, the average building height and value of lambdaP will be zero and the urban canopy flow field calculations will not be run, so upstream flow parameters will be used in this cell.

The LambdaF angle thresholds have the following restrictions:

- The upper ('to') threshold of one sector must be equal to the lower ('from') threshold of the next (sectors must be in order).
- The first 'from' threshold must be equal to the last 'to' threshold, to describe a complete range of wind directions. Wind directions will be considered to belong to the sector where they are greater than or equal to the 'from' threshold and less than the 'to' threshold.
- Each threshold value must be greater than or equal to 0 and less than 360 degrees.
- Exactly one sector may have a 'from' threshold greater than or equal to its 'to' threshold (crossing 0 degrees).
- The number of sectors must be between one and forty.

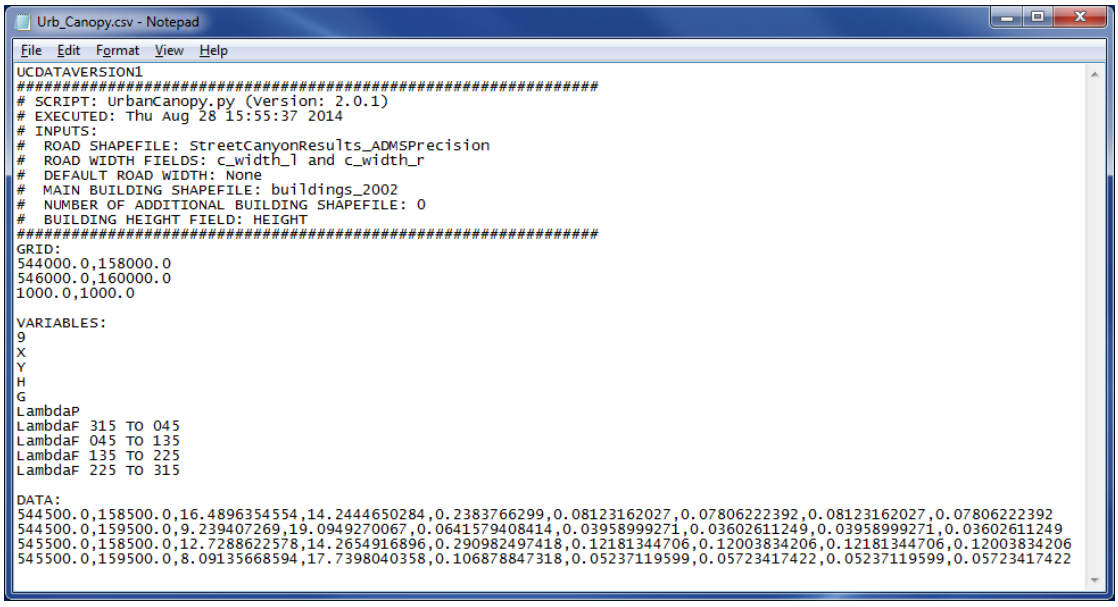


Figure 4.59 Example Urban canopy data input file

The file format is as follows, where all starred items are required:

- 'UCDATAVERSION1' Version string, must be the first line in the file
- (Comments may be included before the first keyword)
- Grid information section
  - 'GRID:' Keyword
  - XMIN, YMIN Minimum x,y coordinates of the urban canopy grid in units of metres, in a coordinate system consistent with all other spatial inputs to the ADMS 5 run
  - XMAX, YMAX Maximum x,y coordinates of the urban canopy grid in units of metres, in a coordinate system consistent with all other spatial inputs to the ADMS 5 run
  - XSPACING, YSPACING The resolution of the urban canopy grid in metres
- (Comments may be added between sections)
- Variable information section
  - 'VARIABLES:' Keyword
  - nVar number of variables included in the file
  - List of nVar variable names, each on a separate line: must include X, Y, G, H, LambdaP, and at least one sector of LambdaF, may include other variables eg. labels. Refer to additional information above about definitions of variables and restrictions on LambdaF sectors
- (Comments may be added between sections)
- Data section
  - 'DATA:' Keyword

* One line per urban canopy grid cell of comma-separated data values for nVar variables, in the order listed in the Variable information section

Blank lines or comments may be included before the first section, between sections or after the last section. There should be no blank lines or comments within each section.

4.24.2 Restrictions

The Urban canopy flow field option cannot be run in combination with the **Buildings**, **Complex terrain**, **Plume visibility**, **Coastline**, **Puffs**, **Fluctuations**, **Gamma dose**, **Marine boundary layer**, **Temperature and humidity output** and **Wind turbine** options. The Urban canopy flow field option cannot be used with jet sources.

4.25 Extend output files

The output data in the short-term *.bld*, *.dep*, *.gst*, and *.levels.gst* files are usually restricted to the first 24 lines of meteorological data in order to prevent the output files from becoming too large. This limit can be removed using an *.aai* file option.

To extend the short-term output files for a run, first create a *.aai* file or edit the existing *.aai* file (described in Section 3.1.6) so that the **Extend output files** option (keyword LONGOP) is enabled, as shown in **Figure 4.60**.

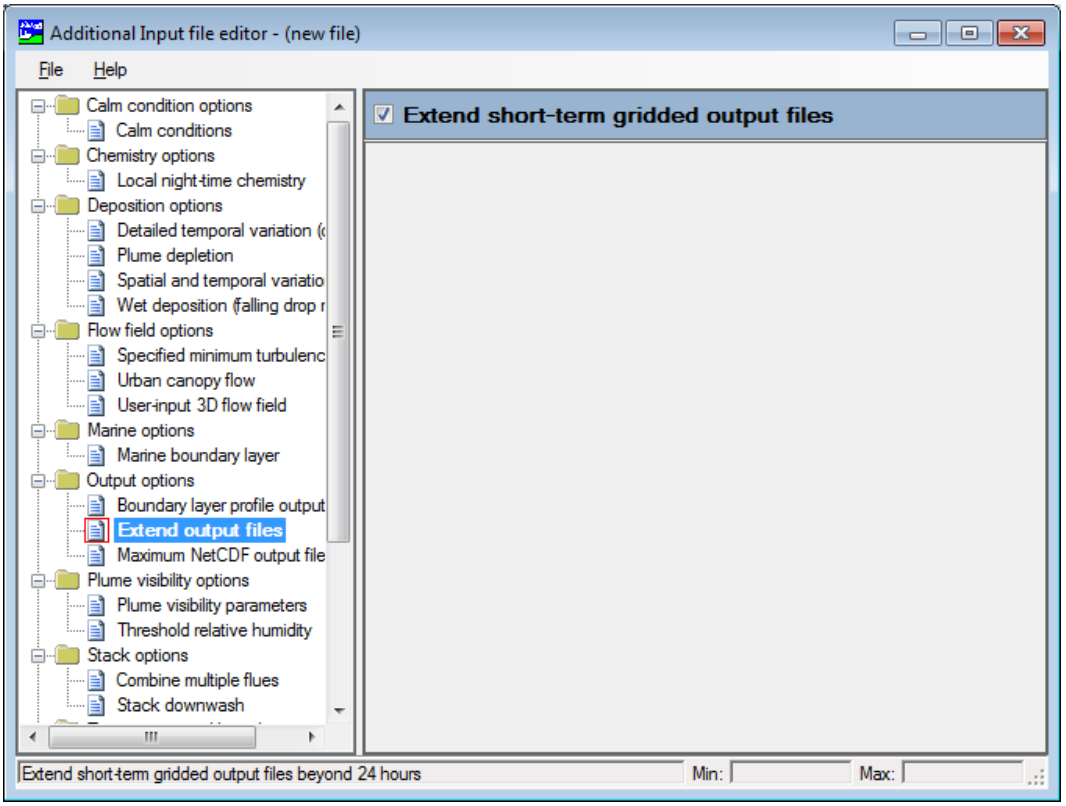


Figure 4.60 The **Extend output files** option enabled in the **Additional Input file editor**

BEWARE: If running a large number of lines of meteorological data (for example one year of data), the output files may become very large, particularly the .gst *and* .levels.gst *files.*

Data for every hour of a long-term run can be obtained in a more compact format using the Comprehensive Output File option, described in Section 3.6.3.

SECTION 5 Import and Export

Sections 3 and 4 of this User Guide have described how to set up ADMS 5 model runs when there is a small amount of source data involved by typing data into boxes in the interface. However, if you have a large amount of source data, entering data in this manner can be time-consuming. ADMS 5 has an import facility allowing for source data to be imported into the ADMS 5 interface from a set of comma-separated variable files. In addition to source data, pollutant and buildings data can also be imported in this way. An export facility is also provided, allowing source, pollutant and buildings data to be exported from the ADMS 5 interface to comma-separated variable files.

A description of the files used with the import and export facility is given in Section 5.1. Section 5.2 provides details on importing data and Section 5.3 on exporting data.

5.1 File formats

The import and export facility uses a set of comma-separated files each containing different aspects of the source, pollutant and buildings data. Each of these files has the same file stem and an extension indicating its contents:

- The *.spt* file contains the main properties of the sources including the efflux parameters and the source type, i.e. most of the information entered into the **Source** screen of the ADMS 5 interface.
- The *.vgt* file contains the vertex information for all sources.
- The *.eit* file contains the pollutant emissions for all sources.
- The *.tft* file contains the traffic flow information for road sources. This file is not used by ADMS 5.
- The *.gpt* file contains the group membership information for all sources.
- The *.ptt* file contains information about pollutant properties, i.e. the information contained in the **Palette of pollutants**.
- The *.bpt* file contains information about buildings.

Each of these files consists of a version string followed by a header row and then the data. The columns of data can be listed in any order and additional columns can be entered but will not be used during the import process. Full descriptions of the formats of each of these files are given in Sections 5.1.1 to 5.1.6. A facility to create a set of templates for these files is described in Section 5.1.7.

5.1.1 .spt file

The *.spt* file contains information about the sources, including the source type and efflux properties. The first line of the *.spt* file must contain the version information `SPTVersion1`. The header line details and data required are described in **Table 5.1**. A separate row of data must be entered for each source.

Many of the parameters in the .spt file only apply for certain source types, or if certain model options are in use. For parameters which are not required fill in 0 or n/a.

Header	Description
`Source name`	Source name.
`Use VAR file`	Is a *.var* file is being used for this source? Enter the keyword `Yes` or `No`.
`Specific heat capacity (J/kg/K)`	Specific heat capacity of the release in Joules per kilogram per Kelvin. The ADMS 5 default value is 1012.
`Molecular mass (g)`	Molecular mass of the release in grams. The ADMS 5 default value is 28.966.
`Temperature or density?`	Is the release temperature or density specified, or is it at ambient conditions? Enter the keyword `Temperature`, `Density` or `Ambient`.
`Temperature (Degrees C) / Density (kg/m3)`	Temperature or density of the release in appropriate units.
`Actual or NTP?`	Is the efflux specified at the temperature or at NTP? Enter the keyword `Actual` or `NTP`.
`Efflux type keyword`	There are 4 options for specifying the efflux. Enter the keyword `Velocity`, `Volume`, `Momentum` or `Mass`.

`Velocity (m/s)`
`Volume flux (m3/s)`
`Momentum flux (m4/s2)`
`Mass flux (kg/s)`
	Efflux in appropriate units.

`Heat release rate (MW)`	Indication of buoyancy flux (Fb) in mega Watts if `Momentum` is selected as the efflux type.
`Source Type`	The type of the source. Enter the keyword `Point`, `Line`, `Area`, `Volume` or `Jet`.
`Height (m)`	Height of the source, in metres.
`Diameter (m)`	Diameter of a point or jet source, in metres.

`Line width (m) /`
`Volume depth (m) /`
`Road width (m) /`
`Grid depth (m)`
	Width of a line source, or vertical extent of a volume source, in metres.

`Canyon height (m)`	Not used in ADMS 5.
`Angle 1 (deg)`	Vertical jet angle, in degrees.
`Angle 2 (deg)`	Horizontal jet angle, in degrees.
`Mixing ratio (kg/kg)`	Mixing ratio of release, mass of water per unit mass of dry release, in kilogram of water per kilogram of dry air. Only used if plume visibility option is selected.
`Traffic flows used`	Not used in ADMS 5.
`Traffic flow year`	Not used in ADMS 5.
`Traffic flow road type`	Not used in ADMS 5, enter `n/a`.
`Main building`	The selection of main building. Enter the keyword `(Main)`, `(None)` or the name of a building.
`Comments`	Further comments may be added here, which will not be included in the *.apl* file.

Table 5.1 – Data to be entered in the *.spt* file.

5.1.2 .vgt file

The *.vgt* file contains vertex information for all of the sources. The first line of the *.vgt* file must contain the version information `VGTVersion2`. The header line details and data required are described in **Table 5.2**. For point and jet sources one vertex must be entered, for line sources exactly two vertices must be entered and for area and volume sources between three and 50 vertices describing a convex polygon can be entered. Vertex information for each source must be entered in order and the sources must be entered in the same order as the *.spt* file.

Header	Description
`Source name`	Source name
`X (m)`	X coordinate of the vertex, in metres
`Y (m)`	Y coordinate of the vertex, in metres

Table 5.2 – Data to be entered in the *.vgt* file.

5.1.3 .eit file

The *.eit* file contains pollutant emissions information for all sources. The first line of the *.eit* file must contain the version information `EITVersion1`. The header line details and data required are described in **Table 5.3**. Separate lines should be entered for each pollutant emitted from each source. Data for each source must be kept together and the sources must be entered in the same order as in the *.spt* file.

Header	Description
`Source name`	Source name
`Pollutant name`	Pollutant name
`Emission rate`	Emission rate of the pollutant in g/s for point sources, g/m/s for line sources, $g/m^2/s$ for area sources, $g/m^3/s$ for volume sources

Table 5.3 – Data to be entered in the *.eit* file.

5.1.4 .gpt file

The *.gpt* file contains group membership information for all sources. The first line of the *.gpt* file must contain the version information `GPTVersion1`. The header line details and data required are described in **Table 5.4**. Each group is represented by a column and one row of data should be entered for each source. Sources must be entered in the same order as in the *.spt* file.

Header	Description
`Source name`	Source name
`Source type`	Source type, either Point, Jet, Line, Area or Volume
`Group|<Group name>`	Is the source a member of this group? Enter the keyword `Yes` or `No`.

Table 5.4 – Data to be entered in the *.gpt* file. Where `<Group name>` should be replaced with the name of the group, so for a group called 'IndustrialSite' the header for that column would be `Group|IndustrialSite`.

5.1.5 .ptt file

The *.ptt* file contains property data for all pollutants and corresponds to data entered in the pollutant **Palette**, refer to Section 3.1.4 for further details. The first line of the *.ptt* file must contain the version information `PTTVersion1`. The header line details and data required are described in **Table 5.5**. For gaseous pollutants a single line of data must be entered for each pollutant, for particulate pollutants separate lines of data should be entered for each particulate component.

Header	Description
`Pollutant name`	Name of the pollutant.
`Pollutant type`	Type of pollutant. Enter the keyword `Gas` or `Particulate`.
`Deposition vel. known`	Is the deposition velocity known? Enter the keyword `Yes` or `No`.
`Terminal vel. known`	For particulate pollutants, is the terminal velocity known? Enter the keyword `Yes` or `No`.
`Washout coeff known`	Is the washout coefficient known? Enter the keyword `Yes` or `No`.
`Washout coeff.`	Washout coefficient independent of precipitation rate.
`Washout coeff. A`	Parameter A in expression for washout coefficient dependent on precipitation rate.
`Washout coeff. B`	Parameter B in expression for washout coefficient dependent on precipitation rate.
`Conversion factor (ug/m3 -> ppb)`	For gaseous pollutants, the conversion factor from micrograms per metre cubed to parts per billion by volume (ppb).
`Nature of gas`	For gaseous pollutants, the nature of the gas. Enter the keyword `Non-reactive`, `Reactive` or `Inert`.
`Deposition velocity (m/s)`	Rate at which material is deposited to the ground due to diffusion in metres per second.
`Terminal velocity (m/s)`	For particulate pollutants, the rate at which material is deposited to the ground due to gravitational settling in metres per second.
`Particle diameter (m)`	For particulate pollutants, the diameter of the particles in metres.
`Particle density (kg/m3)`	For particulate pollutants, the density of the particles in kilograms per metre cubed.
`Mass fraction`	For particulate pollutants, the mass fraction of this diameter/density combination. The total mass fraction for each pollutant must add up to 1.

Table 5.5 – Data to be entered in the *.ptt* file.

5.1.6 .bpt file

The *.bpt* file contains data for all buildings, refer to Section 4.9 for further details on the buildings option. The first line of the *.bpt* file must contain the version information `BPTVersion1`. The header line details and data required are described in **Table 5.6**. Each building is represented by a separate row of data.

Header	Description
`Name`	Building name.
`Shape`	Shape of the buildings. Enter the keyword `Rectangular` or `Circular`.
`X (m)`	X coordinate of the building centre, in metres.
`Y (m)`	Y coordinate of the building centre, in metres.
`Height (m)`	Height of the building, in metres.
`Length (m) / Diameter (m)`	Length of a rectangular building or diameter of a circular building, in metres.
`Width (m)`	Width of a rectangular building, in metres.
`Angle (degrees)`	Angle the length of a rectangular building makes with north, measured clockwise in degrees.

Table 5.6 – Data to be entered in the *.bpt* file.

5.1.7 Import templates

A set of blank template files can be created by selecting **Create import templates** from the **Utilities** menu of the ADMS 5 interface. This option creates a blank set of template files with the appropriate headers in a location of the user's choosing. These files can be edited in a spreadsheet package and then imported into ADMS 5 using the **Import** option from the **File** menu.

5.2 Import

To import source, pollutant and/or buildings data from a set of *.csv* files, select **Import** from the **File** menu to start the **Import wizard**, shown in **Figure 5.1**. The **Import wizard** consists of a series of screens guiding you through the import process, some of these screens only appear if certain import options are selected. Navigate through the screens using the **Next >** and **< Previous** buttons. Sections 5.2.1 to 5.2.8 describe each of the screens of the **Import wizard**.

5.2.1 Select files to import

The first screen of the **Import wizard**, shown in **Figure 5.1**, is used to select the files to be imported.

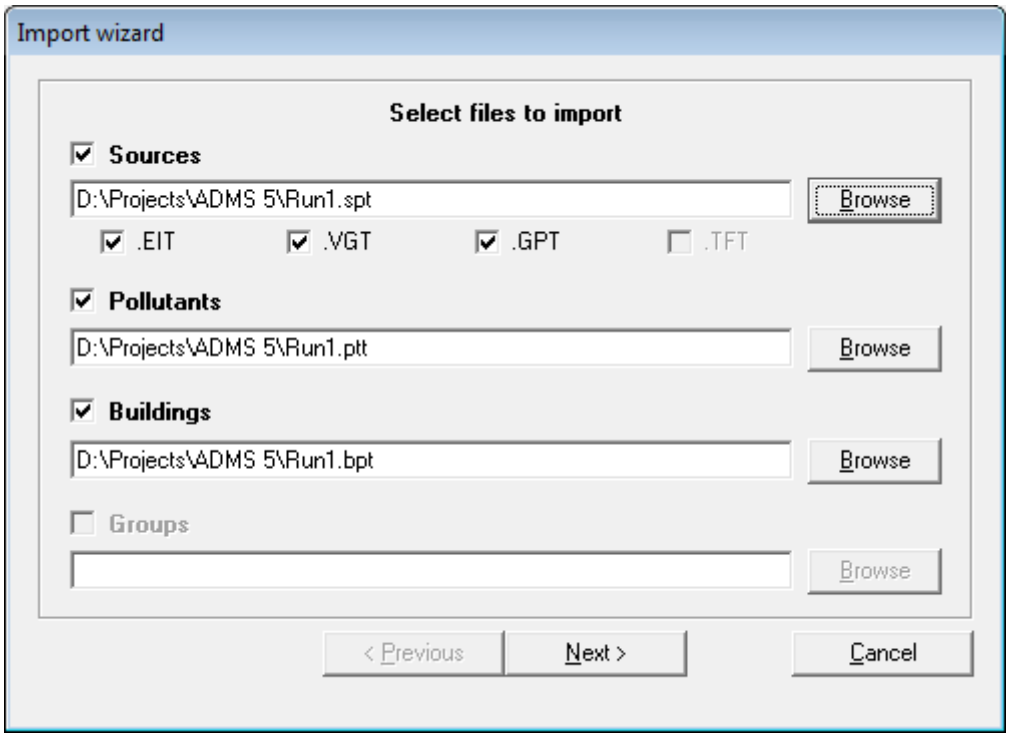


Figure 5.1 –Select files to import screen of the **Import wizard**.

There are four main file types which can be selected for import:

- **Sources** – select this option to import sources. When an *.spt* file is selected the other files corresponding to this are also automatically selected for import. The presence of an *.eit*, *.vgt* or *.gpt* file is indicated by the check boxes below, these can be deselected to prevent importing the data from that file.
- **Pollutants** – select this option to import pollutants.
- **Buildings** – select this option to import buildings. The **Buildings** option is only available if **Buildings** is selected in the **Setup** screen of the ADMS 5 interface.
- **Groups** – select this option to import group membership information for sources already defined in the ADMS 5 interface. This option is only available if **Sources** is not selected. If **Sources** is selected the availability of group

membership information for the sources being imported is indicated by the **.GPT** checkbox.

Once the files to be imported have been selected click **Next >** to move onto the next screen of the **Import wizard**.

5.2.2 Filter sources by type

If sources are selected for import the next screen of the **Import wizard** is the **Filter sources by type** screen, shown in **Figure 5.2**.

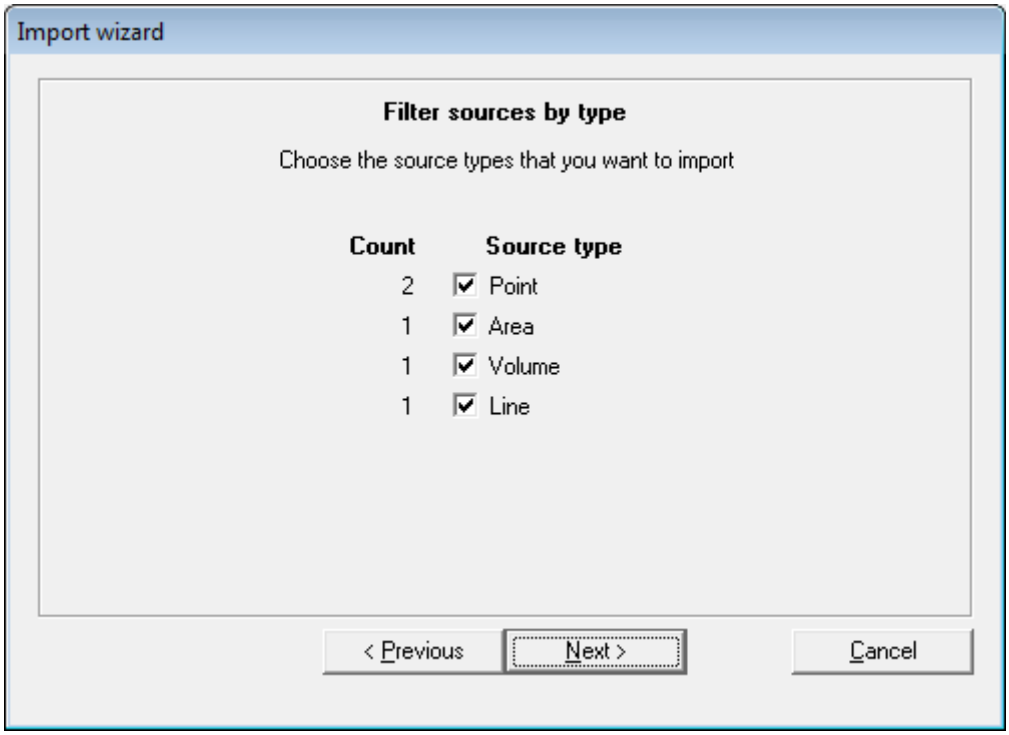


Figure 5.2 – **Filter sources by type** screen of the **Import wizard**.

This screen shows how many sources of each source type have been found in the *.spt* file. Deselect the checkbox against a source type if no sources of that type are to be imported. Source types which are greyed out cannot be imported into ADMS 5.

Once the required source types are selected click **Next >** to move to the **Select sources** screen.

5.2.3 Select sources

The **Select sources** screen allows for selection of individual sources to be imported, shown in **Figure 5.3**.

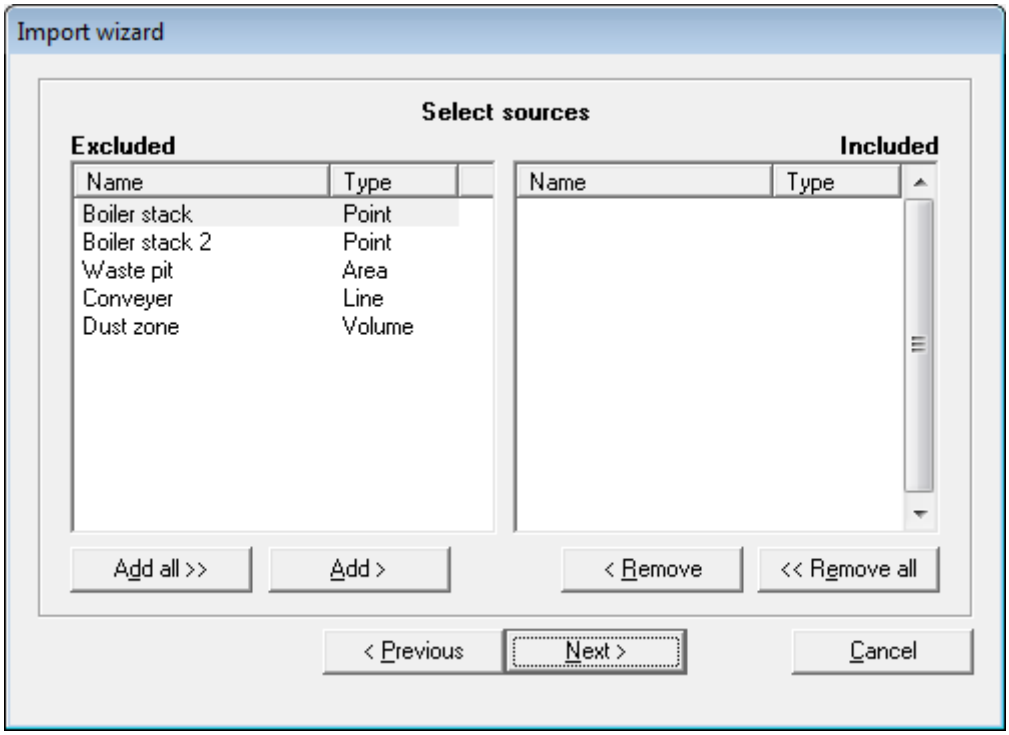


Figure 5.3 – **Select sources** screen of the **Import wizard**.

Sources listed on the right hand side will be imported whereas sources on the left will not be imported. The **Add >** and **Add All >>** buttons can be used to include sources in the import, the **< Remove** and **<< Remove All** buttons can be used to exclude sources from the import. Sources can also be **Included**/**Excluded** for import by double-clicking on the source name or dragging them to the appropriate side.

Once the required sources are selected for import, click the **Next >** button.

5.2.4 Source settings

If sources are being imported and some of the sources selected for import have the same name as sources already defined in the ADMS 5 interface the **Source settings** screen will appear, shown in **Figure 5.4**.

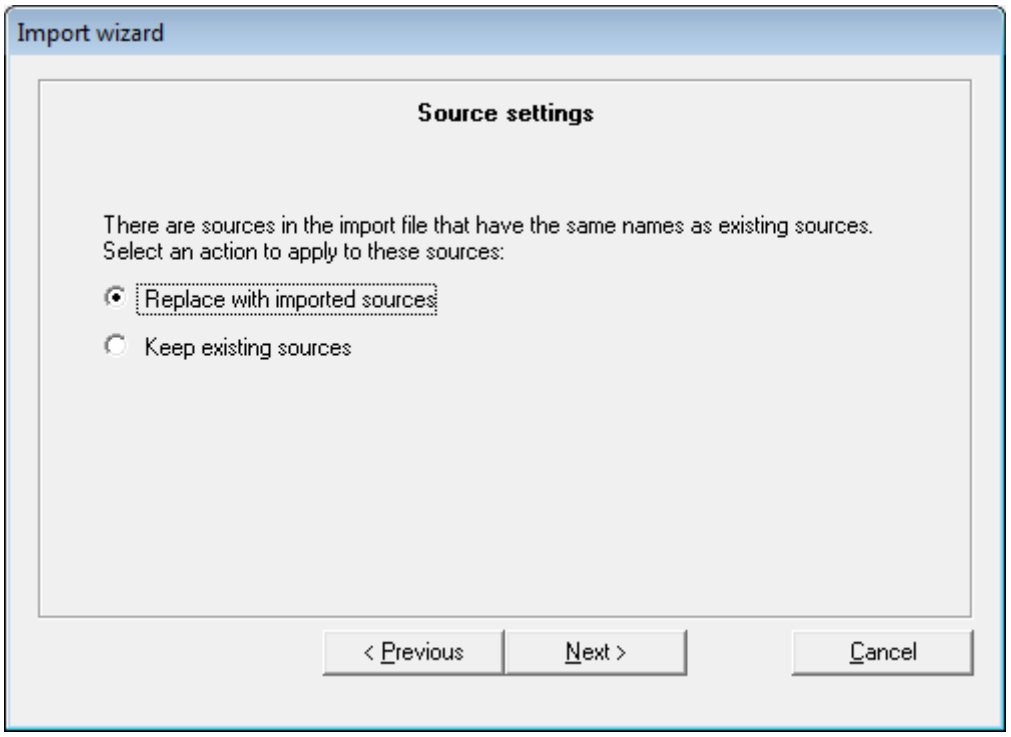


Figure 5.4 – **Source settings** screen of the **Import wizard**.

This screen has two options:

- **Replace with imported sources** – select this option to replace the existing source definitions with those contained in the import files, any other sources without matching names already defined in the ADMS 5 interface will be kept
- **Keep existing sources** – select this option to keep the source definitions already in the ADMS 5 interface, any other sources without matching names will still be imported.

Click **Next >** to move to the next screen of the **Import wizard**.

5.2.5 Pollutant settings

If pollutants are being imported the **Pollutant settings** screen will appear, shown in **Figure 5.5**.

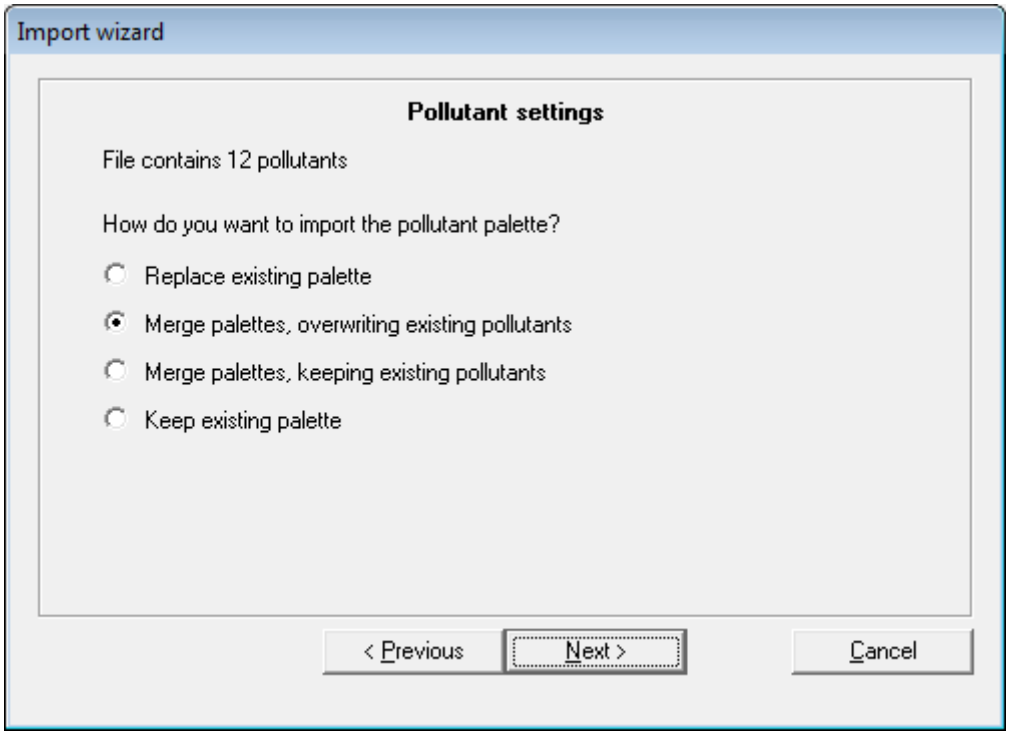


Figure 5.5 – **Pollutant settings** screen of the **Import wizard**.

This screen has four options:

- **Replace existing palette** – all pollutants from the *.ptt* file will be imported, all existing pollutants, except for NOx, NO2, NO, O3 and VOC, will be deleted.
- **Merge palettes, overwriting existing pollutants** – all pollutants from the *.ptt* file will be imported, existing pollutants will be kept providing their names don't match any in the *.ptt* file.
- **Merge palettes, keeping existing pollutants** – all existing pollutants will be kept, any pollutants not already in the palette will be imported.
- **Keep existing palette** – no pollutants will be imported.

Click **Next >** to move to the next screen of the **Import wizard**.

5.2.6 Building settings

If buildings are being imported the **Building settings** screen will appear, shown in **Figure 5.6**.

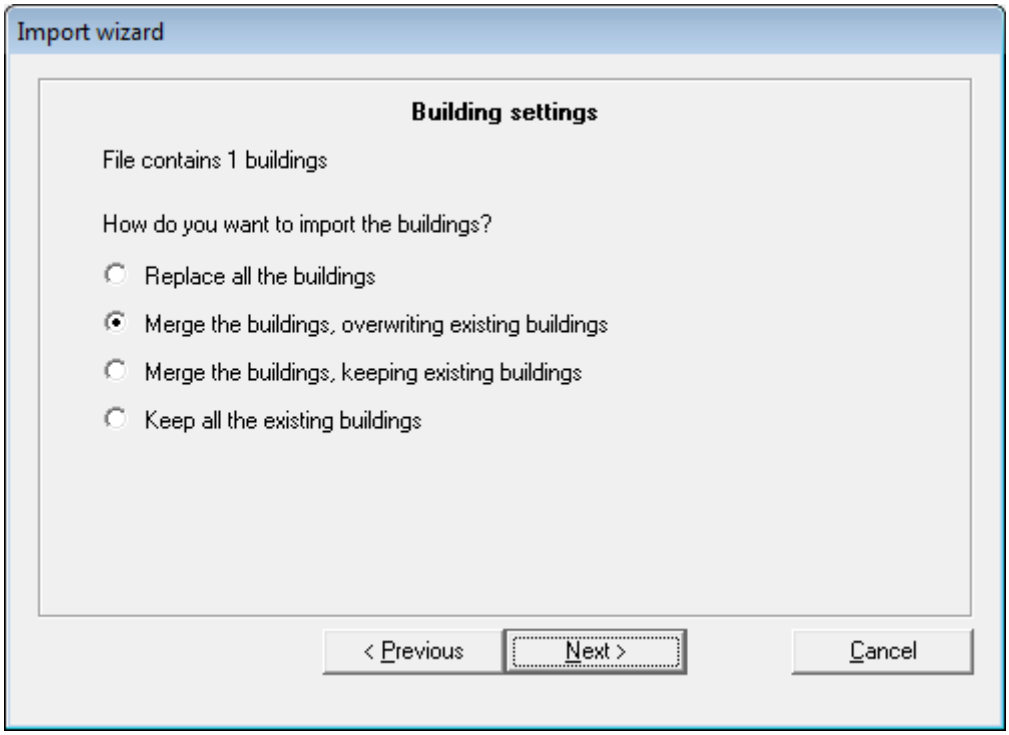


Figure 5.6 – Building settings screen of the **Import wizard**.

This screen has four options:

- **Replace all the buildings** – all buildings from the *.bpt* file will be imported, all existing buildings will be deleted.
- **Merge the buildings, overwriting existing buildings** – all buildings from the *.bpt* file will be imported, any existing buildings will be kept providing their names don't match any in the *.bpt* file.
- **Merge the buildings, keeping existing buildings** – all existing buildings will be kept, any buildings whose names don't match an existing building will be imported.
- **Keep all the existing buildings** – no buildings will be imported.

Click **Next >** to move to the next screen of the **Import wizard**.

5.2.7 Group settings

If group membership information is being imported without other source data the **Group settings** screen will appear, shown in **Figure 5.7**.

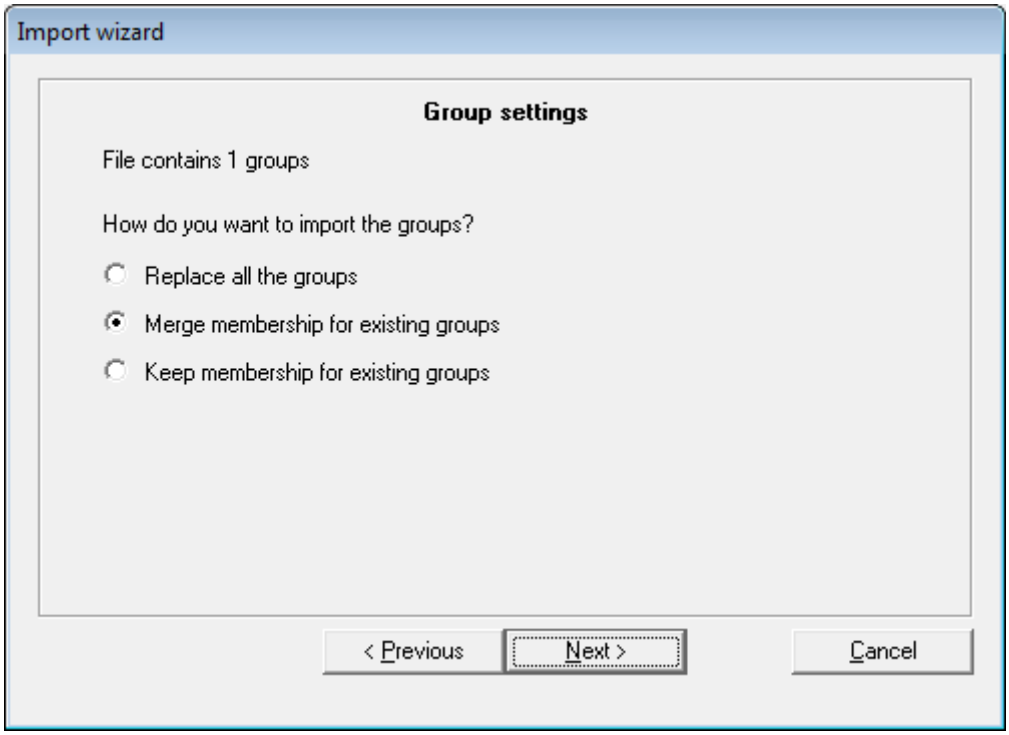


Figure 5.7 – **Group settings** screen of the **Import wizard**.

This screen has three options:

- **Replace all the groups** – all existing group membership information will be deleted and replaced with that in the *.gpt* file.
- **Merge membership for existing groups** – all group membership information will be imported, for existing groups not in the *.gpt* file group membership information will be kept.
- **Keep membership for existing groups** – all existing group membership information will be kept, group membership will be imported for new groups contained in the *.gpt* file.

Click **Next >** to move to the next screen of the **Import wizard**.

5.2.8 Check import

The final screen of the **Import wizard** is the **Check import** screen, shown in **Figure 5.8**.

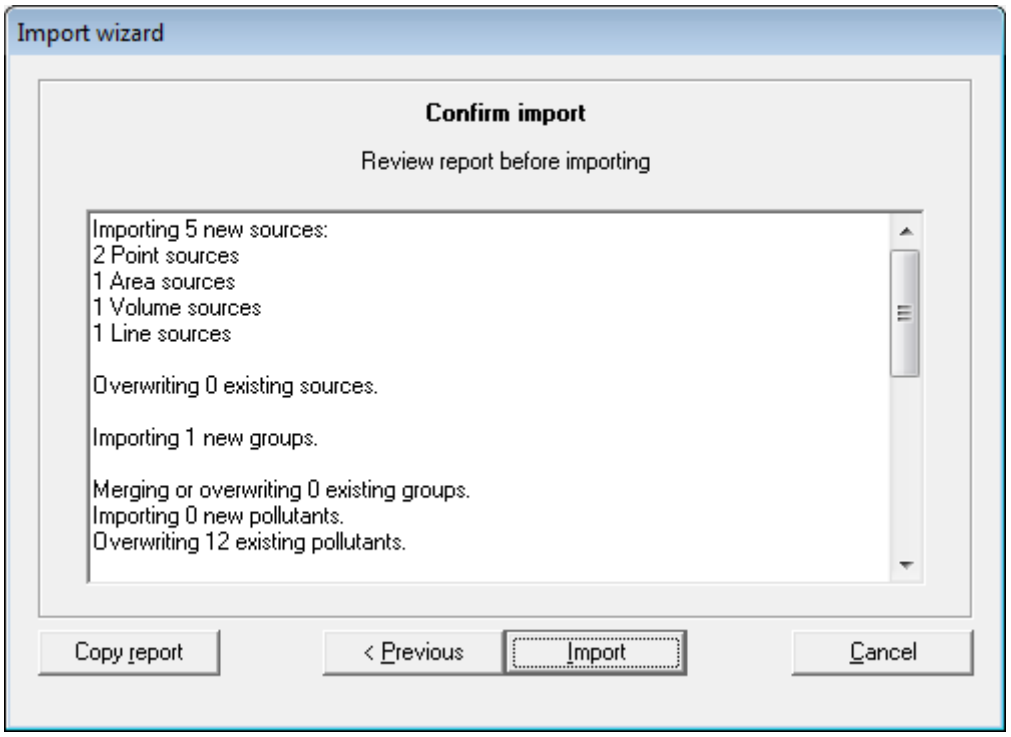


Figure 5.8 – **Check import** screen of the **Import wizard**.

This screen provides a report indicating the numbers of sources, pollutants, buildings and/or groups which will be imported or overwritten along with other information messages. The **Copy report** button can be used to copy the report data to the clipboard. To import the data click on **Import** or click **< Previous** to alter what should be imported.

5.3 Export

It is also possible to export source, pollutant and building data from ADMS 5 into a set of comma-separated variable files. You can use this method to transfer data to other *.apl* files, or to manipulate the data in a spreadsheet package before re-importing.

To export data from the ADMS 5 interface select **Export** from the **File** menu to open the **Export** screen, shown in **Figure 5.9**.

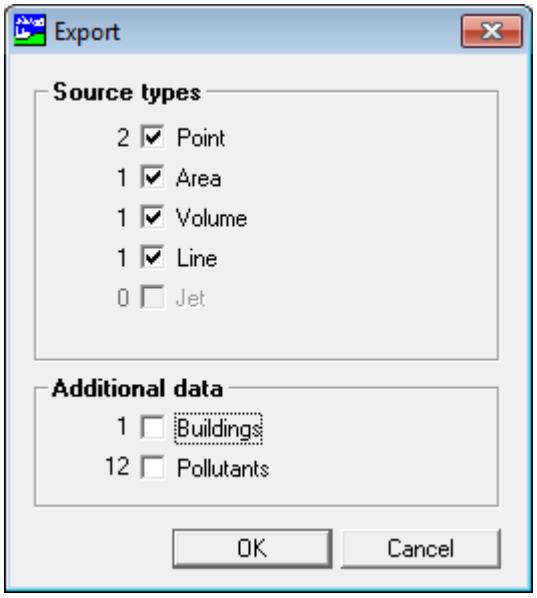


Figure 5.9 – **Export** screen.

The **Export** screen allows for the selection of the **Source types** to be exported along with options to export the **Buildings** and **Pollutants** data. Data about source groups are automatically exported alongside the corresponding sources. Once the items to be exported have been selected click **OK** to open the **Save** dialogue. Enter a directory and filename stem then click **Save**, a set of comma-separated variable files will be created, see Section 5.1 for details of which files will be created based on the options selected.

SECTION 6 Model Output

A variety of output data is produced by ADMS 5 according to the run configuration. This section describes each possible output and presents the tools available in ADMS 5 to visualise the results of a run.

Section 6.1 describes the different types of files produced by ADMS 5. Section 6.2 describes the different types of output files produced when running AERMOD. Sections 6.3 to 6.6 describe some of the first five elements of the **Results** menu of the interface of ADMS 5, as shown in **Figure 6.1**.

The first three options of the menu are features for visualising the results: plots of concentrations or flow fields in Surfer (**Plot in Surfer**), plots of plume centreline properies (**Line Plot**) and numerical results. Plots of concentration and flow fields can also be produced using the ADMS Mapper, refer to Section 3 of the *ADMS Mapper User Guide* for further details.

The fourth element is for examining the output contained in a comprehensive output file. More details of this option are given in the *Comprehensive Output Processor User Guide*.

The fifth option is a utility for visualising the error, warning, log and report files produced during the run.

The sixth option opens an Explorer window containing the directory of the currently loaded *.apl* file.

The last two options on the **Results** menu start up Excel templates containing macros to automate the creation of contour plots in Surfer and 'slideshows' of *.gif* images.

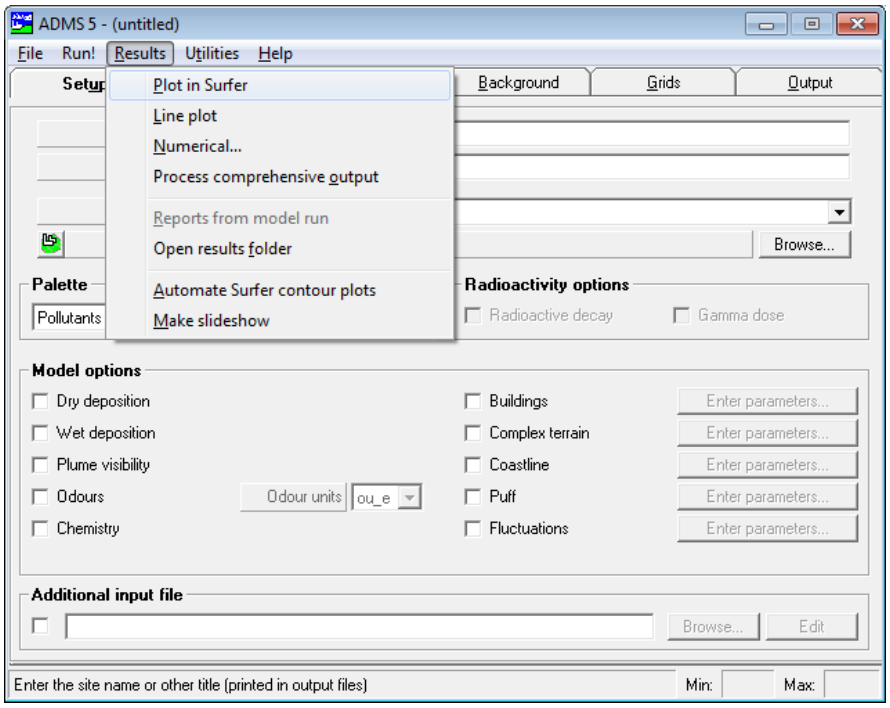


Figure 6.1 – Results menu.

6.1 ADMS 5 output files

During an ADMS 5 model run, a series of output files is created. These files contain the results of the run and are stored in the same directory as the corresponding *.apl* run file. The number of output files produced depends on the run configuration, for example the type of input file and the selected options.

All output files for a model run have the same file stem as the *.apl* file stem and are associated with a different extension that indicates the type of output data contained in the file. For instance, should the *.apl* file be *example.apl*, the output files could be as follows:

> *example.!01, example.~01, example.bld, example.cen, example.dep, example.fst, example.glt, example.levels.glt, example.gst, example.levels.gst, example.gtd, example.levels.gtd, example.log, example.max, example.mop, example.nc, example.out.met, example.plt, example.pro, example.pst, example.ptd, example.rad, example.rpt, example.t01, example.tlt, example.vlt, example.vst, example.w01. example.wlt, example.zlt, example.zst*

Each of the possible output files is described in the following sections.

The majority of the ADMS 5 output files are in a text-based format. As an alternative to using the ADMS 5 results utilities, the user can view them directly with a text editor, for example Notepad.

6.1.1 .err, .wng, .log and .rpt files

These files provide useful information about the model run.

The *.err* file (error) is a text file produced if an error occurs during the run that causes the model to stop. In verification mode, the file may contain several errors. Each error must be corrected before the model can run the *.apl* file successfully.

The *.wng* (warning) file contains any warning messages produced during the run. These are non-fatal errors, i.e. they do not cause the model to stop. The user should check that for each of these warnings the required behaviour is being followed, and make adjustments to model input if necessary.

The *.log* file contains a log of the model run. It details the date and time of the start and end of the run, any program messages, and a summary of the meteorological data (including the number of lines of data the model was able to use).

The *.rpt* file is a summary (report) of the model input data (file pathnames, source characteristics, model options, etc.). It is designed to be easy to import into a Microsoft Word document. It should fit widthways on landscape A4, with standard margins, using Courier font, and minimum size 8 pt.

6.1.2 .cen file

The *.cen* file (plume centreline) contains a series of data related to the plume, which are detailed in **Table 6.1**. It is only produced for short-term, single point source

calculations with gridded output.

Output values are usually given for the first 24 lines of meteorological data only, to prevent the output file from becoming too large. However, the file can be extended if required, as described in Section 6.1.22.

When the **Buildings** option is selected, C(Zp), Zp and Zm are the centreline concentration, centreline height and mean height of the elevated plume only. Sig-y and Sig-z also refer to the elevated plume.

Variable	Description
C(0)	ground-level concentration below the plume centreline
C(Zp)	concentration along the plume centreline
Dry	dry deposition below the plume centreline (ground-level)
Wet	wet deposition below the plume centreline (ground-level)
Tot	total deposition below the plume centreline (ground-level)
Sig-y	plume spread parameter in the crosswind direction
Sig-z	plume spread parameter in the vertical direction
Zp	height of the plume centreline
Zm	mean height of the plume
PINV	fraction of the plume that has penetrated the top of the boundary layer
YP	crosswind plume centreline position
Time	travel time from the source
LiqW	liquid water content on the plume centreline
Tmp0	temperature at the ground below the plume centreline *(if temperature output is selected)*
TmpZP	temperature along the plume centreline *(if temperature output is selected)*
RH0	relative humidity at the ground below the plume centreline *(if relative humidity output is selected)*
RHZP	relative humidity along the plume centreline *(if relative humidity output is selected)*
SH0	specific humidity at the ground below the plume centreline *(if specific humidity output is selected)*
SHZP	specific humidity along the plume centreline *(if specific humidity output is selected)*
H	internal boundary layer height *(if the coastline option is selected)*

Table 6.1 − Variables contained in a *.cen* file.

6.1.3 .glt, .levels.glt and .plt files

The *.glt* file (gridded, long-term) contains output values resulting from long-term average calculations for each of the selected source groups at every point on the lowest level of the output grid. The possible output variables are listed in **Table 6.2**.

The *.levels.glt* file contains output values resulting from long-term average calculations for each of the selected source groups, for each height of the output grid. A separate column of output is given in the *.levels.glt* file for each of the output grid heights. Note that certain output, like deposition, is not given in the *.levels.glt* file as it is not height dependent. The *.levels.glt* file is only created if there is more than one height in the output grid.

The *.plt* (points, long-term) file is equivalent to the *.glt* file except that output values

are given for every specified point defined in the **Grids** screen.

Variable	Description
LTConc	mean concentration
If percentiles selected:	
P###.##	percentile (e.g. P99.9)
If exceedences selected:	
EXpaM	number of exceedences per annum of each concentration threshold value
If dry and/or wet deposition modelled:	
LTDry	long-term average dry deposition
LTWet	long-term average wet deposition
LTTot	long-term average total deposition
If plume visibility option selected:	
Vis(fr)	frequency of visible plumes
VisGr(fr)	frequency of grounded visible plumes
If gamma dose option selected:	
LTGam	long-term average gamma dose concentrations
LTGmD	rate of change of gamma dose rate due to deposited material *(if dry deposition also modelled)*
If temperature and/or humidity output selected (plume visibility):	
LTTmp	long-term average temperature
LTRH	long-term average relative humidity
LTSH	long-term average specific humidity
EXT	number of exceedences of threshold value of temperature *(if selected)*
EXRH	number of exceedences of threshold value of relative humidity *(if selected)*
EXSH	number of exceedences of threshold value of specific humidity *(if selected)*
EXTRH	number of times the simultaneous temperature and relative humidity thresholds are exceeded *(if selected)*
EXTSH	number of times the simultaneous temperature and specific humidity thresholds are exceeded *(if selected)*
If fluctuations option selected:	
PDF	probability that the output concentration (including the effect of fluctuations) exceeds each concentration specified in the PDF box (**Fluctuations** screen)
EXpaF	number of times the output concentration (including the effect of fluctuations) exceeds each concentration specified in the PDF box (**Fluctuations** screen)

Table 6.2 − Variables contained in a *.glt* file.

6.1.4 .gst, .levels.gst, .gtd, .levels.gtd, .pst and .ptd files

The *.gst* file (gridded, short-term) contains output values resulting from short-term calculations for each modelled source group at every point of the output grid and for each line of meteorological data. The possible output variables are listed in **Table 6.3**.

Output values are usually given for the first 24 lines of meteorological data only, to prevent the output file becoming too large. However, the file can be extended if required, as described in Section 6.1.22.

The *.levels.gst* file contains output values resulting from short-term calculations for each of the selected source groups, for each line of meteorological data and for each height of the output grid. A separate column of output is given in the *.levels.gst* file

for each of the output grid heights. Note that certain output, like deposition, is not given in the *.levels.gst* file as it is not height dependent. The *.levels.gst* file is only created if there is more than one height in the output grid.

The *.gtd* file (gridded, time-dependent) is equivalent to the *.gst* file but for time-dependent puff emissions.

The *.levels.gtd* file is equivalent to the *.levels.gst* file but for time-dependent puff emissions.

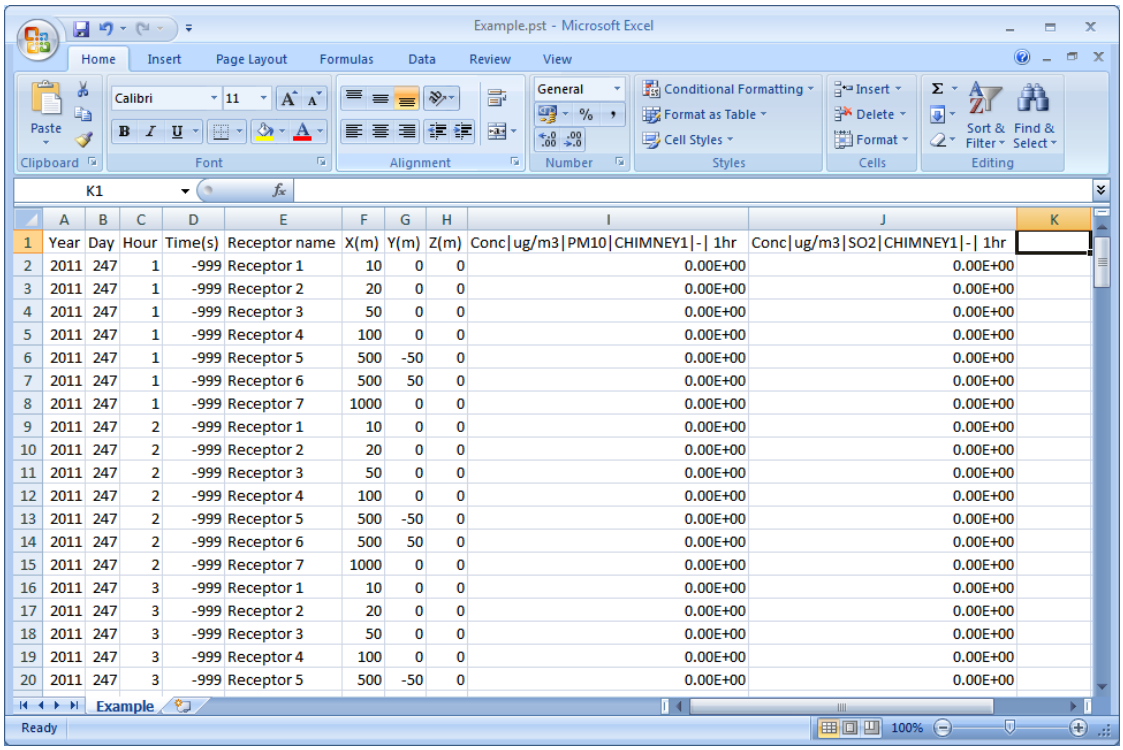
The *.pst* file (points, short-term) is equivalent to the *.gst* file except that output values are given for every specified point defined in the **Grids** screen.

The *.ptd* file (points, time-dependent) is equivalent to the *.pst* but for time-dependent puff emissions.

Examples of *.pst* and *.plt* files (viewed in Microsoft Excel) are illustrated in **Figures 6.2** and **6.3**. Gridded *.glt* and *.gst* files are similar to these files but with a value for every grid point.

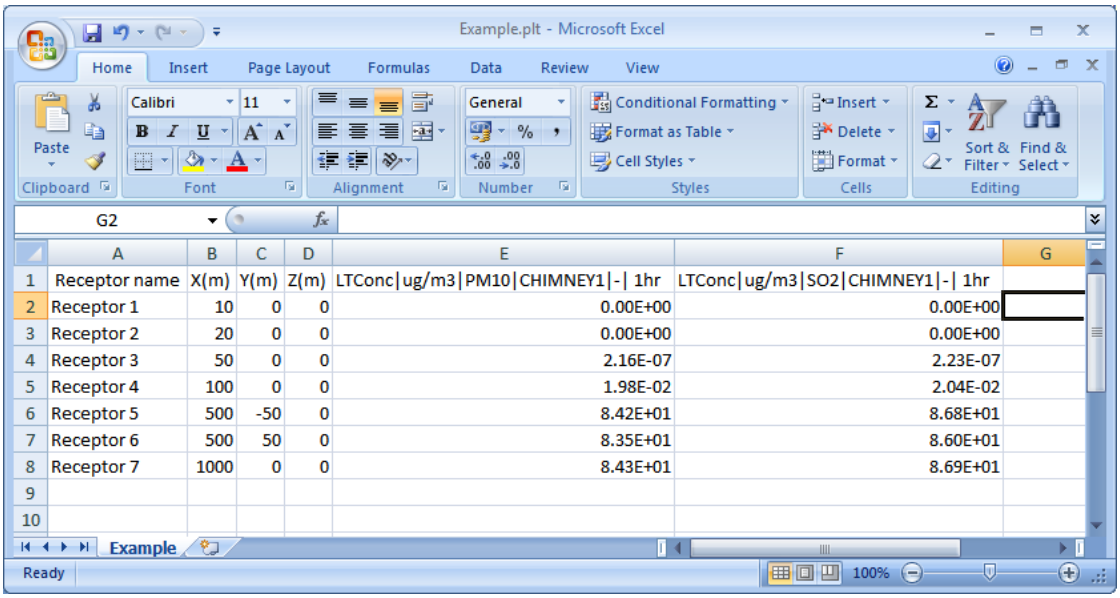
Variable	Description
Conc	mean concentration
If dry and/or wet deposition modelled:	
Dry	dry deposition
Wet	wet deposition
Tot	total deposition
If gamma dose option selected:	
Gam	gamma dose concentrations
GmD	rate of change of gamma dose rate due to deposited material *(if dry deposition also modelled)*
If temperature and/or humidity output selected (plume visibility):	
Temp	temperature
RHum	relative humidity
SHum	specific humidity
If fluctuations option selected	
Sig-C	variance from mean concentrations
F###.##	percentile (e.g. F99.9)
PDF	probability that the output concentration (including the effect of fluctuations) exceeds each concentration specified in the PDF box
Tox.R	toxic response

Table 6.3 − Variables contained in a *.gst* file.



	A	B	C	D	E	F	G	H	I	J	K
1	Year	Day	Hour	Time(s)	Receptor name	X(m)	Y(m)	Z(m)	Conc|ug/m3|PM10|CHIMNEY1|-| 1hr	Conc|ug/m3|SO2|CHIMNEY1|-| 1hr	
2	2011	247	1	-999	Receptor 1	10	0	0	0.00E+00	0.00E+00	
3	2011	247	1	-999	Receptor 2	20	0	0	0.00E+00	0.00E+00	
4	2011	247	1	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
5	2011	247	1	-999	Receptor 4	100	0	0	0.00E+00	0.00E+00	
6	2011	247	1	-999	Receptor 5	500	-50	0	0.00E+00	0.00E+00	
7	2011	247	1	-999	Receptor 6	500	50	0	0.00E+00	0.00E+00	
8	2011	247	1	-999	Receptor 7	1000	0	0	0.00E+00	0.00E+00	
9	2011	247	2	-999	Receptor 1	10	0	0	0.00E+00	0.00E+00	
10	2011	247	2	-999	Receptor 2	20	0	0	0.00E+00	0.00E+00	
11	2011	247	2	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
12	2011	247	2	-999	Receptor 4	100	0	0	0.00E+00	0.00E+00	
13	2011	247	2	-999	Receptor 5	500	-50	0	0.00E+00	0.00E+00	
14	2011	247	2	-999	Receptor 6	500	50	0	0.00E+00	0.00E+00	
15	2011	247	2	-999	Receptor 7	1000	0	0	0.00E+00	0.00E+00	
16	2011	247	3	-999	Receptor 1	10	0	0	0.00E+00	0.00E+00	
17	2011	247	3	-999	Receptor 2	20	0	0	0.00E+00	0.00E+00	
18	2011	247	3	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
19	2011	247	3	-999	Receptor 4	100	0	0	0.00E+00	0.00E+00	
20	2011	247	3	-999	Receptor 5	500	-50	0	0.00E+00	0.00E+00	

Figure 6.2 − An example of a *.pst* file.



	A	B	C	D	E	F	G
1	Receptor name	X(m)	Y(m)	Z(m)	LTConc|ug/m3|PM10|CHIMNEY1|-| 1hr	LTConc|ug/m3|SO2|CHIMNEY1|-| 1hr	
2	Receptor 1	10	0	0	0.00E+00	0.00E+00	
3	Receptor 2	20	0	0	0.00E+00	0.00E+00	
4	Receptor 3	50	0	0	2.16E-07	2.23E-07	
5	Receptor 4	100	0	0	1.98E-02	2.04E-02	
6	Receptor 5	500	-50	0	8.42E+01	8.68E+01	
7	Receptor 6	500	50	0	8.35E+01	8.60E+01	
8	Receptor 7	1000	0	0	8.43E+01	8.69E+01	
9							
10							

Figure 6.3 − An example of a *.plt* file.

6.1.5 .max file

The *.max* file (maximum concentrations) contains the maximum value of the mean concentration and each of the percentiles selected, the position of the maximum values and, in the case of the percentiles, for which line of meteorological data the maximum values occur. It is created when a long-term average calculation is made, with or without percentiles.

In addition, following each maximum percentile value, the concentrations exceeding

that value at that point are listed, alongside the line number of the meteorological data for which they occur. For example, when using one year of data (8760 lines), there will be 2% of 8760 lines (176) where the 98th percentile value is exceeded. This output is only given for the 98th percentile and above.

6.1.6 .mop file

The *.mop* file (meteorological input and output parameters) contains the meteorological input parameters and the calculated meteorological output values for each line of data in the meteorological data file. The frequency of occurrence of each condition is also reported for statistical meteorological data. The input parameters are in the left hand columns of the file, and the calculated output parameters in the right hand columns.

6.1.7 .nc file

The *.nc* file is a netCDF format file containing the concentration and dry or wet deposition (if selected) data for each unique pollutant and averaging time combination selected for long-term output in the **Output** screen of the ADMS 5 interface. For averaging times greater than one hour the averaging time used in creating this file is one hour. Percentile or exceedence data are not output to the netCDF file. As well as containing the concentration (and deposition) data, the netCDF file also contains basic meteorological data relating to each of the lines in the meteorological data file. The *.nc* file is created if the **Comprehensive output file** option is selected on the **Output** screen of the ADMS 5 interface.

*Use the **Process Comprehensive Output** option on the **Results** menu to open the .nc file in the **Comprehensive Output File Processor** in order to view data or extract statistics. Refer to the Comprehensive Output File Processor User Guide for more details.*

The *.nc* file is in netCDF format with the dimensions, variables and attributes as described in **Tables 6.4** to **6.6** below.

Dimension	Description
nPoints_XY	Number of lowest level grid points and all specified points (XY output points)
nPoints_XYZ	Total number of all output points (XYZ output points)
nMetLines	Number of *.met* lines
nGroups	Number of groups
nDatasets	Number of datasets, with one dataset for each long-term pollutant, for each averaging time less than or equal to one hour, for concentration, dry deposition (if selected) and wet deposition (if selected)
OutputOptionsUsed	*Number of possible output options (=3)*
NumberOfTypesOfOutputPoint	*Number of types of output point (=5)*
MetLineStringLength	*Maximum length of a met line description (=11)*
DatasetNameStringLength	*Maximum length of a dataset name (=34)*
GroupStringLength	*Maximum length of a group name (=20)*
PointNameStringLength	*Maximum length of an output point name (=34)*
OutputOptionsStringLength	*Maximum length of an output option name (=14)*
OutputPointTypesLength	*Maximum length of an output point type name (=16)*

Table 6.4 – Dimensions in the *.nc* file. The dimensions in italics are dimensions required as part of the netCDF format and are the same for all files.

Variable	Dimensions	Description
PointX_XY	nPoints_XY	X-coordinate of XY output points
PointY_XY	nPoints_XY	Y-coordinate of XY output points
PointName_XY	nPoints_XY, *PointNameStringLength*	Name of output point for XY output points • "Grid, grid" for Cartesian grid points • "Polar, (***R***, θ)" for polar grid points where ***R*** and θ are the radius and angle of the point respectively • "Receptor, ***name***" for specified points where ***name*** is the name entered by the user for the point
PointX_XYZ	nPoints_XYZ	X-coordinate of XYZ output points
PointY_XYZ	nPoints_XYZ	Y-coordinate of XYZ output points
PointZ_XYZ	nPoints_XYZ	Z-coordinate of XYZ output points
PointName_XYZ	nPointsXYZ	Name of output point for XYZ output points, see PointName_XY. For polar grid points the height is also given in the name.
Output_Points_Type	*NumberOfTypesOfOutputPoint*, *OutputPointTypesLength*	Description of the types of output point
Number_Of_Output_ Points_Of_Each_Type	*NumberOfTypesOfOutputPoint*	Number of output points of each type
Met_Line	nMetLines, *MetLineStringLength*	Date of *.met* line or *.met* line number for each *.met* line
Met_Freq	nMetLines	Frequency for each *.met* line
Met_Uat10m	nMetLines	10 m wind speed for each *.met* line
Met_Phi	nMetLines	Wind direction for each *.met* line
Met_H_over_LMO	nMetLines	Boundary layer height divided by Monin-Obukhov length for each *.met* line
Output_Options_ Description	*OutputOptionsUsed*, *OutputOptionsStringLength*	Names of the output options
Output_Options_Used	*OutputOptionsUsed*	Indicator of whether each output option is used (1=yes, 0=no)
Group	nGroups, *GroupStringLength*	Name of each output group
DatasetNames	nDatasets, *DatasetNameStringLength*	Descriptive name for each dataset
Dataset#	nMetLines, nGroups, nPoints*	Dataset containing concentration or deposition data.

Table 6.5 – Variables in the *.nc* file. Dimensions are listed with the fastest-varying last and the string length dimension is indicated in italics for character variables. * For the datasets (Dataset#) nPoints is either nPoints_XYZ for concentration datasets or nPoints_XY for deposition datasets.

Associated with	Attribute	Description
Global	File_type	Indicator that this is a comprehensive output file
	File_version	Version number of the comprehensive output file format
	Model_name	Name of the model which created the file
	Model_Version	Version number of the model which created the file
	Model_Build_Number	Build number of the model which created the file
	Model_Release_Date	Build date of the model that created the file
	Date_Created	The date the file was created
	Time_Created	The time the file was created
	Site_name	The site name entered in the interface
	Project_name	The project name entered in the interface
	Is_Data_Hourly_ Sequential	Indicates whether **Met. data are hourly sequential** is selected in the interface
	Grid_Description	Description of the output grid
Variables	Units	Units of the variable
Dataset#	Dataset_Name	Name of the dataset
	Pollutant_Name	Name of the pollutant for the dataset
	Output_Type	Type of output for the dataset (concentration, dry deposition or wet deposition)
	Av_Time_s	Averaging time for the dataset in seconds
	Units	Units of the dataset
	ugm3_to_ppb_conv_factor	$\mu g/m^3$ to ppb conversion factor for the pollutant associated with the dataset.

Table 6.6 – Global and variable attributes in the *.nc* file.

6.1.8 .out.met file

The *.out.met* file contains a copy of the meteorological data, cut down to the subset being used, with the effects of the choice of wind sector size taken into account. Refer to Section 9.1 for more details of how the wind sector size affects the wind direction. This is done as follows:

- For wind not in sectors, the data in the *.out.met* file is identical to the input meteorological data file.
- For wind in sectors smaller than 15^o, the data in the *.out.met* file is identical to the input meteorological data, except that the wind directions are altered to take into account the angular cycling applied.
- For wind in sectors larger than 15^o, for each line of input meteorological data there are five lines of data in the *.out.met* file. These lines are a copy of the input meteorological data except that the wind directions are altered to give the five directions used to represent that sector, and the frequency is divided equally between the lines.

6.1.9 Boundary layer profile: .pro file

The *.pro* file contains the boundary layer properties output for a series of heights for each meteorological data line. The *.pro* file is produced if the **Boundary layer profile output** option is enabled using an *.aai* file, refer to Section 4.16 for more details. The possible output variables are listed in **Table 6.7**.

Variable	Description
Met line no.	Met line number
z(m)	Height of output
z/h	Height of output as fraction of boundary layer height
U(z) (m/s)	Mean wind speed
DUDZ(z) ((m/s)/m)	Gradient of mean wind speed
SigU(z) (m/s)	Longitudinal turbulence
SigV(z) (m/s)	Transverse turbulence
SigW(z) (m/s)	Vertical turbulence
Rlv(z) (m)	Transverse turbulence length scale
Rlw(z) (m)	Vertical turbulence length scale
TL(z) (s)	Lagrangian time scale
Eps(z) (m2/s3)	Energy dissipation rate
Pot Temp(z) (K)	Potential temperature
Temp(z) (K)	Temperature
Press(z) (mbar)	Pressure
N(z) (1/s)	Buoyancy frequency
If plume visibility is selected:	
Q(z) (kg/kg)	Specific humidity
RH(z) (%)	Relative humidity

Table 6.7 − Variables contained in a *.pro* file.

6.1.10 Buildings: .bef, .bld and .bwk files

These files are related to the **Buildings** module.

The *.bef* file (effective building file) contains the coordinates of the effective building for each source and line of meteorological data.

The *.bld* file (buildings) contains the dimensions of buildings input to the model as well as some parameters calculated during the run (including the region affected by the presence of buildings, the cavity size and the cavity concentration). Output values are usually given for the first 24 lines of meteorological data only, in order to prevent the file from becoming too large. However, the file can be extended if required, as described in Section 6.1.22.

The *.bwk* file (building wake) contains the cavity and wake height as a function of downstream distance from the upwind edge of the effective building. It is only produced for single (point) source short-term runs with buildings.

6.1.11 Deposition: .dep file

The *.dep* file (deposition) is related to the deposition option. It contains the values of deposition variables (e.g. terminal velocity) input to the model as well as those

calculated during the run.

Output values are usually given for the first 24 lines of meteorological data only, to prevent the output file from becoming too large. However, the file can be extended if required, as described in Section 6.1.22.

6.1.12 Fluctuations: .fst file

The *.fst* file (fluctuations, short-term) is related to the **Fluctuations** module. It contains short-term probability distribution function (PDF) values for each concentration, at each point of the output grid, if the limit values have been automatically selected by the model (by specifying **Auto** in the **Fluctuations** screen).

6.1.13 Flowfield: .W01 files

The *.W01*, *.W02*, etc, files are related to the **Complex terrain, Wind turbines** and **Urban canopy flow** options and are only created for short-term calculations with an output grid if flow field output is selected. The *.W01*, *.W02*, etc. files contain flow field data at the locations defined for the output grid. These files are only created for the first 24 lines of met. data.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **U(m/s)**: Component of wind velocity in west-east direction
- **V(m/s)**: Component of wind velocity in south-north direction
- **W(m/s)**: Vertical component of wind velocity (relative to sea level)
- **Ux(m/s)**: Longitudinal component of wind velocity (i.e. in the direction of the upstream wind; the direction specified for the wind in the meteorological data)
- **Uy(m/s)**: Transverse component of wind velocity (i.e. perpendicular to the direction of the upstream wind)
- **Angle(deg)**: Wind direction (degrees measured anticlockwise, 0^o = wind from west)
- **Magnitude of horizontal wind (m/s)**: Magnitude of horizontal wind velocity ($=\sqrt{U^2+V^2}$)

6.1.14 Flowfield: .wlt file

The *.wlt* file is related to the **Complex terrain, Wind turbines** and **Urban canopy** options and is only created for long-term calculations with an output grid if flow field output is selected. The *.wlt* file contains flow field data, averaged over all the lines of met. data, at the locations defined for the output grid.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **U(m/s)**: Component of mean wind velocity in west-east direction
- **V(m/s)**: Component of mean wind velocity in south-north direction
- **W(m/s)**: Vertical component of mean wind velocity (relative to sea level)
- **Mean horizontal wind speed (m/s)**: Mean value of the horizontal wind speed ($=\overline{\sqrt{U^2+V^2}}$)
- **Magnitude of mean horizontal wind vector (m/s)**: Magnitude of the mean horizontal wind vector ($=\sqrt{\overline{U}^2+\overline{V}^2}$)
- **Angle(deg)**: Direction of the mean horizontal wind vector (degrees measured anticlockwise, 0^o = wind from west).
- **Magnitude of mean horizontal wind vector (m/s)**: Magnitude of the mean horizontal wind vector (Note that this column appears twice in the file – this is for use by the flow field plotting tool described in Section 6.3.2)

6.1.15 Flowfield: .T01 files

The *.T01*, *.T02*, etc, files are related to the **Complex terrain**, **Wind turbines** and **Urban canopy** options and are only created for short-term calculations with an output grid if flow field output is selected. The *.T01*, *.T02*, etc. files contain turbulence data at the locations defined for the output grid. These files are only created for the first 24 lines of met. data.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **Sig-U(m/s)**: Longitudinal turbulence σ_u
- **Sig-V(m/s)**: Transverse turbulence σ_v
- **Sig-W(m/s)**: Vertical turbulence σ_w

6.1.16 Flowfield: .tlt file

The *.tlt* file is related to the **Complex terrain, Wind turbines** and **Urban canopy** options and is only created for long-term calculations with an output grid if flow field output is selected. The *.tlt* file contains turbulence data, averaged over all the lines of met. data, at the locations defined for the output grid.

Variables are:

- **X(m)**: X coordinate of output point
- **Y(m)**: Y coordinate of output point
- **Z(m)**: Height above terrain of output point
- **Sig-U(m/s)**: Mean longitudinal turbulence σ_u
- **Sig-V(m/s)**: Mean transverse turbulence σ_v
- **Sig-W(m/s)**: Mean vertical turbulence σ_w

6.1.17 Flowfield: .zst file

The *.zst* file is related to the **Complex terrain, Wind turbines** and **Urban canopy** options and is only created for short-term calculations with specified points if flow field output is selected. The *.zst* file contains flow field and turbulence data at the specified points for each line of met. data.

Variables are:

- **Year**: Year of the current line of met. data
- **Day**: Julian day number of the current line of met. data
- **Hour**: Hour of the current line of met. data
- **Receptor name**: Name of the specified point
- **X(m)**: X coordinate of specified point
- **Y(m)**: Y coordinate of specified point
- **Z(m)**: Height above terrain of specified point
- **U(m/s)**: Component of wind velocity in west-east direction
- **V(m/s)**: Component of wind velocity in south-north direction
- **W(m/s)**: Vertical component of wind velocity (relative to sea level)
- **Ux(m/s)**: Longitudinal component of wind velocity (i.e. in the direction of the upstream wind; the direction specified for the wind in the meteorological data)
- **Uy(m/s)**: Transverse component of wind velocity (i.e. perpendicular to the direction of the upstream wind)
- **Angle(deg)**: Wind direction (degrees measured anticlockwise, 0^o = wind from west)
- **Magnitude of horizontal wind (m/s)**: Magnitude of horizontal wind velocity $(=\sqrt{U^2+V^2})$
- **Sig-U(m/s)**: Longitudinal turbulence σ_u
- **Sig-V(m/s)**: Transverse turbulence σ_v
- **Sig-W(m/s)**: Vertical turbulence σ_w

6.1.18 Flowfield: .zlt file

The *.zlt* file is related to the **Complex terrain**, **Wind turbines** and **Urban canopy** options and is only created for long-term calculations with specified points if flow field output is selected. The *.zlt* file contains flow field and turbulence data, averaged over all of the lines of met. data, at each of the specified points.

Variables are:

- **Receptor name**: Name of the specified point
- **X(m)**: X coordinate of specified point
- **Y(m)**: Y coordinate of specified point
- **Z(m)**: Height above terrain of specified point
- **U(m/s)**: Component of mean wind velocity in west-east direction
- **V(m/s)**: Component of mean wind velocity in south-north direction
- **W(m/s)**: Vertical component of mean wind velocity (relative to sea level)
- **Mean horizontal wind speed (m/s)**: Mean value of the horizontal wind speed ($=\overline{\sqrt{U^2+V^2}}$)
- **Magnitude of mean horizontal wind vector (m/s)**: Magnitude of the mean horizontal wind vector ($=\sqrt{\overline{U}^2+\overline{V}^2}$)
- **Sig-U(m/s)**: Mean longitudinal turbulence σ_u
- **Sig-V(m/s)**: Mean transverse turbulence σ_v
- **Sig-W(m/s)**: Mean vertical turbulence σ_w

6.1.19 Plume visibility: .vlt and .vst files

These files are related to the **Plume visibility** option.

The *.vlt* file (visibility, long-term) is produced when the **Plume visibility** option is selected and long-term calculations are performed. It contains the variables listed in **Table 6.8**. All distance variables are given in metres.

Variable	Description
Met line number	meteorological data line number
Source	name of the source
Pollutant	pollutant modelled
X first visible	distance from the source at which the plume first becomes visible
Max liquid water content	maximum liquid content of the plume (kg/kg)
X last visible	distance from the source at which the plume is no longer visible
Visible length	length of the visible plume
Plume height where last visible	height of the plume centre line at the point where the plume is no longer visible
X first grounding	distance from the source at which the plume is first 'grounded', where 'grounding' is defined as the plume height being less than the vertical plume spread

Table 6.8 − Variables contained in a *.vlt* or *.vst* file.

The *.vst* file (visibility, short-term) is equivalent to the *.vlt* file but for short-term calculations.

The following special cases may occur:

- If the plume is not visible within the output region then location of the first visible and last visible plumes will be set to -999. The visible plume length in this case is set to 0.
- If the plume becomes visible within the output region, and remains so until the downwind edge of the region, then the last visible plume location will be set to -999. The visible plume length in this case is set to -999.

6.1.20 Radioactivity: .rad file

The *.rad* file (radioactivity) summarises the decay routes of the output radioactive pollutants. It is produced when the **Radioactive decay** option is selected.

6.1.21 .!01 and .~01 extensions

The *.!01, .!02*, etc. files are files used by the **Line Plot** utility of ADMS 5 (see Section 6.4). They are only produced for short-term, single point source calculations with gridded output. The number in the extension refers to the line of meteorological data to which the results correspond.

The *.~01, .~02*, etc. files are equivalent to the *.!01, .!02*, etc. file. They are produced when the release is a time-dependent puff. Similarly, they are created for the **Line Plot** utility of ADMS 5 (**Results, Line Plot** menu command; see Section 6.4).

6.1.22 Increasing the length of output files

As noted above, the output data of the *.bld, .cen, .dep, .gst, .levels.gst, .gtd* and *.levels.gtd* files are usually restricted to the first 24 lines of meteorological data in order to prevent the output files from becoming too large. This limit can be removed

by using an additional input file (*.aai*) option. Please refer to Section 4.25 for details of how to enable this option.

6.2 AERMOD files

When running an ADMS 5 parameter file with AERMOD a series of output files are created. These files contain the results of the run along with intermediate results used in the conversion from an ADMS 5 parameter file to an AERMOD parameter file. The results are all stored in the same directory as the corresponding *.apl* file. The number of output files produced depends on the run configurations, for example the type of output selected and the selected model options.

All output files for a model run have the same file stem as the *.apl* file but with either *_AERMOD*, *_AERMAP*, *_BPIPPRM* or *_AERMET* appended. The different extensions then indicate the type of data contained in the file. For instance, should the *.apl* file be *example.apl*, the output files could be as follows:

example_AERMOD.dem, example_AERMOD.emf, example_AERMOD.inp, example_AERMOD.glt, example_AERMOD.gst, example_AERMOD.log, example_AERMOD.lt, example_AERMOD.mop, example_AERMOD.out, example_AERMOD.pfl, example_AERMOD.plt, example_AERMOD.pst, example_AERMOD.sfc, example_AERMOD_MET.ami, example_AERMAP.inp, example_AERMAP.map example_AERMAP.rcp, example_AERMAP.sou, example_BPIPPRM.inp, example_BPIPPRM.out, example_AERMET1.inp, example_AERMET2.inp, example_AERMET3.inp, example_AERMET_OS.dsk, example_AERMET_SF.dsk, example_AERMET1.rpt, example_AERMET2.rpt, example_AERMET3.rpt

Each of the possible output files is described in the following sections.

6.2.1 Meteorological settings file

The *_AERMOD_MET.ami* file contains information on the choices entered in the **Choose met. data for AERMOD run** screen. If this file is present when **Run AERMOD** is selected from the **File** menu then the information in this file will be used to fill in the **Choose met. data for AERMOD run** screen. This option may be useful when the same meteorological data settings are used for multiple runs. The path of the AMI file is given in the **AMI file** section of the **Choose met. data for AERMOD run** screen, if the path is greyed out this indicates that the file will be created and no existing data are available.

6.2.2 Error, warning and log files

These files provide useful information about the model run:

- *_AERMOD.err*
- *_AERMOD.wng*
- *_AERMOD.log*

The *.err* file (error) is a text file produced if an error occurs during conversion or execution of the run that causes the model to stop. Each error must be corrected before the *.apl* file can be run successfully with AERMOD.

The *.wng* (warning) file contains any warning messages produced during the run. These are non-fatal errors, i.e. they do not cause the model to stop. The user should check that for each of these warnings the required behaviour is being followed, and make adjustments to model input if necessary.

The *.log* file contains a log of the model run. It details the date and time of the start of the run, the conversion process of inputs from ADMS 5 to AERMOD, the AERMOD run and the conversion of the AERMOD output data to ADMS 5 format.

6.2.3 Concentration and deposition output files

These files contain the results for the outputs entered on the **Output** screen of the ADMS 5 interface at the locations entered on the **Grids** screen of the ADMS 5 interface:

- *_AERMOD.glt*
- *_AERMOD.gst*
- *_AERMOD.plt*
- *_AERMOD.pst*

The format of these files is the same as for an ADMS 5 run and more details can be found in Section 6.1. Note that not all of the ADMS 5 output options are available when running AERMOD, refer to Appendix A.4 for more details about which options can be used.

6.2.4 ADMS meteorological data file

The *_AERMOD.mop* file contains meteorological input and output parameters from the ADMS meteorological processor. It is identical to the ADMS *.mop* file, as described in Section 6.1.6. This file is used when creating the *_AERMOD.sfc* and *_AERMOD.pfl* files when running AERMOD with ADMS meteorological input data.

6.2.5 Intermediate files

These are intermediate files created as part of the process of running an ADMS 5 *.apl* file with AERMOD.

- *_AERMOD.inp* – the main input file for AERMOD. This file contains details of the model setup, source details and output options in the format required for input into AERMOD. This file is created based on the details entered into the *.apl* file.
- *_AERMOD.sfc* – the surface meteorological file for AERMOD. This file contains surface meteorological data in the format required for input into AERMOD.
- *_AERMOD.pfl* - the vertical profile meteorological file for AERMOD. This file contains meteorological data at a series of heights (possibly one) in the format required for input into AERMOD.

- *_AERMOD.emf* – the time varying emissions file for AERMOD. This file contains the time varying emissions and source parameters in the format required for input into AERMOD.
- *_AERMOD.out* – the main AERMOD output file. This file is output from AERMOD during the run and contains details of the setup, messages generated during the run and a selection of concentration output
- *_AERMOD.lt* – the AERMOD period average output file. This file is output from AERMOD and contains the average concentration over the meteorological period at each of the receptor points.

6.2.6 Complex terrain files

These are intermediate files created as part of running an ADMS 5 *.apl* file which uses the **Complex terrain** model option with AERMOD.

- *_AERMOD.dem* – the terrain file for AERMAP. This file contains the terrain data being modelled, converted into the correct format for input into AERMAP.
- *_AERMAP.inp* – the main input file for AERMAP. This file contains the source and receptor locations in the format required for input into AERMAP.
- *_AERMAP.map* – the main output file from AERMAP. This file is output from AERMAP during the terrain processing and contains details of the run.
- *_AERMAP.rcp* – the receptor point output file from AERMAP. This file is output from AERMAP during the terrain processing and contains the required terrain parameters at the receptor point locations for AERMOD.
- *_AERMAP.sou* – the source output file from AERMAP. This file is output from AERMAP during the terrain processing and contains the required terrain parameters at the source locations for AERMOD.

6.2.7 Buildings files

These are intermediate files created as part of running an ADMS 5.*apl* file which uses the **Buildings** module with AERMOD.

- *_BPIPPRM.inp* – the main input file to BPIPPRM. This file contains details of the buildings and source locations in the format required for input into BPIPPRM.
- *_BPIPPRM.out* – the main output file for BPIPPRM. This file is output from BPIPPRM during the buildings processing and contains messages created during the run along with the required buildings parameters for each source for AERMOD.

6.2.8 AERMET meteorology files

These are intermediate files created as part of using AERMET to process the meteorological data.

- *_AERMET#.inp* - the input files for each of the three stages of AERMET. These files contain the information required for input into AERMET.
- *_AERMET_OS.dsk* – the onsite data file for input into stage 1 of AERMET. This file may be created from the meteorological data in the ADMS *.met* and *.prf* files, depending on the variables and options chosen.
- *_AERMET_SF.dsk* – the surface data file for input into stage 2 of AERMET. This file is created from the meteorological data in the ADMS *.met* file.
- *_AERMET#.rpt* – the report files from each of the three stages of AERMET. These files are output from AERMET during the meteorological processing and contain a summary of the run including any error and warning messages.

6.3 Plots in Surfer (2-D Output Plotter)

This section outlines how to produce contour and vector plots in Surfer from gridded ADMS 5 output data using the **2-D Output Plotter** utility launched from the ADMS 5 interface. It is also possible to produce contour and vector plots from the ADMS Mapper. Please refer to the ADMS Mapper User Guide for more details.

To launch the **2-D Output Plotter (Plotting in Surfer)** utility, select **Results**, **Plot in Surfer** from the top of the ADMS 5 interface. This brings up the screen shown in **Figure 6.4**.

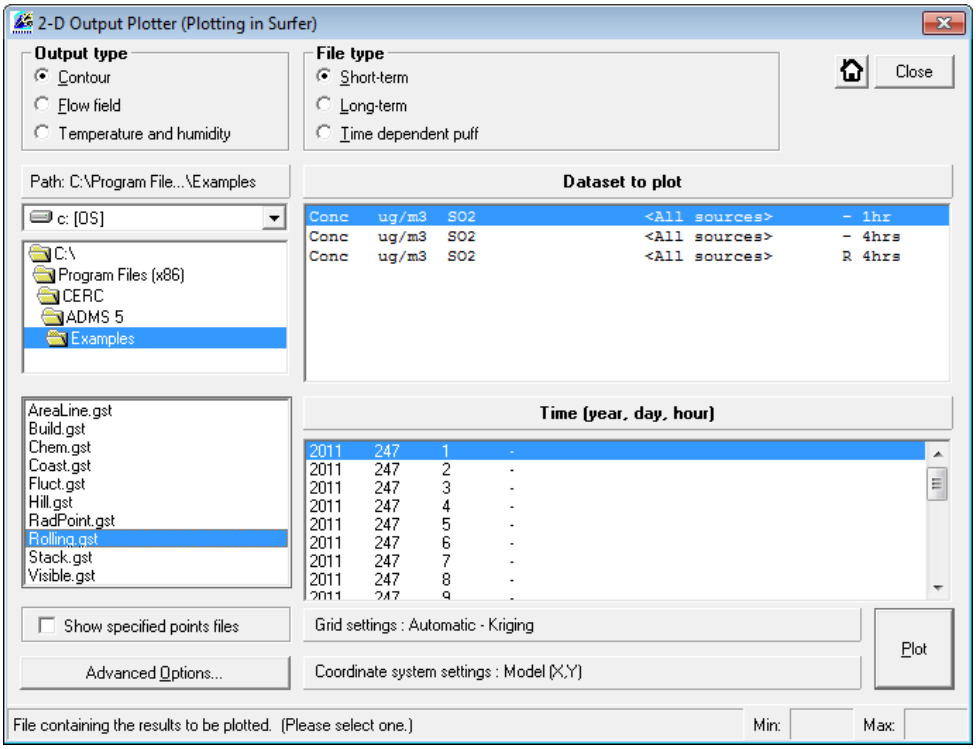


Figure 6.4 − The **2-D Output Plotter** screen.

The **2-D Output plotter** allows for plotting many different types of gridded output from ADMS 5. The **Output type** should first be selected:

- **Contour**: for plotting contour plots of data output to the *.gst* or *.glt* file including: concentration, deposition, temperature and humidity, gamma dose, etc.
- **Flow field**: for plotting contour or vector plots of flow field output from the **Complex terrain**, **Wind turbine** or **Urban canopy flow** options.
- **Temperature and humidity**: This option relates to output from the ADMS-Urban Temperature and Humidity model and is not used with ADMS 5 output. For plotting output from the ADMS 5 temperature and humidity option select **Contour**.

6.3.1 Contour

When **Contour** is selected as the **Output type** the **2-D Contour plotter** will look as shown in **Figure 6.4**. Here are the steps to follow in order to create a contour plot of data:

Step 1 Choose the file type from **Short term**, **Long term** and **Time dependent puff** (at the top of the screen). This defines the types of files listed in the lower left hand box. If **Short term** is chosen, all the files with the extension *.gst* or *.levels.gst* (in the chosen directory) are listed, if **Long term** is chosen, all the files with the extension *.glt* or *.levels.glt* are listed, and if **Time dependent puff** is chosen, all files with the extension *.gtd* or *.levels.gtd* are listed.

*Checking the **Show specified points files** box below the list of files also allows for specified points files (*.pst, .plt, .ptd*) to be plotted, providing they contain sufficient points.*

Step 2 Select the appropriate folder and click on the name of the file containing the data to plot. Right-clicking on **Path:** allows the path of a file or directory to be pasted in. Files or directories can also be dragged from Explorer onto **Path:** to use that file or directory.

Step 3 Click on the dataset to plot (**Dataset to Plot** box), and if relevant also click on the time for which the data are to be plotted (**Time (year, day, hour)** box).

The **Dataset to Plot** box shows a list of all the variables that can be used for the plot. Each record in the list shows the type of output (e.g. 'Conc' for concentration), the units of output (e.g. 'ppm'), the pollutant name (e.g. 'CO'), the source or group name (e.g. 'Power Station'), whether it is a rolling average ('R' or '-'), and the averaging time (e.g. '1hr'). For *.levels.glt* and *.levels.gst* files, an additional vertical height column is displayed (e.g. 'z=10.0m', as shown in **Figure 6.5**).

For short-term runs only, a **Time (year, day, hour)** box is also displayed. The box displays the time (year, day, hour) as given in the meteorological data irrespective of whether these data are entered from a file or by hand. If no time information is given in the meteorological data file, numbers corresponding to each consecutive meteorological condition are used. For example, numbers 1 to 7, corresponding to meteorological conditions A to G respectively, are listed when the example file *R91a-g.met* is used.

Step 4 Click on the **Advanced Options**... button to set some properties of the plot (optional – refer to Section 6.3.3 for details).

Step 5 Click on **Plot** to plot the selected data in Surfer.

The ADMS 5 output file is converted to a grid file (.grd*) for Surfer to plot. Save this in the directory where the* .apl *file is located. Once the Surfer drawing has been saved, the* .grd *file can be deleted as it is no longer required.*

In the upper right hand corner, the **Close** button closes the **ADMS Contour Plotter** and the  button brings the ADMS 5 interface to the front without closing the utility.

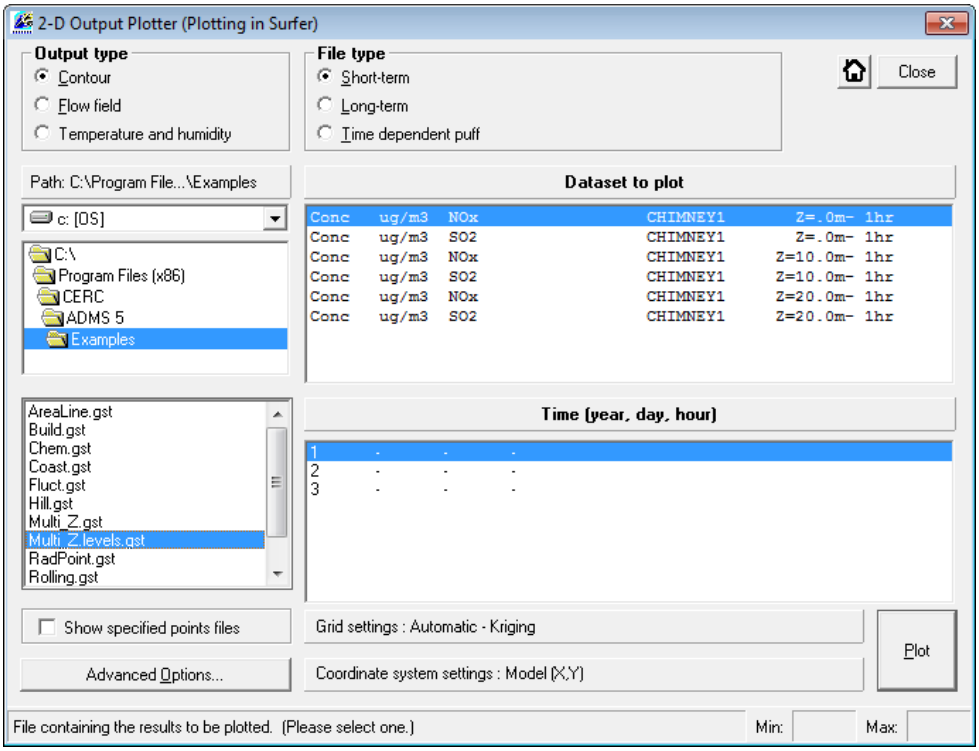


Figure 6.5 − The **2-D Output Plotter** screen showing output at multiple z-levels.

If you are creating many contour plots then you may wish to use the Surfer Automation tool. Details of this tool can be found in the Surfer Automation and Slideshow Creator Tools User Guide (SurferAutomationandSlideshowCreator_UserGuide.pdf) located in the Documents subdirectory of the ADMS 5 installation directory.

6.3.2 Flow field

The **Complex Terrain**, **Wind Turbine** and **Urban canopy flow** options include the ability to output flow fields. The flow field output files contain the wind and turbulence fields:

- for each line of meteorological data if short-term output is selected. These output files are described in Sections 6.1.13 and 6.1.15
- as averages of these quantities over all meteorological data lines if long-term is selected. These output files are described in Sections 6.1.14 and 6.1.16.

These files can be plotted by selecting **Flow field** in the **2-D Output plotter**, shown in **Figure 6.6**.

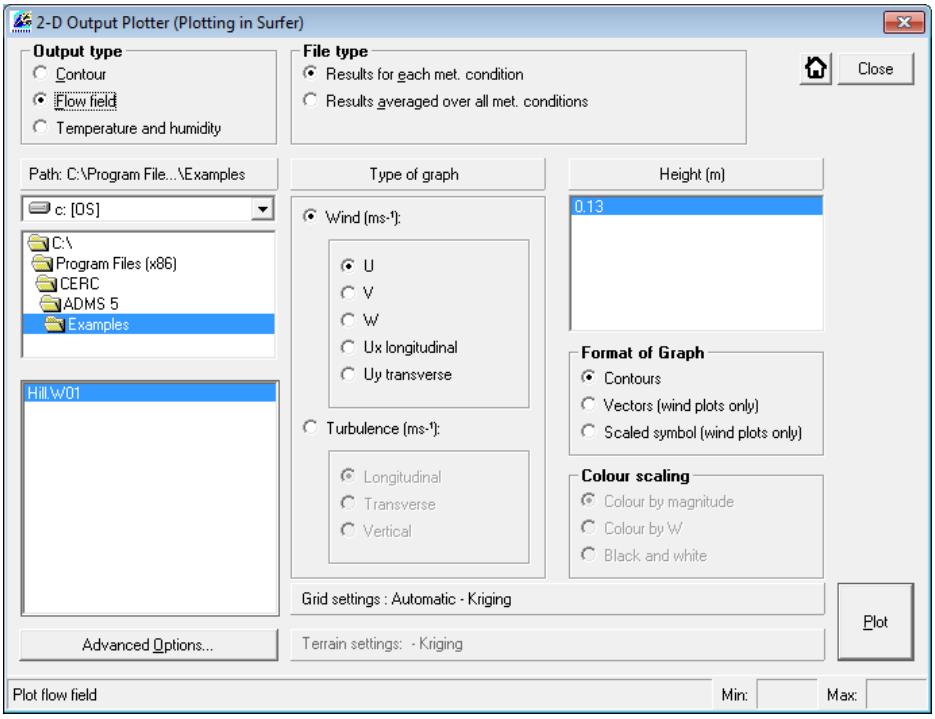


Figure 6.6 – The **2-D Output plotter** utility with **Flow field** selected.

In order to create a flow field plot, proceed as follows:

Step 1 Select whether to plot **Results for each met. condition** (short-term) or **Results averaged over all met. conditions** (long-term).

Step 2 In the **Type of graph** box, select the appropriate data to plot (see further on for a description of all variables available).

Step 3 Select whether a contour plot, a vector plot or a scaled symbol plot is to be made, from the **Format of Graph** box.

Step 4 If making a **Vectors (wind plots only)** plot select the colour scaling to use from the **Colour scaling** box.

Step 5 From the boxes on the left, select the folder and then the file that contains the data to plot. Right-clicking on **Path:** allows the path of a file or directory to be pasted in. Files or directories can also be dragged from Explorer onto **Path:** to use that file or directory.

Step 6 Choose the height from the **Height (m)** box.

Note that these heights are measured from the surface of the terrain.

Step 7 Click on **Plot** to display the plot in Surfer.

An example of a contour plot is shown in **Figure 6.7** and an example of a vector plot in **Figure 6.8**.

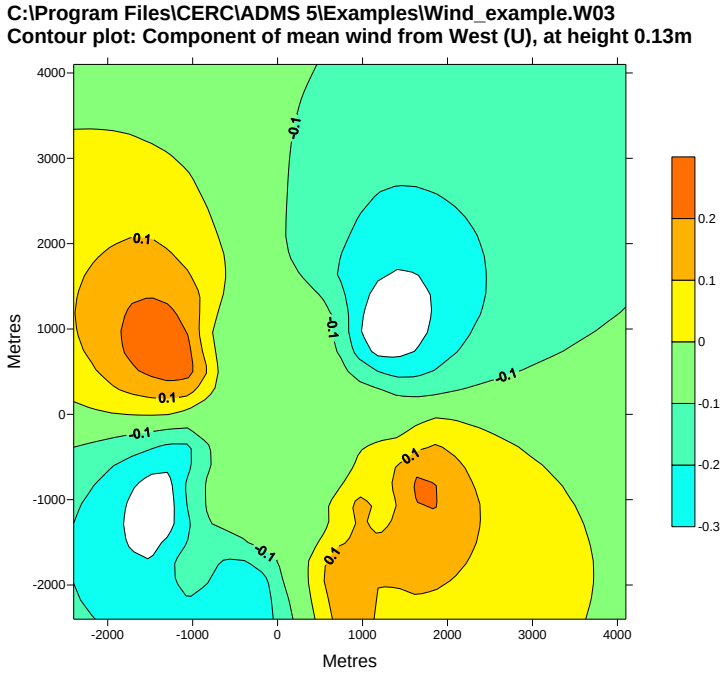


Figure 6.7 − Example flow field contour plot.

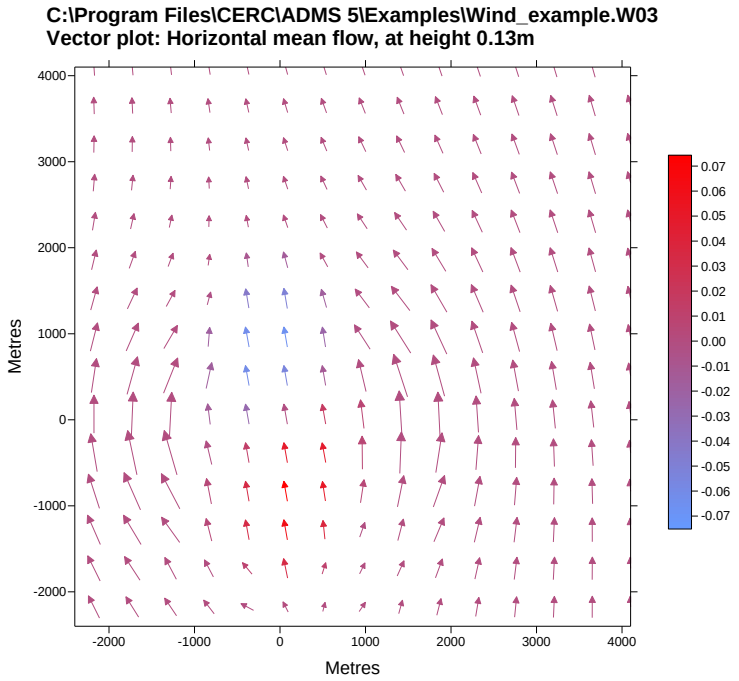


Figure 6.8 − Example flow field vector plot with the colour scaled by vertical velocity.

Type of graph

This section lists all the possible variables that can be plotted:

Wind (ms^{-1})

1. **U**: component of wind velocity in west-east direction.
2. **V**: component of wind velocity in south-north direction.
3. **W**: vertical component of wind velocity (relative to sea level).

4. **Ux longitudinal**: longitudinal component of wind velocity, i.e. in the direction of the upstream wind (the wind direction specified in the meteorological data). Short-term output only.
5. **Uy transverse**: transverse component of wind velocity, i.e. perpendicular to the direction of the upstream wind. Short-term output only.
6. **Mean Speed**: Mean value of the horizontal wind speed. Long-term output only.
7. **Mean vector magnitude**: Magnitude of the mean horizontal wind vector (refer to Section 6.1.14 for definition). Long-term output only.

Turbulence (ms^{-1})

1. **Longitudinal**: longitudinal turbulence parameter (σ_u).
2. **Transverse**: transverse turbulence parameter (σ_v).
3. **Vertical**: vertical turbulence parameter (σ_w).

Vector plot

The horizontal wind, i.e. the vectors (U,V), can also be displayed as a vector plot by selecting **Wind (ms^{-1})** in the **Type of graph** box and **Vectors (only for wind plots)** in the **Format of Graph** box. A choice of colour scaling is then available in the **Colour scaling** box:

- **Colour by magnitude**: the vectors are coloured by the horizontal wind speed
- **Colour by W**: the vectors are coloured by the vertical velocity
- **Black and white**: the vectors are not coloured

An example of a vector plot is shown in **Figure 6.8**.

Scaled symbol plot

The horizontal wind, i.e. the vectors (U,V), can also be displayed as a scaled symbol plot by selecting **Wind (ms^{-1})** in the **Type of graph** box and **Scaled symbol (only for wind plots)** in the **Format of Graph** box. The scaled symbol plot is similar to the vector plot but plots arrows at each of the locations in the output file rather than gridding the data as is carried out in the vector plot. An example plot is shown in Figure 6.9

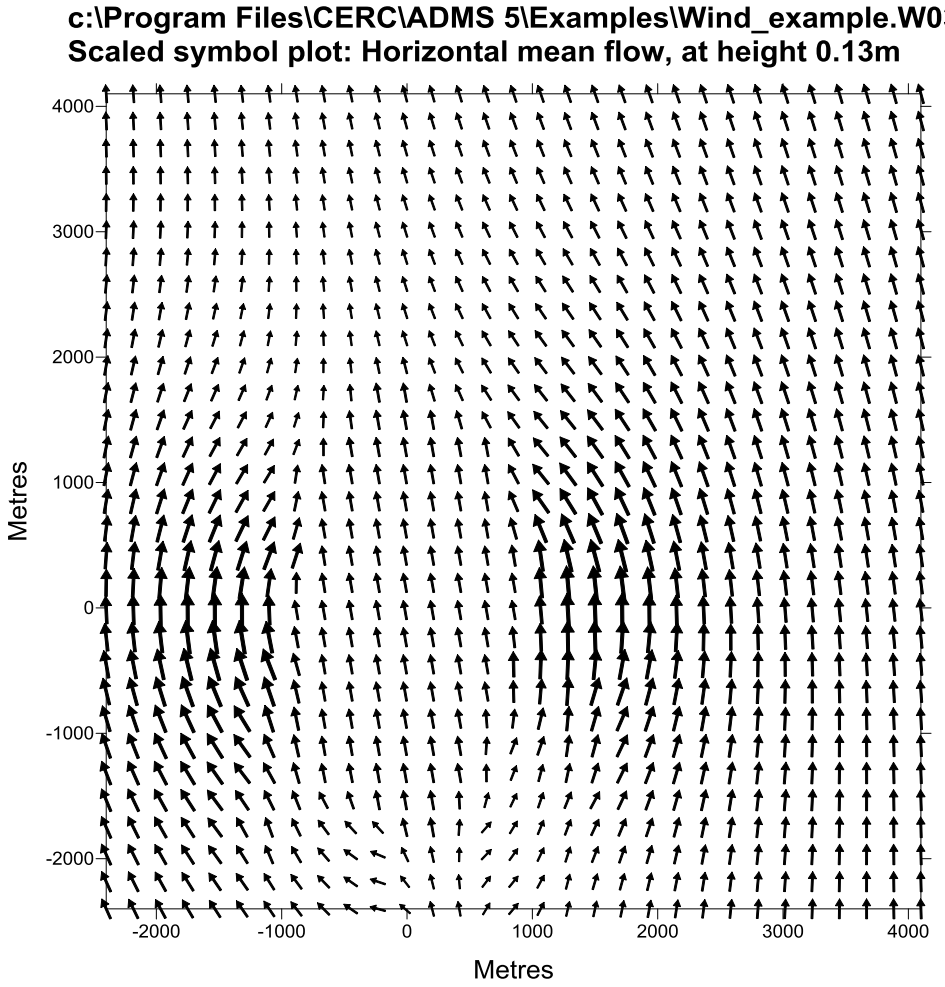


Figure 6.9 − Example scaled symbol plot.

6.3.3 Advanced options

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

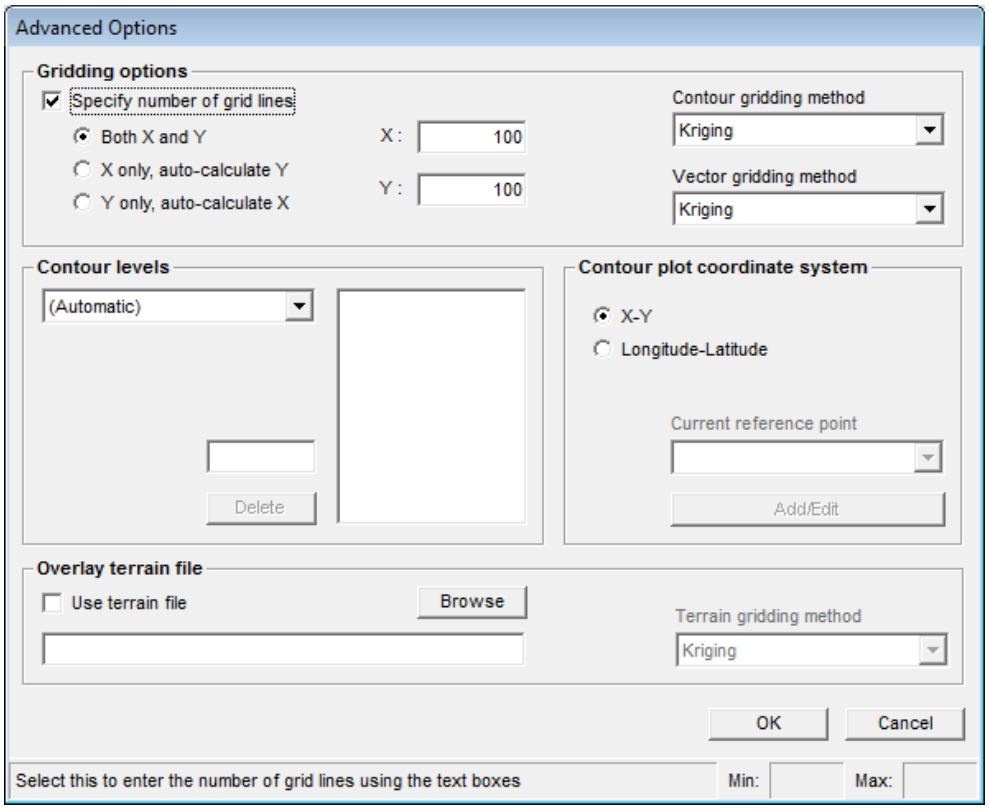


Figure 6.10 − The **Advanced Contour Options** screen.

When you have finished with the **Advanced Options** screen, click on the **OK** button to return to the **2-D Output Plotter (Plotting in Surfer)** main screen. Once the required information has been selected in this screen, click on **Plot** to create the plot.

*Some of the options on the **Advanced Options** screen are only available for some plot types.*

Further help with using Surfer is given in Appendix D.

Gridding options

A number of different gridding methods for a contour plot can be selected from the **Contour 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 results to be plotted.*

Contour levels

To choose the contour levels to plot, click on **(User-specified)** from the drop-down list.

For each level you want to plot (up to a maximum of 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, click on that number in the larger box and click on the **Delete** button.

Contour plot coordinate system

To convert your output from X-Y coordinates to longitude-latitude coordinates, select **Longitude-Latitude** and then click on **Add/Edit 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.

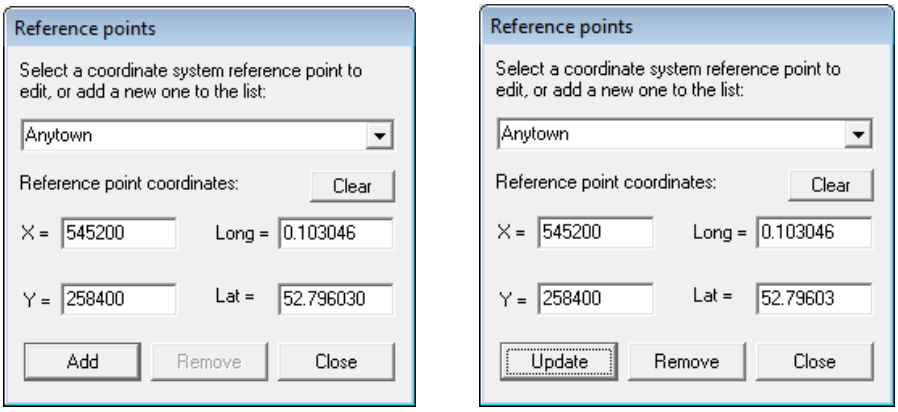


Figure 6.11 − Left: entering coordinates of a datum point. Right: revising and updating coordinates of a datum point.

In the UK, the X-Y coordinates will typically be national grid (OSGB) references.

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

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

Overlay terrain file

If the effects of hills have been included in the model run, it may be appropriate to overlay the plot on the terrain data. To do this, check the **Use terrain file** box, then click on the **Browse** button to locate the *.ter* file. The gridding method for the terrain plot can be chosen from the **Terrain gridding method** list. You should ensure that the terrain file is the right one for the plot being created, i.e. the plot is located within the terrain contours.

The **Grid settings** and the **Coordinate system** or **Terrain settings** selected on the **Advanced Options** screen are displayed at the bottom of the main screen, as shown at the bottom of **Figure 6.6**.

6.4 Line plots (ADMS Line Plotter)

ADMS 5 has an integrated line plot utility called **ADMS Line Plotter**. This can only be used with model runs where short-term gridded output for a single point source is selected. Concentration and plume dispersion data are plotted along a line of points extending in the downstream direction from the source, i.e. along the 'plume centreline', as far as the downstream edge of the output grid. This utility is only available for point or jet sources, because the plume 'centreline' is not well-defined for other source types.

The **ADMS Line Plotter** uses a specific type of file to produce the plots. The files have the extension *.!n* for plumes and time-integrated puffs, and *.~n* for time-dependent puffs, where *n* is a two-digit number corresponding to the line of meteorological data of the *.met* file used in the model run (see Section 6.1.21). These files contain the same information as the *.cen* file but they have been formatted so that they can be read by the graphical interface. *They are not intended to be read and/or interpreted by the user.*

To launch the **ADMS Line Plotter**, select **Results**, **Line Plot** from the main ADMS 5 interface. This brings up the screen shown in **Figure 6.12**.

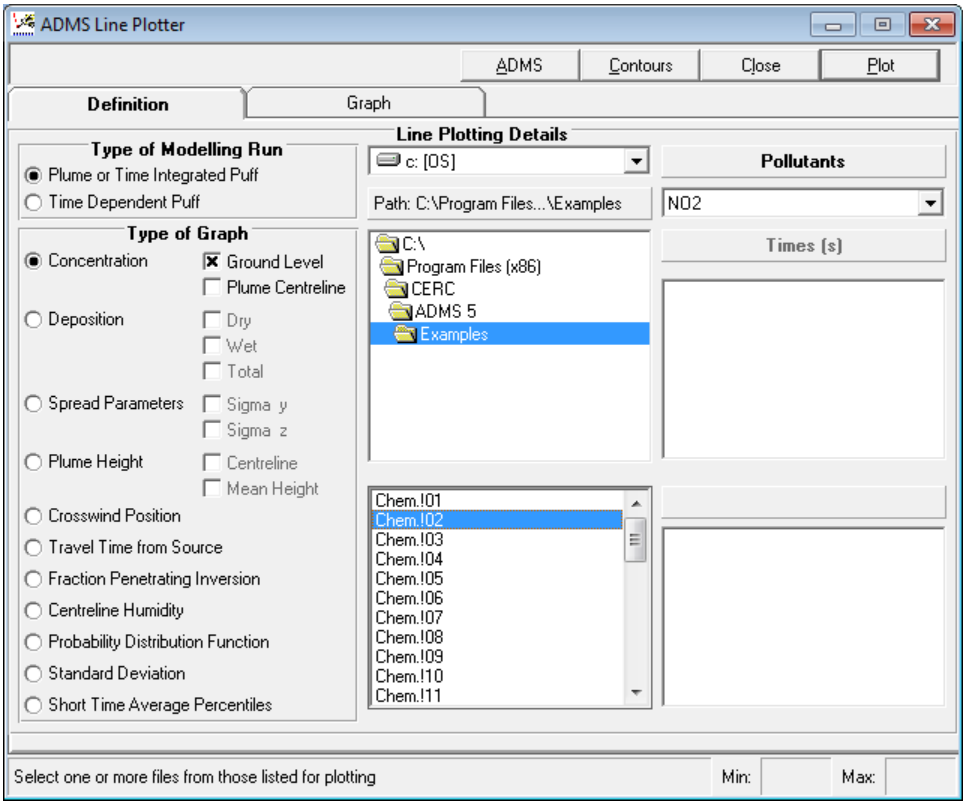


Figure 6.12 − The **ADMS Line Plotter** utility, **Definition** screen.

6.4.1 Main interface

The utility is composed of two screens: the **Definition** screen to select the data to plot, and the **Graph** screen to display and customise the plot. The **Plot** button plots the selected data. The **Close** button closes the utility. The **ADMS** button brings the ADMS 5 interface to the front, keeping the utility opened. The **Contours** button opens the **2-D Output Plotter** utility.

To use the **ADMS Line Plotter**, proceed as follows:

Step 1 Select the required output from the **Definition** screen.

In the **Type of Modelling Run** box, choose the option relevant for the type of run. If the **Plume or Time Integrated Puff** option is chosen, the files with the *.!n* extension in the current directory are displayed in the lower centre box of the screen. If the **Time Dependent Puff** option is chosen, those with the *.~n* extension are displayed.

Step 2 Select the appropriate folder and click on the file containing the data to plot.

*Up to ten output files from the list displayed in the screen can be plotted at the same time. Use the **SHIFT** and **CTRL** keys to select which files to plot.*

Step 3 In the **Type of Graph** area, choose the variable to plot (see further).

More than one variable can be plotted at the same time.

Step 4 Click on the **Plot** button. The screen switches to the **Graph** screen where the data are plotted (see **Figure 6.13**).

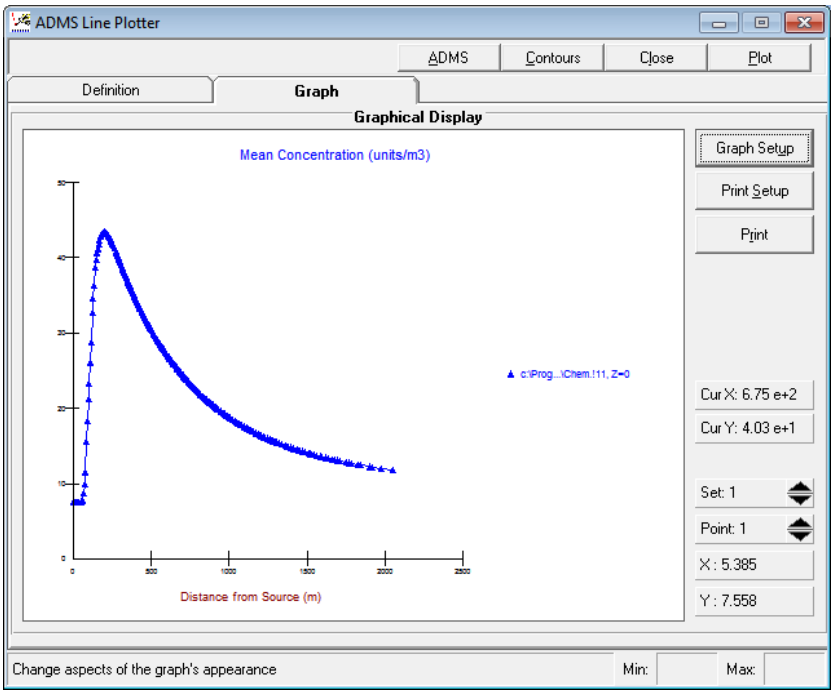


Figure 6.13 − The **ADMS Line Plotter** utility, **Graph** screen. Example of plot.

Types of graph

This section lists all the possible variables that can be plotted.

4. **Concentration** (**Ground Level**): ground-level concentration beneath the plume centreline.

   Concentration (**Plume Centreline**): plume centreline concentration.

If there is an inversion at the top of the boundary layer that splits the plume in two, the plume centreline concentration is for the part of the plume below the inversion. When the **Buildings** option is selected, the centreline concentration of the elevated plume is given.

5. **Deposition** (**Dry**, **Wet**, **Total**): ground-level deposition fluxes (plume or time-dependent puff) or ground-level deposits (time-integrated puff) below the plume centreline.

   These output options should only be used if the **Wet deposition** and/or **Dry deposition** options were selected in the **Setup** screen of the ADMS 5 interface.

6. **Spread Parameters** (**Sigma y**): lateral plume spread

   Spread Parameters (**Sigma z**): vertical plume spread

   If there is an inversion at the top of the boundary layer that splits the plume in two, this output is for the part of the plume below the inversion. When the **Buildings** option is selected, these outputs apply to the elevated plume only.

7. **Plume Height** (**Centreline**): height of the plume centreline,

   Plume Height (**Mean Height**): mean height of the plume.

   If there is an inversion at the top of the boundary layer that splits the plume in two, the output is for the part of the plume below the inversion. When the **Buildings** option is selected, centreline and mean plume heights are given for the elevated plume only.

8. **Crosswind Position**: crosswind position of the plume centreline.
9. **Travel Time from Source**: time taken for plume material to travel from the source to a downstream location.
10. **Fraction Penetrating Inversion**: fraction of the plume that has penetrated above the top of the boundary layer.
11. **Centreline Humidity**: liquid content of the plume at its centreline given as the mass of water vapour per unit mass of dry plume (units kg/kg).

    This output option should only be used if the **Plume visibility** option has been selected in the **Setup** screen of the ADMS 5 interface.

12. **Probability Distribution Function**: when this is selected, all PDF concentrations input in the **Fluctuations** screen are listed in the lower right hand box labelled **PDF**. Select the concentration for which you wish to plot the PDF.

    This output option should only be used if the **Fluctuations** option has been selected in the **Setup** screen of the ADMS 5 interface.

13. **Standard Deviation**: standard deviation of the concentration profile of the plume.

    This output option should only be used if the **Fluctuations** option has been selected in the **Setup** screen of the ADMS 5 interface.

14. **Short Time Average Percentiles**: when this is selected, all percentiles input in the

Fluctuations screen are listed in the lower right hand box labelled **Percentiles**. Choose the percentile you wish to plot by selecting it from this box.

This output option should only be used if the **Fluctuations** option has been selected in the **Setup** screen of the ADMS 5 interface.

6.4.2 Graph customisation

The appearance of the plot can be altered by clicking on the **Graph Setup** button available on the **Graph** screen (see **Figure 6.13**). It brings up the **Graph Design** dialogue box shown in **Figure 6.14**.

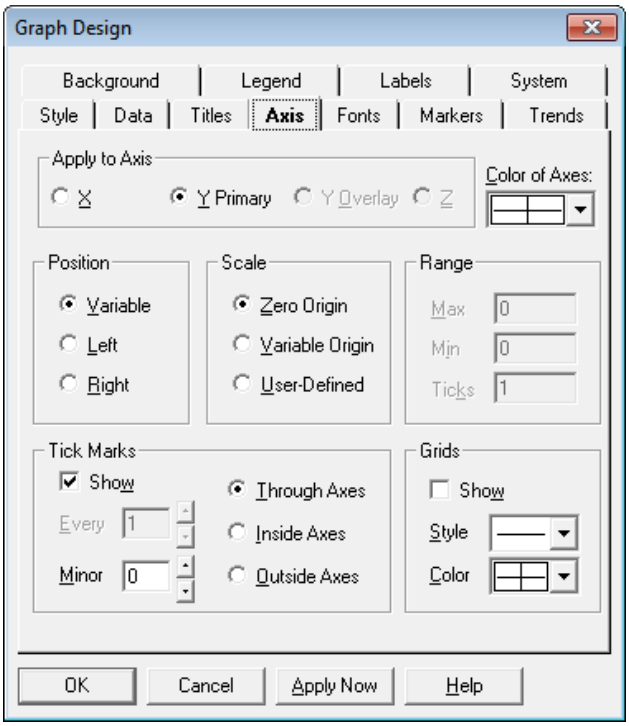


Figure 6.14 − The **Graph Design** box of the **ADMS Line Plotter** utility.

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

1. **Background**: allows the user to alter the background colour of the graph and the style of borders drawn around titles, the legend and the graph.
2. **Legend**: allows the user to select the legend text, size and position.
3. **Labels**: allows the user to turn axis labels and data point labels on and off, and change their format.
4. **System**: allows the user to print a graph and save a graph (data and/or graph layout) to file.
5. **Style**: allows the user to select how the data are to be displayed on the graph (e.g. just points or with lines joining the points), and change the scale of the axes to a log scale.
6. **Data**: allows the user to add or remove data points and control the range of data points plotted.

7. **Titles**: allows the user to write/edit titles and change the orientation of titles. The limit of 80 characters in the title is set by the graphics software.
8. **Axis**: allows the user to alter the minimum/maximum values of axes and the number of ticks. To do this, choose the axis in the **Apply to Axis** box. Click on **User-Defined** in the **Scale** box and enter the appropriate minimum/maximum values and number of ticks in the **Range** box. Click on **Apply Now**. If the graph is correct, click on **OK**. Labels and their orientation are set on the **Labels** screen.
9. **Fonts**: allows the user to set the font of the graph title, axes labels and legends.
10. **Markers**: allows the user to alter the symbol used in the graph to denote the data points, change the size and colour of these symbols, and adjust the colour and thickness of the line and whether it is dashed or solid.
11. **Trends**: allows the user to add limit lines, statistical lines and fit a curve to data points.

6.5 Viewing numerical data

The numerical output files created by ADMS 5 are mainly comma-separated text files. Input and output files can be accessed from the ADMS 5 interface (**View** buttons for the input files, **Results**, **Numerical**... menu option for the output files), and viewed in WordPad, Notepad, Microsoft Excel or some other application of the user's choice. The application used to view files is set in the **Preferences** menu as described in Section 6.5.1.

An alternative way of opening the output files in Microsoft Excel is outlined in Section 6.5.3.

6.5.1 Choosing application to view output

To change the preferred viewing application for input data and output files, select **File**, **Preferences**, **Viewing options** from the menu bar of the ADMS 5 interface. This will bring up the **File Viewing Preferences** screen shown in **Figure 6.15**.

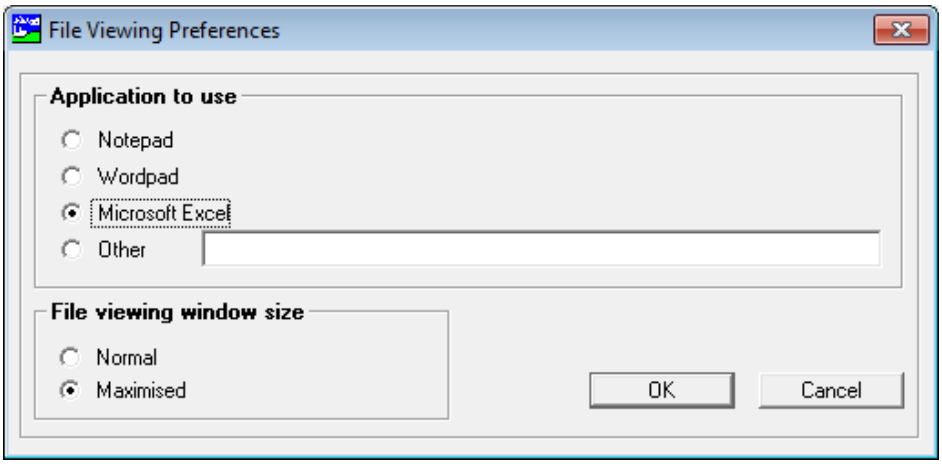


Figure 6.15 − The **File Viewing Preferences** window.

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 last used window) or **Maximised** (full screen).

6.5.2 Viewing output from the ADMS 5 interface

The output files associated with the *.apl* file that is currently open in the interface can be viewed using the **Results**, **Numerical**... menu command of the interface. The output file is then opened with the application selected in the **File Viewing Preferences**, as described in Section 6.5.1. A dialogue box will appear showing all of the output files for the current *.apl* file.

If the file is being opened in Microsoft Excel, it will automatically be opened as a comma-separated file.

6.5.3 Use of Microsoft Excel to view numerical output

The numerical output files can be opened using Microsoft Excel without using the ADMS 5 interface, proceeding as follows. These instructions relate to Microsoft Excel 2007.

Step 1 Start Microsoft Excel.

Step 2 Click on the Office button and select **Open**, and browse through directories to locate the relevant directory.

Step 3 Type the file name extension of the required output file (e.g. **.glt* if searching for *OneDay.glt*) in the **File name** box.

Click on the file to open and then click **Open**. This starts the Microsoft Excel **Text Import Wizard** shown in **Figure 6.16**.

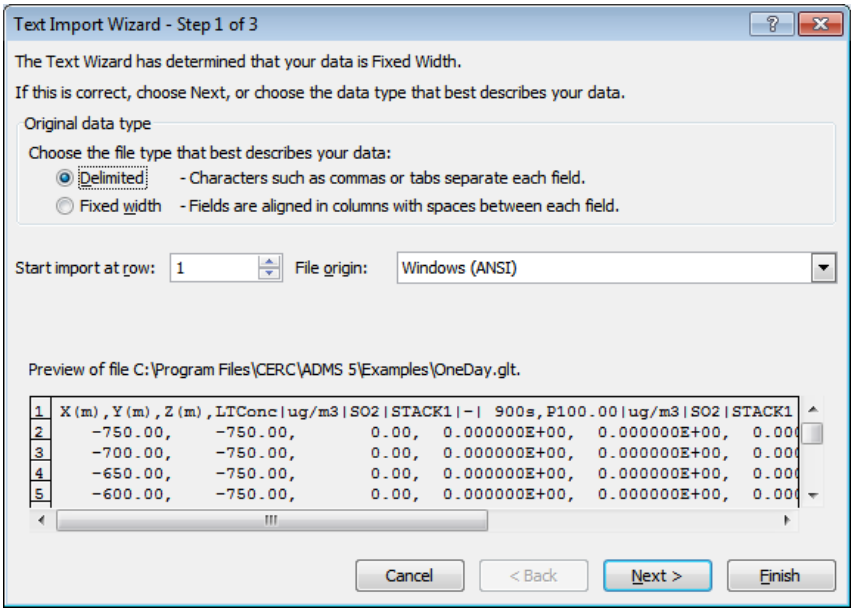


Figure 6.16 − Step 1 of the Microsoft Excel **Text Import Wizard**.

Step 4 In the screen shown in **Figure 6.16**, select the **Delimited** option, ensure that **File origin** is set to **Windows (ANSI)** and click on **Next >** to move to the next step (see **Figure 6.17**).

Double-click on the option to select it and to move directly to the next step.

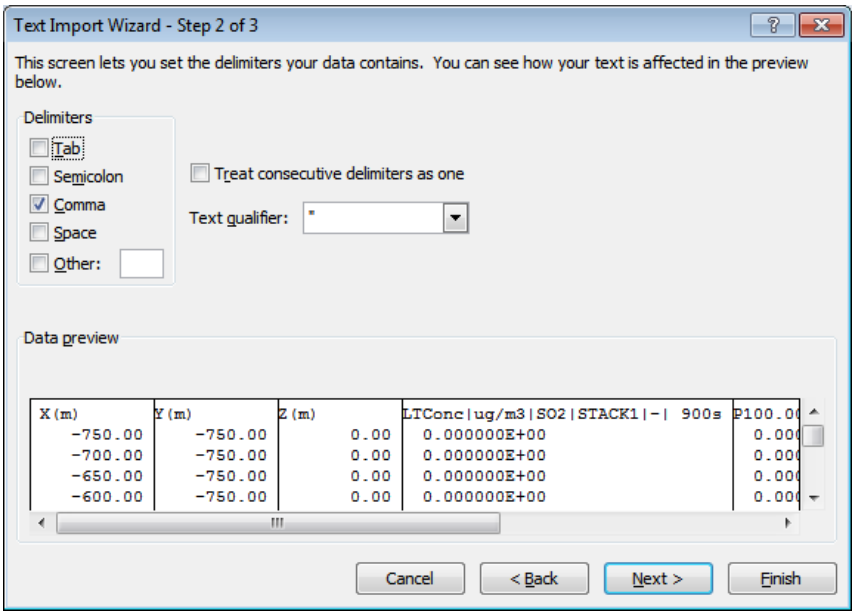


Figure 6.17 − Step 2 of the Microsoft Excel **Text Import Wizard**.

Step 5 In the **Delimiters** box, select the **Comma** check box and click on **Next >** to move to the last step (see **Figure 6.18**).

Double-click on the option to select it and go directly to the next step.

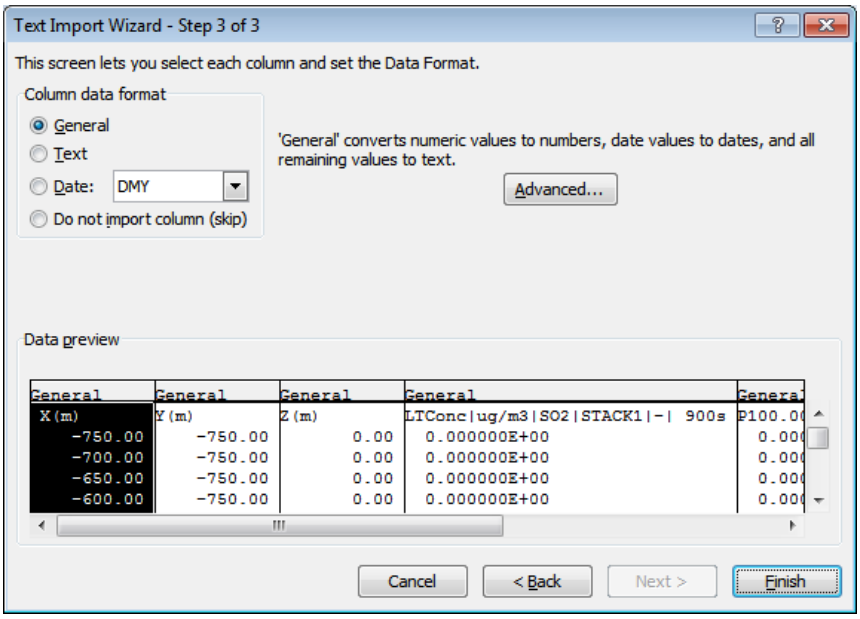
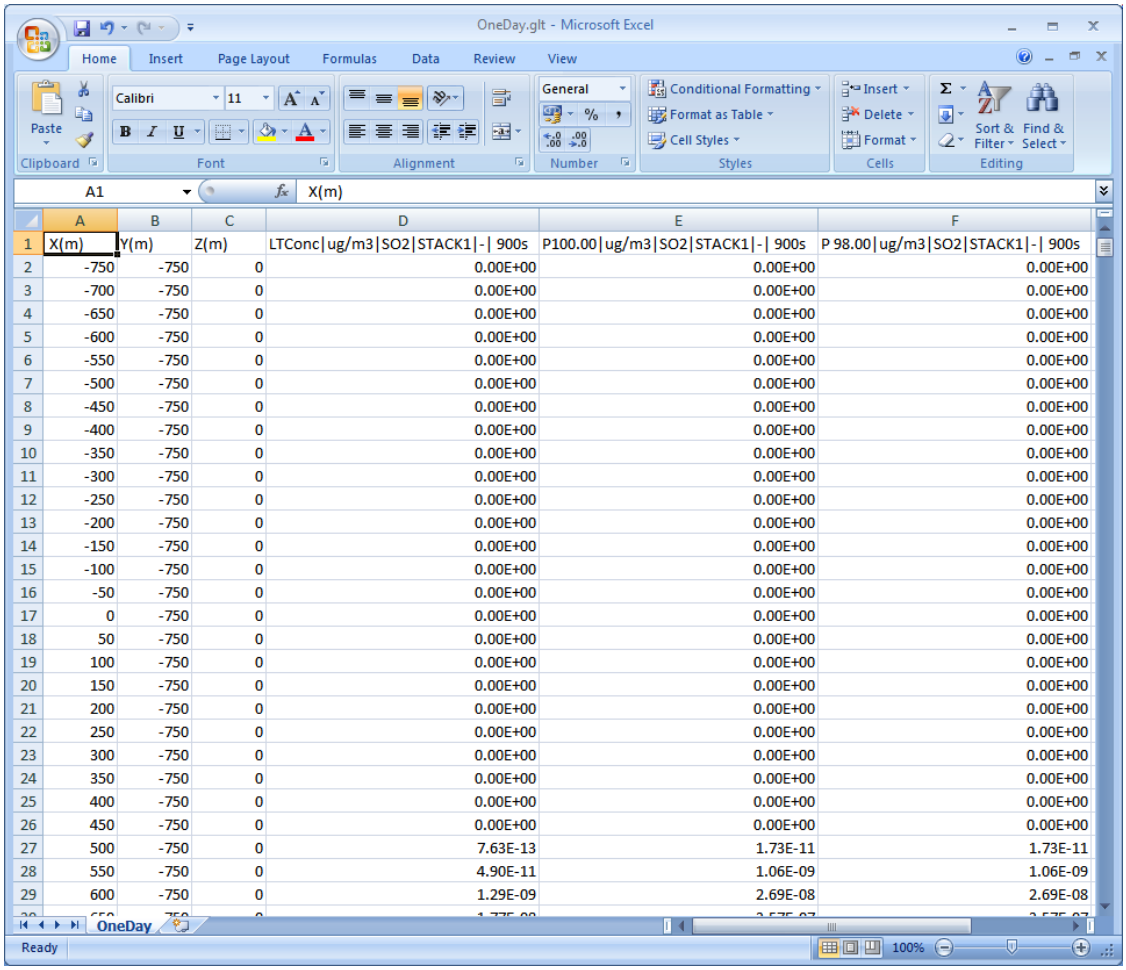


Figure 6.18 − Step 3 of the Microsoft Excel **Text Import Wizard**.

Step 6 In the **Data Preview** box, ensure that the file has been properly separated and click on **Finish** to import the data.

Imported data in Microsoft Excel are shown in **Figure 6.19**.

If the column headings are not visible because the columns are too narrow, select all the columns containing data and click on the **Format** button in the **Cells** section of the **Home** tab on the ribbon and select **Autofit Column Width** to increase the column widths to fit the longest entry in each column.



	A	B	C	D	E	F
1	X(m)	Y(m)	Z(m)	LTConc|ug/m3|SO2|STACK1|-| 900s	P100.00|ug/m3|SO2|STACK1|-| 900s	P 98.00|ug/m3|SO2|STACK1|-| 900s
2	-750	-750	0	0.00E+00	0.00E+00	0.00E+00
3	-700	-750	0	0.00E+00	0.00E+00	0.00E+00
4	-650	-750	0	0.00E+00	0.00E+00	0.00E+00
5	-600	-750	0	0.00E+00	0.00E+00	0.00E+00
6	-550	-750	0	0.00E+00	0.00E+00	0.00E+00
7	-500	-750	0	0.00E+00	0.00E+00	0.00E+00
8	-450	-750	0	0.00E+00	0.00E+00	0.00E+00
9	-400	-750	0	0.00E+00	0.00E+00	0.00E+00
10	-350	-750	0	0.00E+00	0.00E+00	0.00E+00
11	-300	-750	0	0.00E+00	0.00E+00	0.00E+00
12	-250	-750	0	0.00E+00	0.00E+00	0.00E+00
13	-200	-750	0	0.00E+00	0.00E+00	0.00E+00
14	-150	-750	0	0.00E+00	0.00E+00	0.00E+00
15	-100	-750	0	0.00E+00	0.00E+00	0.00E+00
16	-50	-750	0	0.00E+00	0.00E+00	0.00E+00
17	0	-750	0	0.00E+00	0.00E+00	0.00E+00
18	50	-750	0	0.00E+00	0.00E+00	0.00E+00
19	100	-750	0	0.00E+00	0.00E+00	0.00E+00
20	150	-750	0	0.00E+00	0.00E+00	0.00E+00
21	200	-750	0	0.00E+00	0.00E+00	0.00E+00
22	250	-750	0	0.00E+00	0.00E+00	0.00E+00
23	300	-750	0	0.00E+00	0.00E+00	0.00E+00
24	350	-750	0	0.00E+00	0.00E+00	0.00E+00
25	400	-750	0	0.00E+00	0.00E+00	0.00E+00
26	450	-750	0	0.00E+00	0.00E+00	0.00E+00
27	500	-750	0	7.63E-13	1.73E-11	1.73E-11
28	550	-750	0	4.90E-11	1.06E-09	1.06E-09
29	600	-750	0	1.29E-09	2.69E-08	2.69E-08

Figure 6.19 − ADMS 5 output data imported into Microsoft Excel.

6.5.4 Use of Microsoft Excel to create a time series graph

Microsoft Excel can also be used to produce a time series graph of short-term concentration or other outputs, at each of the specified output points. To produce such a graph, follow these steps, again these instructions relate to Microsoft Excel 2007:

Step 1 Open a *.pst* output file in Microsoft Excel as described in Section 6.5.3.

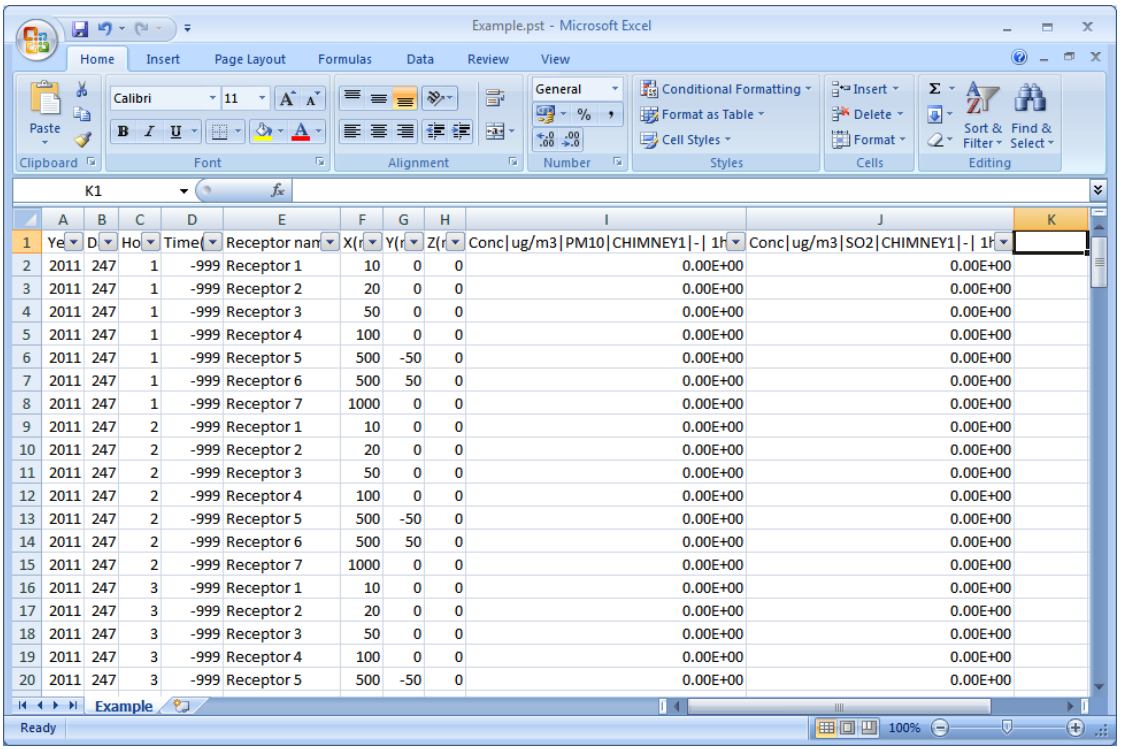
Step 2 Select all columns containing data (e.g. columns A:J in **Figure 6.20**) and choose **Filter** from the **Sort & Filter** section of the **Data** tab of the ribbon.

Filters are activated and represented by the arrow box in each of the heading cells.

Step 3 Filter the data as required (e.g. all values for a specific receptor) by clicking on the appropriate arrows. For example, the 'Receptor 3' value as shown in **Figure 6.21**.

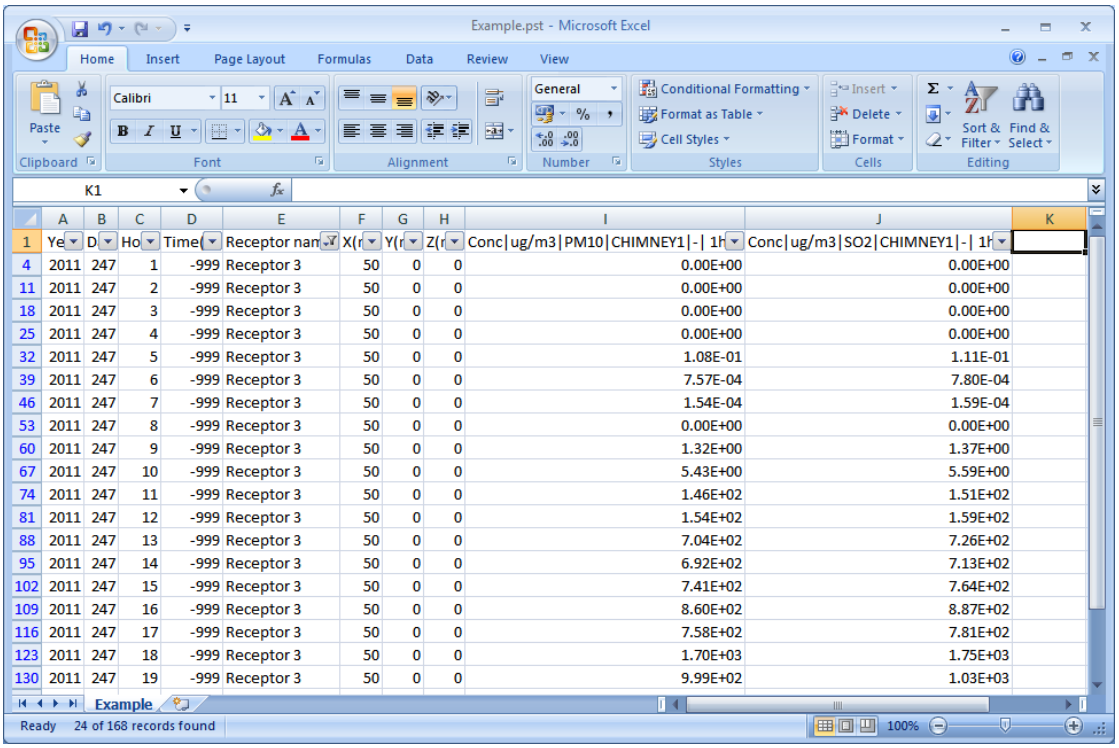
Step 4 Select the columns to be plotted by clicking on these columns while holding down the Control key. Then choose the chart type you want from the **Charts** section of the **Insert** tab of the **Ribbon**.

Step 5 The chart will appear and can be customized by first selecting the chart and then using the three tabs that appear under **Chart Tools** on the ribbon.



	A	B	C	D	E	F	G	H	I	J	K
1	Ye	D	Ho	Time(	Receptor nan	X(r	Y(r	Z(r	Conc|ug/m3|PM10|CHIMNEY1|-| 1h	Conc|ug/m3|SO2|CHIMNEY1|-| 1h	
2	2011	247	1	-999	Receptor 1	10	0	0	0.00E+00	0.00E+00	
3	2011	247	1	-999	Receptor 2	20	0	0	0.00E+00	0.00E+00	
4	2011	247	1	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
5	2011	247	1	-999	Receptor 4	100	0	0	0.00E+00	0.00E+00	
6	2011	247	1	-999	Receptor 5	500	-50	0	0.00E+00	0.00E+00	
7	2011	247	1	-999	Receptor 6	500	50	0	0.00E+00	0.00E+00	
8	2011	247	1	-999	Receptor 7	1000	0	0	0.00E+00	0.00E+00	
9	2011	247	2	-999	Receptor 1	10	0	0	0.00E+00	0.00E+00	
10	2011	247	2	-999	Receptor 2	20	0	0	0.00E+00	0.00E+00	
11	2011	247	2	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
12	2011	247	2	-999	Receptor 4	100	0	0	0.00E+00	0.00E+00	
13	2011	247	2	-999	Receptor 5	500	-50	0	0.00E+00	0.00E+00	
14	2011	247	2	-999	Receptor 6	500	50	0	0.00E+00	0.00E+00	
15	2011	247	2	-999	Receptor 7	1000	0	0	0.00E+00	0.00E+00	
16	2011	247	3	-999	Receptor 1	10	0	0	0.00E+00	0.00E+00	
17	2011	247	3	-999	Receptor 2	20	0	0	0.00E+00	0.00E+00	
18	2011	247	3	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
19	2011	247	3	-999	Receptor 4	100	0	0	0.00E+00	0.00E+00	
20	2011	247	3	-999	Receptor 5	500	-50	0	0.00E+00	0.00E+00	

Figure 6.20 − Filtering ADMS 5 output files in Excel.



	A	B	C	D	E	F	G	H	I	J	K
1	Ye	D	Ho	Time(	Receptor nan	X(r	Y(r	Z(r	Conc|ug/m3|PM10|CHIMNEY1|-| 1h	Conc|ug/m3|SO2|CHIMNEY1|-| 1h	
4	2011	247	1	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
11	2011	247	2	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
18	2011	247	3	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
25	2011	247	4	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
32	2011	247	5	-999	Receptor 3	50	0	0	1.08E-01	1.11E-01	
39	2011	247	6	-999	Receptor 3	50	0	0	7.57E-04	7.80E-04	
46	2011	247	7	-999	Receptor 3	50	0	0	1.54E-04	1.59E-04	
53	2011	247	8	-999	Receptor 3	50	0	0	0.00E+00	0.00E+00	
60	2011	247	9	-999	Receptor 3	50	0	0	1.32E+00	1.37E+00	
67	2011	247	10	-999	Receptor 3	50	0	0	5.43E+00	5.59E+00	
74	2011	247	11	-999	Receptor 3	50	0	0	1.46E+02	1.51E+02	
81	2011	247	12	-999	Receptor 3	50	0	0	1.54E+02	1.59E+02	
88	2011	247	13	-999	Receptor 3	50	0	0	7.04E+02	7.26E+02	
95	2011	247	14	-999	Receptor 3	50	0	0	6.92E+02	7.13E+02	
102	2011	247	15	-999	Receptor 3	50	0	0	7.41E+02	7.64E+02	
109	2011	247	16	-999	Receptor 3	50	0	0	8.60E+02	8.87E+02	
116	2011	247	17	-999	Receptor 3	50	0	0	7.58E+02	7.81E+02	
123	2011	247	18	-999	Receptor 3	50	0	0	1.70E+03	1.75E+03	
130	2011	247	19	-999	Receptor 3	50	0	0	9.99E+02	1.03E+03	

Figure 6.21 − Filtering ADMS 5 output files in Excel, filter on the receptor name.

6.6 Model run reports

The reporting files produced during a model or verification run for the currently open *.apl* file can be viewed from the ADMS 5 interface at any time by selecting **Reports from model run** from the **Results** menu. This will bring up the file verification results screen, as shown in **Figure 6.22**.

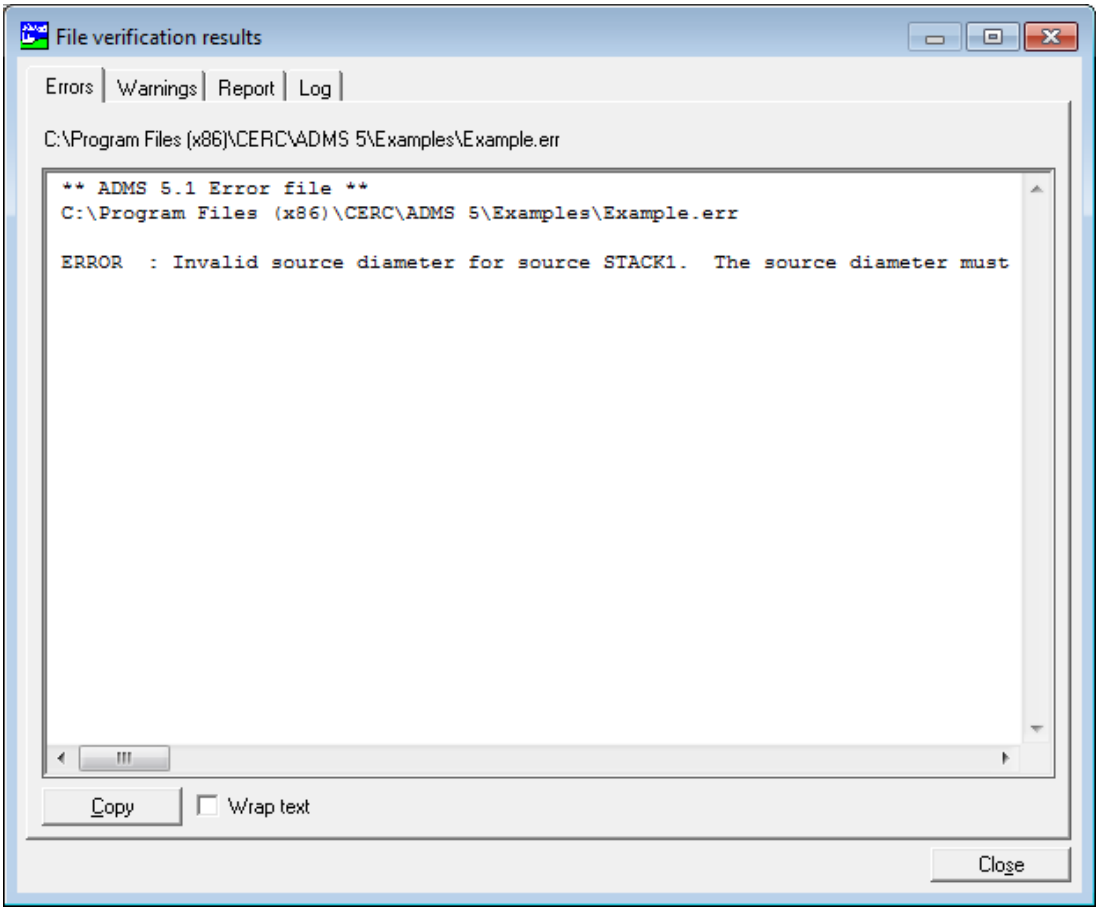


Figure 6.22 − The **File verification results** screen.

This window shows each of the four different reporting files in separate tabs (these files are described in more detail in Section 6.1.1). If a file does not exist for the current *.apl* (for instance, if there were no errors found in verification) then the corresponding tab will not appear.

On each tab the horizontal and vertical scroll bars can be used to view the information outside the current visible area. Alternatively, checking the **Wrap text** box will cause the text in each tab to be wrapped (each line broken to fit horizontally in the window) so that only vertical scrolling is required. The **Copy** button can be used to copy all of the text in the current tab to the clipboard.

SECTION 7 Utilities

This section describes the utilities listed in the menu **Utilities** of the ADMS 5 interface (shown in **Figure 7.1**), i.e.:

1. **Open Mapper**
2. **Visualise input in Surfer...**
3. **Convert met. data**
4. **Create terrain file**
5. **Download terrain data**
6. **Make/Run batch file**
7. **View a wind rose**
8. **Create import templates**

These utilities are divided into visualisation tools (numbers 1, 2 and 7), input data processing tools (numbers 3, 4 and 6) and an option for creating template files for **Import** (8, refer to Section 5.1.7 for more details). A link is also provided to a website where SRTM[1] terrain data can be downloaded (5). SRTM data can be converted into a format suitable for use with ADMS 5 by using the ADMS Mapper, refer to Section 5.6 of the ADMS Mapper User Guide for more details. There are also options to **Start Excel** and **Start Surfer** which launch Microsoft Excel and the Surfer application. Help and tips on the use of Surfer are provided in Appendix D.

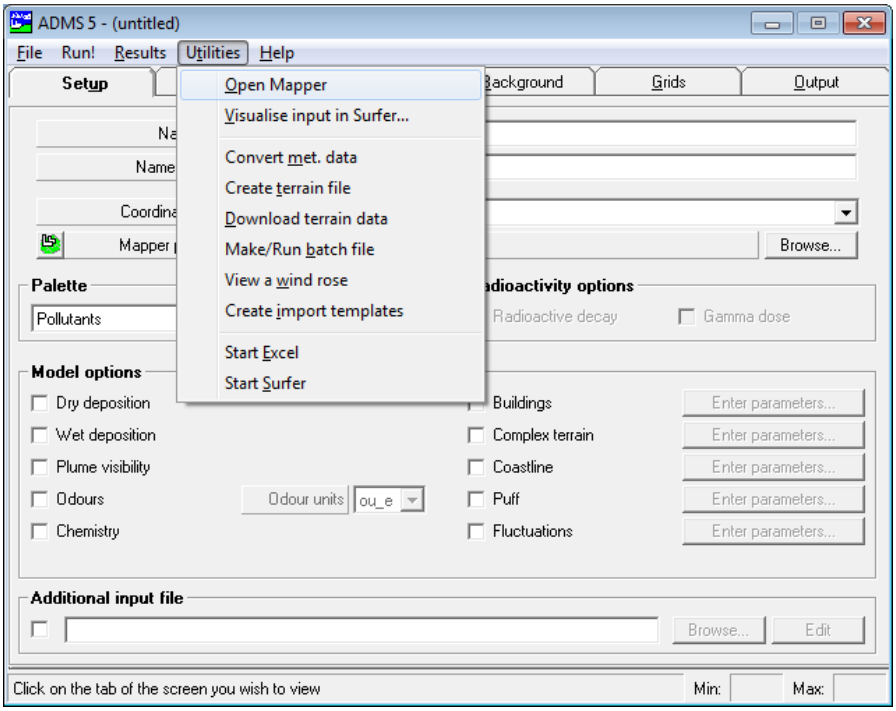


Figure 7.1 – The **Utilities** menu.

[1] Shuttle Radar Topography Mission data on a near global scale from 56° S to 60° N.

7.1 ADMS Mapper

The **ADMS Mapper** is an integrated mapping tool for displaying and editing source data, building and receptor locations and viewing results. It is launched with the **Utilities**, **Open Mapper** menu command or the button on the **Setup** screen and illustrated in **Figure 7.2**.

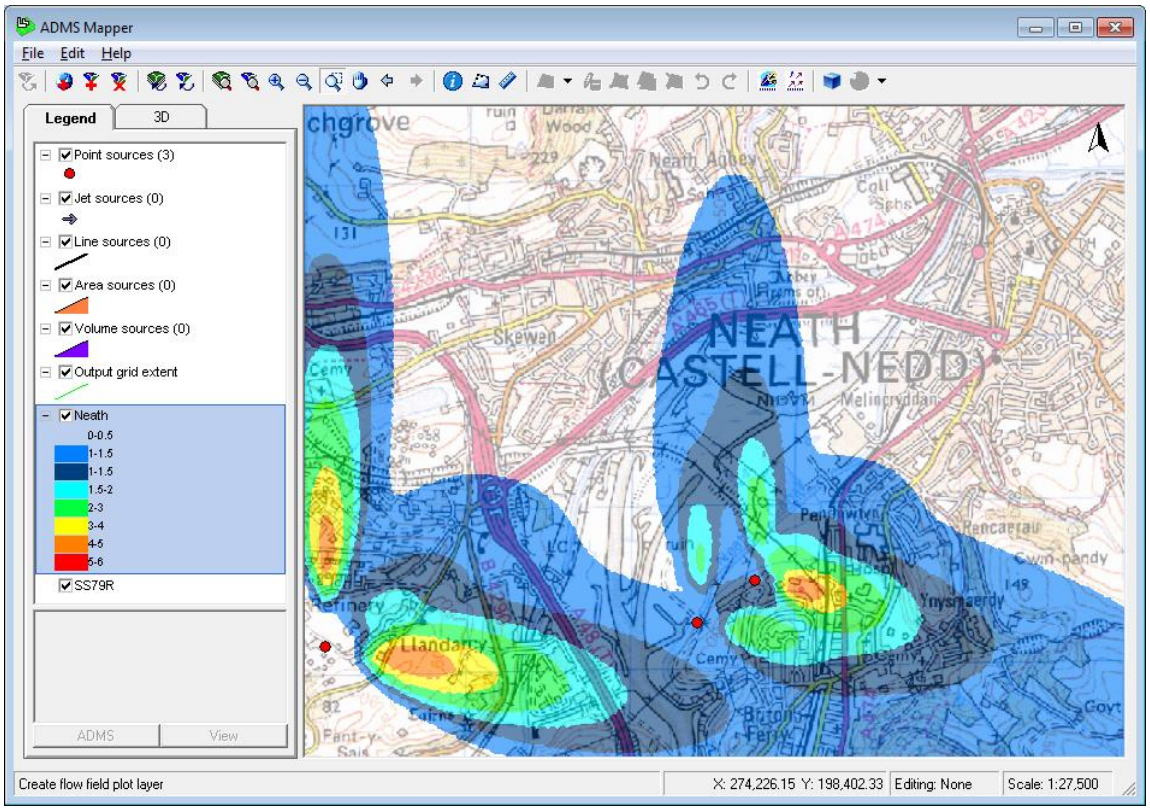


Figure 7.2 – The **ADMS Mapper** interface.

Some of the useful features in the **ADMS Mapper** allow the user to:

- create new sources of every type by drawing;
- create buildings and receptor points by drawing;
- edit the geometry of existing sources, buildings, and receptor points;
- view receptor point (*.asp*), terrain (*.ter*) and surface roughness (*.ruf*) files;
- create contour and vector plots;
- create terrain files, e.g from OS Terrain 50 or SRTM data;
- export sources or buildings to shape files or MapInfo MIF files;
- create images for inclusion in reports showing contour plots, positions of sources and buildings; and
- view map image tiles, shape files, MapInfo files, AutoCAD DXF files, etc.

Please refer to the *ADMS Mapper User Guide*, available from the **Help\User Guide** menu option in the ADMS Mapper, for instructions.

7.2 Visualising input data in Surfer

ADMS 5 includes a visualisation utility, which is accessed from the **Utilities**, **Visualise Input in Surfer**... menu (as illustrated in **Figure 7.3**).

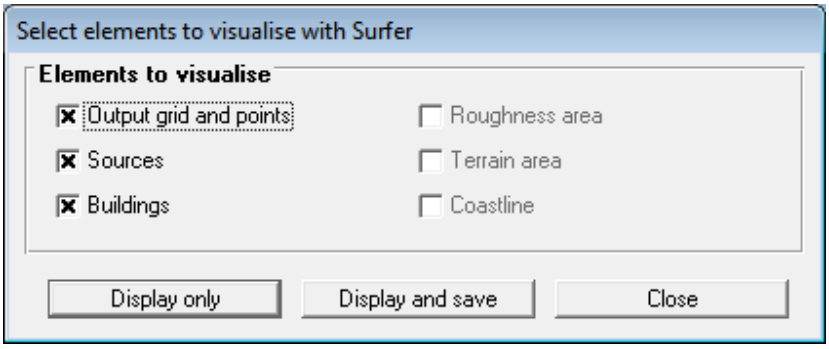


Figure 7.3 – Dialogue box to select the elements to visualise.

This utility allows the user to visualise the positions of various features and their relation to one another. This provides a very useful mechanism for checking the model setup before clicking on the **Run!** command.

The output is produced in Surfer, can be saved as a Surfer file (*.srf*) and imported into report documents. These files can also be combined with ADMS 5 output contour plots to create a Surfer overlay.

The features that can be visualised are:

- the output grid as specified in the **Grids** tab,
- specified points as specified in the **Grids** tab,
- source locations as specified in the **Source** tab,
- building positions, dimensions and orientations as specified in the **Buildings** screen,
- the area covered by a surface roughness file (*.ruf*) as specified in the **Hills** screen,
- the area covered by the terrain file (*.ter*) as specified in the **Hills** screen,
- the location and the orientation of the coastline as specified in the **Coastline** screen.

Only the features *currently defined in the interface (or loaded* .apl *file)* can be visualised. By default, all the available features are selected. You can narrow the selection by unchecking the box of any feature you do not want to visualise.

If it appears after the first visualisation that, say, a source position has been specified incorrectly, then return to the main ADMS 5 interface, make the appropriate changes and then click on **Utilities**, **Visualise input in Surfer**... again until the input data are as required.

7.2.1 Display and/or save the visualisation

The user can choose **Display only** to make a quick check of the data at any stage during the creation of an *.apl* file or **Display and Save** to save the output as a Surfer file (*.srf*).

Clicking **Display only** displays the visualisation in Surfer. This visualisation can be

modified and saved afterwards in Surfer if needed.

On clicking **Display and save**, the user is prompted for a Surfer file name (*.srf*). The visualisation will then be created in Surfer and saved using the file name given. A number of so-called "post files" (*.dat*) will also be created, based on the Surfer file name provided.

For guidance on customising the Surfer visualisation, please refer to Appendix D.

7.2.2 Layout of visualised features

A legend is provided as a key for all the elements in the visualisation. This legend should appear below the map. If the default paper size in Surfer is not A4, the legend may be incorrectly positioned. The mouse can be used to select the legend and move it to a new position.

Output grid (grid and specified points)

Regular and variable Cartesian grids are shown on the visualisation as a grid of green lines with the X and Y coordinates shown along each axis.

If specified points are also chosen as output, these are shown as green crosses with their labels plotted to the right. An example of a regular Cartesian grid with four specified points is shown in **Figure 7.4**.

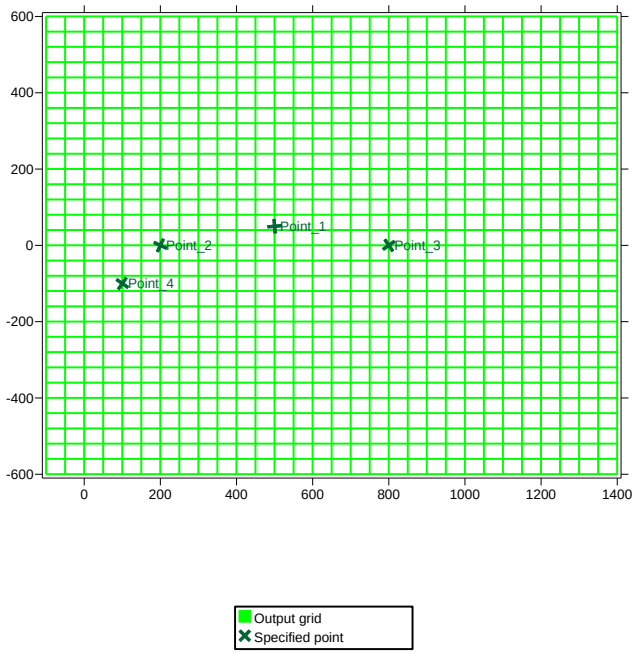


Figure 7.4 – A regular Cartesian output grid and four specified points.

Both regular and variable polar grids are shown as green crosses at the vertices of the grid, since Surfer cannot draw circles or arcs. An example of a regular polar grid is shown in **Figure 7.5**.

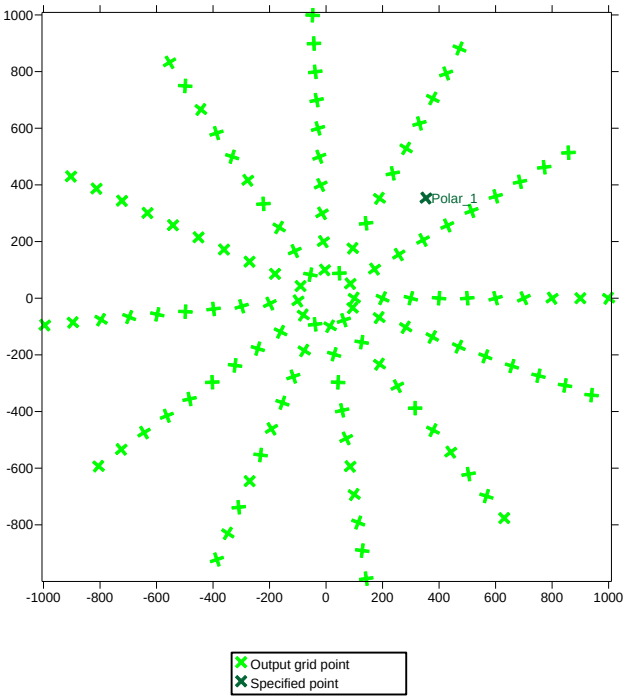


Figure 7.5 – A regular polar output grid and one specified point.

Sources

Each type of source has a different symbol and is labelled with the name assigned to it in the interface, as detailed below and shown in **Figure 7.6**.

Point and jet sources are shown as five-pointed red stars, labelled to the right. Area and volume sources are shown as solid red polygons, labelled at the centre. Their height is not indicated. Line sources are shown as solid lines, labelled at the centre. The width of these lines is the width given to the line sources in the interface.

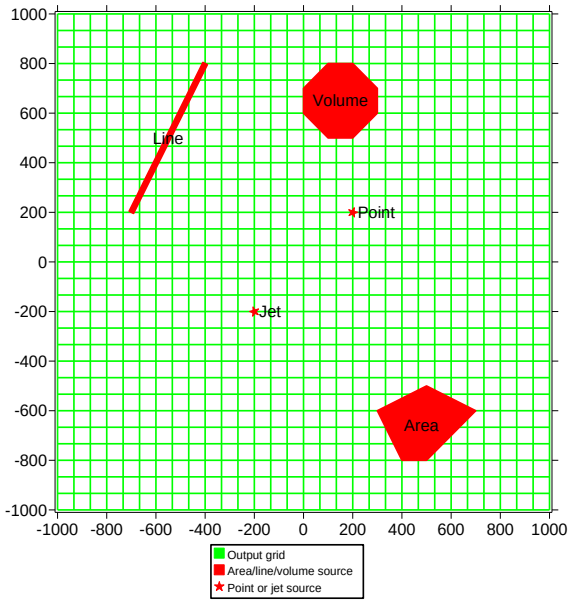


Figure 7.6 – Different types of source available in ADMS 5.

Buildings

Buildings are represented as grey areas labelled at the centre with the names assigned to them in the interface. The main building has an asterisk (*) appended to its name. The height of the buildings is not indicated. This is shown in **Figure 7.7** below.

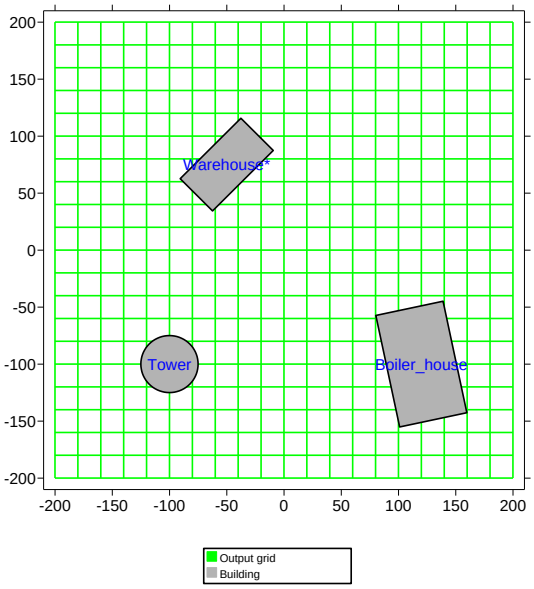


Figure 7.7 – Rectangular and circular buildings.

Roughness area

The area covered by the roughness file is represented as an open blue rectangle (**Figure 7.8**).

Terrain area

The area covered by the terrain file is drawn as an open brown rectangle (**Figure 7.8**).

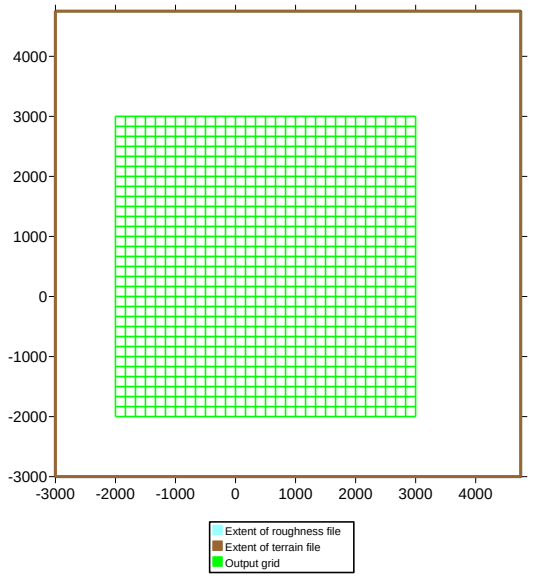


Figure 7.8 – Output grid and extent of terrain and roughness data. (In this example the roughness and terrain data cover the same area, so the roughness open blue rectangle is hidden by the terrain open brown rectangle.)

Coastline

The coastline (position and orientation) is represented as a blue line labelled with the word “Coastline” at its centre. The coastline is defined as an infinitely long line, but only the portion of this line, if any, that falls inside the output grid is shown. The coastline can only be shown for scenarios with a regular, Cartesian, output grid. An example is shown in **Figure 7.9**.

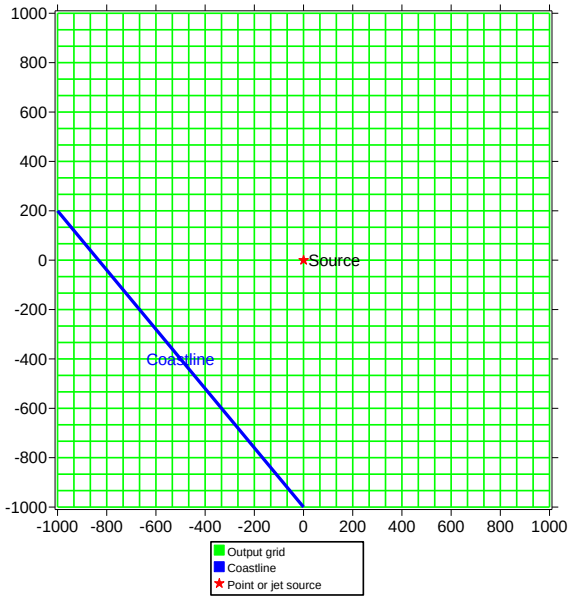


Figure 7.9 – A regular Cartesian grid, a source and a coastline.

7.3 Converting meteorological data

Selecting the **Convert Met. Data** option in the **Utilities** menu opens the **ADMS Meteorological Data File Converter utility** (shown in **Figure 7.10**), which converts two types of meteorological data used in the United States into the ADMS 5 format. The two accepted types of data are:

1. NOAA/NCDC files (*.dat*) created using the HUSWO program to extract data from a NOAA/NCDC Hourly Weather Observations archive CD.
2. Surface data files (*.sfc*) used as input files for the American AERMOD model.

In addition it is possible to incorporate upper air files with the NOAA/NCDC files, or to incorporate an upper air file into an existing ADMS 5 format meteorological data file (*.met*). There are two accepted formats of upper air files:

1. FSL Rawisond data format from NOAA.
2. Upper air data format used as input to Stage 2 of the American AERMET meteorological processor.

---

*The meteorological converter will **not** handle blank lines or missing data. Input data must be complete otherwise output data will be incorrect.*

Some examples of input files are supplied with ADMS 5 as well as the corresponding meteorological output files produced. Refer to the the files *<install_path>\Examples\ex_dat.** and *<install_path>\Examples\ex_sfc.**.

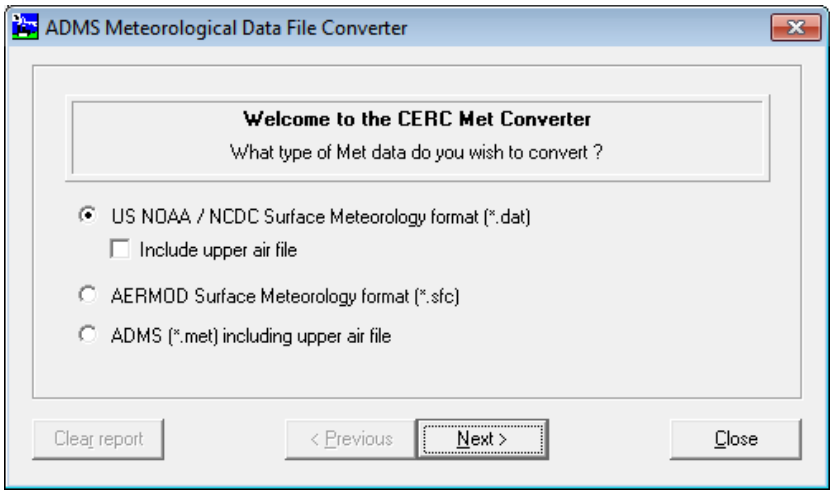


Figure 7.10 – **Convert Met. Data** utility.

7.3.1 Running the utility

To launch the utility, select the **Utilities, Convert Met. Data** menu command to access the screen shown in **Figure 7.10**. Use the **< Previous** and **Next >** buttons to navigate between the screens.

Step 1 Select the type of meteorological data you want to convert.

Step 2 Indicate successively the path and name of the file to convert, the upper air file (if one is being used), the meteorological output file and the report file.

By default, the output and report files are located in the same directory as the input file. They use the same filename stem with different extensions (.met *and* .rpt *respectively).*

Step 3 Once you are ready, click on **Convert** in the screen shown in **Figure 7.11**. The converted meteorological data are written to the output file (*.met*) and a report file (*.rpt*) is created with information on the conversion.

Beware that any previously existing files with the same name will be overwritten when you click on ***Convert****, without warning message.*

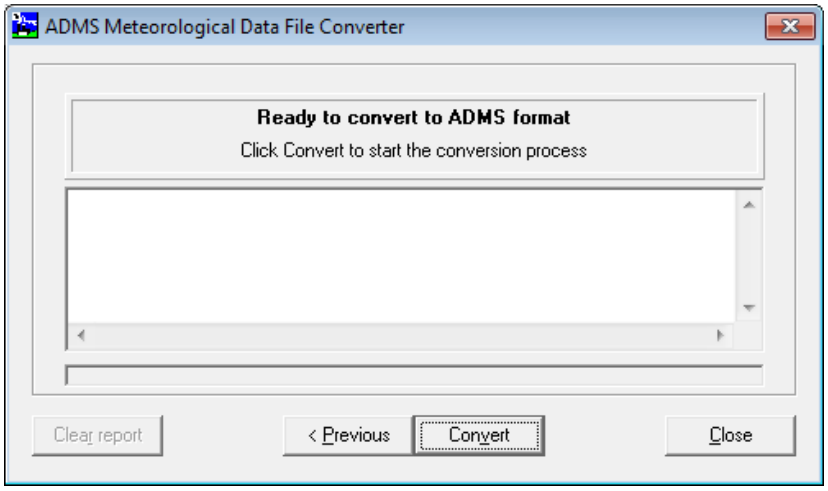


Figure 7.11 – Convert Met. Data utility, **Ready to convert to ADMS format** screen.

Step 4 If the upper air file supplied does not contain a longitude value then a screen to enter this will appear (as shown in **Figure 7.12**).

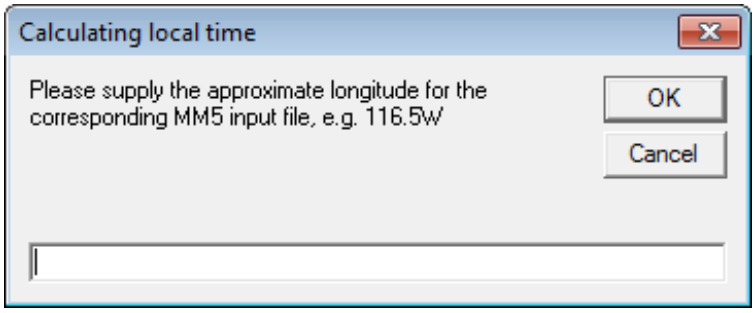


Figure 7.12 – Convert Met. Data utility, **Calculating local time** screen.

Step 5 During the conversion, data are written to the screen and to the report file (*.rpt*) (refer to section 7.3.3 for a full description). When the conversion is finished (successfully or not), a processing completed screen appears, click **OK** to return to the **Convert Met. Data** utility (as shown in **Figure 7.13**).

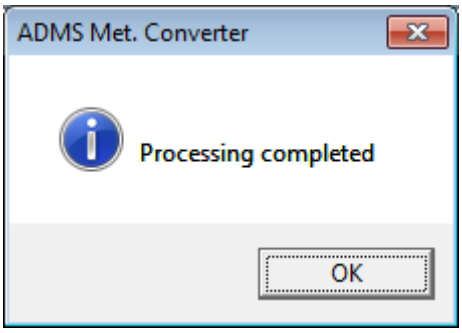


Figure 7.13 – **Convert Met. Data** utility, conversion completed screen.

Step 6 Click on **Close** to exit the utility. Alternatively, click on **< Previous** to return to earlier screens to convert another file. **Clear report** can be used to clear the report data shown on the screen.

7.3.2 Format of the output meteorological data file (.met)

Conversion from NOAA/NCDC .dat *files*

In order to convert the NOAA/NCDC *.dat* files into the ADMS 5 format, a minimum number of variables must be present in the file and given in metric units. These are the station identifier, the time (both automatically included when the *.dat* file was created), the sky cover, the dry bulb temperature, the relative humidity, the station pressure, the wind direction, the wind speed, and the hourly precipitation. Additional variables present in the file will not be included in the output *.met* file produced by the conversion utility.

The header line of the output *.met* file contains the station identifier. Then follows a list of the variables included in the *.met* file. Finally the corresponding data are listed. The variables are the year (YEAR), the month (MONTH), the day of the month (DATE), the Julian day (DAY), the hour of the day (HOUR), the wind speed (U), the wind direction (PHI), the cloud cover (CL), the surface temperature (T0C), the relative humidity (RHUM), and the precipitation rate (P). If an upper air file is included in the conversion the buoyancy frequency above the boundary layer (NU) will also be included in the *.met* file. A full description of the ADMS 5 meteorological data is given in Section 9.1.

The month and the day of the month are given for information only, and are not used by ADMS 5.

Conversion from AERMOD .sfc *files*

The header line of the *.sfc* file is reproduced at the start of the output *.met* file. Then follows a list of the variables included in the *.met* file. Finally the corresponding data are listed. The variables are the year (YEAR), the month (MONTH), the day of the month (DATE), the Julian day (DAY), the hour of the day (HOUR), the surface heat flux (FTHETA0), the reciprocal of the Monin-Obukhov length (RECIPLMO), the surface albedo (ALBEDO (M)), the wind speed (U), the wind direction (PHI), and the surface temperature (T0C). A full description of the ADMS 5 meteorological data is given in Section 9.1.

The month and the day of the month are given for information only, and are not used by ADMS 5.

Hours labelled 24 in the .sfc file are labelled as 0 on the following day in the .met file.

Conversion from ADMS .met files

The output *.met* file is a copy of the input *.met* file with the addition of the buoyancy frequency above the boundary layer (NU). If the input *.met* file already contains the buoyancy frequency above the boundary layer the conversion will not take place.

7.3.3 Format of the report file (.rpt)

Conversion from NOAA/NCDC .dat *files*

The report file lists the path and name of the NOAA/NCDC input file, the meteorological output file and the report file, and reproduces the station identifier from the input file.

Conversion from AERMOD .sfc *files*

The report file lists the path and name of the AERMOD input file, the meteorological output file and the report file. It reproduces the header line from the input file, and provides the ranges of roughness lengths Z_0, the heights of the wind measurements Z_u and the Monin-Obukhov lengths L_{MO} of the input file. Z_0 and Z_u are not included in ADMS 5 *.met* files, since they have to be given as single representative values in the **Meteorology** tab of the ADMS 5 interface. The range of values of L_{MO} provides useful information when using the ADMS 5 option to restrict the values of L_{MO} to avoid unrealistically stable boundary layers.

Conversion from ADMS .met files

The report file lists the path and name of the meteorological input file, the upper air file, the meteorological output file and the report file.

7.4 Making a terrain file

To run the **Complex terrain** model option, ADMS 5 requires data to be input describing the topography surrounding the dispersion site. Section 4.10 describes the **Complex terrain** model option in detail and includes a description of the format of ADMS 5 terrain files.

Terrain files can be created using the ADMS Mapper, refer to Section 5.6 of the ADMS Mapper User Guide for details. The ADMS Mapper can be used with a variety of input data files including OS Terrain 50 and SRTM data.

For compatibility with older data formats ADMS 5 also includes a utility called the **ADMS Terrain Converter** (shown in **Figure 7.14**) to convert the following formats of digital terrain data into ADMS 5 format:

1. OS Landform PROFILE NTF (Great Britain only),
2. OS Landform PANORAMA NTF (Great Britain only),
3. IGN XYZ (France only),
4. OS Northern Ireland DTM (Northern Ireland only),
5. OS Northern Ireland XYZ (Northern Ireland only),
6. OS Ireland XYZ (Eire only).

For options (1) and (2) listed above, the **ADMS Terrain Converter** also informs the user which map tiles they need to purchase or download in order to include their chosen area of complex terrain in an ADMS 5 calculation.

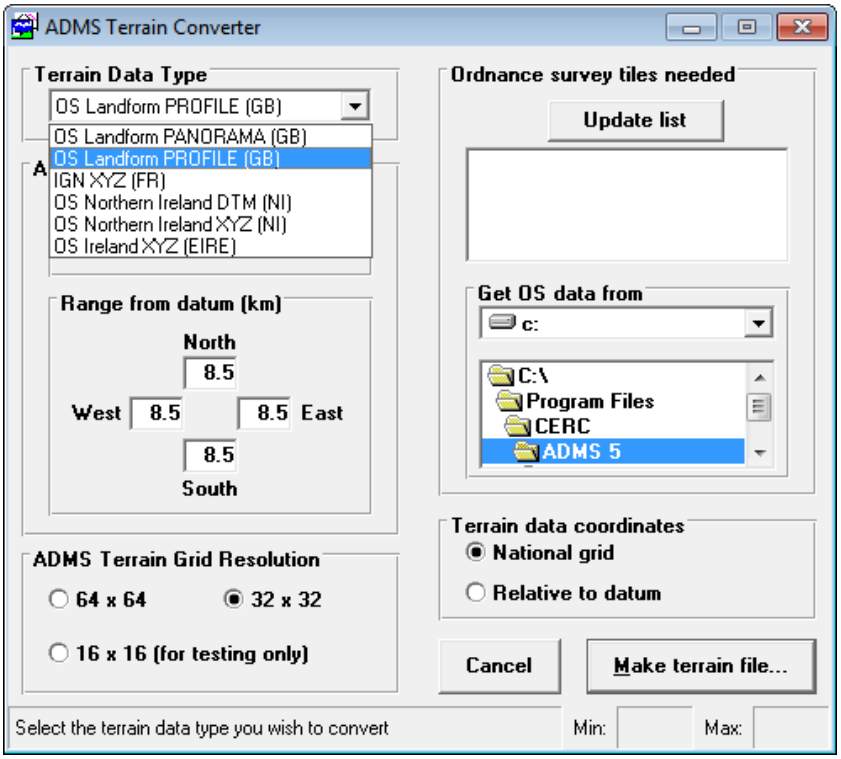


Figure 7.14 – Main screen for **ADMS Terrain Converter**.

To start the utility, click on **Utilities**, **Create Terrain File** from the menu bar at the top of the

main ADMS 5 interface. This will open the screen shown in **Figure 7.14**. To change between different digital data input formats, use the drop-down list in the top left corner of the screen.

For more information regarding the file formats listed above and contact details for suppliers of these data, please refer to Appendix E.

7.4.1 Creating an ADMS 5 terrain file from OS Landform NTF (GB) data

Ordnance Survey (OS) data for England, Scotland and Wales are available in NTF tiles. **Landform PANORAMA** NTF data are supplied in 20 km × 20 km tiles with 50 m resolution. **Landform PROFILE** NTF data are supplied in 5 km × 5 km tiles with 10 m resolution. These tiles use the National Grid coordinate system, where the whole region is divided into 100 km × 100 km 2-lettered squares (see **Figure 7.15**).

Each 2-lettered square is made up of 25 smaller squares each with an area of 20 km × 20 km. Each 20 km × 20 km square is further divided into 16 smaller squares, each with an area of 5 km × 5 km. The coordinate (0,0) is the bottom left-hand corner of the square SV.

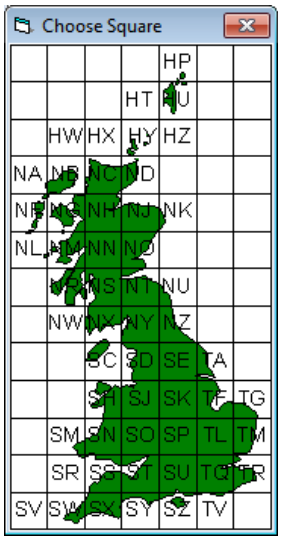


Figure 7.15 – Reference map of UK showing National Grid 100 km squares.

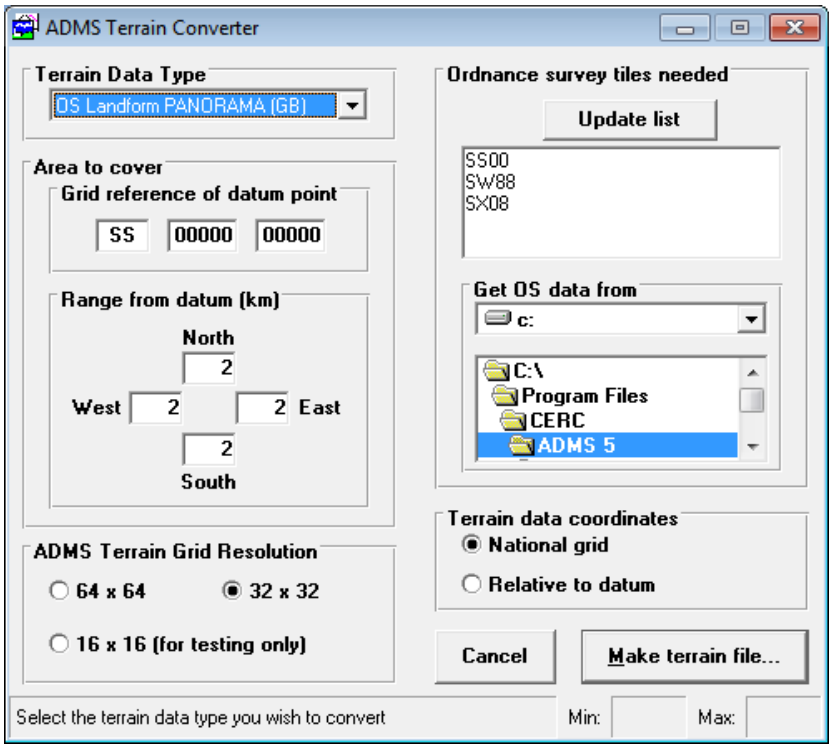


Figure 7.16 – The **ADMS Terrain Converter** screen for Great Britain data.

Area to cover

In the **Grid reference of datum point** box, enter the datum point for the area of interest in 5-figure National Grid Coordinates accompanied by a 2-letter code. Click on the following in order:

- **SV**: this is a default National Grid Reference. Double-click on this box to display the National Grid map shown in **Figure 7.15**. Click on the grid square that contains the source(s) for your modelling scenario, and the corresponding letters will automatically be entered in this box. Alternatively, if you know them, type the letters directly into the box.
- **00000,00000**: the first is the default Easting, the second the default Northing. Note that this grid reference is relative to the south-west corner of each lettered square. Enter the Easting and Northing of the datum, e.g. the primary source position, in metres. An accuracy of 100 m may be sufficient, except for near-field calculations when 10 m resolution is required. The minimum value is 0 m, while the maximum is 99999 m.
- **Range from datum (km)**: in each box, enter the distance from the primary source, in the specified direction, to be included in the terrain file.
- **ADMS Terrain Grid Resolution**: the following resolutions for the terrain grid are available:

  64 × 64 for accurate results in larger domains
  32 × 32 for accurate results
  16 × 16 to be used for testing only

  These grid sizes refer to the resolution of the terrain data in the *.ter* file; the

resolutions listed in Section 4.10 refer to the ADMS 5 internal calculation grid. For best results users might select a high resolution for both the terrain data and the internal calculation grid but this will increase model run times.

Ordnance survey tiles needed

Once the **Area to cover** box has been completed, click on the **Update list** button. The Ordnance Survey tiles required to make the terrain file for the chosen area will be listed in the box.

Then use the **Get OS data from** box to specify where the Ordnance Survey data files are stored. If the Ordnance Survey tile data have already been saved on the hard disk, they can be accessed by specifying drive *C:*. When loading data from disk, choose the drive, e.g. *D:*, from which the Ordnance Survey tile data are to be loaded.

In the **Terrain data coordinates** box, choose either the **National grid** option or the **Relative to datum** option to specify in which format the coordinates are to be given in the created terrain file. If **National grid** is selected, the data will be in 6-figure numbers relative to the origin at SV 000000, 000000. If **Relative to datum** is selected, the data will be in metres relative to the datum.

When the required information has been input, click on the **Make terrain file**... button. You will be prompted to name the terrain file to be created and specify where it is to be stored.

When you click on the **OK** button, the program will start loading Ordnance Survey data and will display a progress window (**Figure 7.17**).

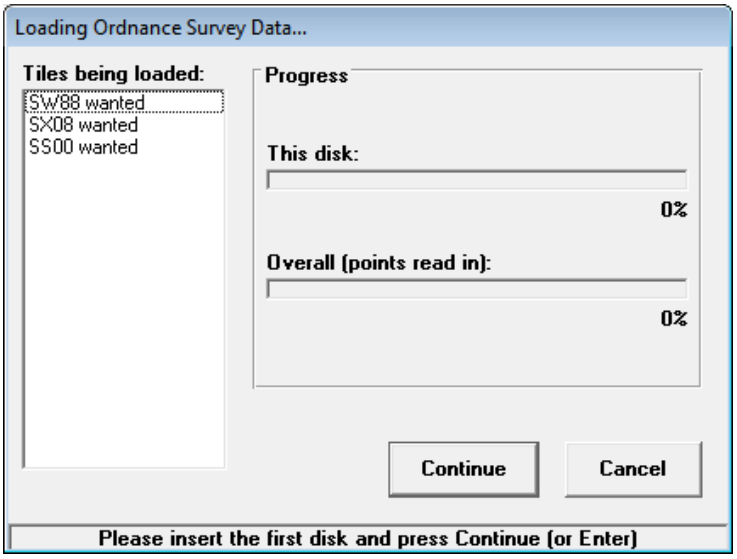


Figure 7.17 – The **Loading Ordnance Survey Data**... progress window.

The name of the file being loaded is shown at the bottom of the screen. A 'beep' will sound after the successful loading of each file, and the message shown in **Figure 7.18** will appear on the screen when the terrain tile is complete.

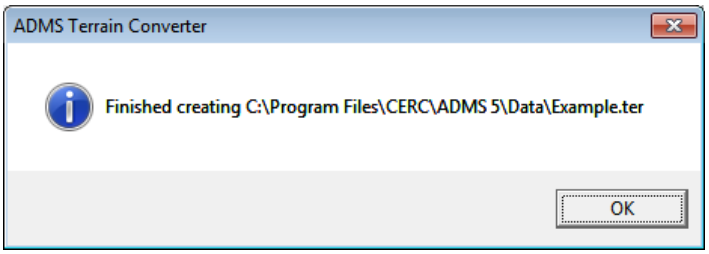


Figure 7.18 – Message to show terrain file has been successfully created.

7.4.2 Creating an ADMS 5 terrain file from IGN XYZ (FR) data

Selecting the area to cover

When choosing the area that you wish to cover with ADMS 5 terrain data, you must think about the boundaries of your IGN data.

If the area you wish to cover overlaps a coastline, and therefore contains missing data over the sea, then proceed as normal. The **ADMS Terrain Converter** will prompt you to identify the missing data as sea and will then set the elevation of the missing data points to be zero. If this is not the case, then the area you choose to cover should lie within the area covered by your IGN data. The **ADMS Terrain Converter** will detect any missing data, for instance at international boundaries, and will prompt you either to select your area manually, or to use the **ADMS Terrain Converter** facility to re-calculate the extent of the area to cover to avoid missing data at the edges.

Creating the terrain file

To convert French IGN XYZ data to ADMS 5 terrain data, follow the steps below:

Step 1 Select **IGN XYZ (FR)** from the drop-down list in the top left corner of the **ADMS Terrain Converter** interface.

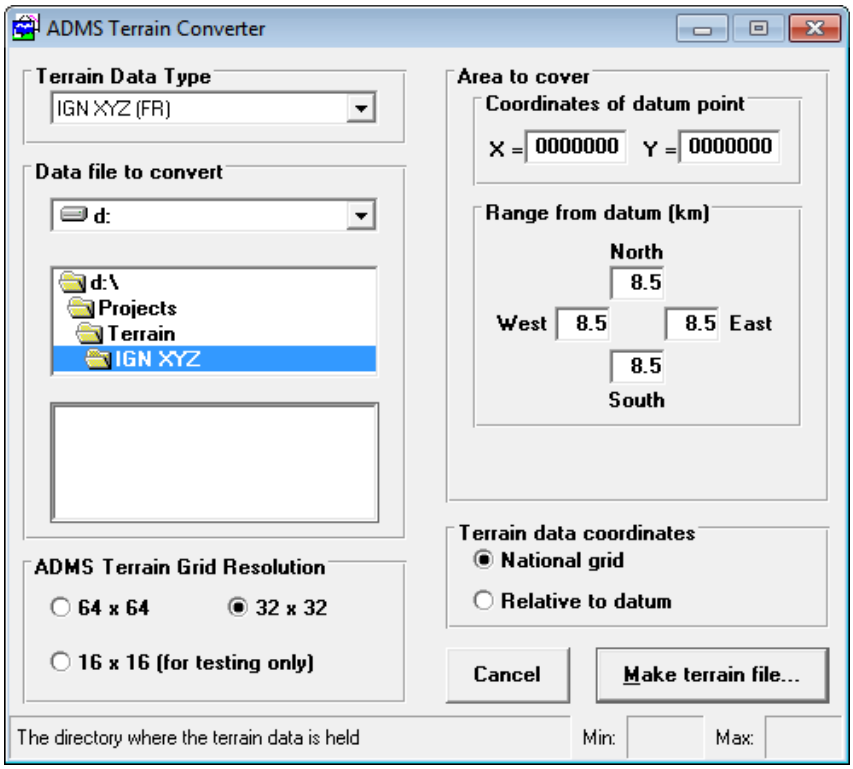


Figure 7.19 – **ADMS Terrain Converter** screen with **IGN XYZ (FR)** data selected.

Step 2 In **Data file to convert**, select the drive, directory and file name of the IGN terrain file that you wish to convert to ADMS 5 format.

Step 3 Select the **ADMS Terrain Grid Resolution** you require. The following resolutions for the terrain grid are available:

64 × 64 for accurate results in larger domains
32 × 32 for accurate results
16 × 16 to be used for testing only

These grid sizes refer to the resolution of the terrain data in the *.ter* file whereas the resolutions listed in Section 4.10 refer to the ADMS 5 internal calculation grid. For best results users might select a high resolution for both the terrain data and the internal calculation grid but this will increase model run times.

Employing the finest internal calculation grid significantly increases the run time for each problem, but also increases the accuracy of the solution.

Step 4 In **Area to cover**, enter the coordinates of the datum point, in metres. This may be any point.

Step 5 Also in **Area to cover**, enter the range from the datum point (in kilometres) that you wish the ADMS 5 terrain grid to cover.

Step 6 Select the **Terrain data coordinates**. If you choose **National grid** then the coordinates in the ADMS 5 terrain file will be the same as the coordinate system of the IGN terrain data. If you choose **Relative to datum** then the coordinate system used in the ADMS 5 terrain file will have the origin at the datum point.

Step 7 You are now ready to make the ADMS 5 terrain file. Click **Make the terrain file**... button.

Step 8 Specify a directory and file name for the newly created ADMS 5 terrain file.

Step 9 When the ADMS 5 terrain file has been successfully created, the **ADMS Terrain Converter** will beep and display the following message:

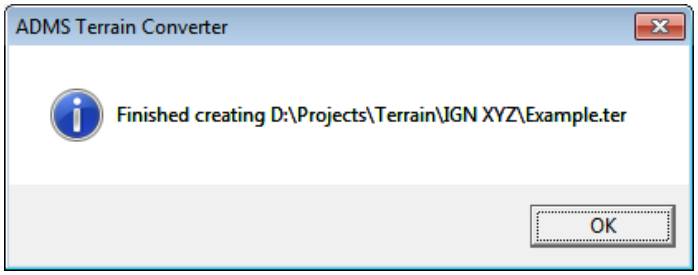


Figure 7.20 – Message to show terrain file has been successfully created.

7.4.3 Creating an ADMS 5 terrain file from OS Northern Ireland DTM (NI) data

Northern Ireland terrain data are supplied as 20 km × 20 km DTM data tiles in the Irish National Grid format, which is shown in **Figure 7.21**.

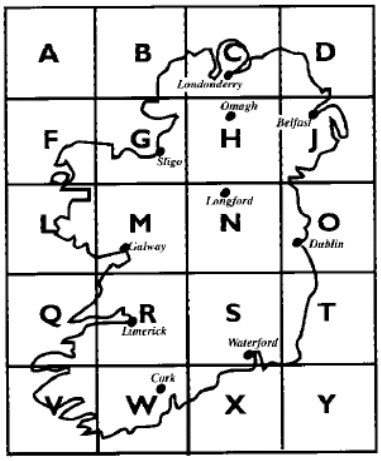


Figure 7.21 – The Irish National Grid.

Each lettered grid square covers an area of 100 km × 100 km; the point in the bottom left corner of grid square V has coordinates (0,0).

If a source is positioned in the centre of the H square i.e. at 50,000 m, 50,000 m, the actual 6-figure National Grid coordinate is (250000, 350000). It is up to you to find the coordinates of the south-west and north-east corner of the 20 km × 20 km square area around the source that you are considering. These coordinates should be supplied to OS-NI when ordering data.

This option is different to the OS Landform options in that it does not tell you which tiles you need to buy for a certain area of terrain, but rather you add all the NI terrain data you have to an index. When you choose a distance from the source point in every direction that you want to include, the utility will either create that terrain file from the

data you have in the index or tell you if you do not have the required amount of data.

To convert Northern Irish DTM data to ADMS 5 terrain data, follow these steps:

Step 1 Select **OS Northern Ireland DTM (NI)** from the drop-down list in the top left corner of the **ADMS Terrain Converter** interface.

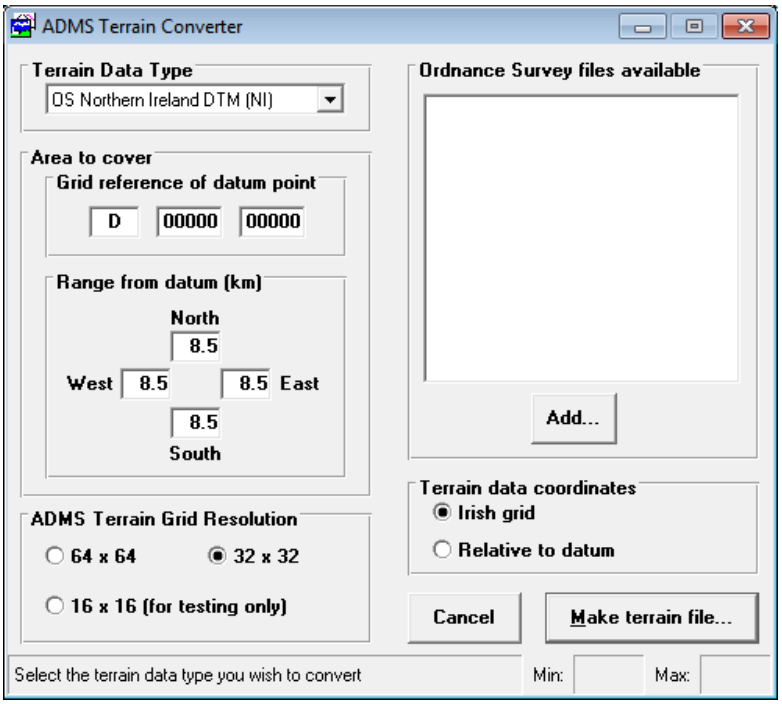


Figure 7.22 – The **ADMS Terrain Converter** screen for Northern Ireland DTM data.

Step 2 Once you have purchased the data you require, it must be added to an index in the utility. Firstly, ensure the data is copied onto the hard disk of your machine. Click on the **Add** button and browse for your files. Locate all the NI DTM terrain data and add each file that you have in turn.

The utility will build the index progressively as each new file is added. All the data you add will be shown in the **Ordnance Survey files available** box. The data need only be added once and the utility will always look for the data in the same place each time it is used.

Step 3 Enter the letter of the grid square e.g. **D** within which your source or datum point is positioned. Then enter the coordinates (easting then northing) of that point within the chosen square (0-99999 m).

Step 4 The next step is to enter the distance in kilometres from the point that you want to include in the terrain file in each direction.

Step 5 The user must then choose the grid size required. This must be 16 × 16, 32 × 32 or 64 × 64. Increasing the grid resolution gives a more accurate representation of the terrain but can increase ADMS 5 run times. A 32 × 32 grid resolution is typical, but if the domain is large or if the terrain is particularly complex, a grid resolution of 64 × 64 should be selected. A 16 × 16 grid resolution should only be used for testing purposes.

Step 6 Choose whether the resultant terrain file should give coordinates in the

Irish National Grid Coordinate system or relative to the datum. The default is the former.

Step 7 When all of the above steps have been completed, click on **Make terrain file**.... You will be prompted to name and save the new file in a particular directory of your choice.

Step 8 Once you have chosen the name and location for the file and selected **Save** one of two things may happen:

1) You have the necessary data to complete the file in the index and the terrain file is completed. If this is the case the utility will indicate that it has found the data. It will then show the window from **Figure 7.23** to say that the terrain file has been successfully created.

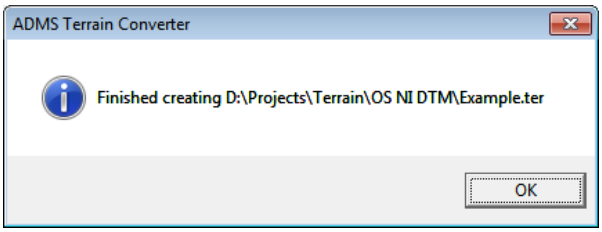


Figure 7.23 – Finished creating terrain file.

2) The data required to make the specified terrain file are not available. In this case a window such as that shown in **Figure 7.24** will appear detailing which data are missing. The green dots refer to areas for which you do have data; blue dots refer to known areas of sea (the OS data only records these very close to the coast) and the red dots refer to areas where no data are available i.e. in the sea or where you need to buy more data. **Figure 7.24** shows that the area of terrain chosen extends across into the Republic of Ireland and was therefore not included in the index.

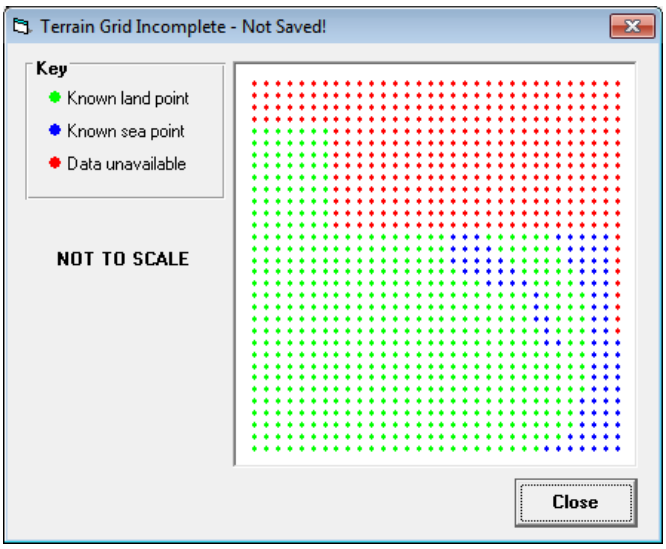


Figure 7.24 – Data not available for the terrain file.

If this window appears then the terrain file has not been created. You are required to change the extent of the area you want to cover in the interface so that it covers the available data, or purchase more data from OS-NI. (It

is important to note that when the file has not been completed, the window does not draw the area to scale. It will always be a square of the same number of dots so images may appear compressed or elongated.)

7.4.4 Creating an ADMS 5 terrain file from OS Northern Ireland XYZ (NI) data

Terrain data for Northern Ireland are also available in XYZ format, which lists the X, Y and Z coordinates of each data point separated by commas. The Z coordinates are in decimetres and there is a header line at the start of the file. Coordinates that are in the sea are given the value -1000. Files of this type should have the extension *.txt*.

To create an ADMS 5 terrain file from this type of data, first select **OS Northern Ireland XYZ (NI)** from the drop-down list in the top left hand corner of the **ADMS Terrain Converter** interface and then follow Steps 2 to 9 from the instructions for French IGN XYZ data (see Section 7.4.2).

7.4.5 Creating an ADMS 5 terrain file from OS Ireland XYZ (EIRE) data

Data for the Republic of Ireland are based on the same National Grid as data for Northern Ireland (see **Figure 7.21**). Data files have the extension *.dtm* but are in XYZ format. This is a simple text file that lists the X, Y and Z coordinates separated by spaces. The Z coordinates are in metres and there is no header. Coordinates that are in the sea are given the value 0.

To create an ADMS 5 terrain file from this type of data, first select **OS Ireland XYZ (EIRE)** from the drop-down list in the top left hand corner of the **ADMS Terrain Converter** interface and then follow Steps 2 to 9 from the instructions for French IGN XYZ data (see Section 7.4.2).

7.5 Making and running a batch file

It is often useful to be able to run several files consecutively, without opening each *.apl* file in the ADMS 5 interface and using the **Run** menu option. This can be done using a batch file. A batch file is a plain text file with the extension *.bat*, containing DOS commands.

7.5.1 Format of batch files

The most common syntax for running ADMS 5 is:

```
<model path name> <file path name> /e2
```

where `<model path name>` is the full path name of the ADMS 5 executable file (*ADMSModel.exe* located in the model installation directory),

`<file path name>` is the full path name of the *.apl* file you wish to run, both enclosed in inverted commas (`"`),

`/e2` is an option to cancel the prompt window at the end of the model run

For example, if ADMS 5 is installed in directory *C:\Program Files\CERC\ADMS 5* and the *.apl* file to run is *D:\My Work\Test.apl*, the command will be:

```
"C:\Program Files\CERC\ADMS 5\ADMSModel.exe" "D:\My Work\Test.apl" /e2
```

Repeat this line for each *.apl* file you are running. The text in a *.bat* file is not case-sensitive. To start the model run(s), double-click on the *.bat* file in Explorer.

7.5.2 Batch File Creator utility

Batch files may also be created and run using the **Batch File Creator** utility of ADMS 5.

To launch it, select the **Utilities**, **Make/Run batch file** menu command. It opens the screen shown in **Figure 7.25**.

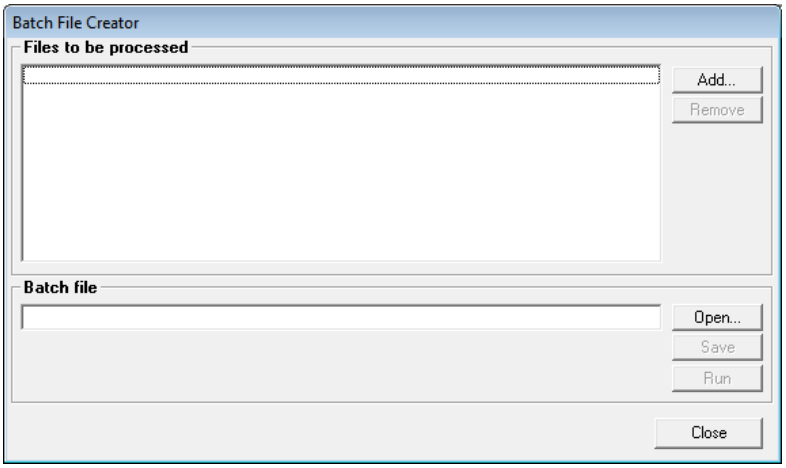


Figure 7.25 – **Batch File Creator** utility.

To set up and run a batch file, proceed as follows.

Step 1 Click on **Add**... and browse for the *.apl* file to be run. The path and name of the selected file is added to the **Files to be processed** box.

Repeat until all the files you wish to run are included in the list (**Figure 7.26**). Files may be removed from the list with the **Remove** button.

Multiple .apl *files can be selected together in the browse window and added. In such case, they are listed alphabetically.*

Step 2 Save the batch file by clicking on **Save**. The file is saved with the extension *.bat*.

The batch file must always be saved prior to running.

Step 3 Run the batch file immediately (**Run**) or exit the utility (**Close**).

The .apl *files are run in the listed order.*

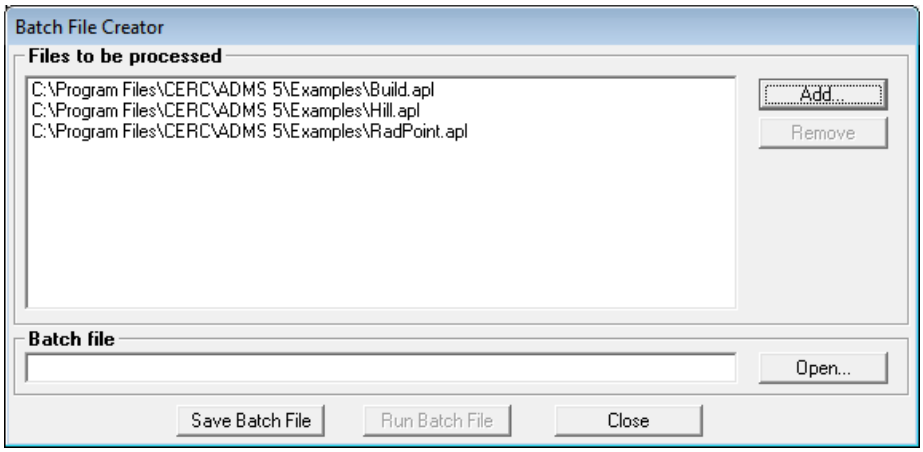


Figure 7.26 – Selection of files to be included in the batch file.

To open and edit or run a batch file which you have created earlier, click **Open** and browse for the batch file. The *.apl* files to be run will be listed in the **Files to be processed** box.

7.6 Viewing a wind rose

The **Wind rose viewer** utility allows the data in a meteorological data file (*.met*) to be visualised graphically. The visualisations can be viewed on the screen, printed, copied to another Windows application or exported to a file. An example of a copied image is shown below in **Figure 7.27**.

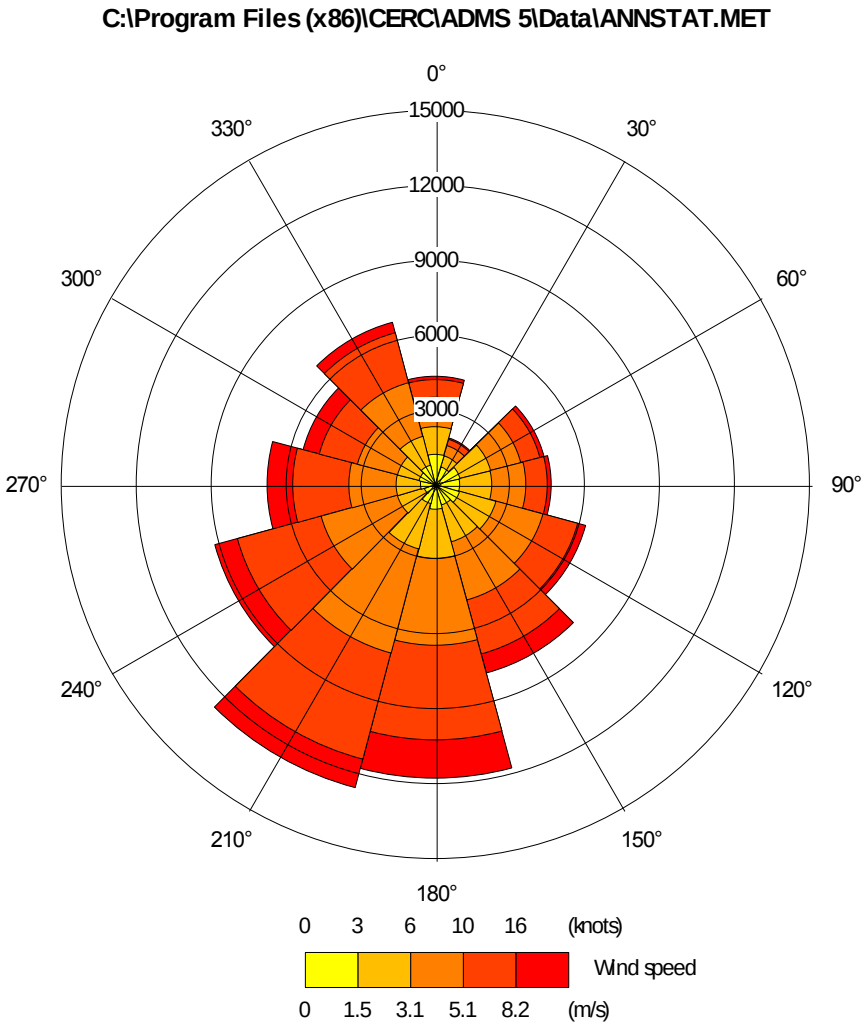


Figure 7.27 – Example of a wind rose produced with the ADMS 5 **Wind rose viewer** utility.

7.6.1 Categories for the data

To produce these visualisations, the data in the meteorological file is divided into categories by wind speed and wind direction. This information is displayed in a polar plot, in which the angle of an element represents the wind direction and the radial distance from the centre represents the frequency of occurrence.

For statistical meteorological data, the frequency is taken from the data in the file. For sequential meteorological data, the frequency of each line is taken to be 1.

The wind speed categories are those used by the UK Met Office in their statistical meteorological data, namely 0-3 knots, 3-6 knots, 6-10 knots, 10-16 knots and 16 knots or above. Meteorological lines with wind speeds of less than 0.01 m/s are not included in the plot.

The **Wind rose viewer displays** the number of **Lines used** and the number of **Lines skipped**. Lines may be skipped if they contain invalid wind speed/direction data, have wind speeds which are too low or are outside of the defined subset period.

7.6.2 Using the wind rose utility

The wind rose utility is accessed from the **Utilities, View a wind rose** command or from the **Wind rose** button on the **Meteorology** screen and is shown in **Figure 7.28**.

*If the **Wind rose** button on the **Meteorology** screen is used the meteorological data file currently selected in the interface will automatically be loaded into the **Wind rose viewer**. Launching from the **Wind rose** button will allow the selected sector size and Met. Data subset to be taken into account.*

Opening a meteorological file (.met)

To open a meteorological file, go to the **File, Open**... command and browse for the *.met* file.

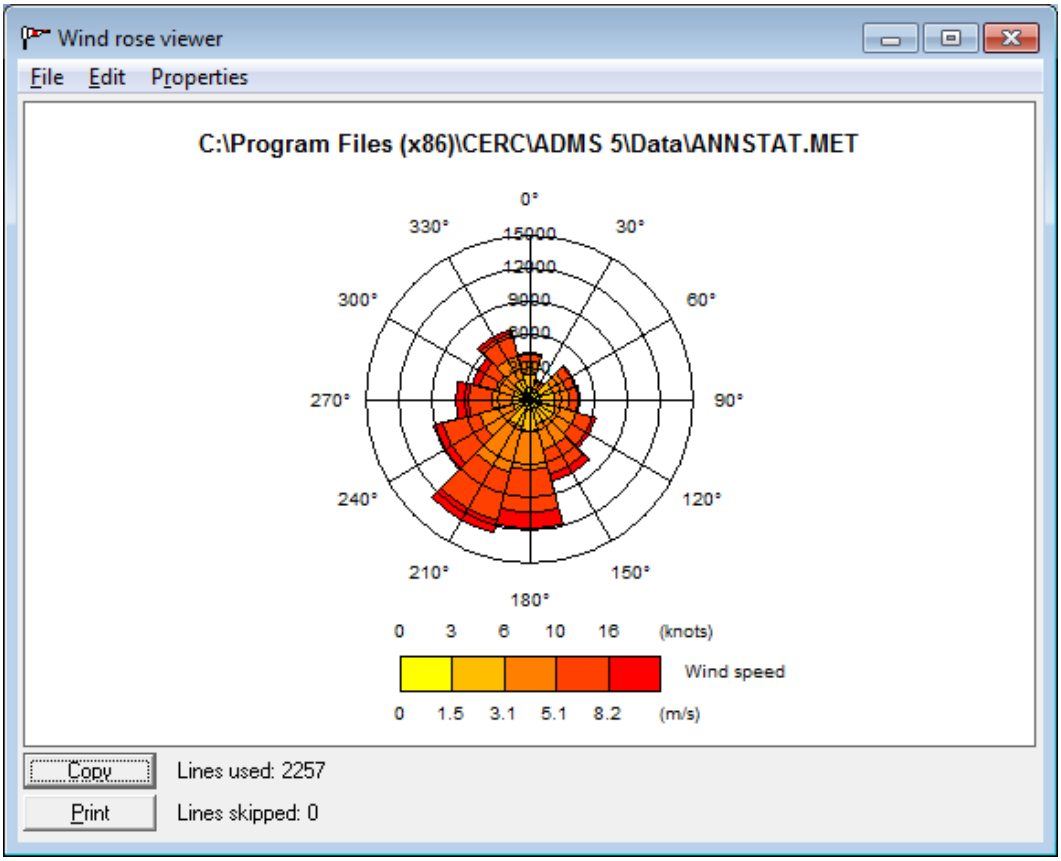


Figure 7.28 – Main screen of the **Wind rose viewer** Utility.

Customising the appearance of the wind rose

The appearance of the wind rose can be altered using the various options from the **Properties** menu:

- The colours used in the wind rose can be set by selecting the **Colour for highest wind speed**... and **Colour for lowest wind speed**... options.
- The title can be shown or removed by checking or clearing **Show Title**. The title can be altered by choosing **Set Title**....
- The fonts used in the wind rose display can be altered as follows: choose **Label Font**... to alter the font for the wind speeds and angles, and choose **Title Font** to alter the font for the title.
- There are several different visualisation styles for the wind rose. These can be selected as follows: choose **Arc segments** to display each wind sector as a pie slice, choose **Polygon** to join points along neighbouring wind sector centrelines to form the sides of a polygon, or choose **Triangles** to display each wind sector as a triangle.
- The angle interval into which the wind speeds are "binned" for display can be altered by choosing one of **10° intervals**, **22.5° intervals** or **30° intervals**. Note that changing the binning interval can take several seconds.

Printing the wind rose

To print the wind rose, click the **Print** button or choose **File, Print** from the menu. To use a different printer or to change the settings of the current printer choose **File, Print Setup**... from the menu. The wind rose will occupy a complete printed page.

Copying the wind rose to the clipboard

The wind rose can be copied to the clipboard by clicking the **Copy** button or by choosing **Edit**, **Copy** from the menu. It can then be pasted into another Windows application, such as Microsoft Word.

Exporting the wind rose to a file

The wind rose image can be exported as an enhanced metafile file (*.emf*) by choosing **File**, **Export Picture**... from the menu. This file may then be imported into other Windows applications, such as Microsoft Word. The imported image may be resized without reducing the resolution quality.

SECTION 8 Worked Examples

In order to help the user to learn how to use ADMS 5, a few worked examples have been set up to cover the main functions of the model. They illustrate how to set up single and multiple source scenarios, how to calculate long- and short-term average results under different meteorological conditions, and three different ways to look at the calculated output.

The *<install_path>\Examples* directory contains additional example *.apl* files that are useful for further illustrating the model functionalities.

You can launch ADMS 5 in several different ways:

- double-click on the icon for the shortcut created earlier (see Section 2.2).
- use the Windows **Start** menu and select **Programs**, **ADMS 5**;
- go to the main ADMS 5 directory *<install_path>* and double-click on the file *adms.exe*;

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

New users of ADMS 5 are advised to carry out these worked examples before attempting to set up runs including any of the advanced modelling options described in Section 4.

8.1 Example 1: Single point source under different meteorological conditions

The object of this example is to demonstrate how to set up and run the simplest case of a single point source problem and to look at the different types of output available from ADMS 5.

In particular, this example looks at emissions from a 30 m high stack. The ground-level concentration and the plume spread will be compared for seven different meteorological conditions, which correspond *approximately* to the Pasquill-Gifford stability categories A to G.

8.1.1 Setting up the run

To model a single source in a flat terrain, proceed as follows.

Step 1 Open ADMS 5 and click on **File**, **New** to create a new input file, i.e. a default input file or "clean sheet" in which to enter data.

Step 2 Enter a **Name of site** and **Name of project** of your choice.

These are optional but sensible titles should be entered as they help to

differentiate between model runs. The titles will be written out to the log file of the model run.

There are no complex effects in this calculation. Compare your input to **Figure 8.1**. This completes input to the **Setup** screen.

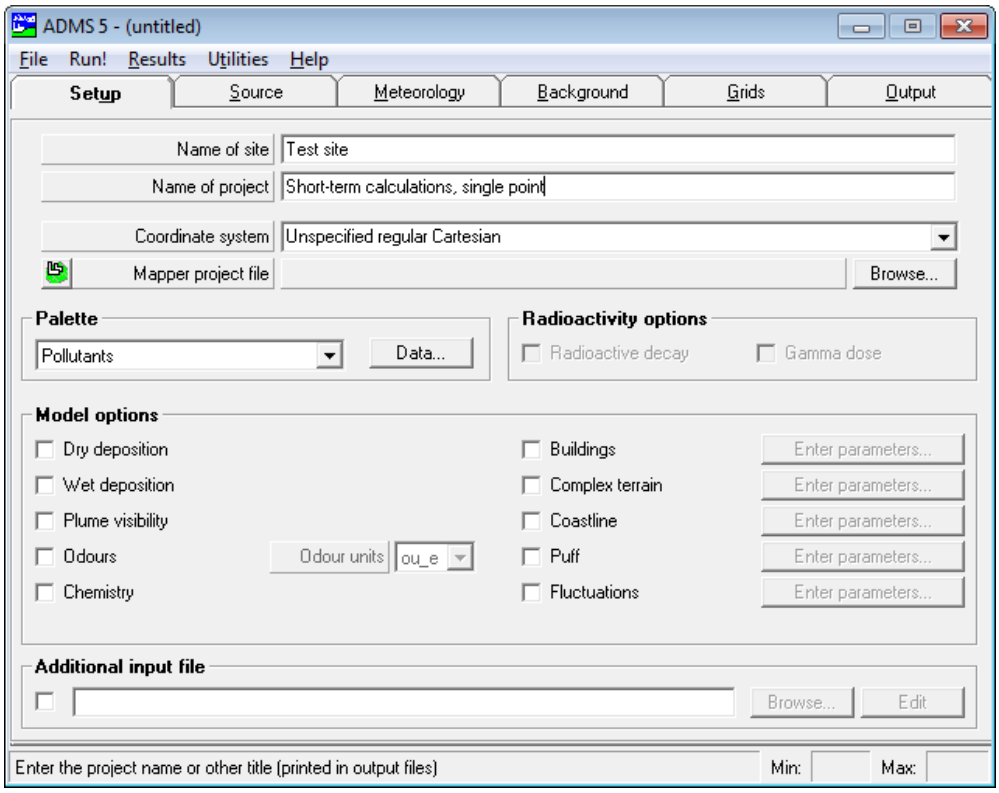


Figure 8.1 − Example 1: the **Setup** screen.

Step 3 Go to the **Source** screen. Most time is spent entering input on this screen when setting up a modelling run with several sources. To enter a new source click on **New** to add a line of data to the table.

The default source is a point source with a height of 50 m. The scroll bar along the bottom of the table indicates that there is more information in the table that is currently out of view. The data in the first three columns (**Cp**, **Mol. mass** and **Density**) are the default values and should not be changed for this example. (The default values are for air, so that provided the concentration of pollutant in the emission is low, which is commonly the case, these default values are acceptable.)

Step 4 Give the source a name such as "Boiler stack".

Step 5 Change the height of the source to 30 m and the velocity to 0 m/s.

If the velocity or volume flow rate is set to zero, then plume rise effects are ignored by the model, even if the emission temperature is greater than ambient. This is a useful mechanism to turn off plume rise. The **Source** screen should now look like **Figure 8.2**.

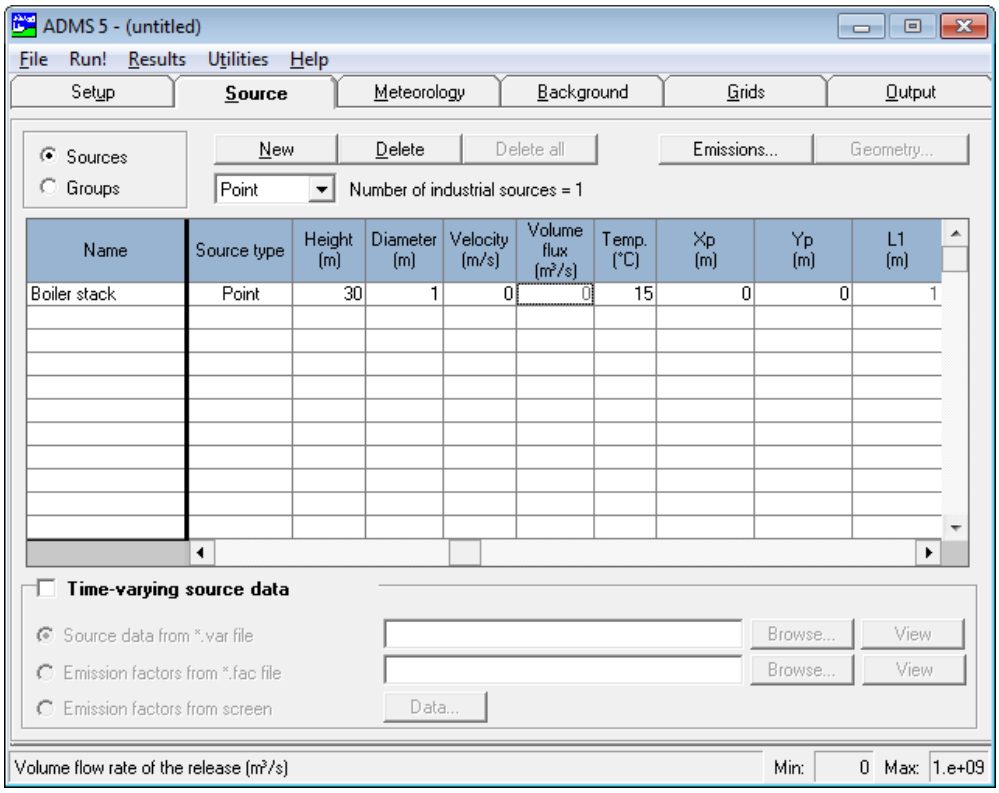


Figure 8.2 − Example 1: the **Source** screen.

Each source needs to have one or more pollutants associated with it. These are entered via the **Emissions** screen. Make sure that the cursor is in the correct row of the table and click on the **Emissions**... button to enter the **Emissions** screen. The pollutants and mass emission rates for the source are entered here. On first entering the **Emissions** screen, the units for the emission rate are shown as g/s, and NO_X appears as the default pollutant.

Step 6 Click on the down arrow shown to the right of the word "NOx" in the **Pollutant name** cell and select SO_2 from the drop-down list (the ten pre-defined pollutants appear). Enter an emission rate of 1 g/s, so that this screen now matches that shown in **Figure 8.3**. Click **OK** to close the **Emissions** screen.

Click on the **Meteorology** tab to access the screen in which meteorological data and site data are entered. First we will enter data about the dispersion site.

Step 7 Leave the **Latitude** as 52°. Using the drop-down arrow, select a **Surface roughness** at the dispersion site for **Agricultural areas (min)**, this corresponds to a value of 0.2 m. Leave all of the other options in the **Site data** box unchanged.

Step 8 In the **Met. data** box select **Enter on Screen**. Leave the rest of the options in this box unchanged. The screen should look like **Figure 8.4**.

We now need to enter the meteorological data.

Step 9 To do this click on the **Data**... button next to the **Enter on screen** option. This will bring up the **Meteorological Data** screen.

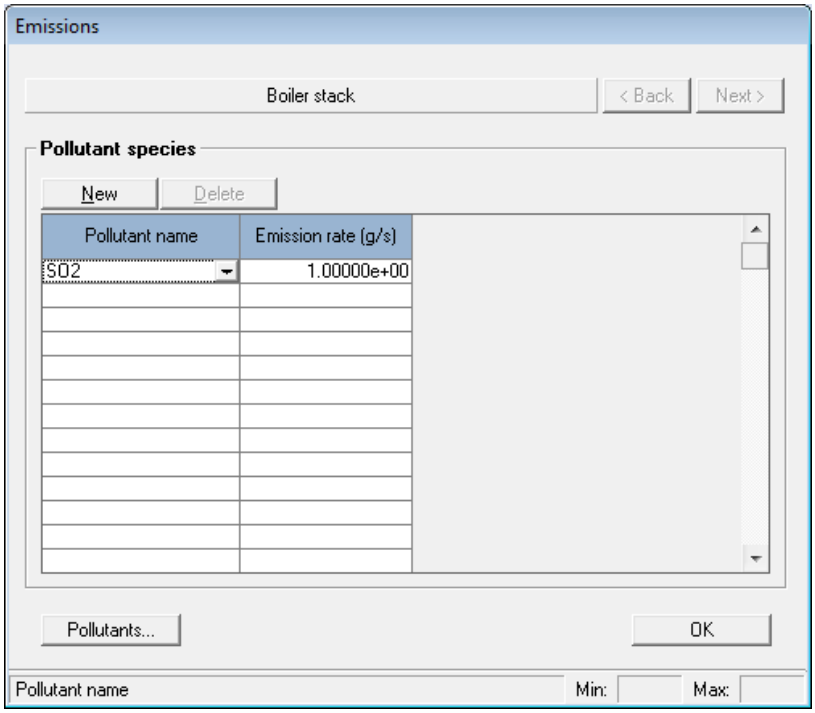


Figure 8.3 − Example 1: the **Emissions** screen.

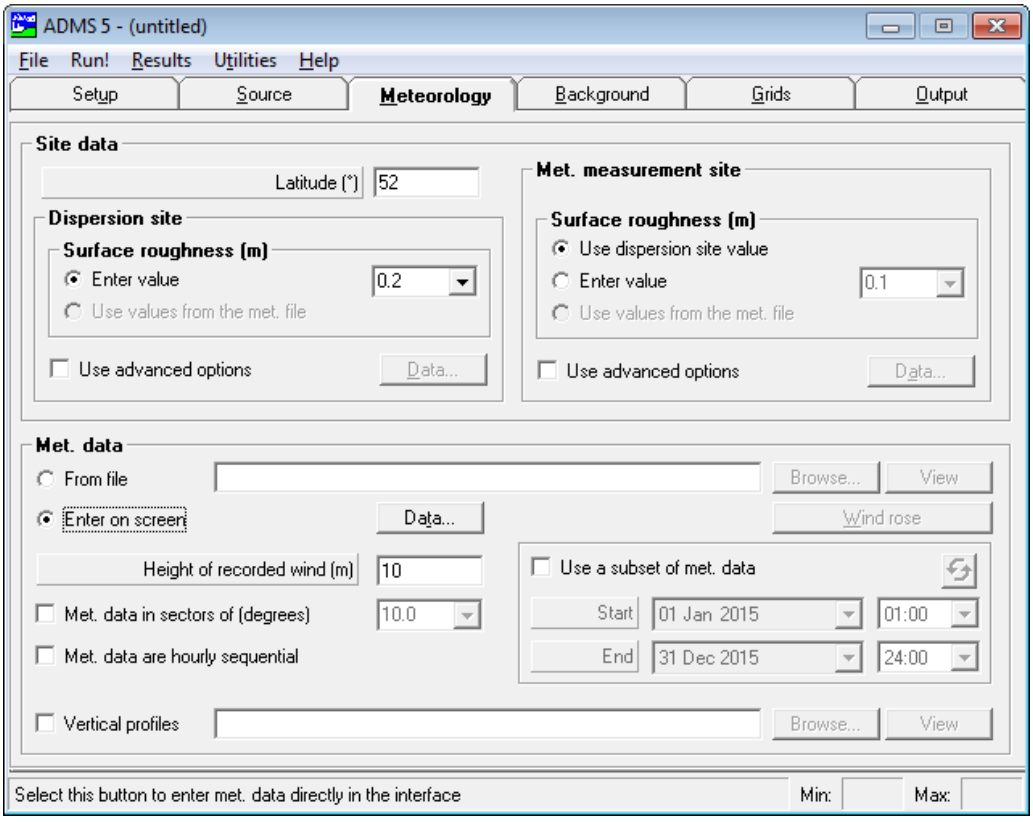


Figure 8.4 − Example 1: the **Meteorology** screen.

The minimum data required are either the *wind speed, wind direction, year, Julian day number, time of day and cloud cover*, OR the *wind speed, wind direction and surface heat flux*.

If any of the other meteorological parameters shown on the right of the screen are known, they can be added by clicking in the appropriate check boxes. If you have no available data to enter here it is better to let the model calculate parameters from the existing data rather than make an inaccurate assumption.

For this example we want to look at the influence on dispersion of meteorological conditions that correspond *approximately* to the Pasquill-Gifford stability categories A to G. However, it is important to understand that there is no exact equivalence between these stability categories and the description of boundary-layer meteorology in terms of boundary layer height, *h*, and the Monin-Obukhov length, L_{MO}, used in ADMS 5 (Section 9.2).

In this example we will enter wind speed, wind direction and surface heat flux plus the boundary layer height.

Step 10 Enter the data as shown in **Figure 8.5.**

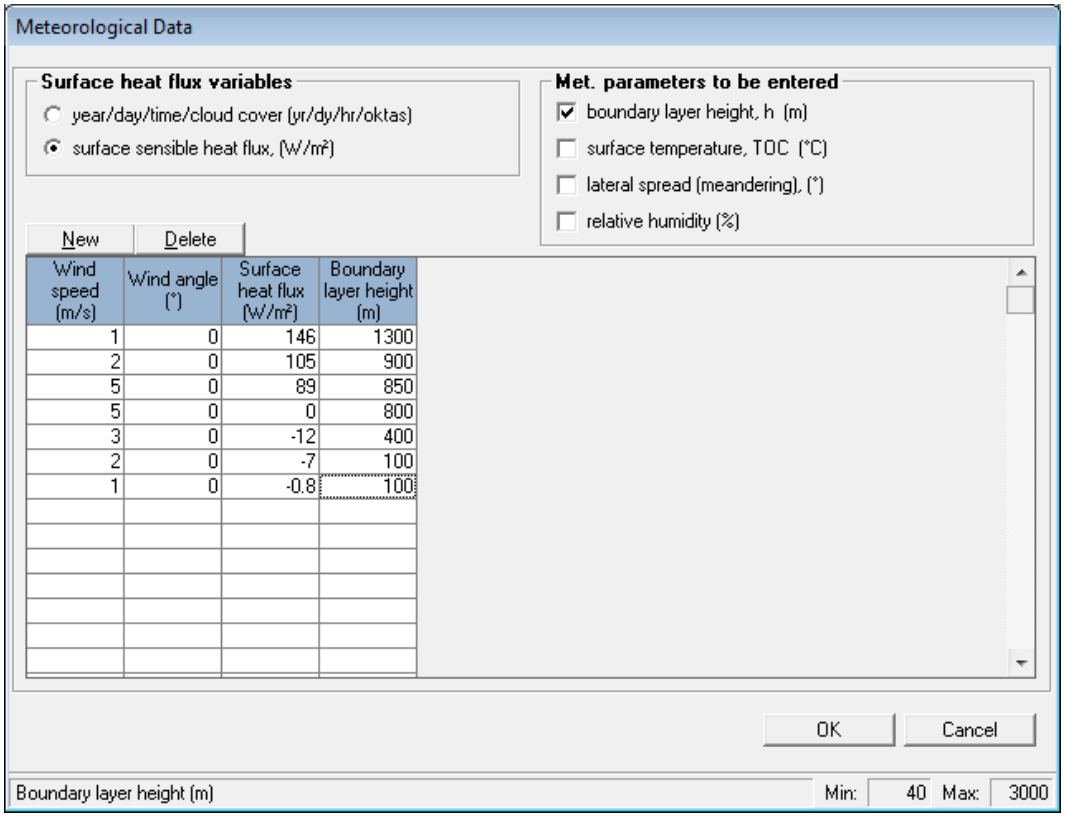


Figure 8.5 − Example 1: the **Meteorological Data** screen.

The next screen is the **Background** screen. As we don't want to enter any background concentrations for this example leave this set to none as shown in **Figure 8.6**.

Now the area over which we want to calculate concentrations must be defined. This is carried out in the **Grids** screen.

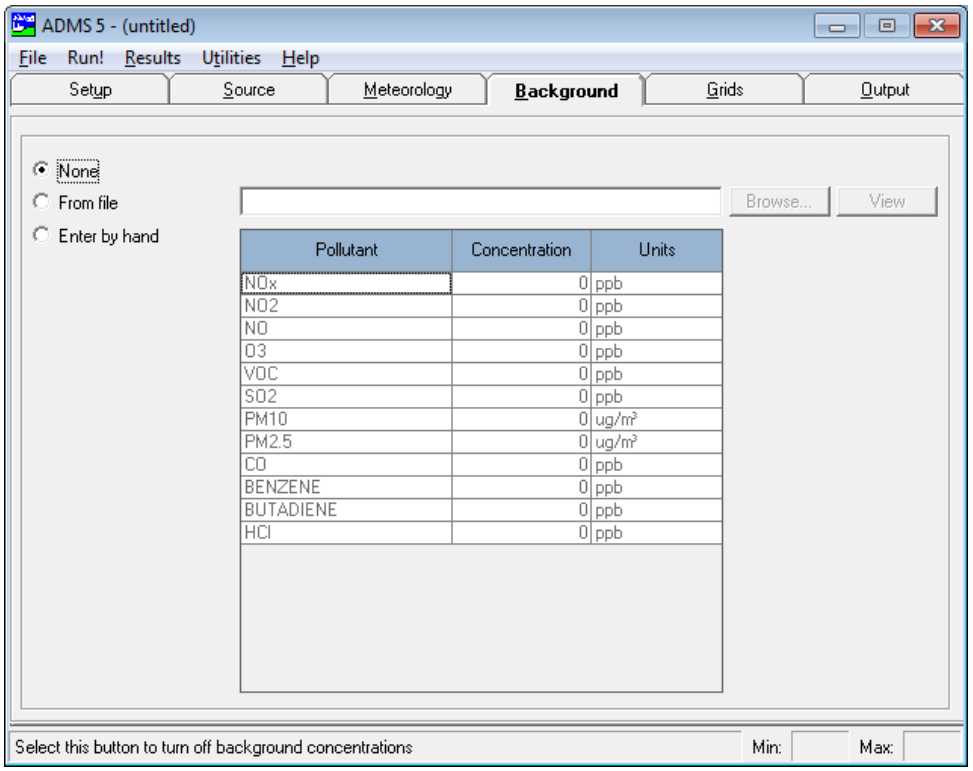


Figure 8.6 − Example 1: the **Background** screen.

The regular Cartesian grid is set up by specifying the minimum and maximum X, Y and Z coordinates of your calculation grid and the number of points in each direction. The maximum number of points in each horizontal direction is 2001 and the maximum in the vertical direction is 501. It is often a good idea, especially when there is one main source, to use a 31×31 point grid or a 101×101 point grid so that concentrations will be calculated at an equal number of points in both directions from a central reference point (e.g. (0,0)) as well as *at* that reference point.

The default grid is a ground level Cartesian grid with 31 regularly spaced points in the X and Y directions with a width of 2 km in each direction. The grid is centred on (0,0) which was the source position given in the **Source** screen.

Step 11 Set the minimum X and Y values to be -500 m and the maximum to be 500 m and select the number of points in the X and Y directions to be 21, as shown in **Figure 8.7**.

This will create a ground level calculation grid extending to 500 m in each direction from the central (0,0) location with a grid spacing of 50 m.

Click on the **Output** tab to enter the **Output** screen (**Figure 8.8**). The default screen shows a **Source** output for the "Boiler stack" selected, but no pollutant output is specified. Selecting the **Source** option for output ensures that the line plotting utility may be used. (This type of output is not available with multiple source runs or long-term average calculations.)

Step 12 Click on **New** in the **Pollutant output** box to add a line to the table.

The default is for the model to calculate short-term (**ST**) concentrations of NO_X with an averaging time of 1 hour, giving the results in units of $\mu g/m^3$.

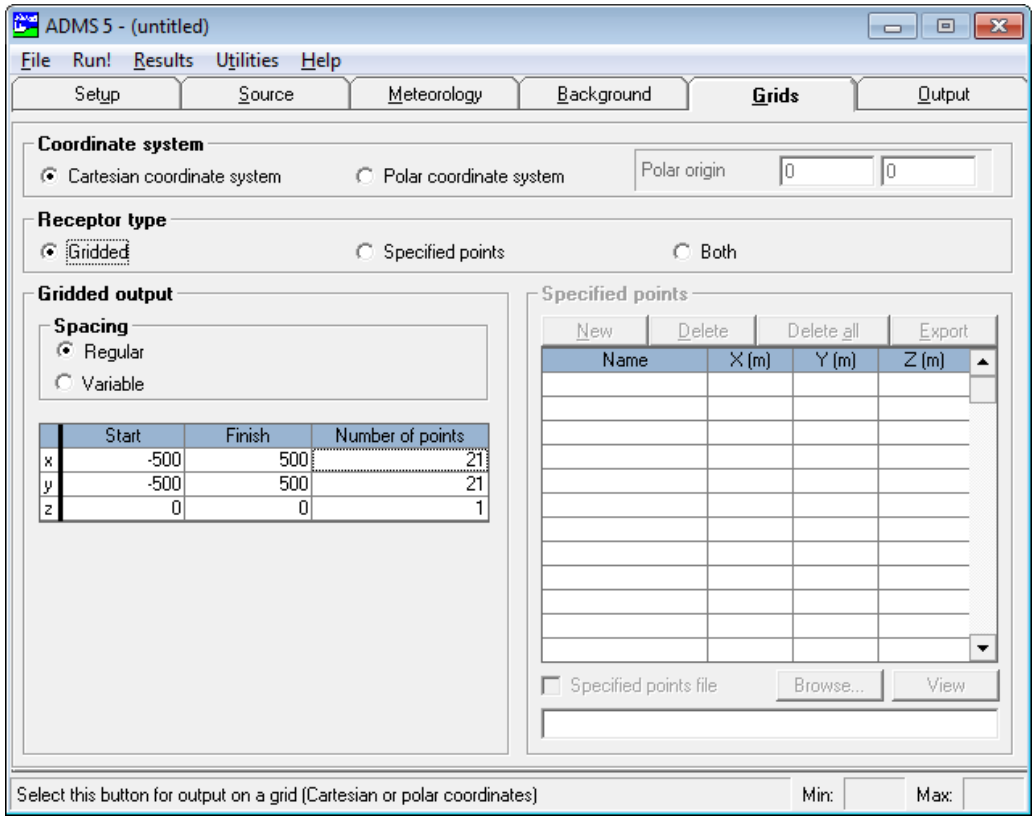


Figure 8.7 − Example 1: the **Grids** screen.

Step 13 Change the output pollutant to SO_2 by clicking on the down arrow in the pollutant name cell and selecting SO_2 from the drop-down list of pollutants that appears. *Ensure that the output pollutant chosen is the pollutant emitted from the stack.*

Make sure that there are red ticks in the **Include** column and the **Source** output table (next to the "Boiler stack"). If not, click on the appropriate cell or enter **Y** in the cell.

---

*If no pollutants are selected in the **Pollutant output** table, the program will not run successfully. You MUST choose at least one.*

Your **Output** screen should look as shown in **Figure 8.8**.

Having now specified all the input required, you just need to save the configuration before running the model.

Step 14 Click on **File**, **Save** or **Save As**... and save the file as *Ex1_point.apl*.

If you are asked about verification click on **Yes** to run the verification checks; refer to Section 2.5 for more details.

Step 15 Click on **Run!** to run the model.

The run will only take a few seconds for this example. A run window similar to **Figure 8.9** 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 related to the meteorological data.

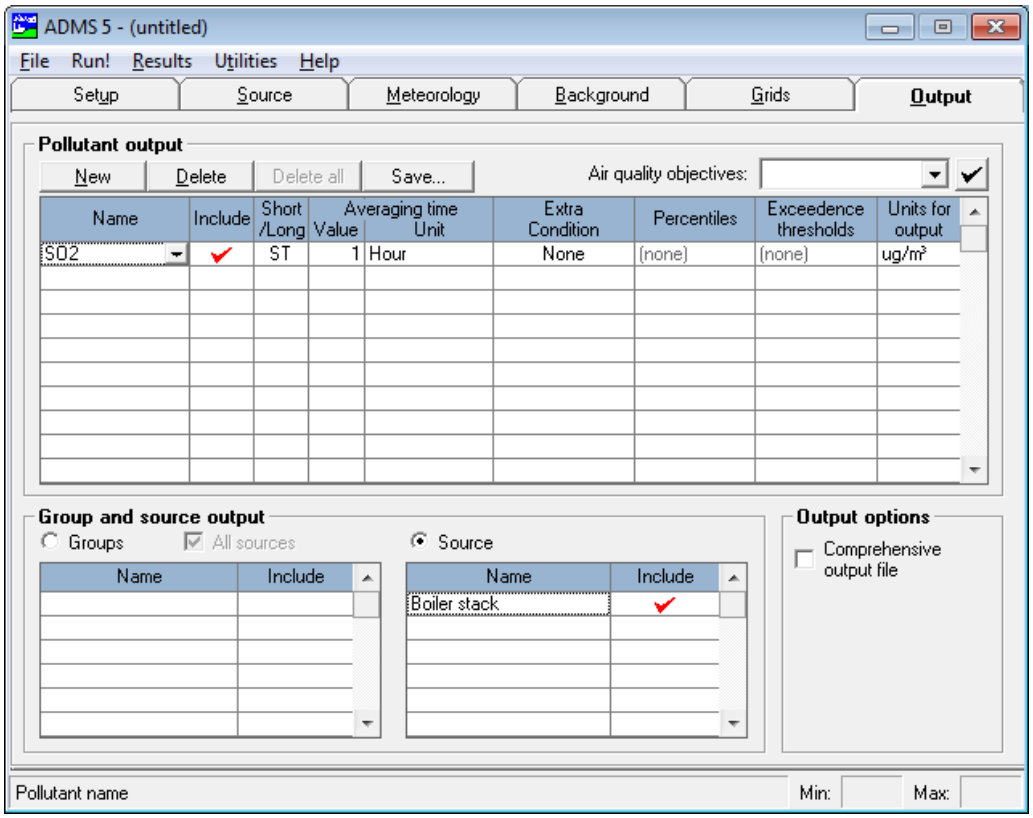


Figure 8.8 − Example 1: the **Output** screen.

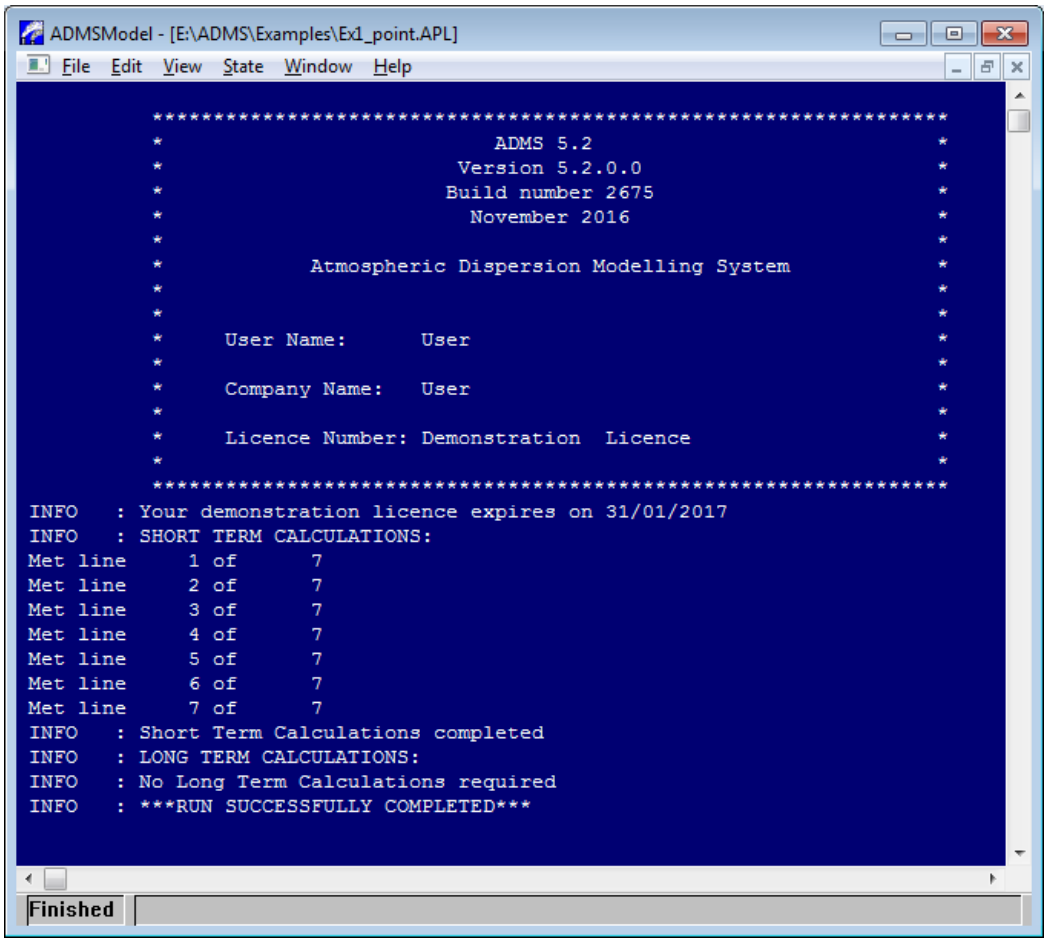


Figure 8.9 − Example 1: the run window.

At the end of the run the dialog box shown in **Figure 8.10** will appear. This means that the run has completed successfully.

Step 16 Click on **Yes** to close the window shown in **Figure 8.10**.

Alternatively, click on **No** to leave the window open and view the screen messages. Use the **File**, **Exit** command from the menu bar of the run window to return to the main interface of ADMS 5.

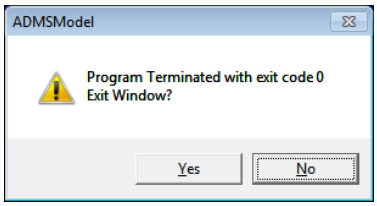


Figure 8.10 − Example 1: end-of-run dialog box.

8.1.2 Viewing output results

In ADMS 5, the results can be viewed in three ways, namely line plots, contour plots and numerically. Go to the main **Results** menu at the top of the interface to see these three options.

The line plotting utility

This utility plots line graphs of quantities such as the ground-level concentration, plume parameters, etc., and is described in Section 6.4.

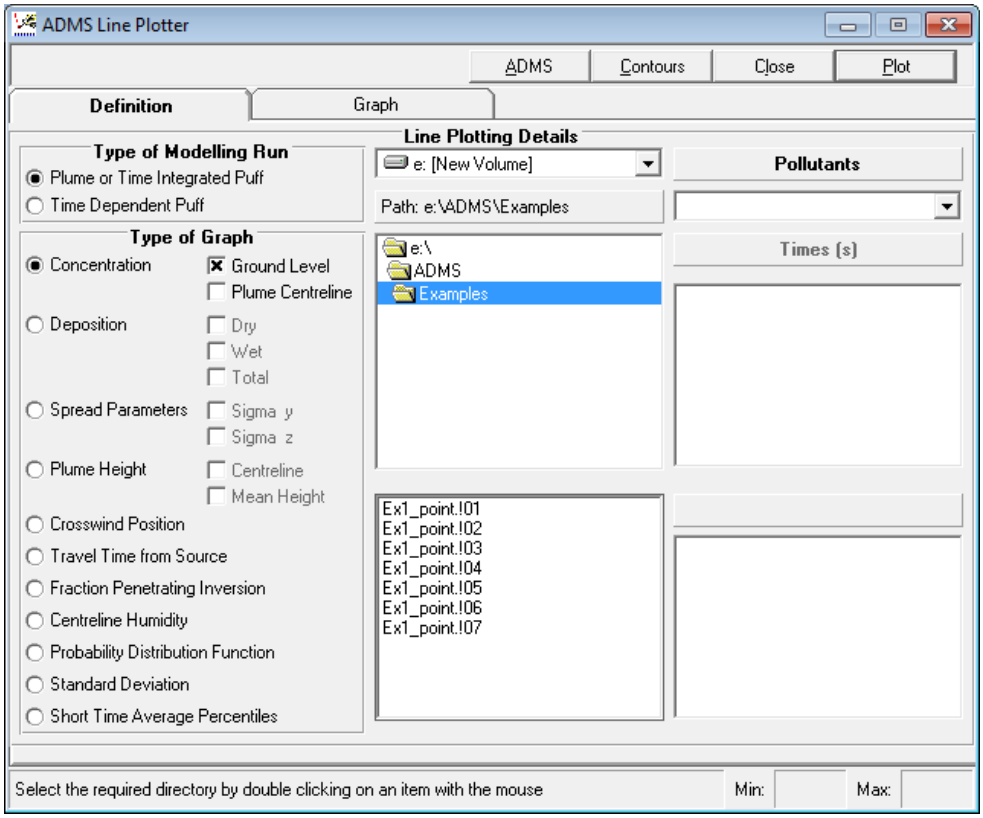


Figure 8.11 − Example 1: the **ADMS Line Plotting** utility.

Step 1 Launch the utility by selecting the **Results, Line Plot** menu command to access the screen shown in **Figure 8.11**.

For our example, there are 7 individual output files, one for each line of meteorological data entered in the **Meteorology** screen. These have the extension *.!n*, where *n* is the number of the line of meteorological data. Thus, in this case, the results in the file *point.!01* are for the highly convective meteorological conditions of the first line of data, which corresponds approximately to a category A stability. Similarly the results in the file *point.!07* are for the extremely stable meteorological conditions of the seventh line of data, which corresponds approximately to a category G stability.

The default line plots that are created are the ground level concentrations underneath the plume centreline.

Step 2 To look at such results for any one output file, click on that file name and press **Plot**.

To look at such results for all output files, select all seven output files (using the **SHIFT** key) and press **Plot**, so that we can compare the ground level concentrations for the modelled range of meteorological conditions on one graph.

This composite graph is shown in **Figure 8.12**. Different colours and symbols are used to distinguish data from each file plotted.

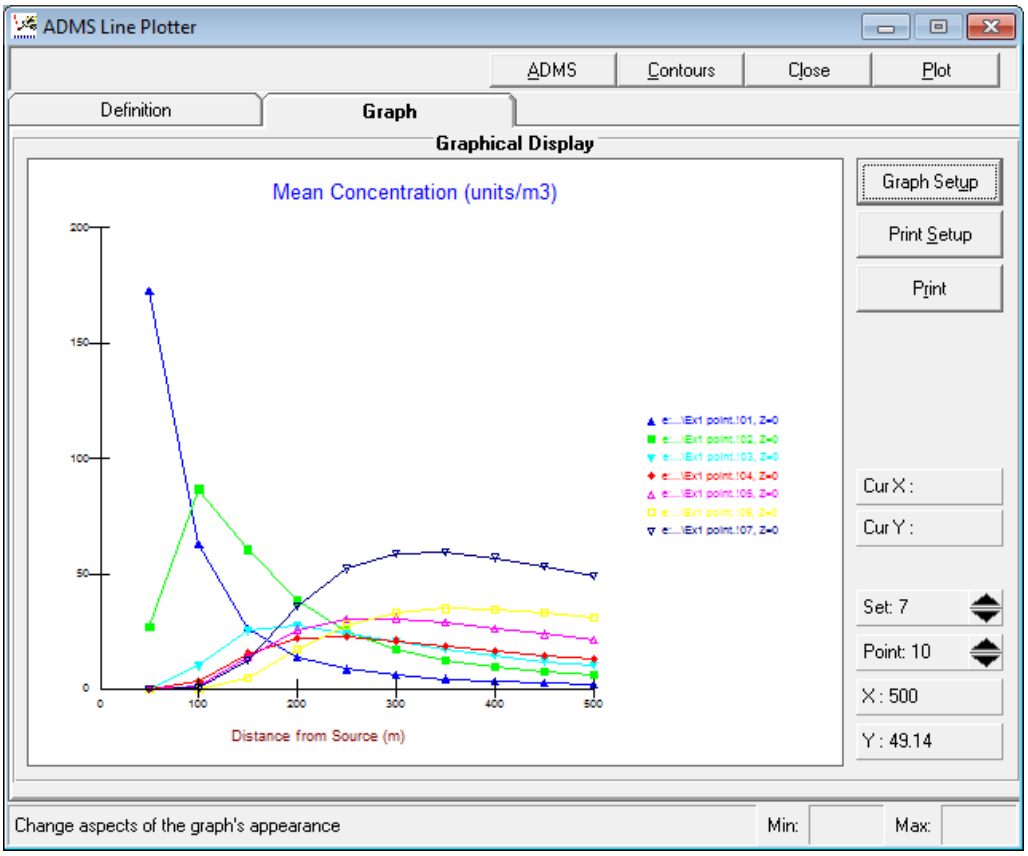


Figure 8.12 − Example 1: ground-level concentrations for all met. data lines.

From the graph, it can be seen that the maximum ground-level concentration occurs under conditions corresponding to the first meteorological line, i.e. in very convective conditions, and occurs very close to the source. The lowest maximum ground-level concentration occurs with the fourth meteorological line, i.e. under neutral stability conditions.

---

*Concentrations are calculated at distances starting 50 m downwind from the source, as this is the grid spacing defined in the **Grid** screen. To calculate concentrations at shorter downwind distances, a finer output grid or the **Specified points** option on the **Grids** screen should be used.*

Other results can be plotted, such as deposition when it has been calculated. Here, in order to plot the plume spread parameters, go back to the **Definition** screen.

Step 3 Click on the **Definition** tab to return to the screen shown in **Figure 8.11**.

Step 4 Click on **Spread Parameters** and check the **Sigma z** option.

Step 5 Select all the files again and click on **Plot**.

This time the vertical spread of the plume is shown, for each meteorological condition, as shown in **Figure 8.13**. The figure shows the vertical plume spread as a function of distance downstream. This illustrates that the greatest vertical spread occurred with the most convective conditions and the least vertical spread with stable conditions.

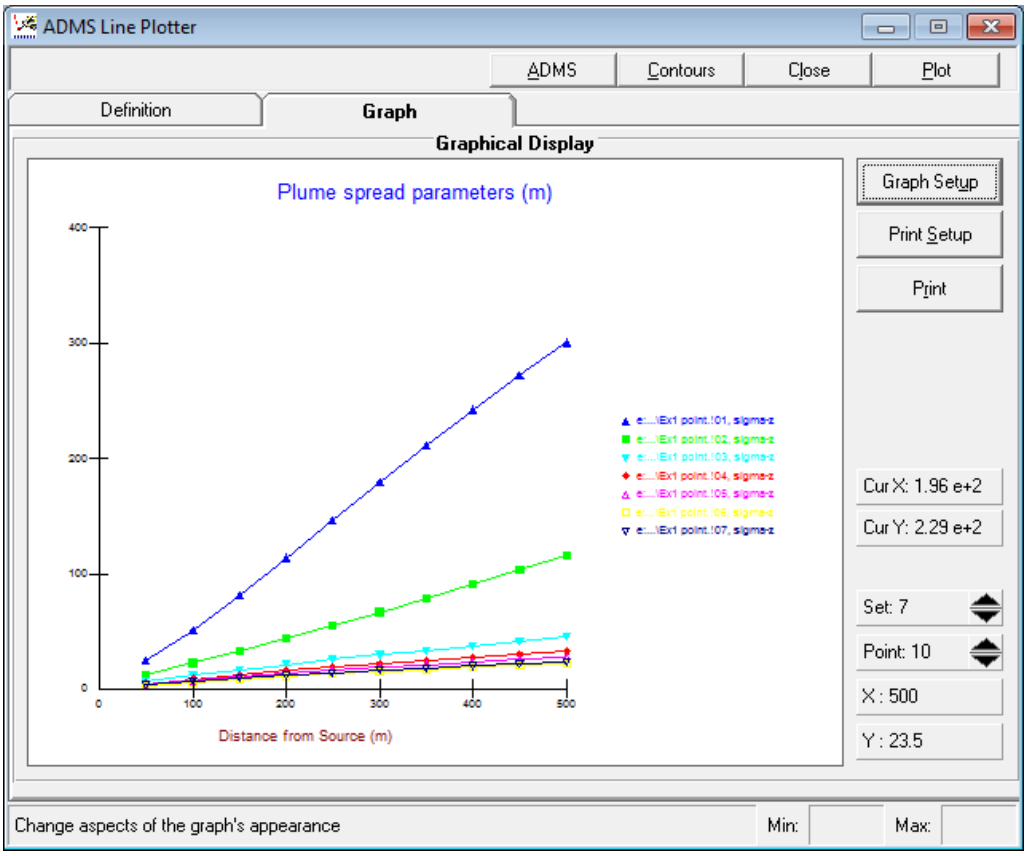


Figure 8.13 − Example 1: vertical plume spread parameters for all met. data lines.

The legends, titles and symbols in a line plot can be edited and/or enlarged or reduced by clicking on the ***Graph Setup*** *button. The graphs can also be saved and imported into documents for later reporting. For further details, please refer to Section 6.4.*

Contour Plotting

The concentration results can also be viewed as contour plots over the output grid specified on the **Grids** screen. The contour plots are produced in Surfer with the **2-D Output Plotter** utility (see full details on the utility in Section 6.3).

To create a contour plot for the run that has just completed, proceed as follows. All the information for contour plotting from this run is contained in a single output file, *Ex1_point.gst*. This file contains SO_2 concentrations for each point in the output grid, for each line of meteorological data.

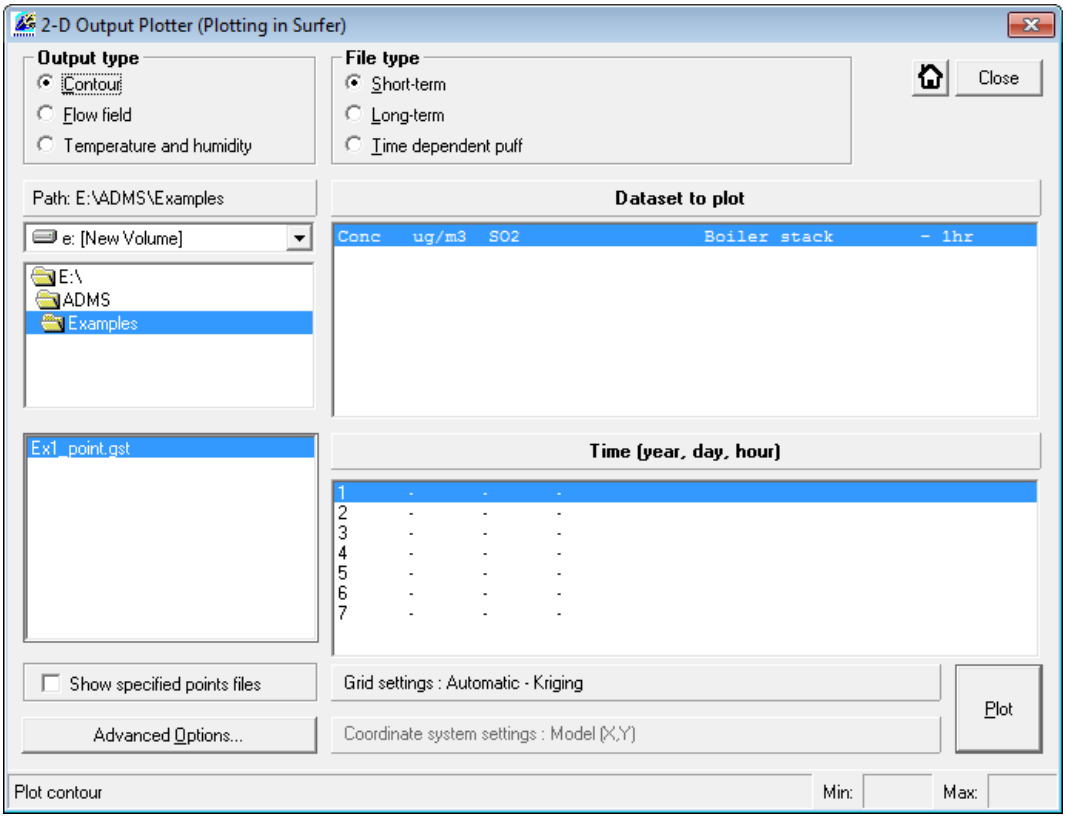


Figure 8.14 − Example 1: the **2-D Output Plotter** utility.

Step 1 Go to **Results, Plot in Surfer** to launch the **2-D Output Plotter** utility. Select **Contour** and then **Short term** to produce a screen like that shown in **Figure 8.14**.

Step 2 Highlight the **Time** and the **Dataset to plot** for the file you want to plot. (You can only make one contour plot at a time.)

Line 1 in the **Time** box at the bottom right of the screen corresponds to the first line of meteorological data (very convective) and line 7 corresponds to the last line of data (very stable).

Step 3 Click on **Plot**.

This will first launch the **Save Surfer Grid File As**... dialogue box – save with the default name, in this case *Ex1_point.grd*. Surfer is then launched and a contour plot of the ground-level concentration for that particular meteorological condition created in a separate window. **Figure 8.15** shows the automatically generated contour plot for line 2 (convective conditions).

The title of the plot clearly indicates which file, source and pollutant the results correspond to. The plot can be edited to show more contour levels, change colours, etc.

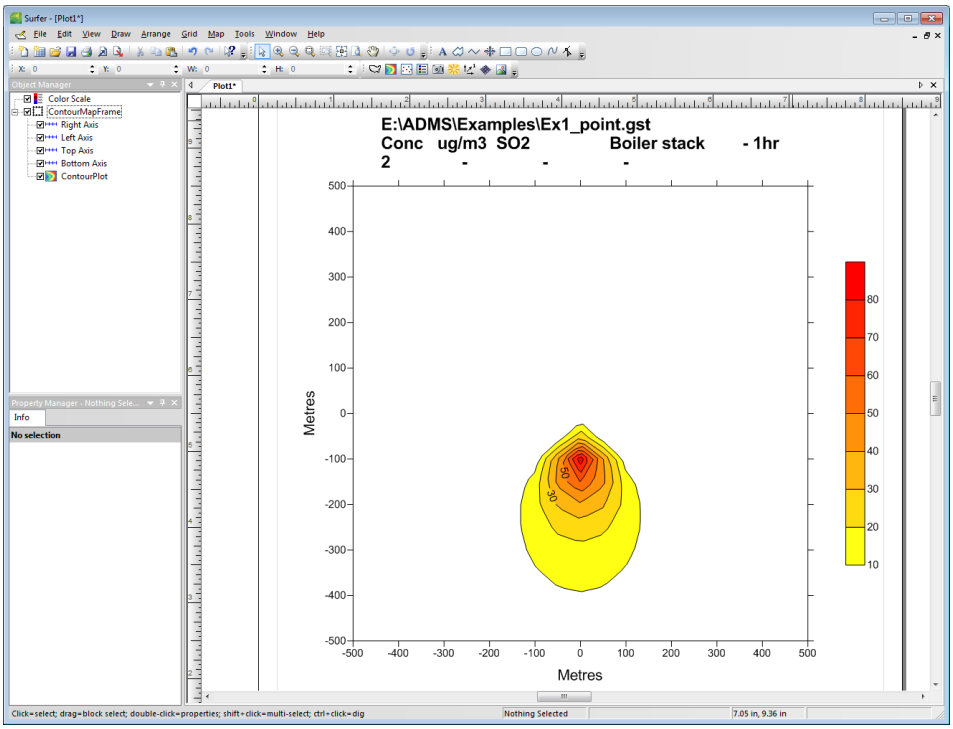


Figure 8.15 − Example 1: concentration contour plot (line 2 of met. data).

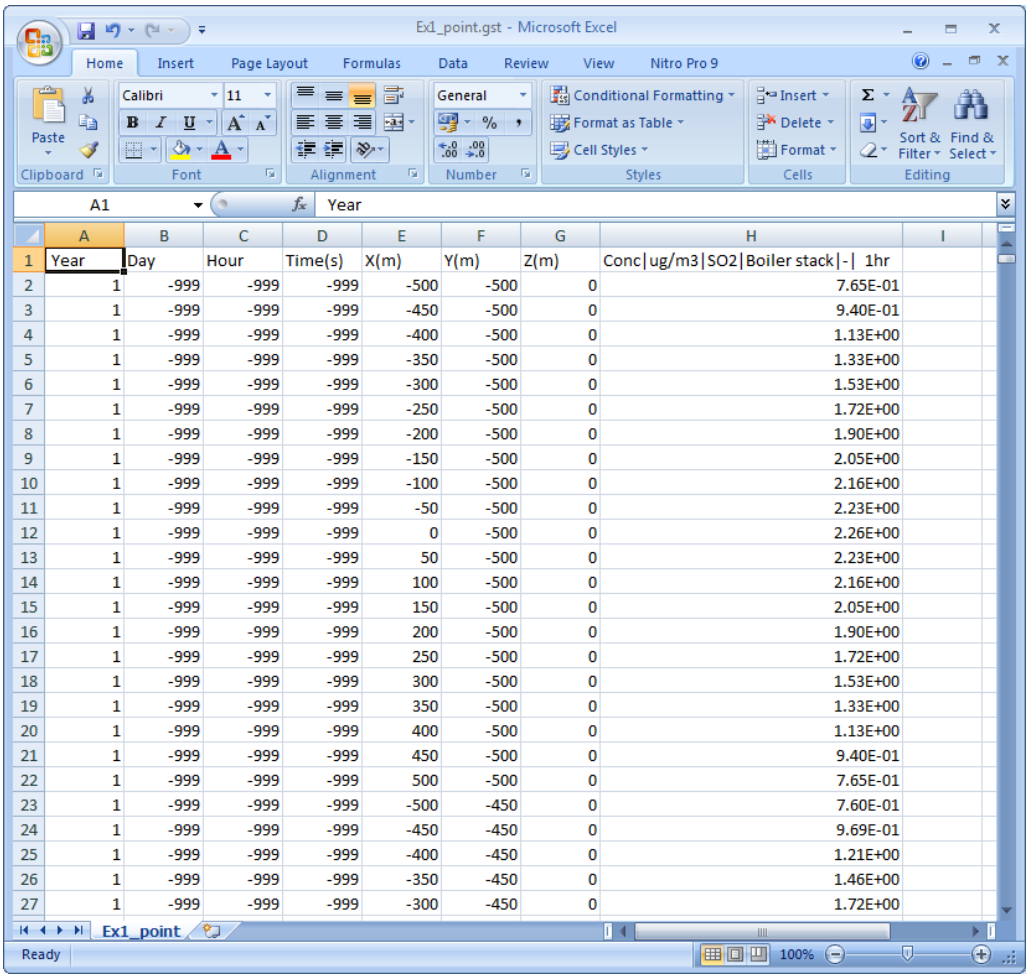
You can return to the **Contour Plotting** screen of ADMS 5 and select another dataset or time to plot. The contour plot will be produced in a new window, so that Surfer may be used to compare graphically the results for two different files.

Numerical results

A number of numerical output files are created for each run. These numerical files can be viewed by clicking on the **Results**, **Numerical**... menu command. This automatically opens an editor on your machine such as Notepad, WordPad or Microsoft Excel and lists the output files that are available for this run.

Users may find the files easiest to read when opened in Excel.

An example of a numerical output file is shown in **Figure 8.16** below. The user can read the concentration at any particular grid point for a particular meteorological data line (indicated by the number in the first column). If the year is a meteorological variable (i.e. either `YEAR` is defined in the *.met* file or **Year** has been entered on the **Meteorology** screen), then the first column contains the year, otherwise it contains the number of the meteorological data line.



	A	B	C	D	E	F	G	H	I
1	Year	Day	Hour	Time(s)	X(m)	Y(m)	Z(m)	Conc|ug/m3|SO2|Boiler stack|-| 1hr	
2	1	-999	-999	-999	-500	-500	0	7.65E-01	
3	1	-999	-999	-999	-450	-500	0	9.40E-01	
4	1	-999	-999	-999	-400	-500	0	1.13E+00	
5	1	-999	-999	-999	-350	-500	0	1.33E+00	
6	1	-999	-999	-999	-300	-500	0	1.53E+00	
7	1	-999	-999	-999	-250	-500	0	1.72E+00	
8	1	-999	-999	-999	-200	-500	0	1.90E+00	
9	1	-999	-999	-999	-150	-500	0	2.05E+00	
10	1	-999	-999	-999	-100	-500	0	2.16E+00	
11	1	-999	-999	-999	-50	-500	0	2.23E+00	
12	1	-999	-999	-999	0	-500	0	2.26E+00	
13	1	-999	-999	-999	50	-500	0	2.23E+00	
14	1	-999	-999	-999	100	-500	0	2.16E+00	
15	1	-999	-999	-999	150	-500	0	2.05E+00	
16	1	-999	-999	-999	200	-500	0	1.90E+00	
17	1	-999	-999	-999	250	-500	0	1.72E+00	
18	1	-999	-999	-999	300	-500	0	1.53E+00	
19	1	-999	-999	-999	350	-500	0	1.33E+00	
20	1	-999	-999	-999	400	-500	0	1.13E+00	
21	1	-999	-999	-999	450	-500	0	9.40E-01	
22	1	-999	-999	-999	500	-500	0	7.65E-01	
23	1	-999	-999	-999	-500	-450	0	7.60E-01	
24	1	-999	-999	-999	-450	-450	0	9.69E-01	
25	1	-999	-999	-999	-400	-450	0	1.21E+00	
26	1	-999	-999	-999	-350	-450	0	1.46E+00	
27	1	-999	-999	-999	-300	-450	0	1.72E+00	

Figure 8.16 − Example 1: *.gst* file viewed numerically.

Further work

Try running this example for different stack heights (e.g. 0 m, 150 m) to see which meteorological conditions will lead to the highest maximum ground-level concentrations as the stack height is varied.

8.2 Example 2: Long-term average and percentile concentrations

The previous example looked at the effect of seven individual meteorological conditions on the dispersion of a plume from a single stack. These were specific conditions designed to approximate different classes of atmospheric stability.

Most commonly, we are interested in the effect of *real recorded* meteorological data on the dispersion of plumes in the atmosphere. We want to know the maximum short-term concentrations together with the long-term average concentrations that are most likely to occur for a particular process in a particular part of the country, over a period of, for instance, a day, a year or 10 years.

In this next example, we will model a single source with a height of 45 m that represents a boiler stack. We are interested in the ground-level concentrations of SO_2 and NO_X that occur over a calculation grid area of 1.5 km in each direction from the stack, paying particular attention to the concentrations at specific receptor points in the area.

We may want to compare the modelled results with regulatory standards, notably the NAQS 99.9^{th} percentile of 15 minute averages for SO_2, and the 99.79^{th} percentile of 1 hour averages for NO_X. This normally requires a long-term average concentration calculation to be carried out, either with a single year of hourly sequential meteorological data or with 10 years of statistically analysed meteorological data for that site. However, to save time for this example, we will use one day of hourly sequential data.

We want to know the long-term average concentration (in this case the daily average) together with percentile concentrations recorded at each point over the *whole day*.

---

*If we were using a meteorological data file containing a year of data, the long-term average would be the **annual** average concentration.*

Although the two pollutants have different averaging times, ADMS 5 has the ability to combine different averaging times in a single run. Hence we need only perform one run to calculate long-term average concentrations for SO_2 with a 15-minute averaging time and for NO_X with a 1-hour averaging time.

8.2.1 Setting up the run

The meteorological data file to be used in this example is *oneday.met*, which is a sample data set provided with the model (it may be found in the directory *<install_path>\Data*). This file may be viewed by opening it in an editor such as Notepad, WordPad or Microsoft Excel as a comma-separated variable file. A portion of the file is shown below in **Figure 8.17**.

This file contains a single day of hourly sequential meteorological data. It is for day number 247 of the year (4^{th} September in a non-leap year) and it can be seen that the file contains data for hour 1 to hour 24 for that day.

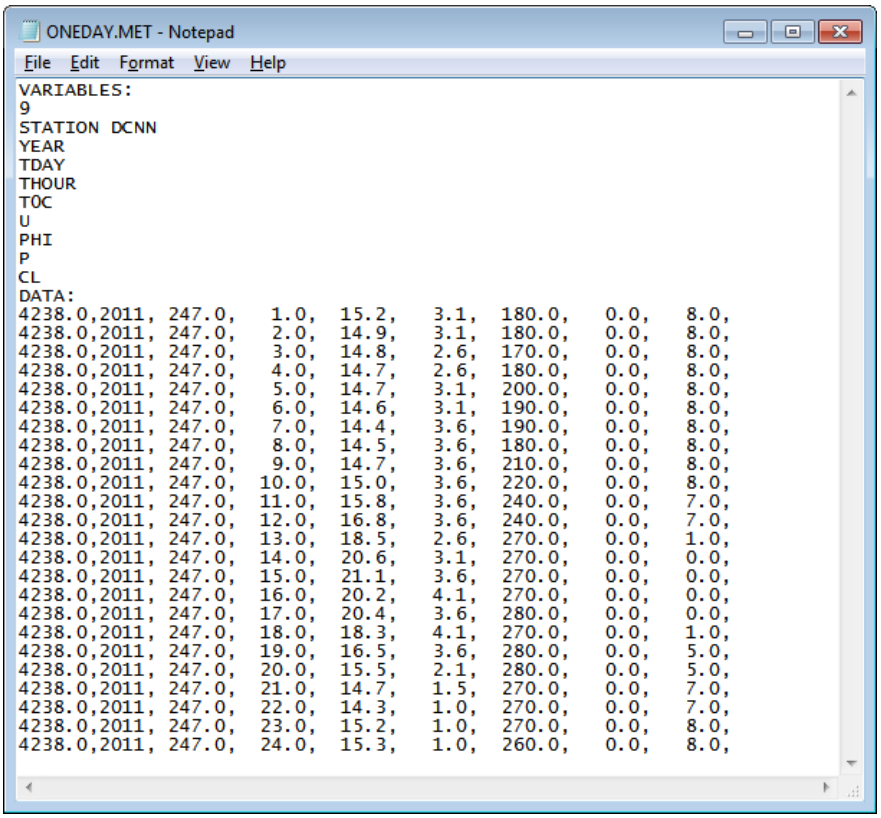


Figure 8.17 − Example 2: The meteorological data file *oneday.met* viewed in Notepad.

To set up the example, proceed as follows.

Step 1 Open ADMS 5 and click on **File, New** to create a new input file, i.e. a default input file or "clean sheet" in which to enter data.

Step 2 In the **Setup** screen, add entries to the **Name of site** and **Name of project** boxes.

Leave the rest of the screen as it is, as we will not be using any of the additional model options in this example. Compare with the screen shown in **Figure 8.18**.

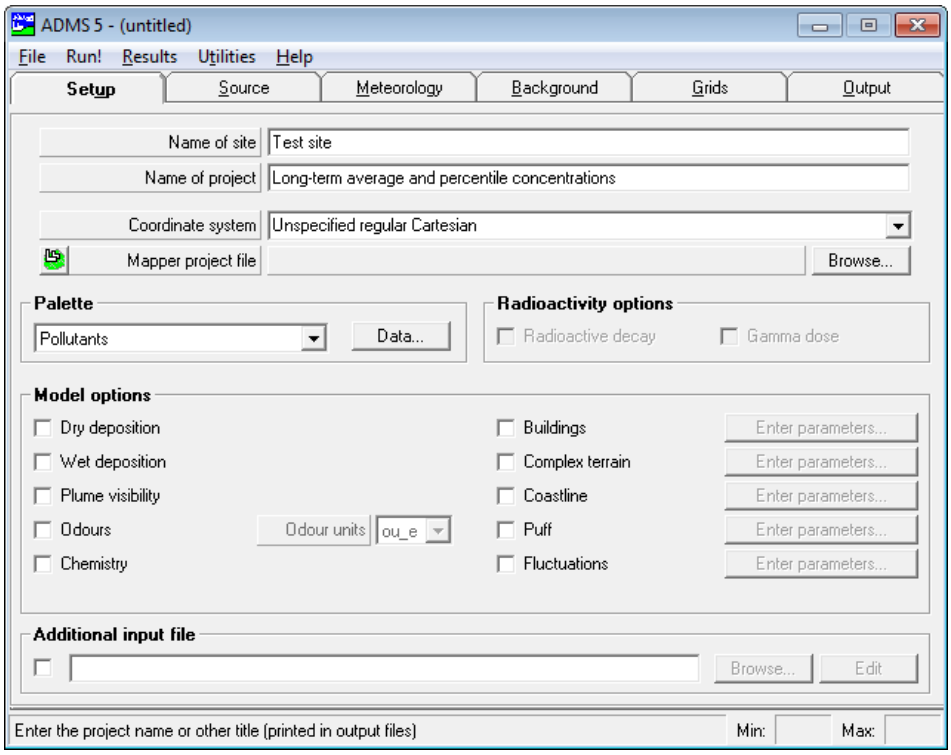


Figure 8.18 − Example 2: the **Setup** screen.

Step 3 Go to the **Source** screen and enter a new point source. Call it "Boiler Stack" and specify a height of 45 m, a diameter of 0.5 m, an exit velocity of 20 m/s and a temperature of 65°C. The stack is positioned at (0,0).

Step 4 Click on **Emissions**... and click on **New** to include two pollutants in the table. Select NO_X from the drop-down menu as the first pollutant species with an emission rate of 7.5 g/s and SO_2 as the second species with an emission rate of 10 g/s.

Step 5 In the **Meteorology** screen, leave the **Latitude** as 52° and set the dispersion site surface roughness to **Parkland, open suburbia**, corresponding to 0.5 m. Now choose the **From file** option for the meteorological data and click on the **Browse**... button to locate the file *oneday.met*. Click on **Met. data are hourly sequential**. Click on **Met. data in sectors of (degrees)** and select **10.0** from the drop-down box, to indicate that the wind directions in the meteorological data were recorded to the nearest 10°.

The **Meteorology** screen should match that shown in **Figure 8.19**. We will also add background data using the file *background.bgd* that again is located in *<install_path>\Data*.

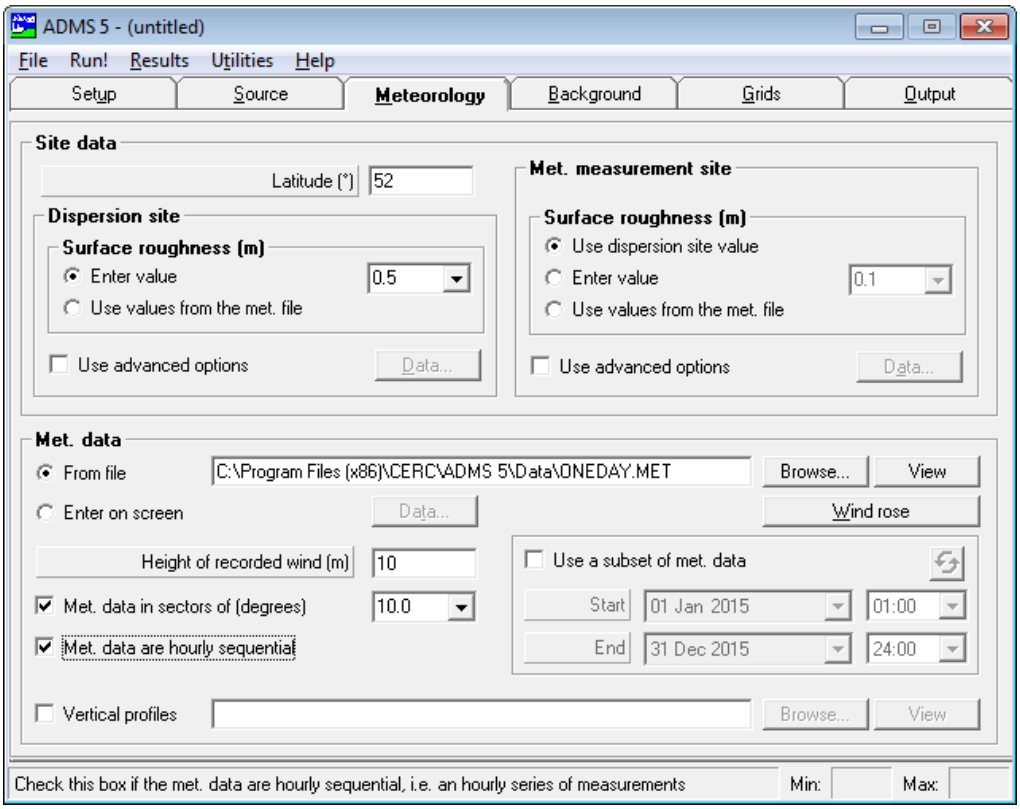


Figure 8.19 − Example 2: the **Meteorology** screen.

Step 6 Go to the **Background** screen, select the **From file** option and then click on the **Browse**... button to locate the file *background.bgd*.

The output will be calculated on a regular Cartesian grid over our chosen area *and* at specific receptor locations where there may be a school, a site boundary or some other sensitive area. To create a Cartesian grid extending to 1.5 km from the source in each direction we want to set up as detailed a grid as possible, with sensible grid spacing in order to capture the maximum concentrations. A good grid for this problem would be as shown in **Figure 8.20**.

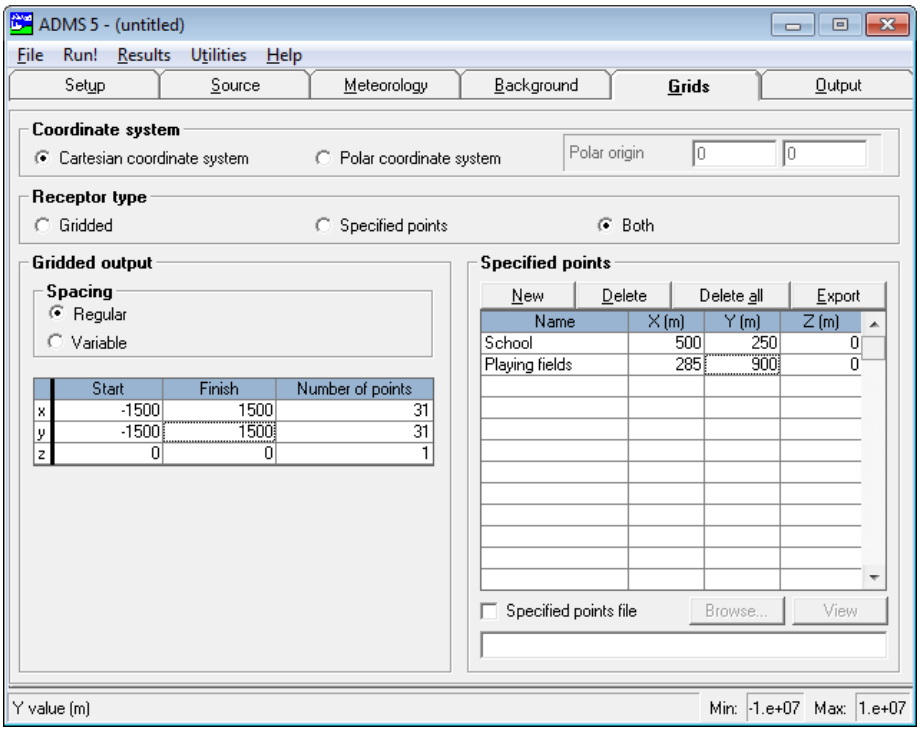


Figure 8.20 − Example 2: the **Grids** screen.

Step 7 Go to the **Grids** screen and click on the **Both** option under **Receptor type**.

Step 8 Set up the grid as shown in **Figure 8.20**. To enter (up to 100) specified points, click on **New** for each point. Each point requires X, Y and Z coordinates relative to (0,0,0). (By default the Z coordinate is zero, i.e. at ground level, but it can be at any elevation required.)

It is now time to define the required output.

Step 9 In the **Output** screen click on **New** to add a line to the **Pollutant output** table.

The default is for the model to calculate short-term **ST** concentrations of NO_X with an averaging time of 1 hour, giving the results in units of $\mu g/m^3$.

Step 10 Select **LT** by clicking in the **Short/Long** cell to select long-term calculations. Go to the **Percentiles** cell and click on the drop-down menu. Choose **Edit as table** and add '99.79' to calculate the 99.79^{th} percentile of hourly means for NO_X.

Step 11 Click on **New** to add a second line to the pollutant output table. Change the output pollutant to SO_2 by clicking on the down arrow in the pollutant name cell and selecting SO_2 from the list of pollutants that appears. Select **LT** and change the averaging time to 15 minutes. Click on **Edit as table** in the **Percentiles** cell in this row and add the 99.9^{th} percentile of 15 minute means for SO_2.

Step 12 Make sure that there are red ticks in the **Include** cells of the pollutant output table (next to NO_X and SO_2) and the source output table (next to "Boiler stack"). If not, click on the appropriate cell or enter **Y** in that cell.

Compare your input with **Figure 8.21** below.

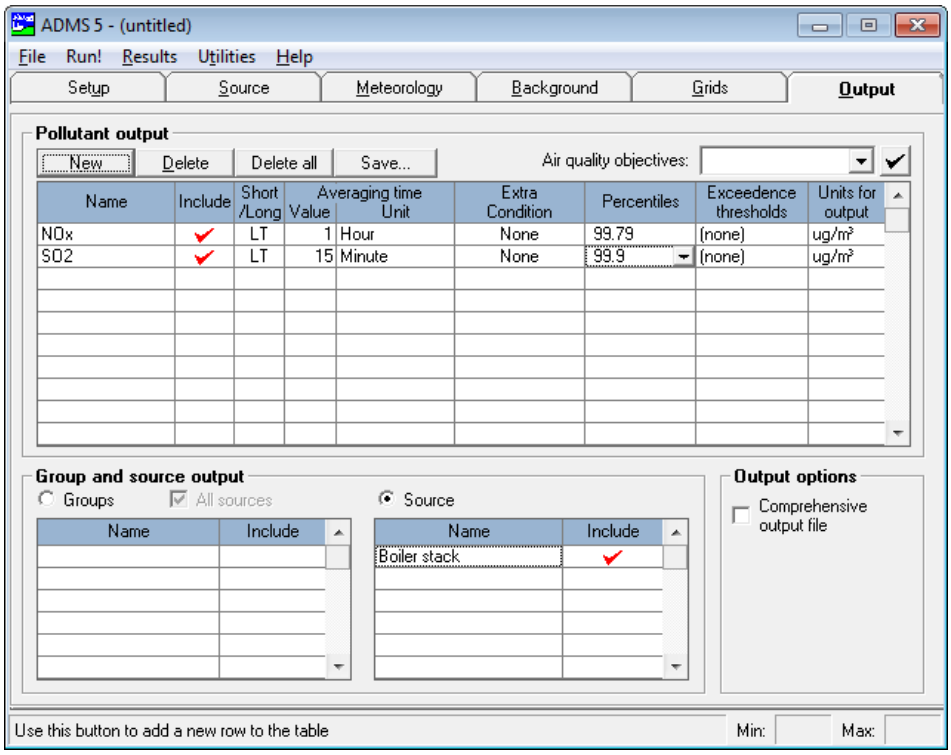


Figure 8.21 − Example 2: the **Output** screen.

Step 13 Save the file as *Ex2_longterm.apl* and run the model.

For the long-term average calculation there is an output file created called *Ex2_longterm.glt* containing four datasets, namely long-term average concentration results and percentile results for each of the two pollutants, together with a file called *Ex2_longterm.plt* giving the same results for each of the specified points.

8.2.2 Viewing output results

There is no line plot for a long-term average calculation. However, contour plots are available. Go to **Results**, **Plot in Surfer** and select **Contour** and **Long term** as shown in **Figure 8.22** below.

Plot the four data sets in turn. The contour plots for the long-term average and the 99.79th percentile of hourly average concentrations of NO_X are shown in **Figure 8.23**. From the contour plot the maximum average NO_X concentration is about 25 μg/m^3.

For accurate numerical values, always read the actual numerical maximum from the appropriate numerical file.

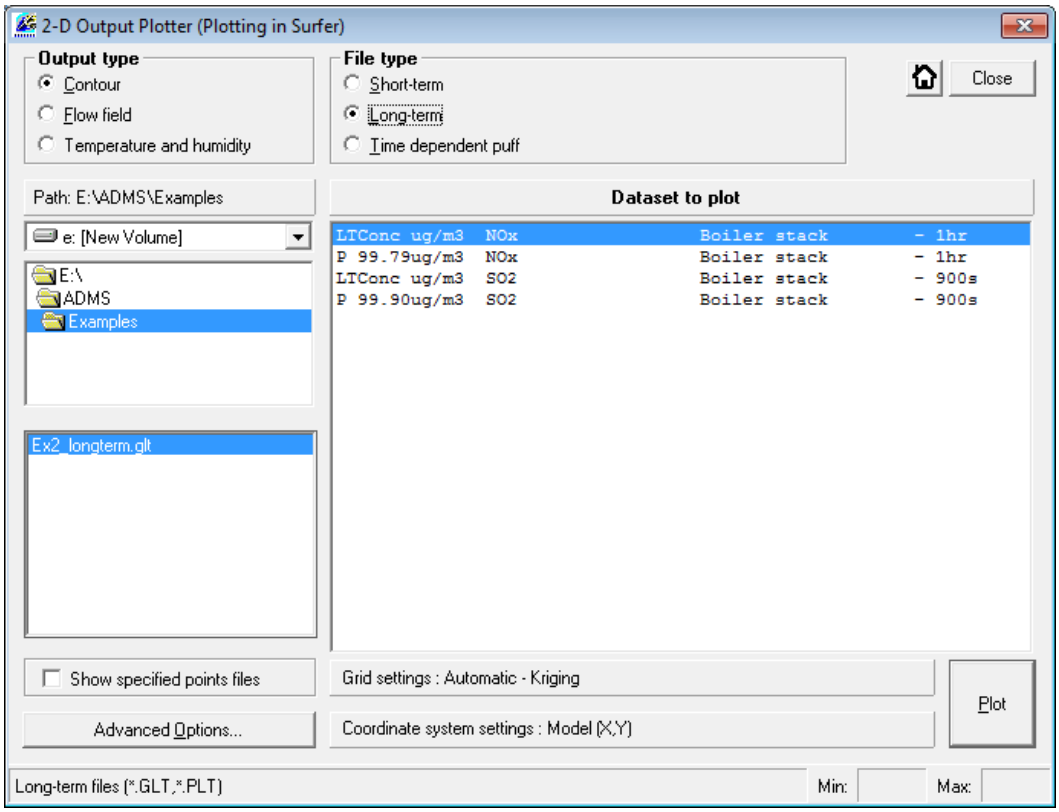


Figure 8.22 − Example 2: the **2-D Output Plotter** screen.

For the purposes of illustration, suppose this were an *annual* average concentration. Assuming all the NO_X consisted of NO_2, it would then be compared to the NAQS annual mean limit of 40 $\mu g/m^3$ NO_2. Note that NO_2 concentrations could be modelled directly using the **Chemistry** option, refer to Section 4.8 for details.

The 99.79th percentile plot shows that the 99.79th percentiles of hourly means are much higher than the daily average concentrations at each grid point. If it were for a whole year, then the maximum calculated value of about 90 $\mu g/m^3$ would be compared to the NAQS limit of 200 $\mu g/m^3$.

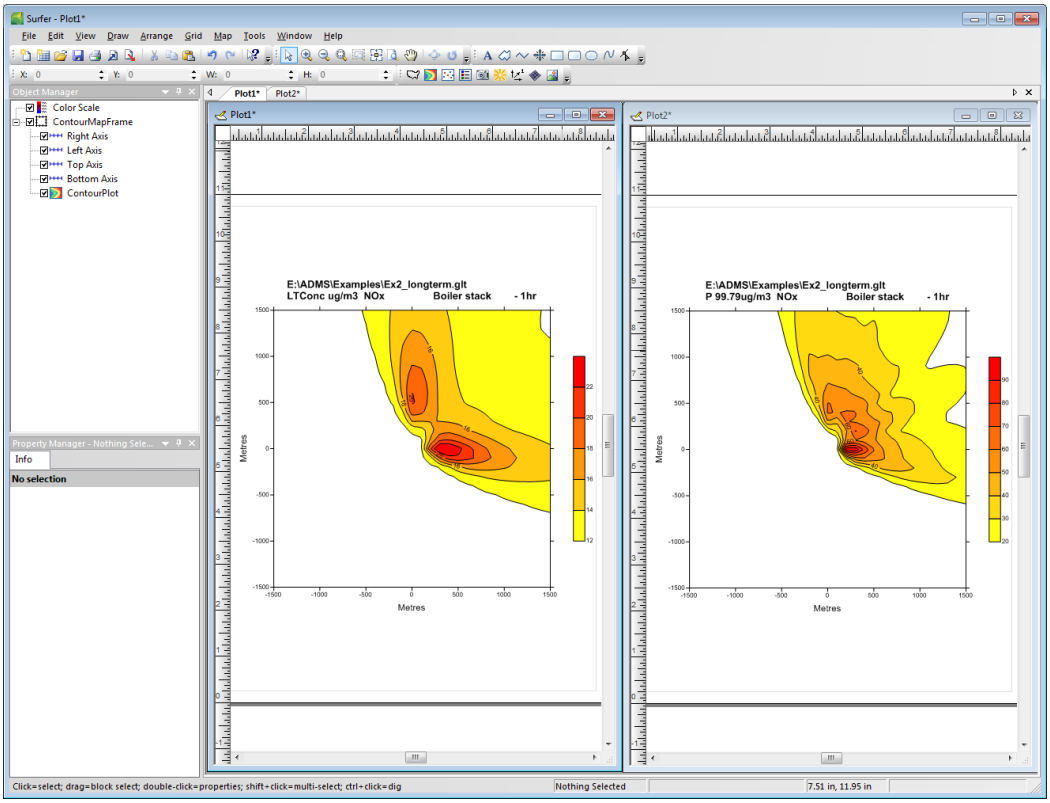


Figure 8.23 − Example 2: contour plots in Surfer for the NO_X.

Look at the long-term SO_2 results in the same way as NO_X. In this case if the run were for a year or more of meteorological data, the percentile results would be comparable with the NAQS limit of 266 $\mu g/m^3$ for the 15-minute average concentrations.

8.3 Example 3: Time-varying emissions with sequential meteorological data

Modelling of the atmospheric dispersion of pollutants is often based on the assumption that the release rate is constant with time. In reality, however, this may not be the case. For example, pollutants may be released from a plant at a higher rate during the night or in the winter months.

ADMS 5 is able to take into account the variation of emission characteristics with time. In order to do this, the emission must be specified for each hour of the period to be modelled, with the data either entered directly into the interface or listed in a file.

If entered directly into the interface, then an *emission factor* is entered for each hour of the day according to whether it is a weekday, Saturday or Sunday. The constant release rate specified in the **Emissions** screen is multiplied by the appropriate emission factor to give the release rate for that hour; other emission characteristics are assumed constant throughout the period. Meteorological data **must contain the year, day and hour** if emission factors are entered directly into the interface.

If the hour by hour time variation is specified in a *.var* file, then the file contains the *emission rate* and other source properties *for each time-varying source*. Statistical or sequential meteorological data may be used with this option. The exact format of the time-varying source file depends on the form of the meteorological data and is described in Section 4.1.

Alternatively, diurnal and monthly profiles of emissions factors along with wind-dependent operational periods can be entered for each source using a *.fac* file. Each source can be assigned any combination of a diurnal profile, a monthly profile and a sector of operational wind directions.

This worked example shows how to calculate concentrations arising from a time-varying source using sequential meteorological data supplied in a file.

8.3.1 Setting up the run

Three sources are considered, 'Stack1' and 'Stack2' emitting two pollutants each at a rate that varies and 'Stack3' emitting a single pollutant with a constant release rate. Ground-level concentrations are calculated separately for Stack2 and Stack3 for a 24-hour period in the summer.

Stack2 is located at (0,0), is 50 m high and the two pollutants emitted are sulphur dioxide (SO_2) and nitrogen oxides (NO_X). The release rate for SO_2 varies from 50 to 200 g/s, with the highest emission rates occurring in the afternoon. The release rate for NO_X also varies from 50 to 200 g/s, but the highest emission rates are at night. The average release rate over the 24-hour period for both SO_2 and NO_X is 120 g/s. Stack1 is located at (0,100) and is also 50 m high. Stack3 is located at (100,0), is also 50 m high, and is emitting sulphur dioxide at a constant rate of 120 g/s. Stack 3 has a diameter of 5 m, exit velocity of 6 m/s and temperature of 120 ^{o}C.

The results demonstrate that ground-level concentrations predicted from time-varying emissions may be significantly different to those with constant release rate, and will

also depend on the meteorology.

Step 1 Open ADMS 5 and click on **File, New** to create a new input file, i.e. a default input file or "clean sheet" in which to enter data.

Step 2 Enter suitable **Name of site** and **Name of project** titles for the run.

Compare with the **Setup** screen shown in **Figure 8.24**.

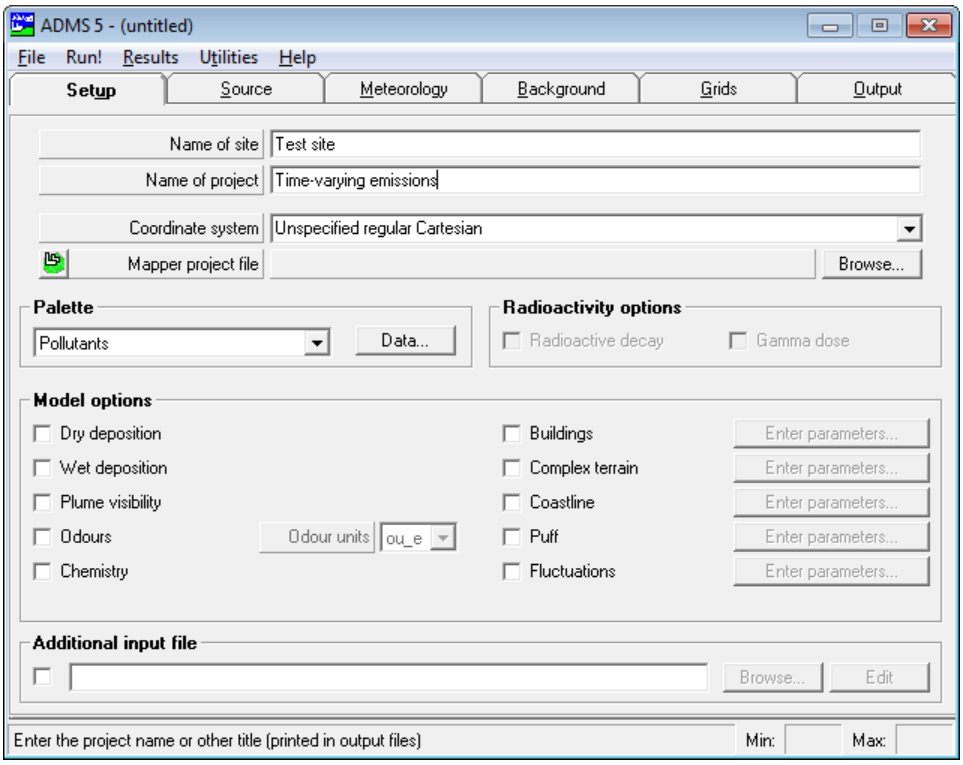


Figure 8.24 − Example 3: the **Setup** screen.

Step 3 Go to the **Source** screen, click **New** three times to add three new sources to the table.

Edit the source names to be 'Stack1', 'Stack2' and 'Stack3'. Check the box marked **Time-varying source data**, select the **Source data from *.var file** option and click on the **Browse**... button to locate the time-varying file *<install_path>\Data\TimeVary.var* (Section 4.1 describes the *.var* file formats). Make sure ticks appear in the **Use VAR file** column next to Stack1 and Stack2 but not next to Stack3. Edit the source details for Stack3 and location of Stack1.

Compare with the **Source** screen shown in **Figure 8.25**.

Step 4 Click on the **Emissions**... button. For Stack3, change the pollutant name to SO_2 from the drop-down list and set the release rate as 120 g/s as shown in **Figure 8.26**.

The emissions for Stack1 and Stack2 will be read from the time-varying file and so are not entered here.

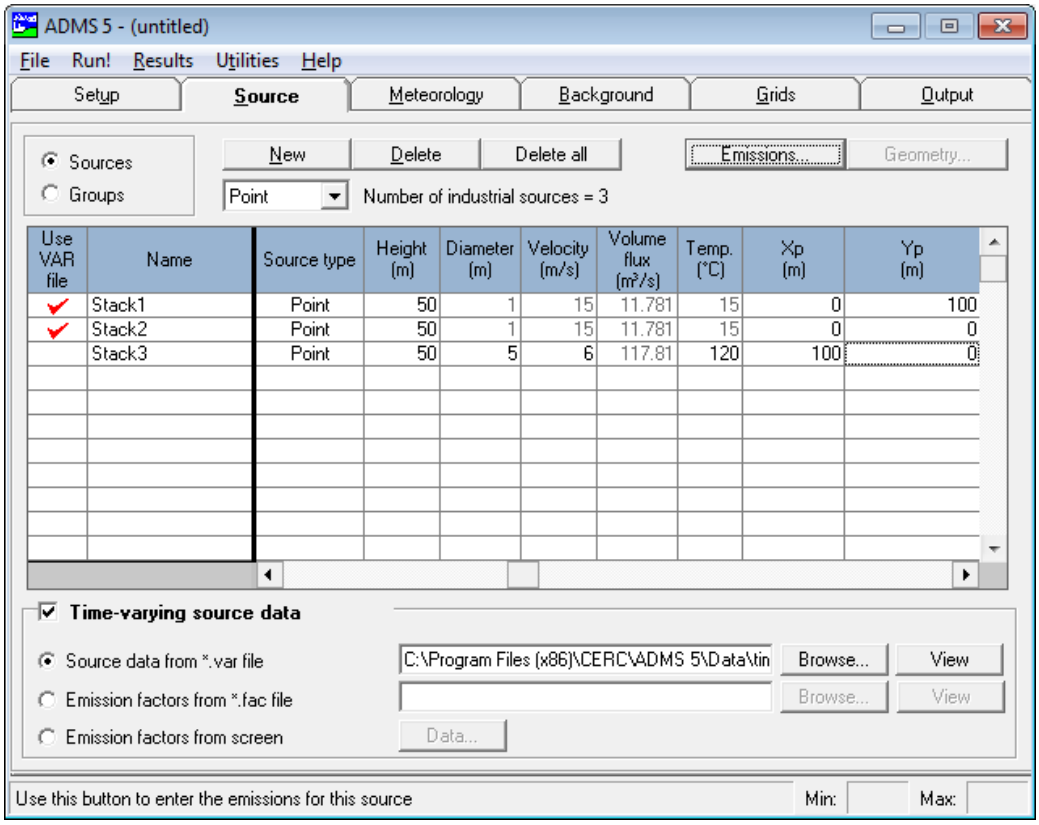


Figure 8.25 − Example 3: the **Source** screen.

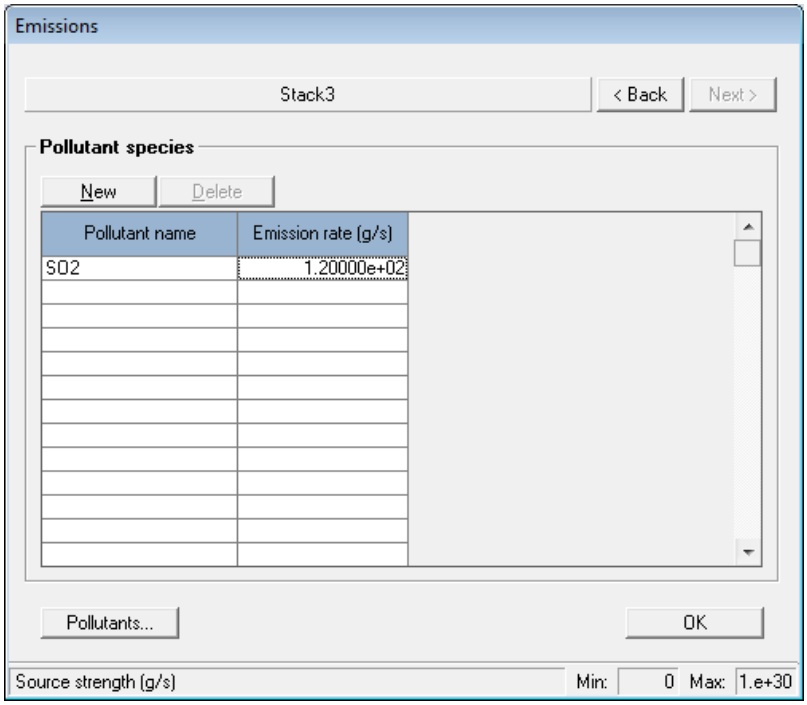


Figure 8.26 − Example 3: setting emission rate for Stack3.

Step 5 Select **Groups**. By default no groups are defined, but an option to output a group containing all of the sources is always available on the **Output** screen. Set up a new group called 'Group_Stack3' by clicking **New**. Enter a name for the group when prompted (**Figure 8.27**). In the table on the right of the **Source** screen, place a tick in the cell next to Stack3 by either double-clicking with the mouse or typing **Y**.

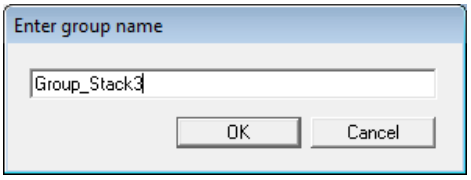


Figure 8.27 − Example 3: naming a new source group.

Compare with the **Source** screen shown in **Figure 8.28**.

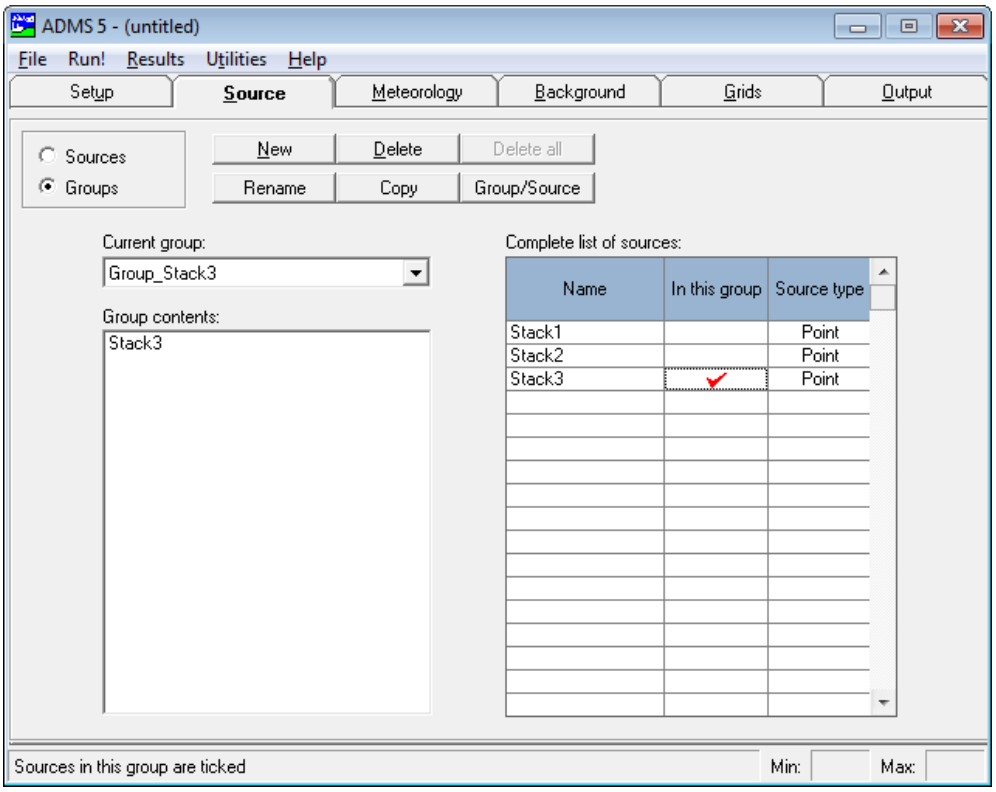


Figure 8.28 − Example 3: one of the source groups.

Step 6 Add another new group 'Group_Stack2' in the same way, this time placing a tick in the box next to Stack2 instead of Stack3.

Step 7 Go to the **Meteorology** screen and change the surface roughness length to **Agricultural areas (max)** corresponding to 0.3 m.

Step 8 Select the meteorological data to be entered **From file** and use the **Browse**... button to choose the file *<install_path>\Data\ONEDAY.MET*.

The file *ONEDAY.MET* is provided with ADMS 5 and shown in **Figure 8.29**. The file contains data for a 24-hour period in summer 2011. The wind speed (`U`) is low to moderate and the predominant wind direction (`PHI`) is westerly, although it is southerly at the start of the period. There is no precipitation (`P`) and the cloud cover

(CL) varies from zero in the afternoon to overcast during the night.

This meteorological data file is used for demonstration purposes only.

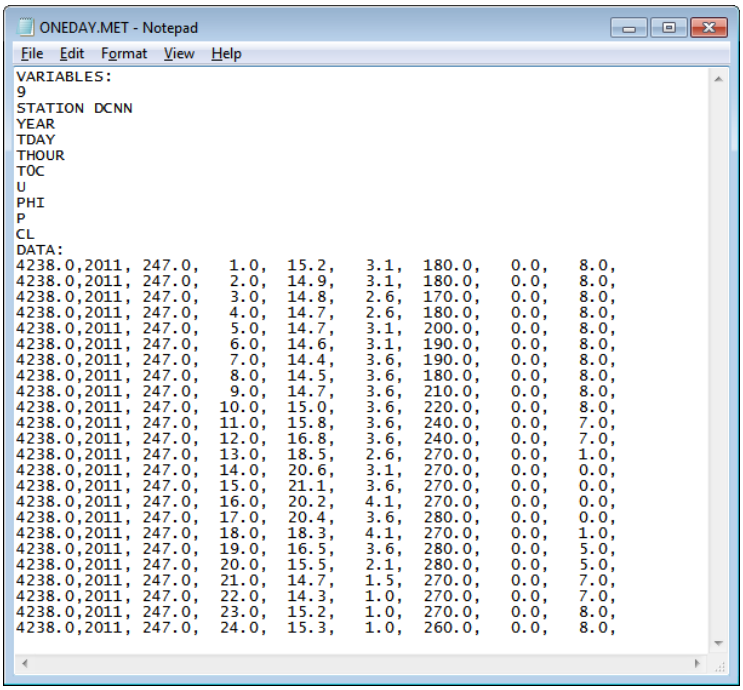


Figure 8.29 − Example 3: the meteorological data file *ONEDAY.MET*.

Step 9 Check the box on the **Meteorology** screen to indicate that the meteorological data are hourly sequential. Check **Met. data in sectors of (degrees)** and select **10.0** from the drop-down list, to indicate that wind direction was recorded to the nearest 10 degrees.

Compare with the **Meteorology** screen shown in **Figure 8.30**.

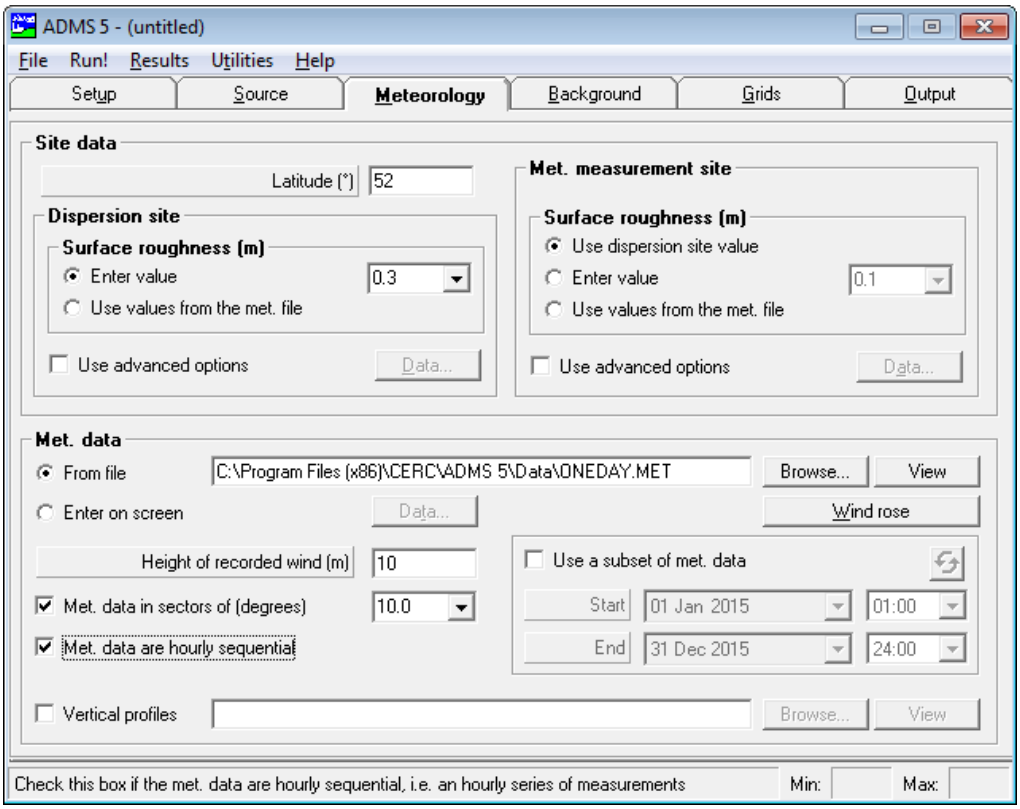


Figure 8.30 − Example 3: the **Meteorology** screen.

Step 10 Leave the background on the **Background** screen set to **None.**

An appropriate output grid for this problem defined on the **Grids** screen is shown in **Figure 8.31**. The grid extends 500 m south and west from the source and 2500 m to north and east.

In general, the wind could be blowing from any direction, so the grid would be set up such that the source was close to the centre of the grid.

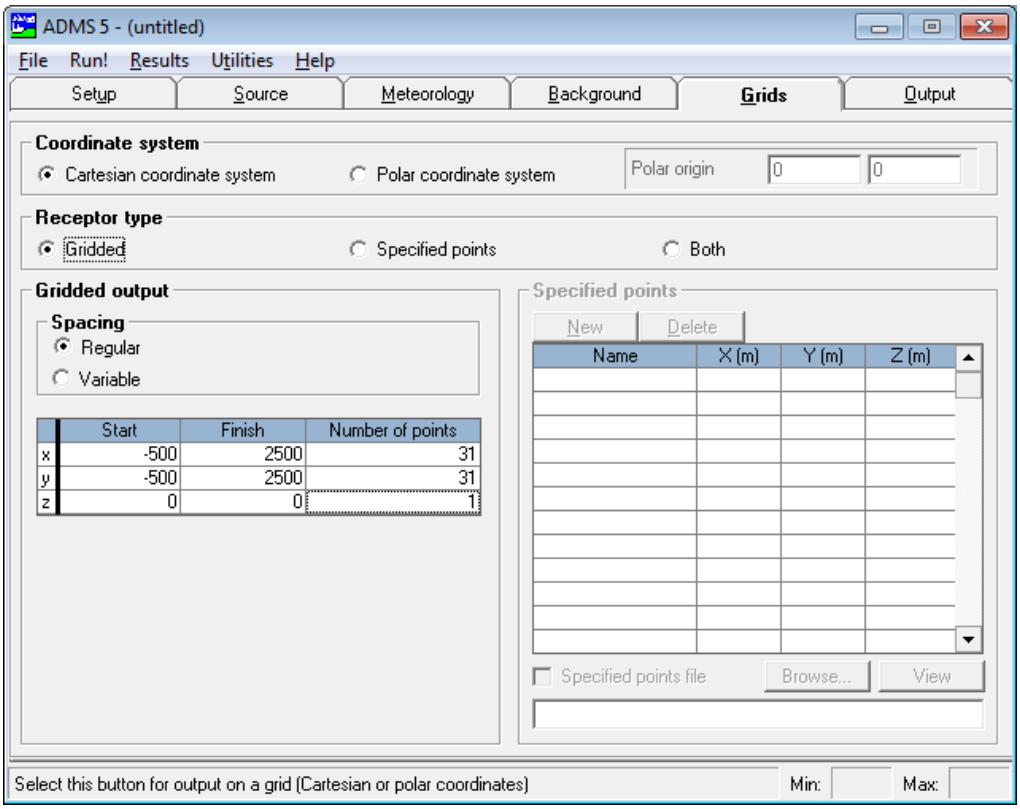


Figure 8.31 − Example 3: the **Grids** screen.

Step 11 Set up the grid as shown in **Figure 8.31**.

In this example, long-term averages will be used to give concentrations averaged over the whole 24 hour period. Results will be produced in a *.glt* output file. Go to the **Output** screen and define the output as follows.

Step 12 Cick **New** to add the default pollutant (NO_X) to the output table.

Step 13 Click **New** again and change the name of the pollutant by clicking on the drop-down list at the right of the cell and selecting SO_2.

Step 14 Set the output to long-term by either double-clicking with the mouse or typing **L** in the cells in the **Short/Long** column.

Step 15 Under **Group and source output**, select **Groups**. Remove the tick next to **All sources** and place a tick in the cells next to 'Group_Stack2' and 'Group_Stack3'. Concentrations will then be calculated separately for the two groups.

Compare with the **Output** screen shown in **Figure 8.32**.

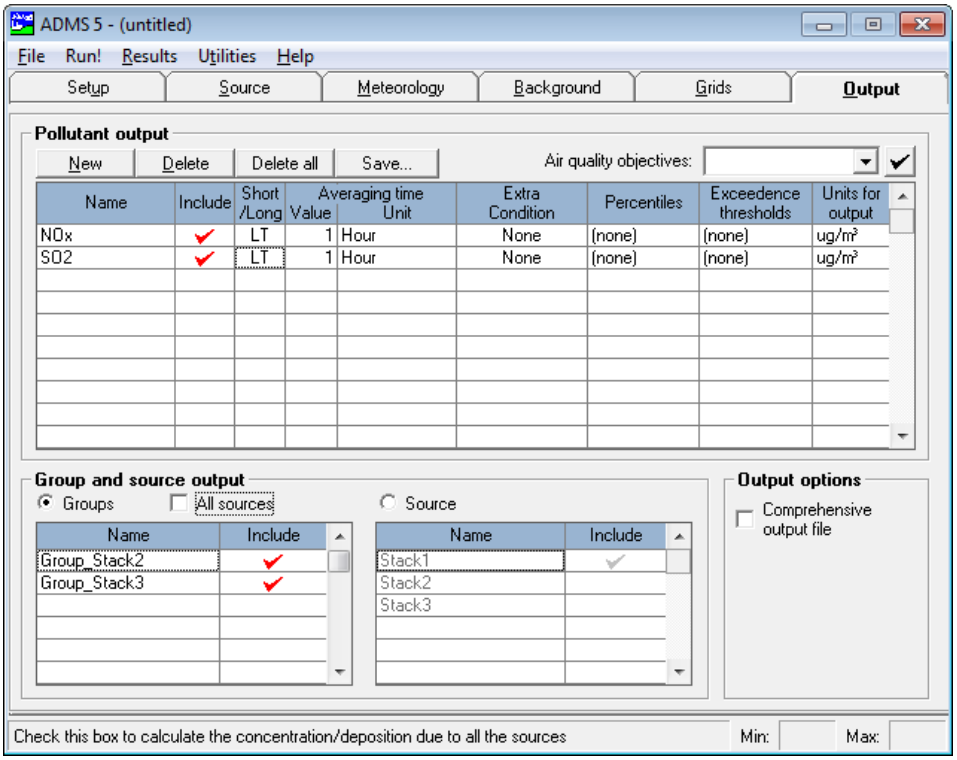


Figure 8.32 − Example 3: the **Output** screen.

Step 16 Click on **File, Save** and save the file as *Ex3_timevary.apl*.

Step 17 Click on **Run!** to run the model. This will take a few seconds.

8.3.2 Viewing output results

The calculation results have been written to a *.glt* file (gridded, long term) and can be visualised with Surfer.

Go to **Results**, **Plot in Surfer** and select **Contour** and **Long term**. The screen will be as shown in **Figure 8.33**. The window on the right-hand side lists the data sets available for plotting.

In turn, plot:

- the concentration of SO_2 for Group_Stack2 and Group_Stack3; and
- the concentration of NO_X for Group_Stack2.

Display the three plots on the screen at the same time.

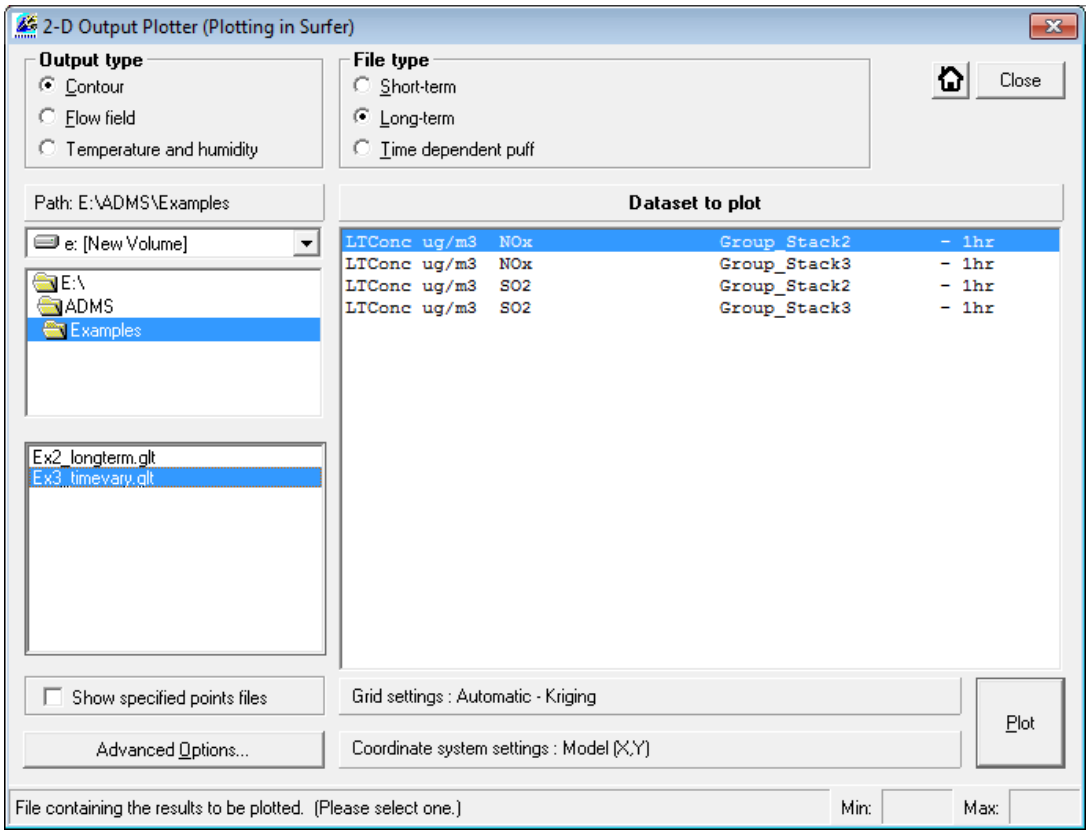


Figure 8.33 − Example 3: the **2-D Output Plotter** screen.

At the moment all the plots are shown on different scales for concentration. To display the results on the same scale, edit the contour levels in Surfer.

Step 1 Click on **Contours** in the **Object Manager** window in the first plot to display the properties in the **Property Manager** window. Click on the **Levels** tab and then click on **Simple** next to the **Level method** and use the drop down menu to change this to **Advanced**. Then click on **Edit Levels...** to access the screen shown in **Figure 8.34.**

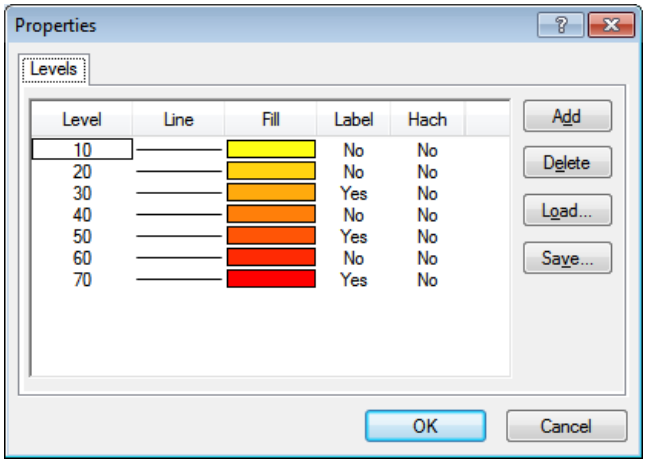


Figure 8.34 − Example 3: the Surfer **Levels Properties** screen.

Step 2 Click on the **Level** button at the top of the **Level** column, and change the minimum and maximum values and the interval to those shown in **Figure 8.35**.

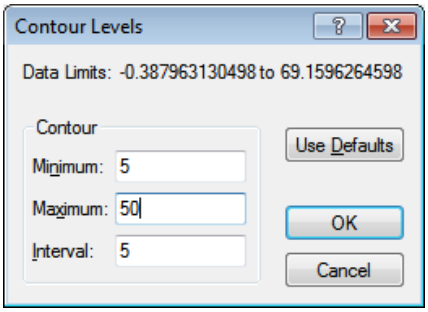


Figure 8.35 − Example 3: user-specified contour levels in Surfer.

This is likely to change the colour scheme to black/grey. To change it back to red/yellow click on the **Fill** button at the top of the **Fill** column in **Figure 8.34**, then click on the **Foreground Color** colour bar and select the **RedHot2** colour scheme from the **Presets** list and adjust the minimum and maximum values in the **Data to Color Mapping** box to those selected in **Figure 8.35**, as shown in **Figure 8.36**.

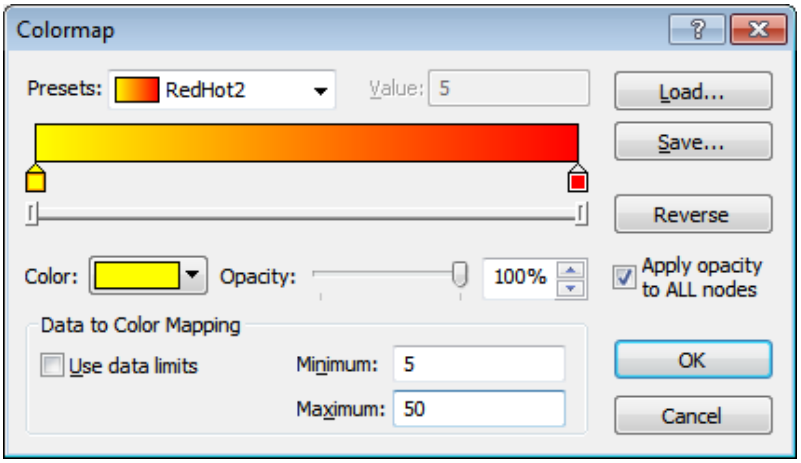


Figure 8.36 − Example 3: user-specified contour colours in Surfer.

Step 3 Return to the screen shown in **Figure 8.34** and click on the **Save**... button to save the level file, as *Ex3_timevary.lvl*.

Step 4 For each of the other two plots, click on **Contours** in the **Object Manager** window and then in the **Property Manager** window click on the **Levels** tab. Access the screen shown in **Figure 8.34** by selecting **Advanced** for the **Levels method** and then clicking on the **Edit Levels...** button. Click on the **Load**... button and load the *Ex3_timevary.lvl* file you have just created.

Additionally, you can modify the contour labels and other features of the plots. For more detail on this, please refer to Appendix D. Your results should now look similar to those shown in **Figure 8.37**.

The concentration contours are different in all three cases and the maximum concentration differs significantly for each plot, although the average release rate is the same. In this example, the maximum SO_2 concentration occurs when the higher release rates occur in the afternoon; the highest NO_X concentrations also occur during the afternoon and late night (when the wind is from the west), not during the early morning when the wind is from the south and the emission rate is highest. Notice also that night-time high emissions, when the wind is blowing from the south, are most

important for concentrations to the north of the source.

The time-varying emission results in a higher maximum daily average ground-level concentration of SO_2, but a lower maximum concentration of NO_X, although the amount emitted during the whole day was the same for both pollutants. This is because the highest emissions of SO_2 occurred during the convective conditions resulting in high ground-level concentrations from this elevated source. By contrast the highest NO_X emissions occurred under stable conditions leading to low ground-level concentrations.

Thus, it is clear that the rate at which a pollutant is released in a particular hour (i.e. under particular meteorological conditions) affects the concentration for that hour and therefore over the whole averaging period.

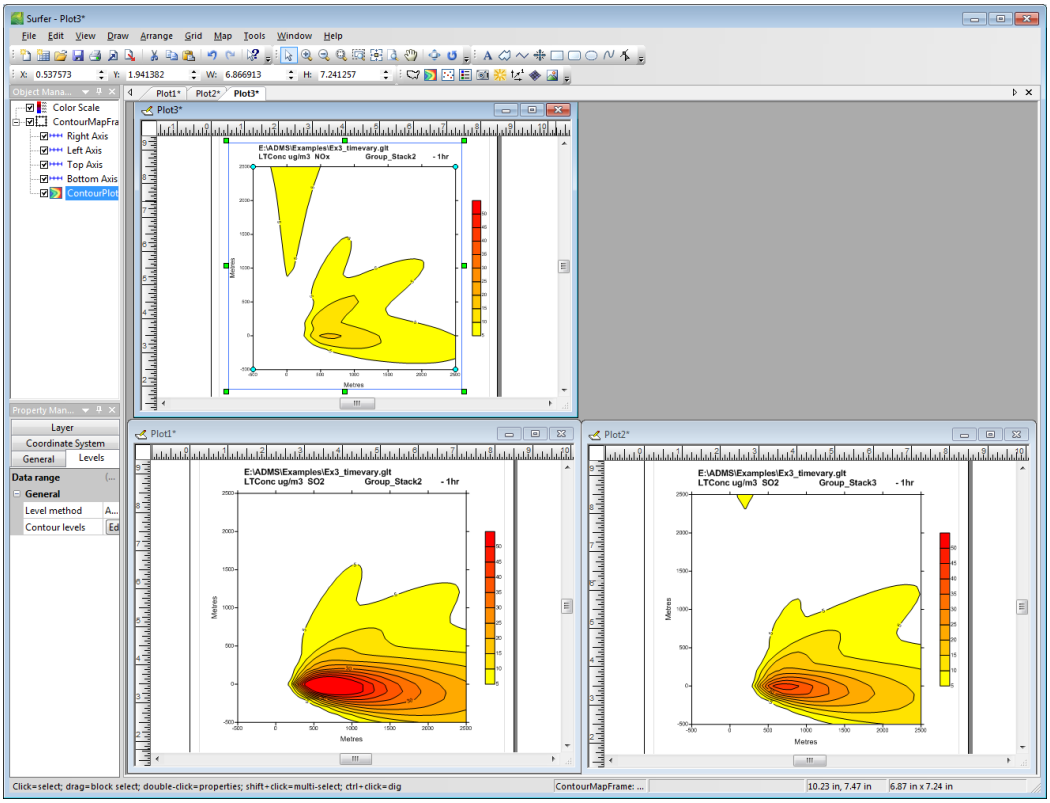


Figure 8.37 − Example 3: concentration contours for NO_X (Group_Stack2) and SO_2 (Group_Stack2 and Group_Stack3).

8.4 Example 4: Radioactive releases

The objectives of this example are: to calculate concentrations and deposition rates of radioactive emissions; to compare the results for isotopes of different half-lives in order to examine the effect of radioactive decay on model results; and to calculate gamma dose due to a plume.

The main pathways by which the public may be exposed to radiation as a result of atmospheric discharges are external exposure, inhalation, contact with contaminated materials or consumption of contaminated foods. Deposition of radioactive substances onto soil and vegetation can result in contamination of food products.

Limits on radioactive emissions are specified in terms of the radiation dose received by members of the public. Radiation dose is calculated from air concentrations and deposition to ground. The contribution to dose is different for different isotopes.

These worked examples show how to calculate concentrations of radioactive material in air and deposition to the ground for radioactive emissions (Section 8.4.1), and how to calculate gamma dose due to a plume (Section 8.4.2). Radiation doses are not calculated, although ADMS 5 output may be used in their calculation.

8.4.1 Example 4a: radioactive emissions and deposition

In this first example, two isotopes are considered in order to illustrate the effects of radioactive decay. One has a long half-life and the other a short half-life.

Setting up the run

The two isotopes emitted are Polonium-210 (Po-210), which has a half-life of 1.196e+7 s (= a few months), and Caesium-126 (Cs-126), which has a half-life of 98.4 s (= a couple of minutes).

The isotopes are released as particulates at a rate of 500 Bq/s from a 100 m high stack with a diameter of 5 m. The release temperature is high (150°C) and the exit velocity is moderate (15 m/s). Results are presented for activity of radioactive material in air and the dry, wet and total deposition rates.

Step 1 Open ADMS 5 and click on **File**, **New** to create a new input file, i.e. a default input file or "clean sheet" in which to enter data.

Step 2 Enter suitable **Name of site** and **Name of project** titles for the run.

Step 3 In the **Palette** drop down select **Isotopes**.

Step 4 Click on the **Data**... button to go to the **Palette of Radioactive Pollutants (Isotopes)** screen shown in **Figure 8.38**.

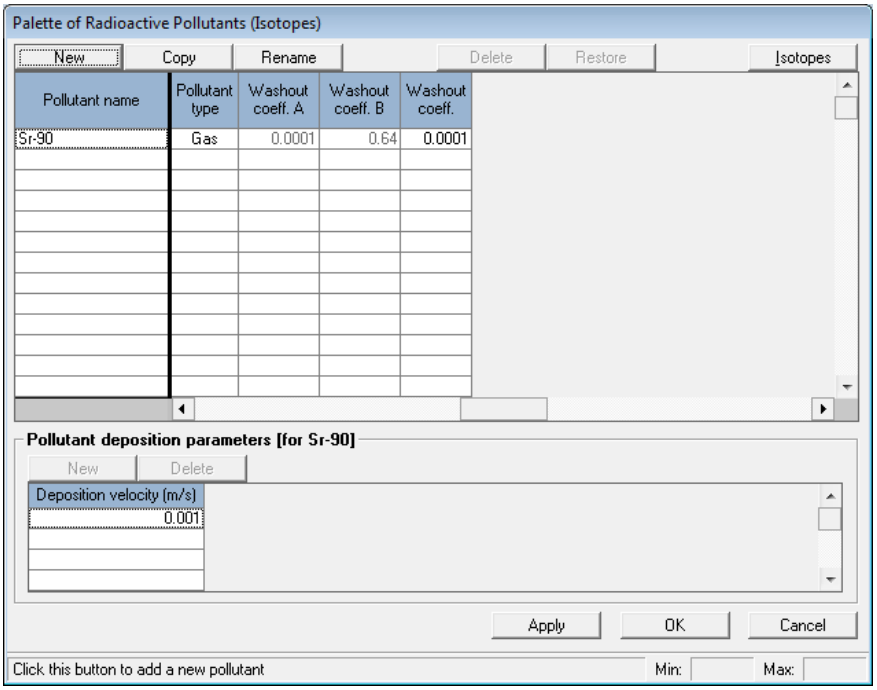


Figure 8.38 − Example 4a: the initial **Palette of Radioactive Pollutants (Isotopes)** screen

Initially, the **Palette of Radioactive Pollutants (Isotopes)** contains only the default isotope of Sr-90. Isotopes may be added to the palette either by selecting an isotope from a drop-down list or by editing the isotope name by hand. A list of all possible isotopes is contained in the file *<install_path>\Isotopes.pdf*, which may be viewed by clicking **Isotopes** in the **Palette of Radioactive Pollutants (Isotopes)**. See Section 4.4 for further details and **Figure 8.39** for an extract of the *Isotopes.pdf* file.

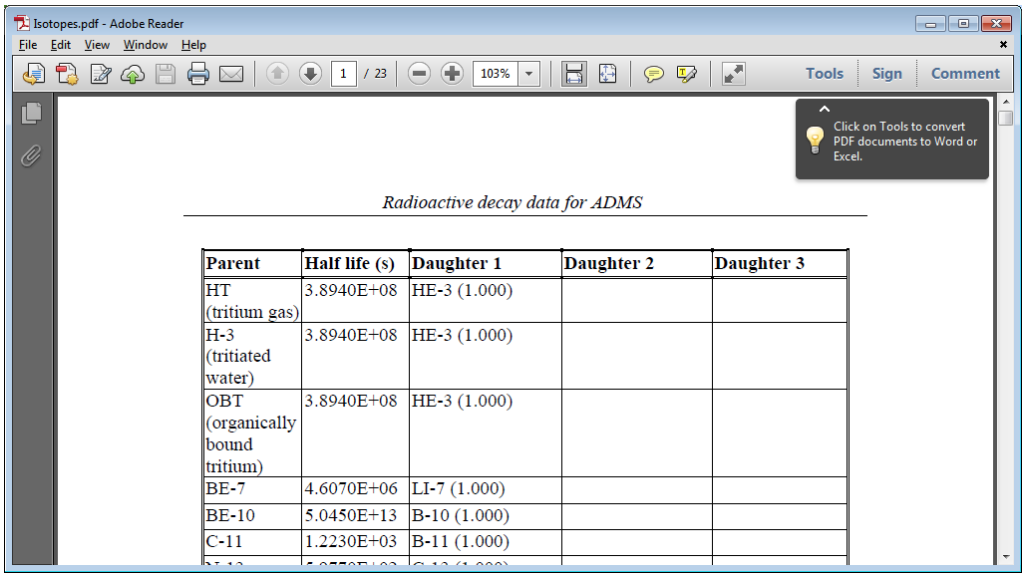


Figure 8.39 − Example 4a: extract of the document *Isotopes.pdf* shown in Acrobat Reader.

Step 5 Click on the cell containing the name Sr-90 and then on **Rename**. Select Po-210 from the drop-down list and click **OK**.

This drop-down list contains isotopes typically emitted in the UK and is much shorter than that in the file *Isotopes.pdf*. Typing one letter moves to a different part of the list (isotopes beginning with that letter).

Step 6 Add a second isotope by clicking **New** and *type* Cs-126 as the isotope name, and then click **OK**.

Although Cs-126 does not appear on the short drop-down list, its half-life and daughters are given in the file Isotopes.pdf*, so it may be modelled by ADMS 5.*

Step 7 Select the pollutant type as **Particle** for both isotopes.

Step 8 Drag the scroll bar at the bottom of the table to the far left to display the first few columns in the table. Select **No** for the first three columns to indicate that the deposition velocity, terminal velocity and washout coefficients are not known. The model will then use default values for these parameters (refer to Sections 4.2 and 4.3 for details of the other options available for deposition parameters).

Compare with the screen shown in **Figure 8.40**.

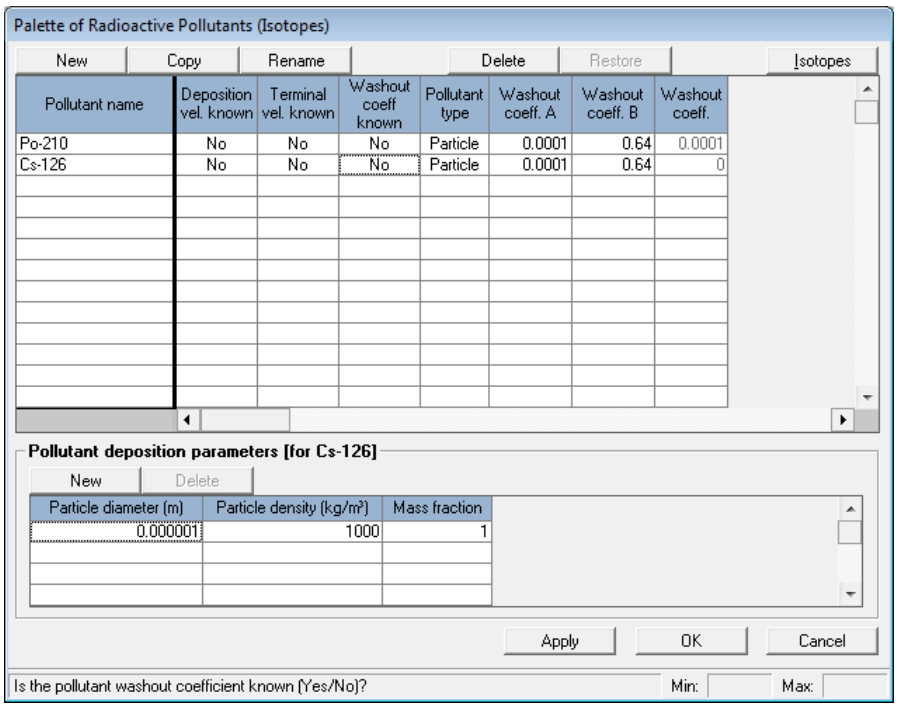


Figure 8.40 − Example 4a: the **Palette of Radioactive Pollutants (Isotopes)** screen.

Once all the required isotopes have been entered into the palette, click **OK** and return to the **Setup** screen.

Step 9 Select **Dry deposition** and **Wet deposition** from the model options and **Radioactive decay** from the Radioactivity options.

Compare with the screen shown in **Figure 8.41**.

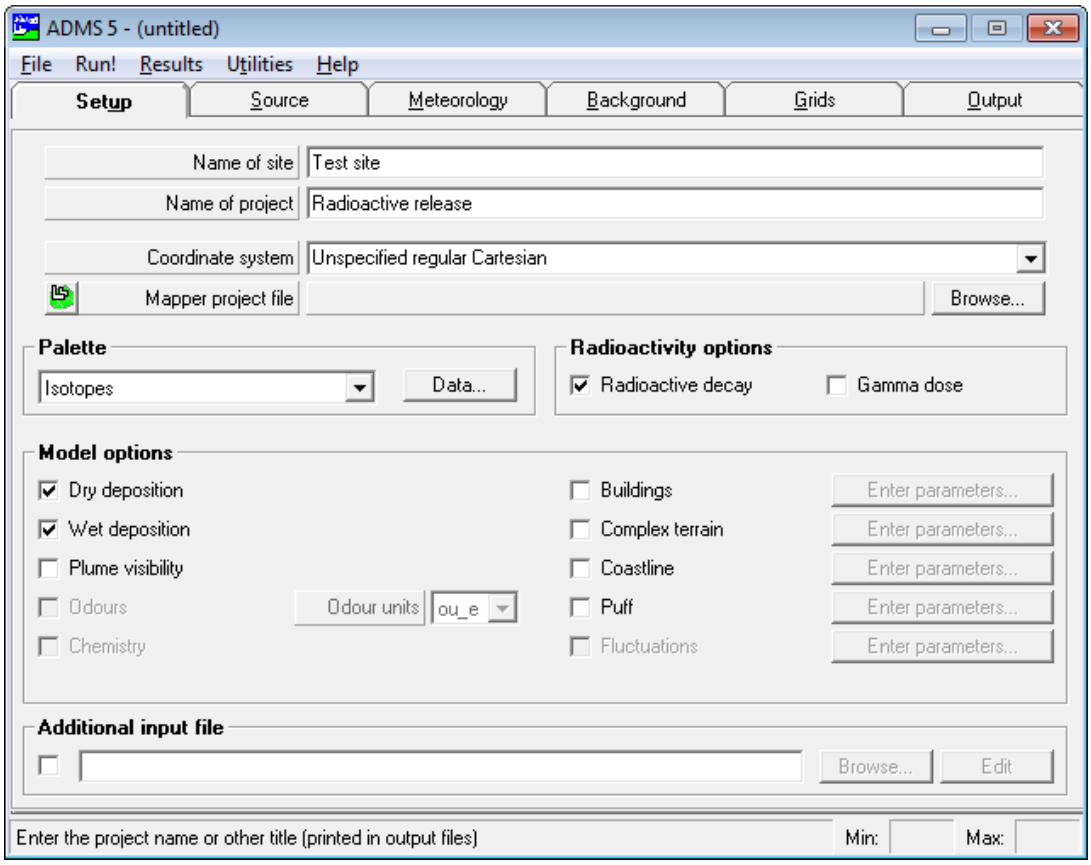


Figure 8.41 − Example 4a: the **Setup** screen.

Step 10 Go to the **Source** screen, click **New** to add a new source to the table and enter the details for the source as detailed at the start of Section 8.4.1.

Compare with the screen shown in **Figure 8.42**.

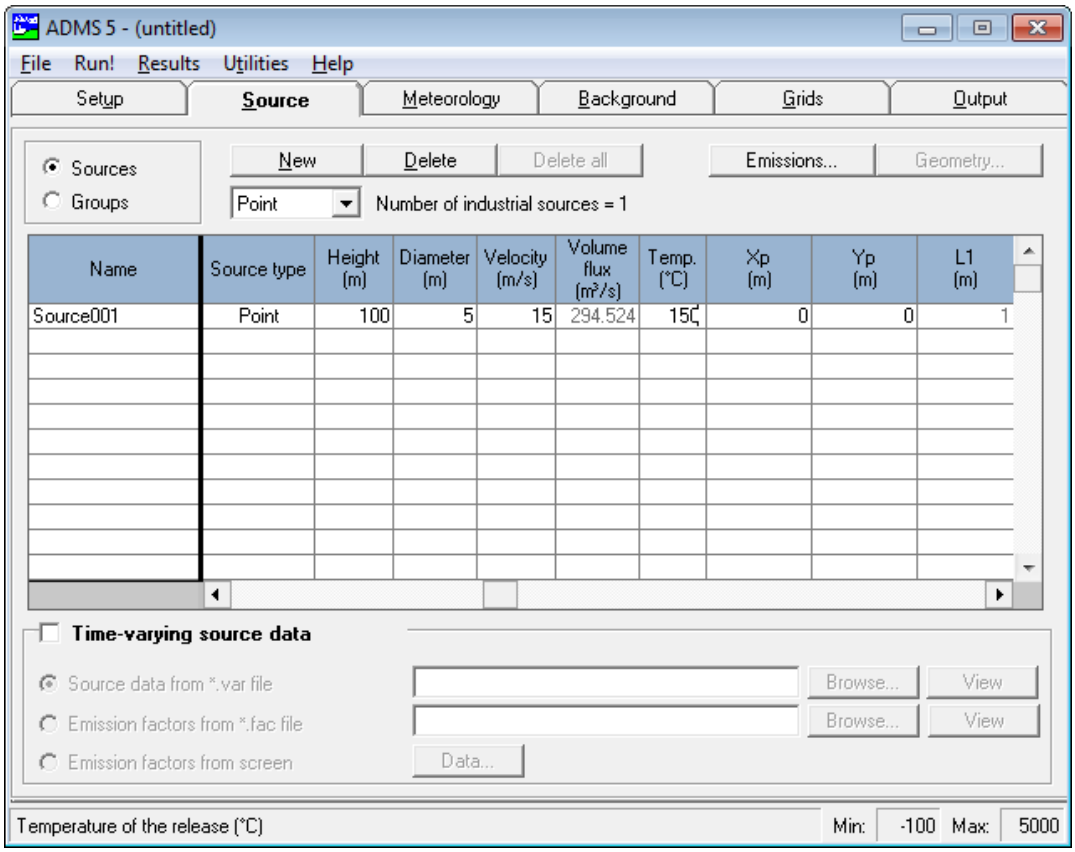


Figure 8.42 − Example 4a: the **Source** screen.

Step 11 Click on the **Emissions**... button.. A default emission rate for Po-210 will have been added to the list of emissions.

Step 12 Alter the emission rate for Po-210 from its default value of 1 Bq/s to 500 Bq/s.

Step 13 Click **New** to add Cs-126 to the table and set the emission rate to 500 Bq/s.

Compare with the screen shown in **Figure 8.43**.

---

*For an isotope to be emitted (i.e. entered in the **Radioactive Emissions (Isotopes)** screen), it must appear in the **Palette of Radioactive Pollutants (Isotopes)**. However, isotopes may appear in the palette even if they are not emitted. Thus, if emissions usually involve some common set of isotopes, these isotopes and their deposition characteristics may be entered into the **Palette of Radioactive Pollutants (Isotopes)** and saved as, say,* base.apl. *Then each time a new scenario is modelled,* base.apl *may be opened, edited and saved under a different name, say* problem.apl. *In this way, the isotopes and their deposition parameters need only be defined once.*

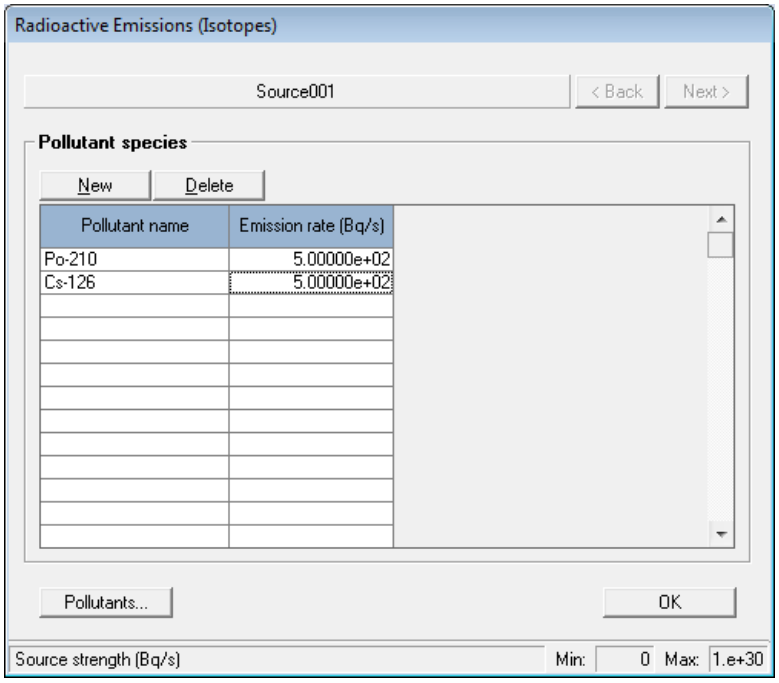


Figure 8.43 − Example 4a: the **Radioactive Emissions (Isotopes)** screen.

Step 14 Go to the **Meteorology** screen and change the surface roughness length to **Agricultural areas (max)**, corresponding to 0.3 m.

Step 15 Set the meteorological data to be **From file** and use the **Browse**... button to choose the file *<install_path>\Data\radDEMO.MET* for this run.

The file *radDEMO.MET* is provided with ADMS 5 and shown in **Figure 8.44**. It contains a single line of meteorological data.

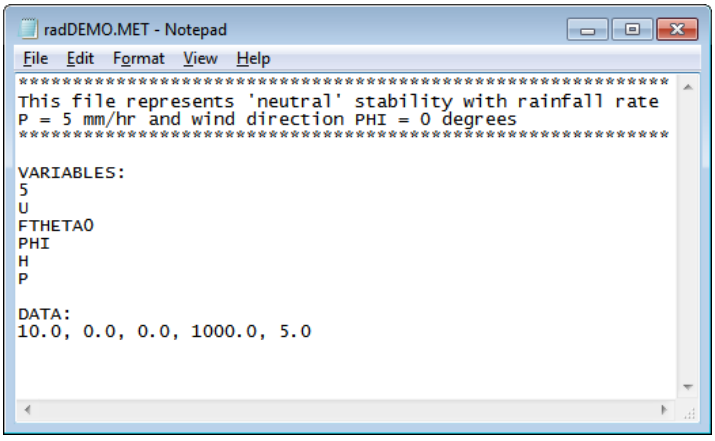


Figure 8.44 − Example 4a: the *.met* file *radDEMO.MET*.

The first few lines contain comments describing the contents of the file, and are optional. The remainder of the file must be in the format shown, although the actual variables used may vary. The file shows that the value of wind speed (`U`) is 10 m/s, surface heat flux (`FTHETA0`) is 0 W/m^2, the wind is blowing from the north (`PHI` = 0), the boundary-layer depth (`H`) is 1000 m and the precipitation rate (`P`) is 5 mm/h.

The file is used for demonstration purposes only. It should not be used for real model calculations. Usually data representing a longer period such as a year would be used.

Background levels cannot be used with the radioactive decay option so we now move on to defining the output grid on the **Grids** screen. An appropriate grid for this problem is shown in **Figure 8.45**, extending south 10000 m from the source and 2000 m to the east and west. (In general, the wind could be blowing from any direction, so the grid would tend to be set up so that the source was at the centre of the grid.)

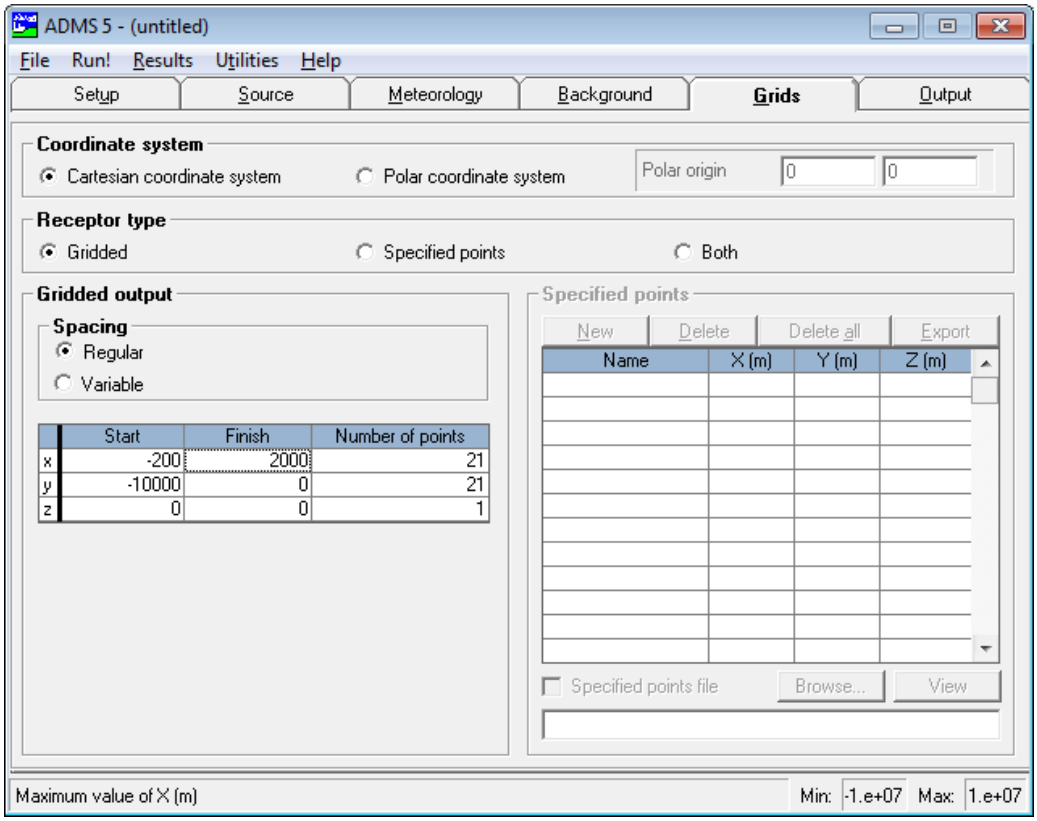


Figure 8.45 − Example 4a: the **Grids** screen.

Step 16 Go to the **Grids** screen and set up a grid as shown in **Figure 8.45**.

Step 17 Go to the **Output** screen and click **New** to add the default isotope to the output table. The default isotope is the first one in the **Palette of Radioactive Pollutants (Isotopes)**, in this case Po-210.

Step 18 Click **New** again and change the pollutant name by clicking on the drop-down list at the right of the cell and selecting Cs-126.

In this example, the default values will be used for all other output parameters. Results will be written to the *.gst* output file for short-term hourly average calculations for a single source. Compare with the screen shown in **Figure 8.46**.

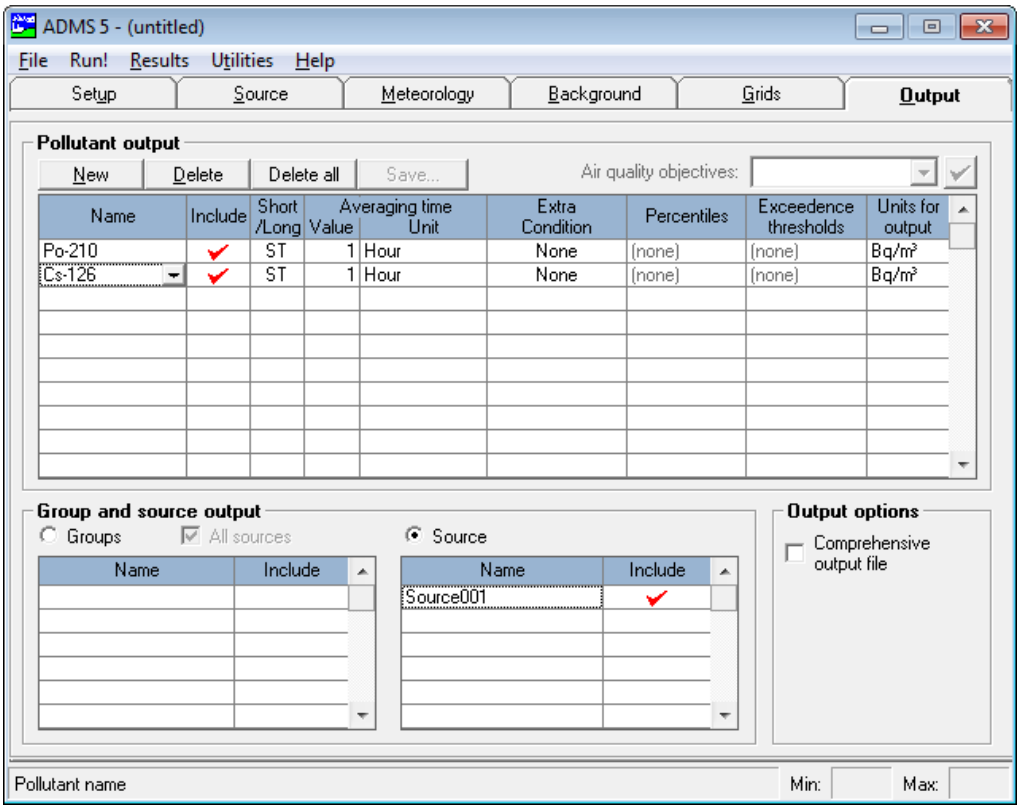


Figure 8.46 − The **Output** screen for Example 4.

Step 19 Click on **File**, **Save** and save the file as *Ex4a_isotopes.apl*.

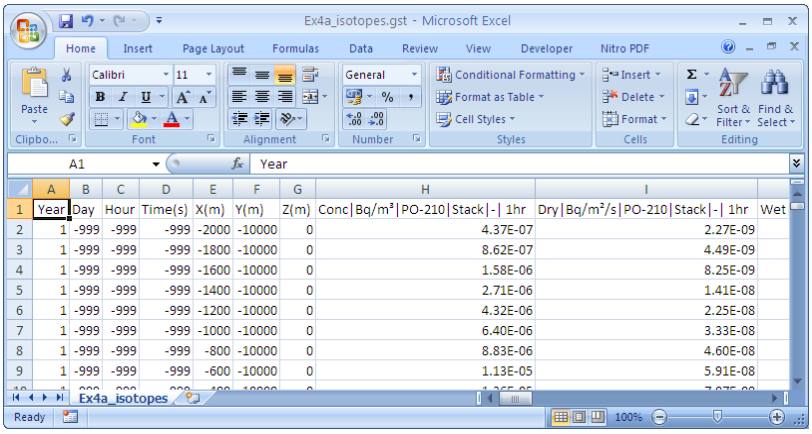
Step 20 Click on **Run!** to run the model. This will only take a few seconds.

Viewing output results

Numerical

The results can be viewed numerically. Select the **Results**, **Numerical**... menu command at the top of the interface. This will open a dialogue box, select *Ex4a_isotopes.gst* and click **Open**. There are 23 columns of data, the first few of which are shown in **Figure 8.47**. To view the additional columns, use the scroll bar at the bottom of the screen. The first four columns contain the year, day, hour and time. In this example these are shown as their default values (1 for the year and -999 for the others) as their values were not specified in the *.met* file (Step 15). The next three columns contain the X, Y and Z coordinates of each output grid point. The remaining columns contain the activity of radioactive material, dry, wet and total deposition for each isotope originally released and all daughter isotopes up to a maximum of 50 parent plus daughter isotopes. Decay chains are modelled until a stable isotope is reached (i.e. until a daughter is produced which has zero activity).

In this example, notice that, as well as the columns of data for the two isotopes released (Po-210 in columns 8 to 11 and Cs-126 in columns 16 to 19), there are columns for two additional isotopes, namely Pb-206 and Xe-126. These are the daughter isotopes of Po-210 and Cs-126, respectively. In this case, the activity of radioactive material and deposition rates of the daughters are zero at all points because Pb-206 and Xe-126 are both stable isotopes.



Year	Day	Hour	Time(s)	X(m)	Y(m)	Z(m)	Conc|Bq/m³|PO-210|Stack|-| 1hr	Dry|Bq/m²/s|PO-210|Stack|-| 1hr	Wet
1	-999	-999	-999	-2000	-10000	0	4.37E-07	2.27E-09	
1	-999	-999	-999	-1800	-10000	0	8.62E-07	4.49E-09	
1	-999	-999	-999	-1600	-10000	0	1.58E-06	8.25E-09	
1	-999	-999	-999	-1400	-10000	0	2.71E-06	1.41E-08	
1	-999	-999	-999	-1200	-10000	0	4.32E-06	2.25E-08	
1	-999	-999	-999	-1000	-10000	0	6.40E-06	3.33E-08	
1	-999	-999	-999	-800	-10000	0	8.83E-06	4.60E-08	
1	-999	-999	-999	-600	-10000	0	1.13E-05	5.91E-08	

Figure 8.47 − Example 4a: the *Ex4a_isotopes.gst* file viewed in Microsoft Excel.

ADMS Line Plotter

The results can also be viewed using the **ADMS Line Plotter** utility (see **Figure 8.48**). Select **Results, Line Plot** from the menu at the top of the interface in order to open the utility.

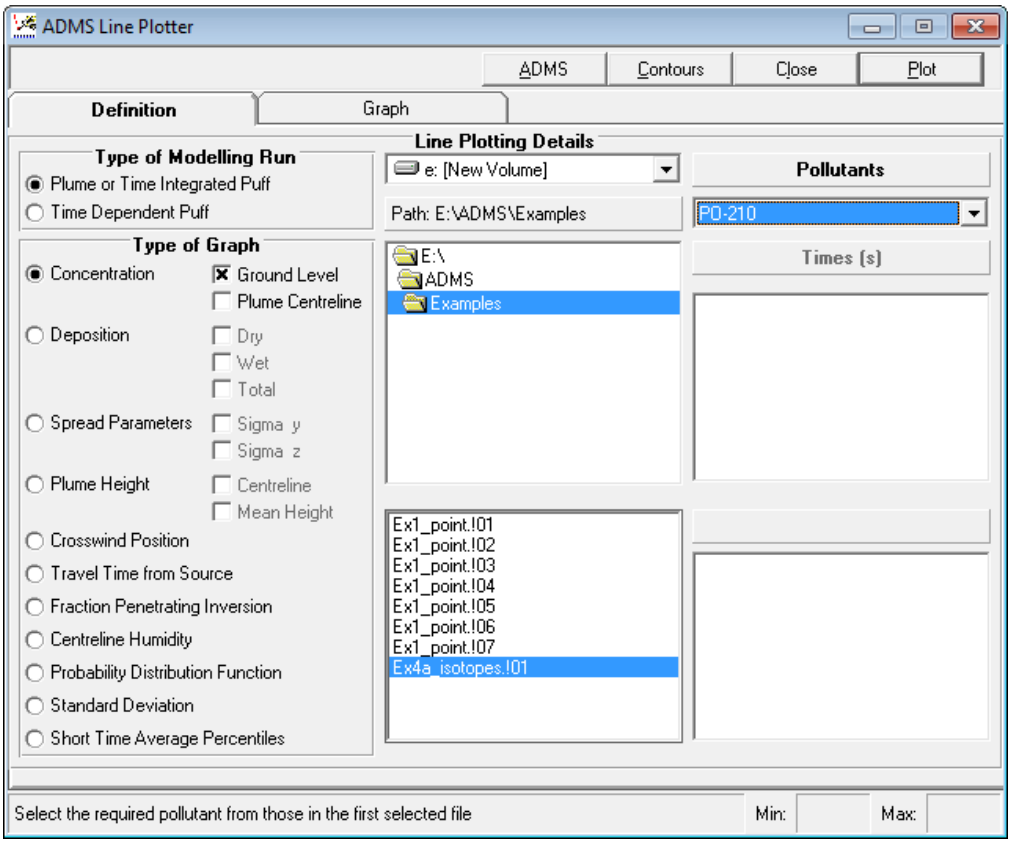


Figure 8.48 − Example 4a: the **ADMS Line Plotter** screen.

Select ground-level concentrations for Po-210 (select from the **Pollutants** list) and then **Plot**. The graph is shown in **Figure 8.49**.

Click on the **Definition** tab to return to the screen in **Figure 8.48**. Select dry, wet and total deposition for Po-210 and click on **Plot**. The results are shown in **Figure 8.50**.

Repeat for Cs-126, with results as shown in **Figures 8.51** and **8.52**.

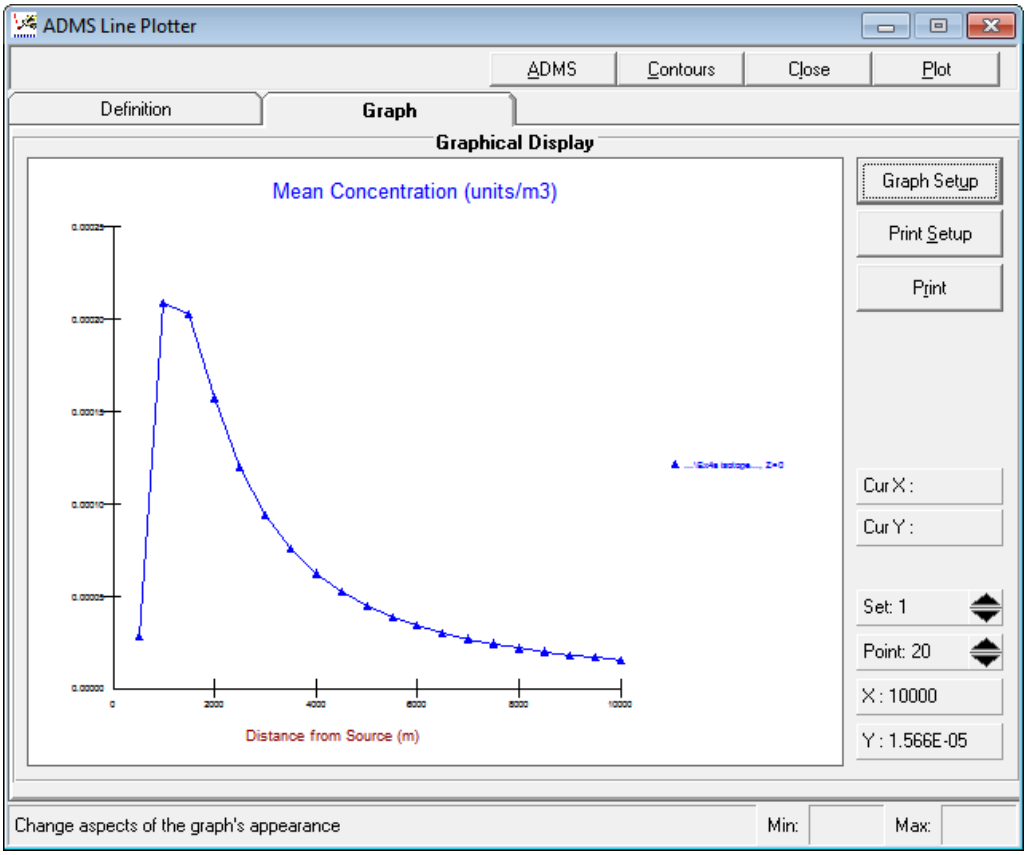


Figure 8.49 − Example 4a: the ground-level concentration for Po-210.

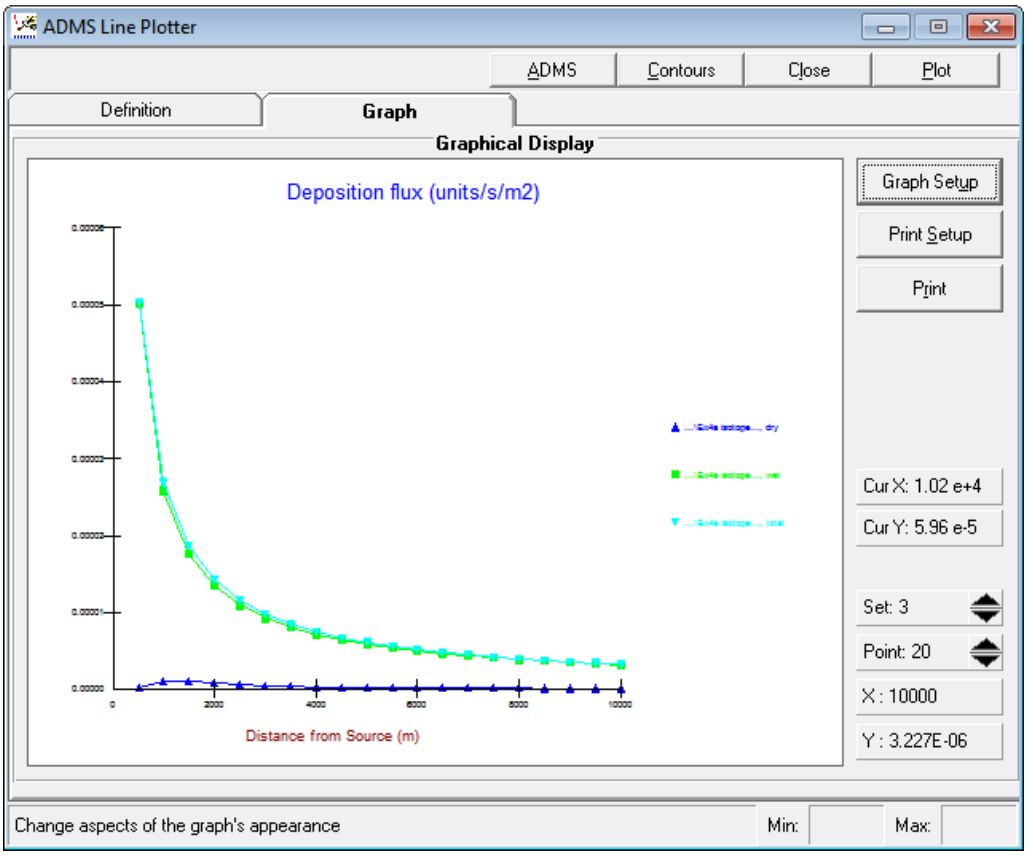


Figure 8.50 − Example 4a: the dry, wet and total deposition fluxes for Po-210.

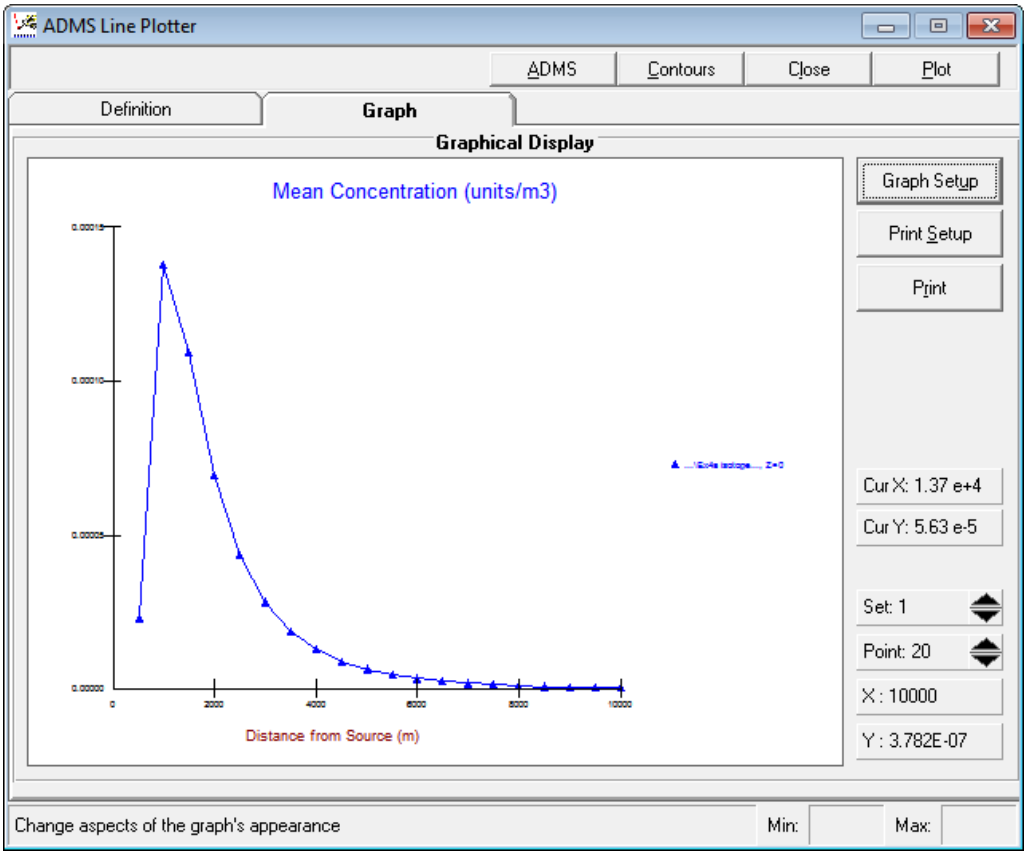


Figure 8.51 − Example 4a: the ground-level concentration for Cs-126.

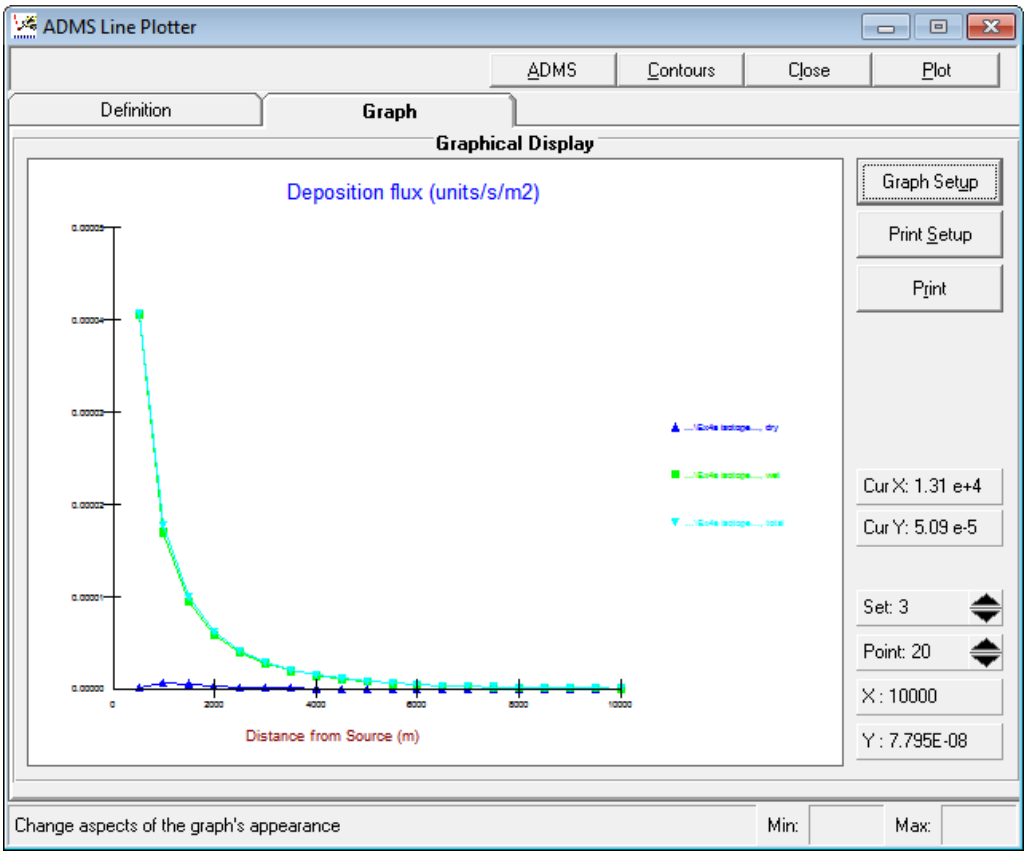


Figure 8.52 − Example 4a: the dry, wet and total deposition fluxes for Cs-126.

ADMS Contour Plotter

Select the **Results, Plot in Surfer** command from the menu at the top of the interface. This opens the **2-D Output Plotter** utility. Select **Concentration** and **Short term** as shown in **Figure 8.53.**

The window at the top right hand side lists the datasets available for plotting. The list contains the activity (**Conc**) of radioactive material, dry, wet and total deposition for the emitted isotopes (Po-210 and Cs-126) and their daughters (Pb-206 and Xe-126).

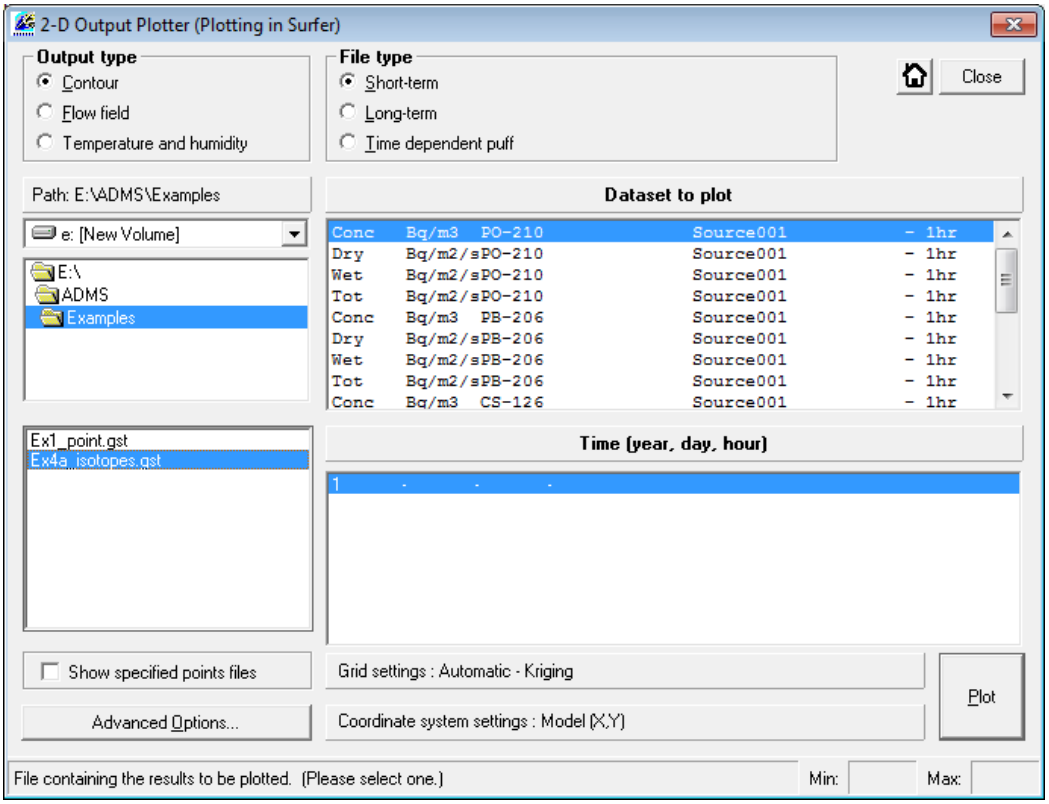


Figure 8.53 − Example 4a: the **2-D Output Plotter** screen.

Plot the activity of Po-210 (Bq/m^3) followed by the total deposition and dry and wet deposition for Po-210 (Bq/m^2/s). The four datasets should be plotted in turn, taking care that a time has been selected.

Use **Window, Tile Horizontal** in Surfer to display the four plots on the screen at the same time.

To display the wet and total deposition and activity plots on the same scale, edit the contour levels in Surfer, as follows.

Step 1 In the plot for the total deposition click on **Contours** in the **Object Manager** window and then on the **Levels** tab in the **Property Manager** window. Select **Advanced** for the **Levels method** and click on **Edit Levels...** to bring up the screen shown in **Figure 8.54**.

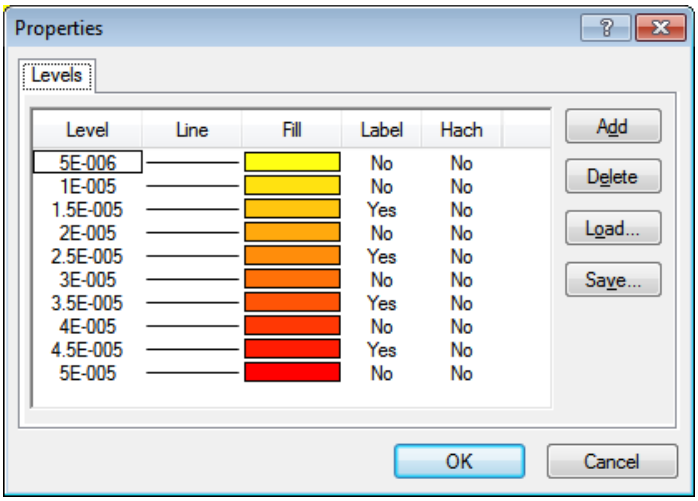


Figure 8.54 − Example 4a: the **ContourPlot Properties** screen in Surfer.

Step 2 Click on the **Level** button at the top of the **Level** column. Change the minimum and maximum values and the interval to those shown in **Figure 8.55**.

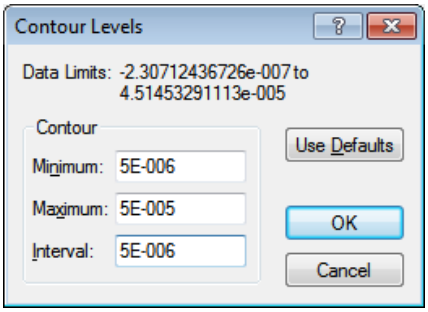


Figure 8.55 − Example 4a: user-specified contour levels in Surfer.

This is likely to change the colour scheme to black/grey. To change it back to red/yellow, click on the **Fill** button at the top of the **Fill** column in **Figure 8.54**. Then click on the **Foreground Color** colour bar. Select the **RedHot2** colour scheme from the **Presets** list and adjust the minimum and maximum values in the **Data to Color Mapping** box to those shown in **Figure 8.56**.

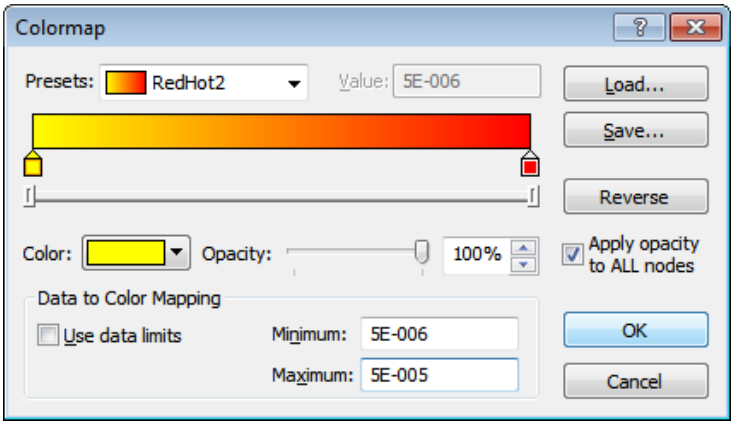


Figure 8.56 − Example 4a: user-specified contour colours in Surfer.

Step 3 Return to the screen shown in **Figure 8.54**. Click on the **Save**... button to save the level file as *Ex4a_isotopes.lvl*.

Step 4 For the wet deposition and activity contour plots, double-click on the contours to access the contour plot properties as shown in **Figure 8.54**, click on the **Load**... button and load the *Ex4a_isotopes.lvl* file you have just created.

The dry deposition contour plot should not be plotted on this scale because the dry deposition rate is much lower.

For more details on editing Surfer plots, see Appendix D.

Repeat for the Cs-126 concentration and deposition rates. Your results should now look similar to those shown in **Figures 8.57** and **8.58**.

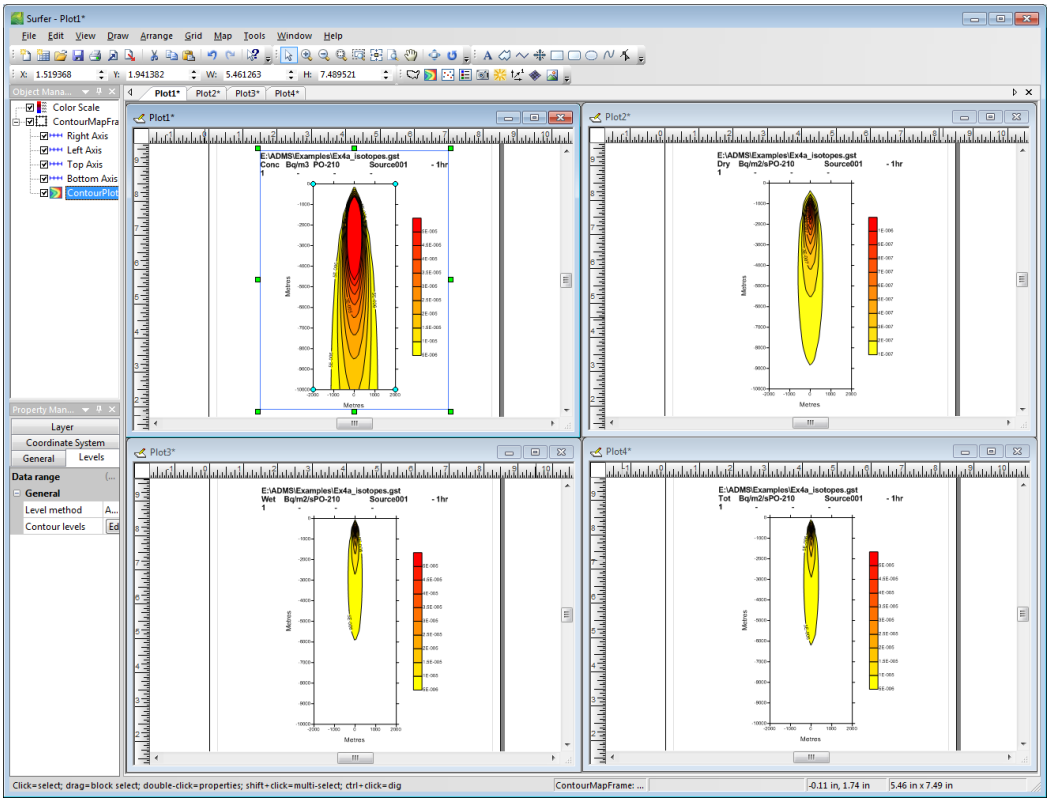


Figure 8.57 − Example 4a: contour plots of the concentration (activity), total, dry and wet deposition rate for Po-210.

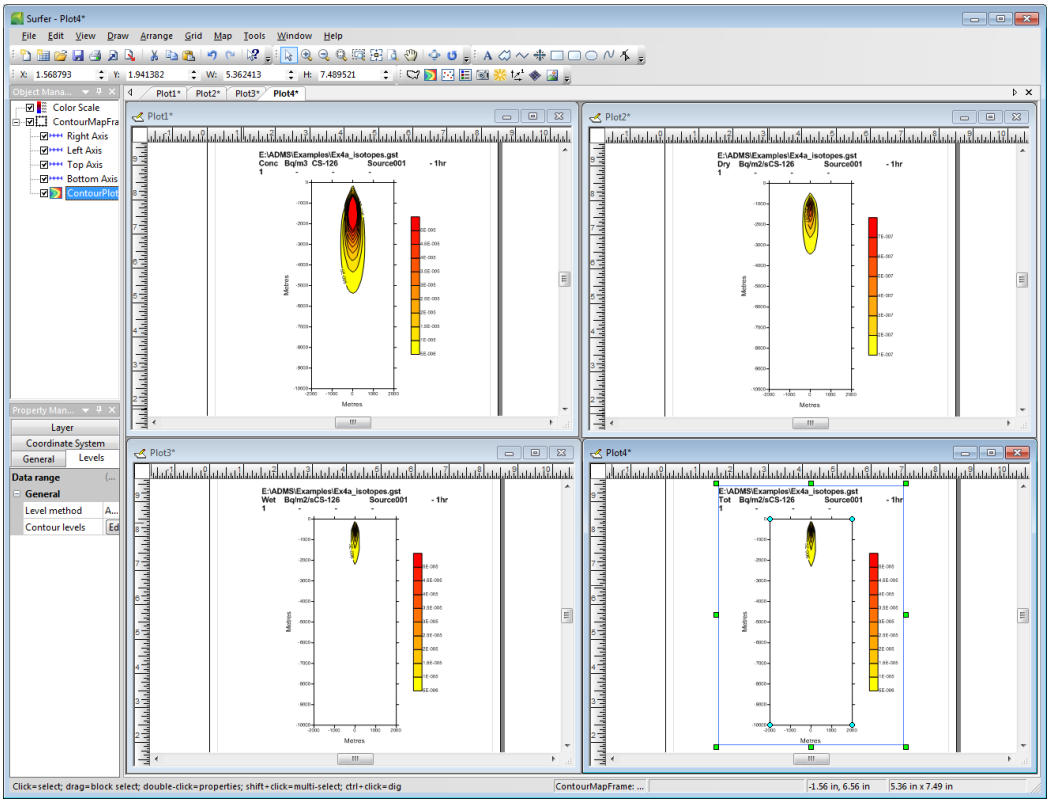


Figure 8.58 − Example 4a: contour plots of the concentration (activity), total, dry and wet deposition rate for Cs-126.

Interpretation

Activity and total deposition

Po-210 has a relatively long half-life (a few months), so there is no significant radioactive decay within the first few kilometres of the stack. Air concentrations of activity and deposition rates have the same values as they would for a non-radioactive release of 500 g/s but the units are different. For radioactive emissions, the release rate is given in Bq/s, and the resulting air concentration of activity is in Bq/m^3, while the deposition to the ground is in $Bq/m^2/s$ (for non-radioactive emissions the units are g/s, g/m^3 and $g/m^2/s$, respectively).

Notice from **Figure 8.57** that the activity contour for 1.0E-5 Bq/m^3 extends past 10,000 m downwind and that the total deposition contour for 1.0E-5 $Bq/m^2/s$ extends to about 2,800 m downwind.

Now look at the results for Cs-126. This has a much shorter half-life (a couple of minutes). There will therefore be significant radioactive decay within a few km of the stack, so the concentration and deposition are much smaller than for Po-210. **Figure 8.58** shows that the activity contour for 1.0e-5 Bq/m^3 of Cs-126 extends to approximately 4,300 m downwind, while the total deposition contour for 1.0e-5 $Bq/m^2/s$ of Cs-126 extends approximately 1,200 m downwind.

Dry and wet deposition

The shapes of the contours for dry and wet deposition are rather different (**Figures 8.57** and **8.58**). The pattern for dry deposition is a similar shape to that of the

ground-level concentration (although the numbers are much smaller), and there is no deposition very close to the source as dry deposition occurs due to contact of the plume with the ground.

Wet deposition occurs due to the contaminant being washed out of the plume. Hence, there is deposition to ground at all points *below* the plume, even if the plume is not touching the ground. Wet deposition therefore gives an indication of the total quantity of pollutant passing over each point on the ground. The highest wet deposition rates occur close to the source, where concentrations within the plume are relatively high as it has not yet dispersed much laterally. Wet deposition rates fall off rapidly with distance, as the plume spreads horizontally and concentrations within the plume decrease.

8.4.2 Example 4b: radioactive emissions and gamma dose

This example calculates the dose of gamma radiation received from a plume containing Sr-91.

Setting up the run

Step 1 Open ADMS 5 and open the *.apl* file created in the previous example by selecting **File**, **Open**... from the menu bar and choosing the file *Ex4a_isotopes.apl*.

Step 2 Save the file as *Ex4b_gammadose.apl* using the **File**, **Save As**... menu command.

Step 3 Amend the project name.

Step 4 On the **Setup** screen check the box marked **Gamma dose** next to the **Radioactive decay** model option and uncheck the **Dry deposition** and **Wet deposition** options.

Compare with the screen shown in **Figure 8.59**.

Step 5 Go to the **Source** screen. Click on the **Emissions**... button and then the **Pollutants**... button. Add the radioactive isotope Sr-91 to the palette of radioactive pollutants, leaving this isotope as a gas and its deposition parameters at their default values.

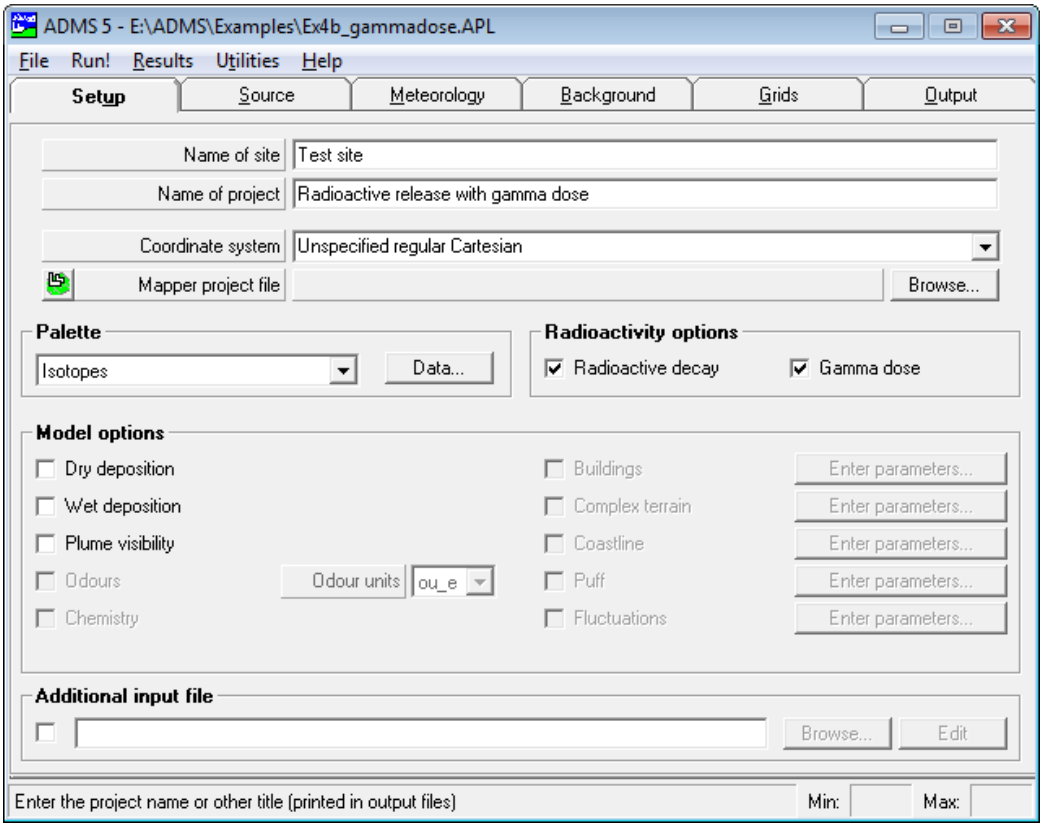


Figure 8.59 − Example 4b: the **Setup** screen.

Step 6 Return to the **Radioactive Emissions (Isotopes)** screen, add Sr-91 as a new pollutant species for the source, with an emission rate of 500 Bq/s.

Compare with the screen shown in **Figure 8.60**.

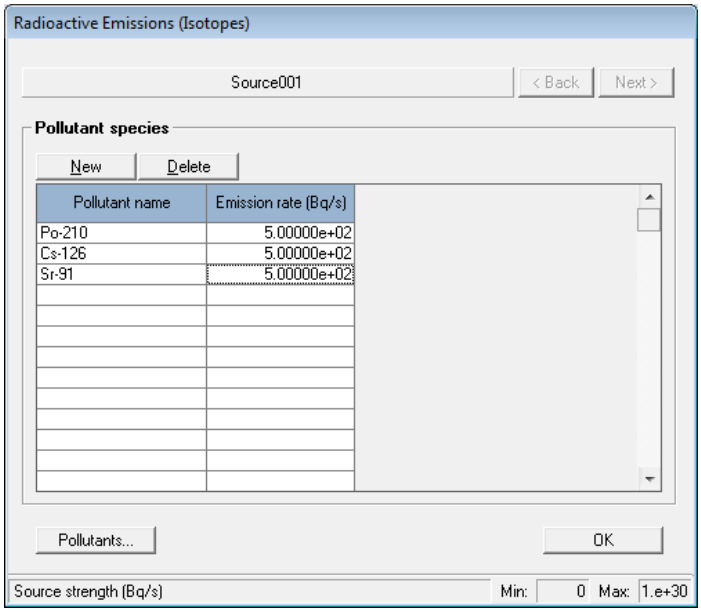


Figure 8.60 − Example 4b: the **Radioactive Emissions (Isotopes)** screen.

Go to the **Grids** screen. The gamma dose calculations take significantly longer than other concentration calculations. Therefore, the grid specifications will be modified.

Step 7 Decrease the **Number of points** on the **Gridded output** table to 7 in the X direction and 9 in the Y direction.

Step 8 Change the **Receptor type** to **Both** and add 5 points to the **Specified points** table as shown in **Figure 8.61.**

Gamma dose will be calculated at *all* the ground level output points entered on this screen. Compare with the screen shown in **Figure 8.61**.

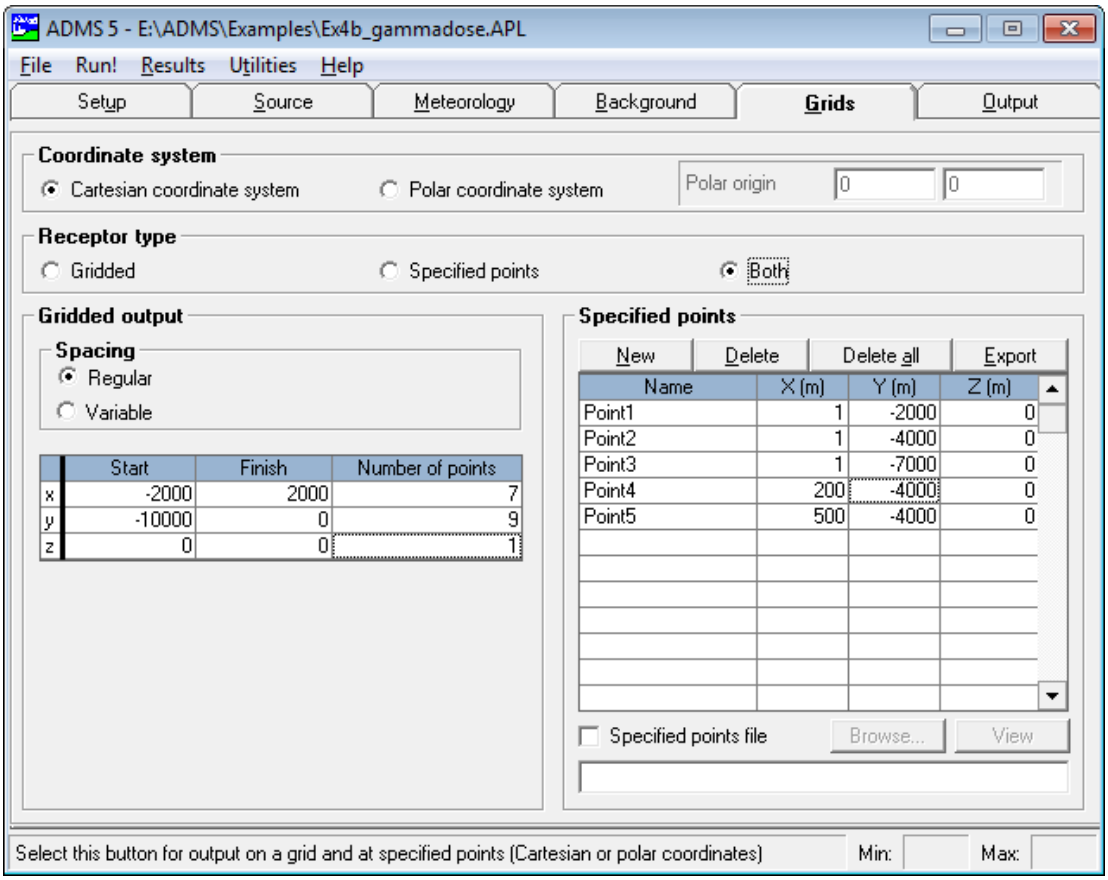


Figure 8.61 − Example 4b: the **Grids** screen.

Step 9 Go to the **Output** screen. In the **Pollutant output** table, double-click or type **N** in the **Include** box for Po-210 to erase the tick from that box; repeat for Cs-126.

Step 10 Add a new line for Sr-91 using **New**.

Compare with the screen shown in **Figure 8.62**.

Step 11 Click on **File, Save**.

Step 12 Click on **Run!** to run the model. This will take a little longer than the first example but will still only take about a minute.

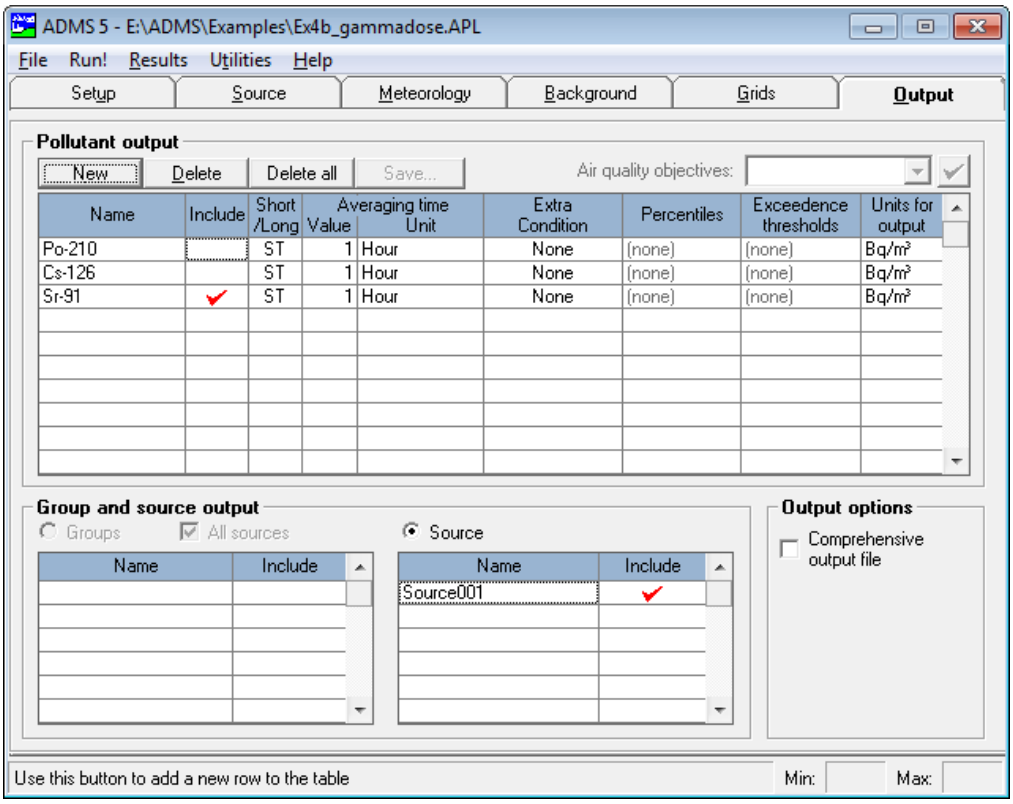


Figure 8.62 − Example 4b: the **Output** screen.

Viewing output results

From the menu bar, go to **Results**, **Plot in Surfer** and select the **Contour** and **Short term** options as displayed in **Figure 8.63**.

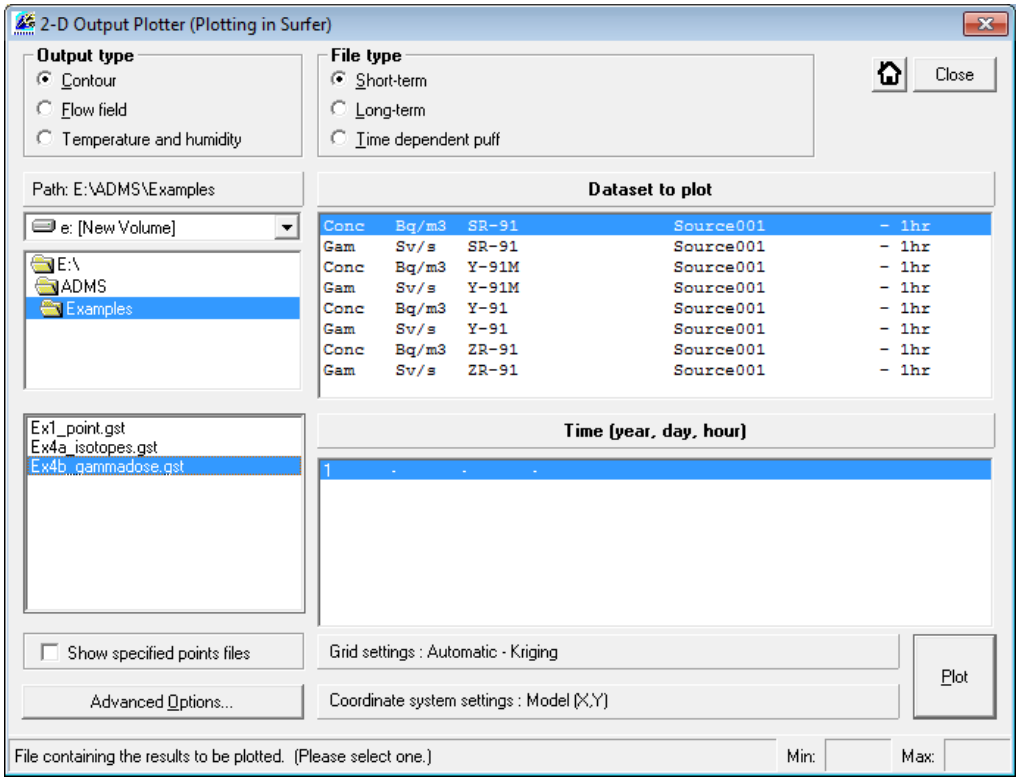


Figure 8.63 − Example 4b: the **2-D Output Plotter** screen.

Because of the low grid resolution used in this example and the small gamma dose values, the gridding method used by the contour plotter needs to be changed.

Click on the **Advanced Options**... button to bring up the **Advanced Contour Options** screen. Change the **Gridding method** to **Inverse distance** and the number of grid lines to match those used in the output grid. Compare with the screen shown in **Figure 8.64**. Click on the **Close** button to return to the contour plotter.

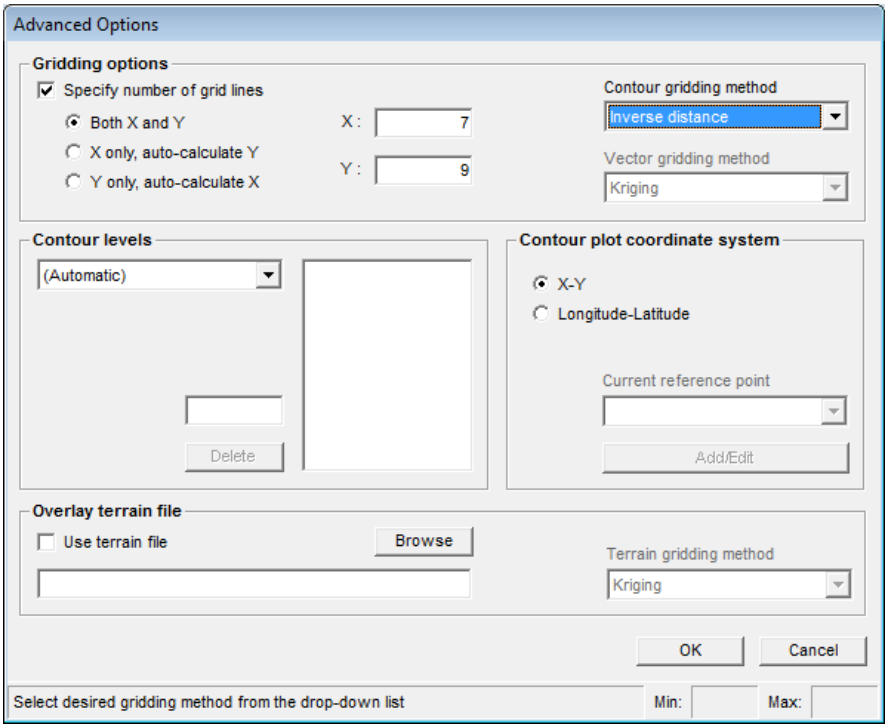


Figure 8.64 − Example 4b: the **Advanced Contour Options** screen.

Plot the gamma dose due to Sr-91 and the gamma dose due to Y-91M. Tile these results in Surfer, as shown in **Figure 8.65** but **do not** display these results on the same scale. You may want to adjust the contour levels. For more details of editing Surfer plots, see Appendix D.

The radioactive isotope Sr-91 has three radioactive daughter isotopes, namely Y-91M, Y-91 and Zr-91 (see the list of data sets in **Figure 8.63**). The half-life of Sr-91 is sufficiently short that some Y-91M and Y-91 are produced over the dispersion site.

The plots in **Figure 8.65** show how gamma dose output due to Sr-91 is in a region downwind of, but relatively close, to the source, while the gamma dose due to the daughter isotope Y-91M has higher gamma dose concentrations further downstream. This is because the activity distribution of Y-91M has a maximum further downstream.

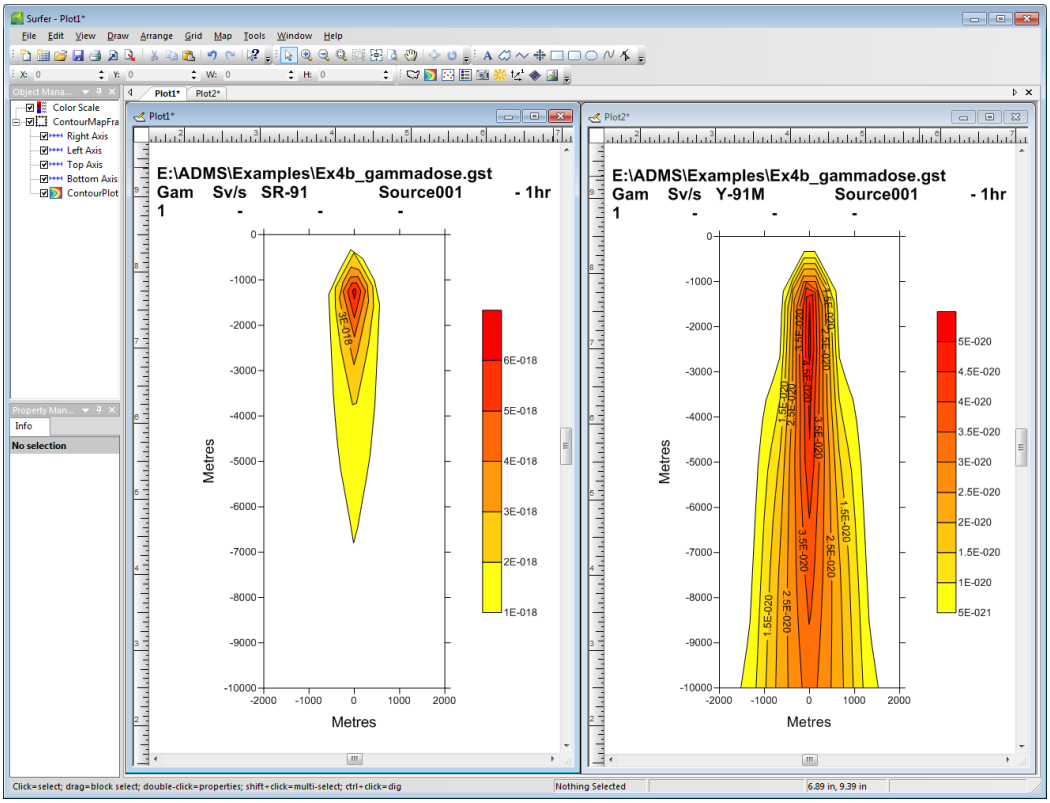


Figure 8.65 − Example 4b: gamma dose for Sr-91 and Y-91M.

8.5 Data files supplied with ADMS 5

This final section lists all the files to be found in the <*install_path*>*Examples* and<*install_path*>*Data* directories supplied with the model.

These files are principally *.apl* and *.met* files, but other auxiliary files such as time-varying source parameter (*.var*) files are also present. For each of the *.apl* files the output produced by running the model is also provided.

A short description of each file supplied in <*install_path*>*Examples* is given in **Table 8.1**. Those supplied in <*install_path*>*Data* are described in **Table 8.2**.

File name	Description
AreaLine.apl	Short-term run including area and line sources
Build.apl	Short-term run including building effects
Chem.apl	Short-term run including background pollutants and chemistry calculations
Coast.apl	Short-term run using the coastline module
Fluct.apl	Short-term run calculating fluctuations of the concentrations
Hill.apl	Short-term run using complex terrain
HillLong.apl	Long-term run using complex terrain
OneDay.apl	Long-term run using one day of hourly sequential data
Puff.apl	Short-term run with time-dependent puff
RadPoint.apl	Short-term run including radioactive decay of radioactive pollutants with dry and wet deposition
Rolling.apl	Short-term run with different averaging times, including rolling averages
Stack.apl	Short-term run with one point source (basic)
Timevary.apl	Long-term run with time-varying emission factors
Visible.apl	Short-term run modelling plume visibility
Ex_dat.dat	Example US NOAA/NCDC meteorological data input file for the ADMS 5 **Convert Met. Data** utility (described in Section 7)
Ex_dat.met	Example converted meteorological data file produced by converting *Ex_dat.dat* with the ADMS 5 **Convert Met. Data** utility (described in Section 7)
Ex_sfc.sfc	Example US AERMOD surface meteorological data input file for the ADMS 5 **Convert Met. Data** utility (described in Section 7)
Ex_sfc.met	Example converted meteorological data file produced by converting *Ex_sfc.sf* with the ADMS 5 **Convert Met. Data** utility (described in Section 7)

Table 8.1 − Example files in the directory *<install_path>\Examples*

File name	Description
Example.asp	Example additional specified point receptor locations file
backgrnd.bgd	Example background concentration file for chemistry calculations
UrbanCanopy.csv	Example urban canopy data file
SpatialVarDep.din	Example spatially varying deposition data file
VarDepFactors.dnf	Example file containing time-varying factors that modify the spatially varying deposition data
EmFactor.fac	Example time-varying emission factor file
ONEDAY.FSL	Example upper air file corresponding to *ONEDAY.MET*
270DEMO.MET	Example meteorological data file with westerly wind direction
ANNSTAT.MET	Example statistical meteorological data file with one year of data
BDFCOAST.MET	Example meteorological data file with convective, neutral and stable conditions (3 lines only) for use with the coastline option
COAST.MET	Example meteorological file with convective conditions for use with the coastline option
CONVEC.MET	Example meteorological file for unstable (convective) conditions
METDEMO.MET	Example meteorological file with explanatory notes of the meteorological data
NEUTRAL.MET	Example meteorological file for neutral conditions
NSC.MET	Example meteorological file with convective, neutral and stable conditions (3 lines only)
ONEDAY.MET	Example meteorological file with one day of hourly sequential data
R91A-G.MET	Meteorological file with data approximately equivalent to the Pasquill-Gifford (R91) stability categories A to G (7 lines)
radDEMO.MET	Example meteorological file including precipitation for use with radioactivity runs
STAT.MET	Example statistical meteorological data file with data representing 120 days
profiles.prf	Example user-entered meteorological vertical profile data file
TemporalVarRs.rsp	Example temporally varying surface resistance data file
ROUGHNES.RUF	Example variable roughness file
Terrain.ter	Example terrain file for complex terrain runs (the hill is a non-symmetric Gaussian hill with maximum height of 300 m)
Ss79r.tfw	World file for *Ss79r.tif*
Ss79r.tif	Example background map file
Ss79r.tif.prj	Coordinate projection file for *Ss79r.tif*
OpenStreetMaps Global TMS.ttkwp	Example Web Map Service file using Open Street Maps Global TMS service
OpenStreetMaps Global WMS.ttkwp	Example Web Map Service file using Open Street Maps Global WMS service
statvary.var	Example time-varying source parameters file for use with statistical met. data
timevary.var	Example time-varying source parameters file for use with sequential met. data
TemporalVarVd.vdp	Example temporally varying deposition velocity data file
Example.wfd	Example wind farm data file for the wind turbine effects module
Example.wtd	Example wind turbine data file for the wind turbine effects module

Table 8.2 − Example data files in the directory *<install_path>\Data*.

SECTION 9 Technical Summary

This section is intended to provide a summary of the mathematical and physical background to ADMS 5, and is essentially an abbreviated form of the Technical Specification documentation (CERC, 2015), to which readers should refer for full details.

9.1 Meteorological input and output

Section 3.3 describes the types of meteorological data that can be input to ADMS 5. For all the data types, each line of input data describes the meteorological conditions for one or more hours, the conditions being assumed to be fixed for any given hour. The meteorology input module reads the data and calculates values of the various meteorological quantities required for running the dispersion model.

Data may be in the form of a chronological record, which is termed *hourly sequential*. Other data may have a certain weight or frequency associated with each line of meteorological data. Such data are usually non-chronological and are termed *statistical* as they have been statistically analysed. It is also possible that the data are neither sequential nor statistical, but some more general collection of one or more meteorological conditions.

9.1.1 Input

The meteorological input dataset can contain a variety of meteorological parameters.

The complete list of possible input variables is shown in **Table 9.1**. The first three columns are alternative forms of the variable names that may be used in creating meteorological data files. As ADMS 5 matches the name to identify a variable the user may want to use short versions of the variable names (columns 1 and 3) in order to avoid making errors in typing. Names from different lists may be used in the same meteorological input file, e.g. using U for wind speed and CLOUD for cloud cover in the same file is acceptable. The fourth column shows the units in which the data should be entered, the fifth and sixth columns show the minimum and maximum values for the data and the seventh column shows whether the variable can be interpolated (see Section 3.3.5).

It is likely that the two most common types of input data will be: sequential data (or data for a single hour) with wind speed, direction, cloud cover, time of day, time of year, temperature and possibly precipitation and sea surface temperature; or statistical data, comprising wind speed, direction, surface sensible heat flux, boundary layer height (or depth) and precipitation. Such data are supplied by the Met Office for the main UK met. stations.

Note that the Pasquill-Gifford stability categories cannot be directly input into the model (although values of U, ϕ, L_{mo} and h may be), nor are they output. In ADMS 5 the boundary structure is characterised by the two parameters, h and L_{MO}. Values of these parameters corresponding *approximately* to the Pasquill-Gifford categories are provided in the data file *r91a-g.met* and shown in **Table 9.2**.

Short name	Long name	Abbreviated name	Units	Min value	Max Value	Can Interpolate?*
WIND SPEED	WIND SPEED	U	m/s	0.0	100.0	R
UG/USTAR	GEOSTROPHIC WIND SPEED/FRICTION VELOCITY	UGSTAR	-	5.0	1000.0	N
WIND DIRN	WIND DIRECTION (DEGREES)	PHI	°	0.0	360.0	R
DIRN CHANGE	GEOSTROPHIC MINUS SURFACE WIND DIRECTION (DEGREES)	DELTAPHI	°	-60.0	60.0	N
HEAT FLUX	SENSIBLE HEAT FLUX	FTHETA0	W/m^2	-100.0	1000.0	N
1/LMO	1/MONIN-OBUKHOV LENGTH	RECIPLMO	m^{-1}	-10.0	10.0	N
BL DEPTH	BOUNDARY LAYER DEPTH	H	m	40.0	4000.0	N
CLOUD	CLOUD AMOUNT (OKTAS)	CL	oktas	0.0	8.0	R
SOLAR RAD	INCOMING SOLAR RADIATION	SOLAR RAD	W/m^2	0.0	1500.0	N
TEMPERATURE	TEMPERATURE (C)	T0C	°C	-100.0	60.0	Y
N ABOVE BL	BUOYANCY FREQUENCY ABOVE BOUNDARY LAYER	NU	s^{-1}	0.0	0.1	Y
DELTA THETA	TEMPERATURE JUMP ACROSS BOUNDARY LAYER TOP	DELTATHETA	°C	0.0	25.0	N
PRECIP	PRECIPITATION RATE (MM/HOUR)	P	mm/h	0.0	500.0	Y
SEA TEMP	SEA SURFACE TEMPERATURE (C)	TSEA	°C	-10.0	40.0	Y
DELTA T	TEMPERATURE OVER LAND MINUS SEA SURFACE TEMPERATURE	DELTAT	°C	-40.0	40.0	Y
SIGMA THETA	SIGMA THETA (DEGREES)	SIGMATHETA	°	0.0	90.0	Y
S HUMIDITY	SPECIFIC HUMIDITY	S HUMIDITY	kg/kg	0.0	0.1	Y
R HUMIDITY	RELATIVE HUMIDITY (PERCENT)	RHUM	%	0.0	100.0	Y
RH ABOVE BL	RELATIVE HUMIDITY ABOVE BOUNDARY LAYER (PERCENT)	RH ABOVE BL	%	0.0	100.0	Y
DRH/DZ	D(RELATIVE HUMIDITY)/DZ ABOVE BOUNDARY LAYER (PERCENT/M)	DRH/DZ	%/m	-10.0	10.0	N
LAT HT FLUX	LATENT HEAT FLUX	LAT HT FLUX	W/m^2	-100.0	1000.0	N
Z0 (M)	ROUGHNESS LENGTH (MET SITE)	Z0 (M)	m	1.0×10^{-10}	100.0	Y
Z0 (D)	ROUGHNESS LENGTH (DISPERSION AREA)	Z0 (D)	m	1.0×10^{-10}	100.0	Y
ALBEDO (M)	ALBEDO (MET SITE)	R	-	0.0	1.0	Y
ALBEDO (D)	ALBEDO (DISPERSION AREA)	ALBEDO (D)	-	0.0	1.0	Y
ALPHA (M)	MODIFIED PRIESTLEY-TAYLOR PARAMETER (MET SITE)	ALPHA	-	0.0	3.0	Y
ALPHA (D)	MODIFIED PRIESTLEY-TAYLOR PARAMETER (DISPERSION AREA)	ALPHA (D)	-	0.0	3.0	Y
HOUR	HOUR	THOUR	-	0	24	R
DAY	DAY	TDAY	-	1	366	R
YEAR	YEAR	YEAR	-	1900	2500	R
FREQUENCY	FREQUENCY	FR	-	0	-	N
FREQUENCY FOR MONTHS xx TO xx, HOURS xxxxx TO xxxxx	FREQUENCY FOR MONTHS xx TO xx, HOURS xxxxx TO xxxxx (GMT + xxxxx)	MONTHS xx TO xx, HOURS xx TO xx	-	-	-	N

Table 9.1 − Variables that may be input into the meteorological input module. *Key for interpolation indicator: R = Required for interpolation, Y = interpolatable, N = not interpolatable.

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

Stability: Stable $h/L_{MO} > 1$
Neutral $-0.3 \leq h/L_{MO} \leq 1$
Convective $h/L_{MO} < -0.3$

U (m/s)	L_{MO} (m)	$1/L_{MO}$ (m^{-1})	h (m)	h/L_{MO}	P-G Category
1	-2	-0.50	1300	-650	A
2	-10	-0.10	900	-90	B
5	-100	-0.01	850	-8.5	C
5	∞	0.00	800	0	D
3	100	0.01	400	4	E
2	20	0.05	100	5	F
1	5	0.20	100	20	G

Table 9.2 − Values of wind speed, Monin-Obukhov length (L_{MO}) and boundary layer height (h), which may be used to represent Pasquill-Gifford categories A-G.

The minimum requirement for meteorological data (for the model to run) is:

- wind speed (this would normally be a near surface wind, but could be a geostrophic wind or friction velocity – in each case the height of wind must be entered, which would typically be 10 m for the near-surface case and should be 1000 m and 0 m for the geostrophic and friction velocity cases, respectively),
- wind direction,

plus *one* of the following:

- reciprocal of Monin-Obukhov length,
- surface sensible heat flux,
- cloud cover, time of day and time of year.

With regard to the latter three parameters, if more than one of these is supplied, they are used in order of preference in the order they are listed above. That is, if the reciprocal of Monin-Obukhov length is specified it will be used, otherwise the surface sensible heat flux will be used. If neither the reciprocal of Monin-Obukhov length nor the surface sensible heat flux are specified then the cloud cover, time of day and time of year will be used.

If cloud cover, time of day and time of year are the *only* data specified, then it is advisable to add temperature and boundary layer height, if a good estimate is known, to the variables in the *.met* file. If the boundary layer height is not known, the estimate for a particular hour will be improved if meteorological data for all the hours from midnight are included, as the ADMS 5 scheme uses this information from previous hours to improve the estimate.

It is preferable to include cloud cover data if available. However, if unavailable,

incoming solar radiation may be specified instead. Incoming solar radiation is measured by a radiometer at ground level, although check with the manufacturer exactly what is measured.

In general, the boundary layer height should always be entered if you think you can provide a better estimate than the meteorological input module. The module will provide a good estimate when the site is in mid latitudes, e.g. UK, and it is either

1. day time with sequential data stretching back at least to dawn, *or*
2. night time.

If however, neither 1 nor 2 apply and you do *not* have information about the boundary layer height, then it is advisable to provide as much information as you have available (e.g. all five of the above meteorological parameters). This may help the model to estimate boundary layer height by enabling it to estimate the values of heat flux occurring prior to the hour currently under consideration. In addition, specifying temperature may also help to improve the module's estimate of the boundary layer height.

Additional meteorological input data that may be required are:

- precipitation rate, if wet deposition is selected (unless a constant value of washout coefficient is specified);
- sea surface temperature and near-surface temperature over land (or the difference between them), if the **Coastline** option is used (see Section 4.12);
- surface relative humidity or specific humidity, plus surface temperature, if the **Plume visibility** option is used (see Section 4.5); and
- frequencies with which particular meteorological conditions occur, if the meteorological data are statistical.

Note that by default, ADMS 5 does not model calm conditions. Any lines of meteorological data for which the wind speed at 10 m is calm (less than 0.75 m/s) will be skipped by the model, and output will be given as -999 for that line of meteorological data.

However, there is an option to enable the modelling of calm conditions using a *.aai* file (see Section 4.18). In this case the lowest value of U at which the model will calculate concentrations is known as *MinU10*. If U is lower than *MinU10* then it will be increased to *MinU10* (with friction velocity u_* and surface heat flux $F_{\theta 0}$ adjusted accordingly). *MinU10* may be input by the user; the default and minimum value is 0.3 m/s.

Further input parameters

In addition to the data in the meteorological input data set, the module also requires certain data that are provided via the interface. These are:

- surface roughness;
- height of recorded wind, i.e. the height above ground at which the wind measurements were made (usually 10 m);

- whether the meteorological data are hourly sequential; and
- whether the wind direction data are in sectors, and if so, the sector size. Wind direction data are often recorded to the nearest 10°, 22.5° or 30°. Inputting the appropriate sector size allows the model to account for the approximation in the wind direction. For small sector sizes (less than 15°) the model will use one of three equally spaced wind directions within the sector. For example, if the data are in 10° sectors, and the wind direction in the meteorological data file is given as 30°, the model will use 26.7°, 30° or 33.3°. For large sector sizes (greater than 15°), the model will carry out calculations for all of 5 equally spaced wind directions within the sector. The results for all wind directions are given equal weighting in the concentration calculations. For example, if the data are in 30° sectors and the wind direction in the meteorological data file is given as 30°, the model will use 18°, 24°, 30°, 36° and 42°. If the data are not in sectors, the wind direction is used exactly as given in the meteorological data file.

Further optional parameters may also be specified via the interface. These include a minimum value of L_{MO} (for urban areas) and values of Priestley-Taylor parameter and surface albedo.

If the met. site is distant from the area of dispersion, the meteorological input module 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. Different values of surface albedo, Priestley-Taylor parameter and minimum value of L_{MO} at the met. site and dispersion site may also be entered. It is also possible to enter a precipitation factor to account for differences in rainfall between the sites (see Section 4.15).

9.1.2 User-input vertical profile data

The user can input vertical profiles for the wind speed, turbulence parameters, temperature and humidity. These data are entered at a sequence of heights for each line of meteorological data. To obtain the value at a particular height, the model will interpolate a value from those entered, using the algorithm outlined below.

The interpolation algorithm is based on a linear interpolation of the input *.prf* data and the standard ADMS 5 profiles; essentially a profile similar in shape to the ADMS 5 profile is fitted to the input *.prf* data. The interpolation algorithm is detailed in the ADMS 5 Technical Specification (CERC, 2015).

Note that the data entered in the *.prf* file override the corresponding values entered in the *.met* file.

9.1.3 Meteorological data pre-processing

The meteorological input module is called once for each hour's data and uses standard algorithms to calculate the boundary layer meteorological parameters required by the dispersion model. Full details can be found in Holtslag and van Ulden (1983) and the ADMS 5 Technical Specification (CERC, 2015).

In addition to processing the data, the module checks that the input data are plausible, i.e. that they lie within certain limits. While it is running, messages may be provided

giving warnings and notification of errors. Where an error occurs which prevents calculations being done for that particular line, output for that line will be given as –999.

9.1.4 Output

The variables output by the meteorological input module are listed in **Table 9.3**.

Variable	Description
u_*	Friction velocity (m/s)
U_g	Geostrophic wind speed (m/s)
U_g^*	Geostrophic wind speed normalised by the friction velocity
ϕ_0	Surface wind direction (angle from which wind blows in degrees measured clockwise from north, e.g. 270° is a westerly wind) (°)
ϕ_g	Geostrophic wind direction (angle from which wind blows in degrees measured clockwise from north) (°)
$\Delta\phi$	Geostrophic wind direction minus surface wind direction (°)
ϕ	Wind direction (as obtained from the meteorological input data) (°)
ϕ_{sec}	Wind direction used by the model for this meteorological data line for long-term calculations, which may differ from the input value if the data are in sectors (°)
w_*	Convective velocity scale (if $F_{\theta_0} > 0$, $w_* = (g\, F_{\theta_0}\, h/\rho\, c_p\, T_0)^{1/3}$; if $F_{\theta_0} \le 0$, $w_* = 0$) (m/s)
F_{θ_0}	Surface heat flux (W/m^2)
K	Incoming solar radiation (W/m^2)
$1/L_{MO}$	Reciprocal of the Monin-Obukhov length (m^{-1})
h	Boundary layer height (m)
N_u	Buoyancy frequency above the boundary layer (s^{-1})
$\Delta\theta$	Temperature jump across the boundary layer top (K)
T_0^c	Near-surface temperature (°C)
P	Precipitation rate (mm/h)
ΔT	Near-surface temperature over land minus sea surface temperature (°C)
σ_θ	Standard deviation of mean wind direction (°)
q_0	Surface specific humidity (kg/kg)
λ_E	Surface latent heat flux (W/m^2)
RH_u	Relative humidity just above the boundary layer (%)
$d(RH_u)/dz$	Relative humidity lapse rate above the boundary layer (%/m)

Table 9.3 − Output variables from the meteorological input module.

The variables P, ΔT, q_0, λ_E, RH_u and $d(RH_u)/dz$ may be missing if insufficient input data are given. The processed met. module data are available for use by the other modules, in particular the boundary layer structure module.

9.1.5 Limitations

In calculating the boundary layer parameters, it is assumed that the boundary layer is self-similar for a given value of h/L_{MO}. However, users of the model should be aware

that there are some situations, such as latitudes near the equator or the poles, where this approximation can lead to significant errors. Full details are given in the ADMS 5 Technical Specification (CERC, 2015).

9.2 Parameterisation of the boundary layer

In ADMS 5 the boundary layer is characterised by the boundary layer height *h* and the Monin-Obukhov length L_{MO} and **not** by a Pasquill-Gifford stability category.

The Monin-Obukhov length is defined as

(9.1)
$$L_{MO} = \frac{-u_*^3}{\left(\frac{\kappa g F_{\theta 0}}{\rho c_p T_0}\right)}$$

in which u_* is the friction velocity at the Earth's surface, κ is the von Karman constant (0.4), g is the acceleration due to gravity, $F_{\theta 0}$ is the surface sensible heat flux, ρ and c_p are, respectively, the density and specific heat capacity of air and T_0 is the near-surface temperature.

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

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

Figure 9.1 shows the different regions of the boundary layer in terms of the parameters h/L_{MO} and z/h where z is height above the ground. **Figure 9.2** shows the same information but with a dimensional vertical scale, z.

In the different regions of the boundary layer different mechanisms are important in generating turbulence. These are:

1. surface heating leading to convectively generated turbulence (the convective eddies increase in energy as they rise through the boundary layer),
2. shear at the surface leading to mechanically generated turbulence,
3. local shear, for instance at the top of the boundary layer, that can be a weak source of turbulence.

This approach to boundary layer stability, whereby the boundary layer structure is defined in terms of two variables (z/L_{MO} and z/h), supersedes the Pasquill-Gifford formulation, and differs crucially from the Pasquill formulation in allowing the variation of boundary layer properties with height to be included. However, it is difficult to make exact comparisons between the two schemes.

Figures 9.1 and **9.2** show Pasquill-Gifford stability categories corresponding *approximately* to ranges of h/L_{MO}. Note that, particularly in stable meteorological conditions, the Pasquill class is not a simple function of h/L_{MO}.

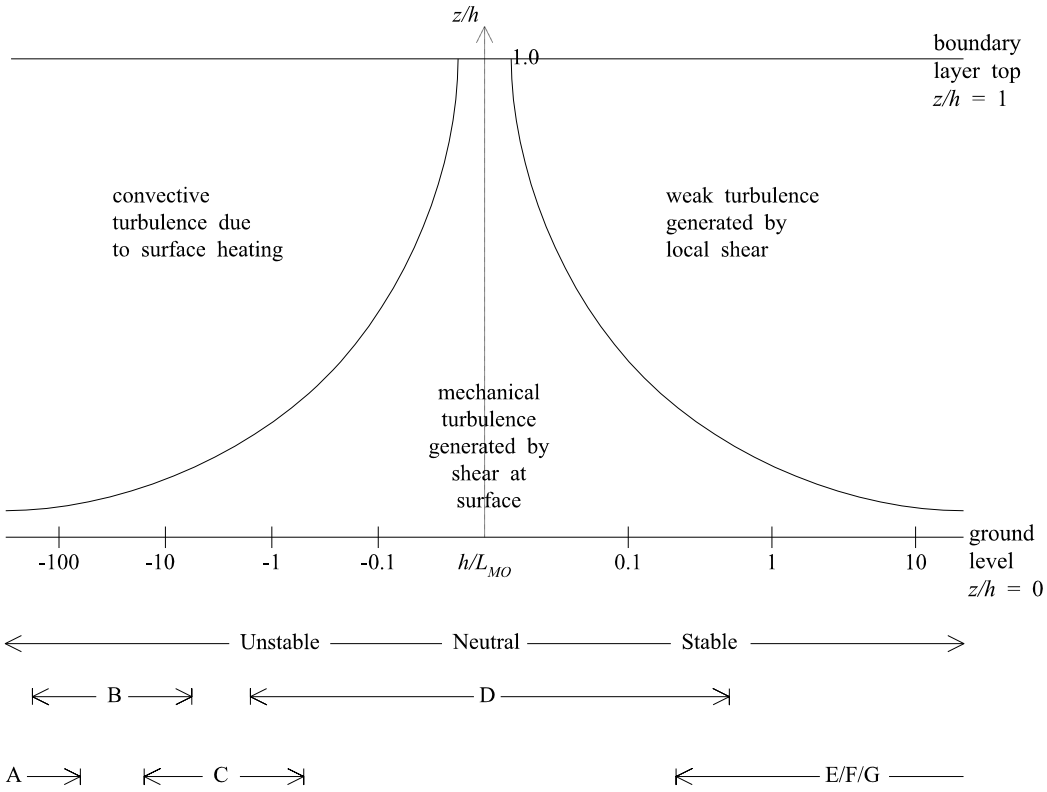


Figure 9.1 – Non-dimensional schematic representation of variation of Monin-Obukhov length with atmospheric stability.

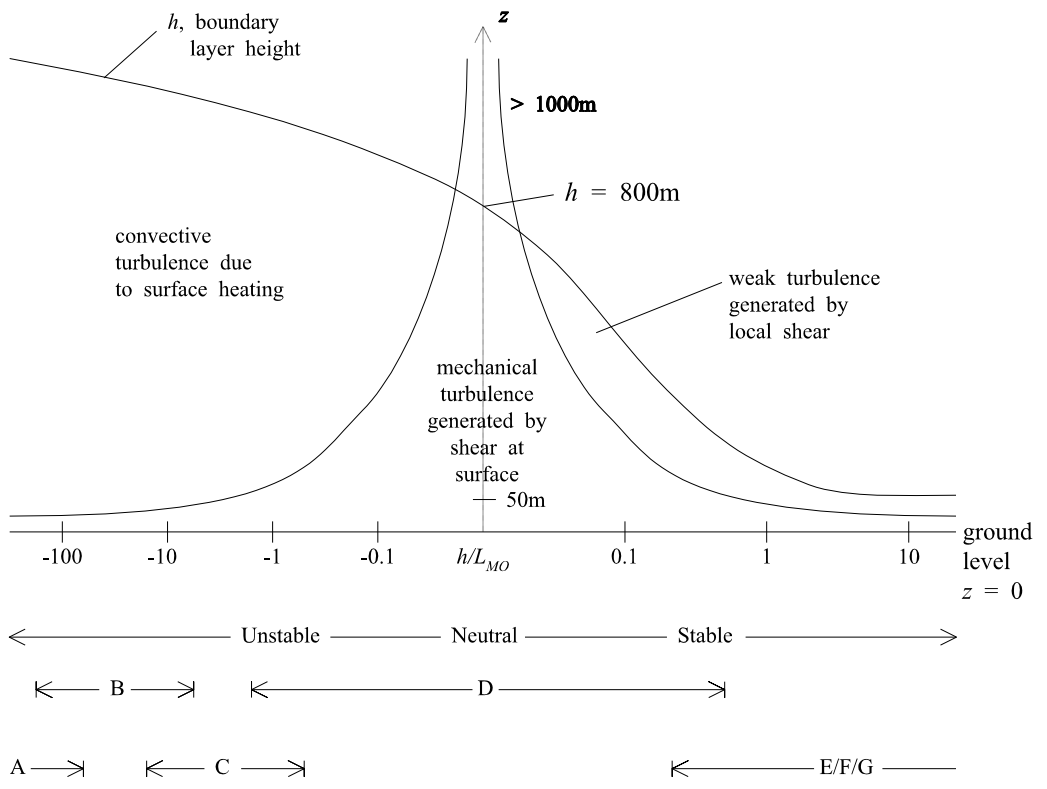


Figure 9.2 – Dimensional schematic representation of variation of Monin-Obukhov length with atmospheric stability.

9.2.1 Boundary layer structure

ADMS 5 calculates the boundary layer variables listed in **Table 9.4** at different heights. Vertical profiles are expressed as functions of z/L_{MO} and z/h and have been derived from experimental data (Caughey and Palmer, 1979; van Ulden and Holtslag, 1985; Hunt *et al*., 1988d). These variables are used, in turn, by other modules.

Variable	Description
$U(z)$, $\frac{dU}{dz}$, $\frac{d^2U}{dz^2}$	Mean wind speed (m/s) and its first (s^{-1}) and second derivatives ($m^{-1}s^{-1}$) with height
$\sigma_u(z)$, $\sigma_v(z)$, $\sigma_w(z)$	Root-mean-square turbulent velocities (m/s)
$\Lambda_v(z)$, $\Lambda_w(z)$	Turbulent length scales (m)
$\varepsilon(z)$	Energy dissipation rate (m^2/s^3)
$T_L(z)$	Lagrangian time scale (s)
$N(z)$	Buoyancy frequency (s^{-1})
$T(z)$	Temperature (K)
$\rho(z)$	Density (kg/m^3)
$P(z)$	Pressure (mbar)

Table 9.4 − Boundary layer variables calculated by ADMS 5.

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

Turbulent velocities σ_u, σ_v and σ_w are subject to a minimum value of turbulence that ranges between 0.01 and 0.2 m/s, depending on the user-input minimum value of L_{MO}. This accounts for the fact that in urban areas (high minimum L_{MO}), where conditions never become very stable, there will always be some turbulence. The minimum turbulence value to be used can also be defined by the user using an *.aai* file, as described in Section 4.22.

9.3 Dispersion over flat terrain

9.3.1 Dispersion parameters

Field experiments and research have shown that the dispersion parameters σ_y and σ_z vary with downwind distance from a point source in a way that depends on the atmospheric boundary layer height (h), the height of the source (z_s) and the height of the plume as it travels downwind. For reviews of this subject, see Hunt *et al*. (1988a), Hanna and Paine (1989) and Weil (1985). This approach is in contrast to older methods, described in the NRPB report R91 (Clarke, 1979) and used in the model ISC (U.S. E.P.A., 1995), in which the effect of the source height is not taken into account when calculating the width and depth of the plume.

There is no general theory or even generally accepted semi-empirical expression that describes the dispersion from a source at all heights within the boundary layer ($0 < z_s < h$), in all conditions of atmospheric stability, and over the complete range of distances from the source to about 30 km downwind. In developing ADMS 5 the approach adopted has been first to use formulae that have been developed and broadly accepted for specific ranges of the parameters z_s/h, h/L_{MO} (stability) and x/h (downwind distance). Interpolation formulae have then been constructed to cover the complete parameter range. The basis for these formulae is set out at length in the report by Hunt *et al*. (1988a).

9.3.2 Stable and neutral boundary layers

All the turbulence in the stable boundary layer is mechanically generated, i.e. there is no generation of turbulence due to convective motions. Usually, the level of turbulence decreases with height, as the relative importance of stratification increases, although it can be enhanced by wave motions at the top of the boundary layer. (The effect of such wave motions is not considered by ADMS 5.)

Concentration distribution

The profile of the concentration distribution is Gaussian with reflections at the ground, and at the top of the boundary layer, if there is an inversion (i.e. sharp increase in temperature with height) at the top of the boundary layer. It is assumed that an inversion is present if conditions are neutral or convective, or if the meteorological pre-processor predicts an inversion. If there is no inversion, the final three terms in the brackets in Equation (9.2), which represent reflections at the top of the boundary layer, are not included.

$$C = \frac{Q_s}{2\pi\,\sigma_y\,\sigma_z\,U} e^{-y^2/2\sigma_y^2} \times \left(e^{-(z-z_s)^2/2\sigma_z^2} + e^{-(z+z_s)^2/2\sigma_z^2} + e^{-(z+2h-z_s)^2/2\sigma_z^2} + e^{-(z-2h+z_s)^2/2\sigma_z^2} + e^{-(z-2h-z_s)^2/2\sigma_z^2} \right) \tag{9.2}$$

Spread parameters

The vertical dispersion parameter (σ_z) at the mean height of the plume (z_m) is linked

directly to the vertical component of turbulence (σ_w) and the travel time from the source (t) by the relationship

(9.3) $$\sigma_z = \sigma_w t \left(\frac{1}{b^2} + \frac{N^2 t^2}{1+2Nt} \right)^{-1/2}$$

where N and σ_w are the buoyancy frequency and root-mean-square vertical turbulent velocity at z_m, respectively (Weil, 1985; Hunt, 1985). The factor b is a function of z_s/h and ensures a smooth transition between the solution for surface releases and elevated releases.

The transverse dispersion parameter (σ_y) is given by

(9.4) $$\sigma_y^2 = \sigma_{yt}^2 + \sigma_{yw}^2$$

The spreading due to turbulence (σ_{yt}) is expressed as:

(9.5) $$\sigma_{yt} = \sigma_v t \left(1 + \sqrt[3]{15.6} \frac{u_* t}{h} \right)^{-1/2}$$

with u_* the friction velocity at the Earth's surface and σ_v the root-mean-square crosswind turbulent velocity crosswind.

The spread due to variations in mean wind direction (σ_{yw}) is equal to $\sigma_\theta x$. The parameter σ_θ is the standard deviation of the mean wind direction. This may either be specified as a measured meteorological input parameter in degrees, or estimated by the meteorological data pre-processor using the following expression:

(9.6) $$\sigma_\theta = 0.065 \sqrt{\frac{7T}{U_{10}}}$$

Here T is the averaging time in hours and U_{10} is the mean wind speed at height 10 m.

Near-field to far-field transition

In neutral conditions the part of the plume that does not have sufficient momentum or buoyancy to penetrate the top of the boundary layer is effectively confined within the boundary layer, because material reaching the top of the layer is reflected downwards. Sufficiently far from the source, after parts of the plume have been reflected at the ground and at the top of the boundary layer, the vertical variation in concentration of the pollutant is so small as to be negligible. This occurs approximately at the downwind distance where $\sigma_z \approx h$ (in fact, where $\sigma_z = 1.5\,h$). Downwind of this point the plume is considered to grow horizontally as a vertical wedge as if from a uniform vertical line source extending from 0 to h, rather than as a cone, so the variation with z in equation (9.2) is ignored.

9.3.3 Convective boundary layer

Field experiments of diffusion from elevated sources in the convective boundary layer

(Briggs, 1985) have confirmed earlier laboratory and computational studies (Lamb, 1982) that the form of the vertical profiles of concentration are skewed and significantly non-Gaussian, thus changing the profile of concentration compared with an assumed Gaussian distribution. Near the ground this is important for modelling processes such as wet or dry deposition. It is also very important for evaluating maximum ground-level concentrations from elevated releases. Ignoring the non-Gaussian distribution can lead to under-estimates of ground-level concentration from elevated sources. Models that use non-Gaussian profiles are referred to as "new generation" models. New generation models include the Hybrid Plume Diffusion Model (HPDM) of Hanna and Paine (1989), the Almanac code of National Power (Moore and Lee, 1982) and the more recent CTDM code of the U.S. E.P.A. (Perry, 1991) and AERMOD (Cimorelli *et al*., 2004).

Concentration distribution

In the convective boundary layer (CBL) the probability distribution of the vertical velocity and, hence, the concentration distribution is **non-Gaussian**, or skewed. The non-Gaussian distribution ensures that, for elevated sources, the height within the plume at which the concentration is a maximum descends as the plume moves downwind, while the plume mean height ascends. After the height of the maximum concentration reaches the ground it can rise again.

Spread parameters

The transverse dispersion parameter (σ_y) is calculated in three parts, the first for dispersion due to convection (σ_{yc}), the second due to mechanically driven turbulence (σ_{yn}), and the third ($\sigma_{yw} = \sigma_\theta x$) is included to allow for the variation in the wind direction:

(9.7) $$\sigma_{yc} = \sigma_{vc} t \left(1 + \frac{t}{h} \sqrt[3]{0.75} w_* \right)^{-1/2}$$

(9.8) $$\sigma_{yn} = \sigma_{vn} t \left(1 + \frac{t}{h} \sqrt[3]{15.6} u_* \right)^{-1/2}$$

σ_{vc} and σ_{vn} are the root-mean-square horizontal turbulent velocities due to convection and mechanically driven turbulence, respectively. σ_θ is as defined in equation (9.6).

The total transverse spread is therefore given by

(9.9) $$\sigma_y^2 = \sigma_{yc}^2 + \sigma_{yn}^2 + \sigma_{yw}^2$$

For the vertical spread σ_z, the skewed nature of the probability distribution function for the vertical turbulent velocity w leads to the definition of σ_{w+} and σ_{w-} for the upwards and downwards velocities, respectively. Then $\sigma_{z+} = \sigma_{w+} t$ and $\sigma_{z-} = \sigma_{w-} t$ are defined and used in the calculation of C_{CBL}. A more detailed description is given in the ADMS 5 Technical Specification (CERC, 2015), P10/01 & P12/01 ("Plume/puff spread and mean concentration module specifications").

The formulae above (equations (9.7) and (9.8)) reduce to forms *suitable* for neutral

conditions as the meteorological conditions tend towards neutral conditions, but the actual forms are not the same as the ADMS 5 Gaussian plume formula for the neutral boundary layer. A smooth transition from the convective boundary layer solution to the neutral boundary layer solution is used.

Near-field to far-field transition

The transition to a far-field model, where the source is equivalent to a vertical line source extending from 0 to h and of uniform strength, is assumed to occur when $\sigma_z = 1.5\,h$.

9.3.4 Special treatment of 'calm' wind conditions

By default, ADMS 5 does not model meteorological data for which the wind speed at 10 m is less than 0.75 m/s, or 'calm'. However, modelling of calm conditions may be enabled through use of an *.aai* file (refer to Section 4.18). This section describes how ADMS 5 treats calm conditions when this option is selected.

The direction of the wind becomes more variable for lower mean wind speeds. In very unstable conditions this can arise because the turbulent fluctuations in the flow are large or comparable with the mean wind even when the geostrophic wind is well defined. In stable conditions, when the geostrophic wind is very small, both the mean and turbulent wind can be very small and immeasurable, but more usually the measured wind at the surface is very light either with variable direction or with consistent direction with the wind being forced by thermal gradients or topography.

The approach used for calm conditions is to calculate the concentration as a weighted average of a normal 'Gaussian' type plume (C_g) and a radially symmetric plume (C_r), where the weighting depends on the wind speed at 10 m (U).

The radially symmetric plume is modelled as a passive source that has a source height equal to the maximum plume height from the standard plume rise calculations, and assumes equal probability of all wind directions. C_r does not include the effect of buildings, if buildings are modelled.

For meteorological data with U **below** $U_{g_{\min}}$ (this value is the lower limit of the validity of the normal Gaussian-type plume), the model calculates C_r only. The default value of $U_{g_{\min}}$ is 0.5 m/s, but this can be altered using the *.aai* file.

For meteorological data with U **above** a critical value U_{crit}, the model calculates C_g only. U_{crit} is determined by

$$U_{crit}^2 = U_{calm}^2 + \frac{\sigma_u^{\,2}}{1+\frac{t}{2T_L}} \tag{9.10}$$

where σ_u is the root-mean-squared longitudinal turbulent velocity at height 10 m, t the time from release and T_L the lagrangian time scale at height 10 m. The default value of U_{calm} is 1.0 m/s, but this can be altered using the *.aai* file.

For meteorological lines with U **between** $U_{g_{\min}}$ and U_{crit}, the model calculates the concentration as a weighted average of C_g and C_r.

If complex terrain or the urban canopy module are modelled in conjunction with the calm conditions module then for wind speeds below 0.75 m/s the flat terrain calm conditions solution is used, for wind speeds above 0.75 m/s the complex terrain or urban canopy solution is used.

9.4 Plume rise

The plume rise module predicts the rise in trajectory and enhanced dilution of a continuous emission of gaseous material that is hot or has momentum. It takes into account the effects of plume buoyancy and momentum in generating plume rise, and includes penetration of inversions.

The underlying theory is based on an integral model, in which integral conservation equations are solved for the fluxes of mass, momentum and heat. The plume is assumed to be of circular cross-section. The plume properties such as velocity, density, etc., are uniform in any cross-section (i.e. a "top-hat" profile) but vary along the plume centreline. Entrainment of ambient air takes place due to the motion of the plume relative to its environment as well as due to atmospheric turbulence. The external velocity and temperature fields vary with height.

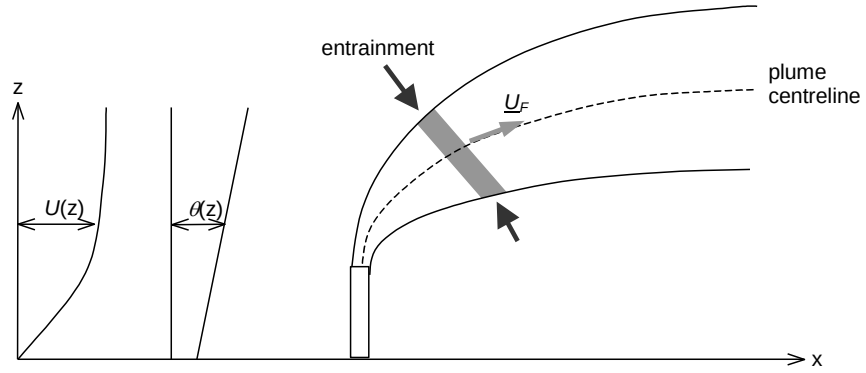


Figure 9.3 – Plume rise model.

The plume rise module is initialised from the following source conditions:

- exit diameter;
- emission velocity or volume flow rate; and
- temperature or density of the release material.

The equations are then solved numerically by a Runge-Kutta numerical scheme with a variable internal time step. If plume visibility is modelled, the plume temperature and hence plume rise is affected by energy transfers relating to the moisture content.

9.4.1 Stack-induced downwash

The flow and pressure fields around a stack can influence the plume rise of the emission by reducing the mean height of the plume just downwind of the source. Only releases of relatively small upward momentum are affected, since all other emissions rise rapidly away from the zone of influence.

The algorithm used to predict stack downwash in ADMS 5 is that used in many other models (Hanna *et al*., 1982; Snyder and Lawson, 1991) and applies to point sources only.

If the emission velocity ratio, w_S/U_H, is less than 1.5, the source height is corrected

(reduced) by Δz_s, where

$$\Delta z_s = \begin{cases} 2\left(\frac{w_S}{U_H} - 1.5\right) D_S & \text{for} \quad \frac{w_S}{U_H} < 1.5 \\ 0 & \text{for} \quad \frac{w_S}{U_H} > 1.5 \end{cases} \tag{9.11}$$

with D_s the outside diameter of the stack, U_H the approach flow speed at the height of the stack top, w_s the emission speed, and Δz_s the correction to the stack height.

If the **Buildings** option is used, the stack downwash correction is limited to ensure that the final source location is not inside one of the user-defined buildings.

ADMS 5 includes an option to disable stack downwash for selected sources; please refer to Section 4.21 for more information.

Limitations

The stack downwash algorithm is quite widely used in dispersion models although it has never been adequately validated.

Strictly speaking, the stack downwash algorithm should only be applied to stacks of circular cross section, though again it could be applied, at the user's discretion, to other cases, such as stacks of square cross section.

9.5 Source types

9.5.1 Point sources

Circular point sources are specified by the location of their centre, their diameter and their height. The circle that defines the point source is assumed to lie in a horizontal plane.

Pollutant emissions are specified in mass units/s.

9.5.2 Area, volume and line sources

Line sources with finite width, and area and volume sources with convex polygonal base areas can be modelled by ADMS 5. All these source types are modelled by ADMS 5 as convex polygons.

Line sources are specified by the location of the midpoints of their ends, the height of both end points and a width. Area and volume sources are defined by the location of the vertices of their base area, a height and, in the case of a volume source, by a depth. The ends of a line source and the base of an area or volume source are assumed to lie in a horizontal plane.

A volume source does not have any plume rise. For instance, fugitive emissions that are continuously emitted from a building with negligible velocity, or the dust generated by quarrying which again is continuously emitted and is mixed throughout a depth at the source location but has no significant emission velocity, might be modelled as volume sources.

Pollutant emissions are specified in mass units/m/s for line sources, mass units/m^2/s for area sources and mass units/m^3/s for volume sources.

Method of calculation

Each source is decomposed into a maximum of 10 source elements, the decomposition being a function of the receptor location. The difference in streamwise distance between source elements and each receptor is constrained not to vary too rapidly, subject to the maximum number of 10 source elements. The concentration is then calculated by summing the contributions from each element.

In the case of a line or area source, the contribution from each element is approximated by a crosswind line source of finite length. The expression for the concentration $\overline{C}(x, y, z)$ from a finite crosswind line source of length L_s is given by equation (9.12). The source strength $\overline{Q}_s$ is in mass units/m/s.

$$\overline{C}(x,y,z)=\frac{\overline{Q}_s}{2\sqrt{2\pi}\sigma_z(x)U}\exp\left(-\frac{(z-z_s)^2}{2\sigma_z^2}\right)\times \left[\operatorname{erf}\left(\frac{y+L_s/2}{\sqrt{2}\sigma_y}\right)-\operatorname{erf}\left(\frac{y-L_s/2}{\sqrt{2}\sigma_y}\right)\right]+\text{reflection terms} \tag{9.12}$$

Similarly, for a volume source, the contribution from each element is approximated by a crosswind vertical slice of finite length and height. The expression for the concentration $\overline{\overline{C}}(x,y,z)$ from a crosswind vertical slice of length L_s and height L_1 is given by equation (9.13). The source strength $\overline{\overline{Q}}_s$ is in mass units/m^2/s.

$$\overline{\overline{C}}(x,y,z)=\frac{\overline{\overline{Q}}_s}{4U}\left[\operatorname{erf}\left(\frac{y+L_s/2}{\sqrt{2}\sigma_y}\right)-\operatorname{erf}\left(\frac{y-L_s/2}{\sqrt{2}\sigma_y}\right)\right]\times \left[\operatorname{erf}\left(\frac{z+L_1/2-z_s}{\sqrt{2}\sigma_z}\right)-\operatorname{erf}\left(\frac{z-L_1/2-z_s}{\sqrt{2}\sigma_z}\right)\right] + \text{reflection terms} \tag{9.13}$$

The calculation is accelerated by discarding those parts of the source that are at sufficiently large crosswind distances from a receptor that they do not contribute significantly to the concentration at that receptor. Redundant parts of the source are considered to be those for which a receptor lies outside the lines of $y = \pm 0.4\, x\, \sigma_y(z = z_m, x = 10\text{ m})$ (as shown in **Figure 9.4**).

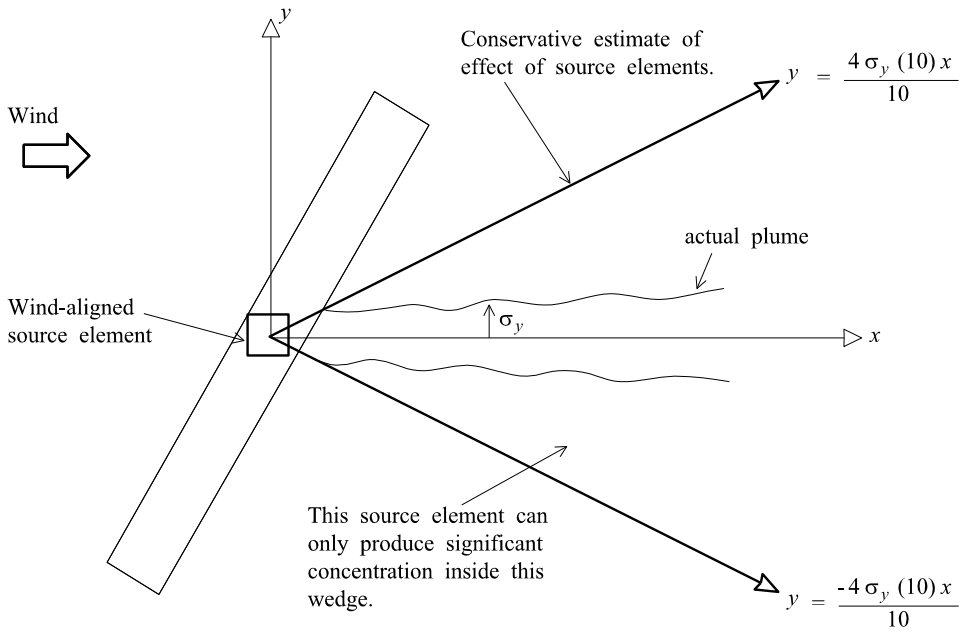


Figure 9.4 – The region of influence of a source element.

9.5.3 Jets and directional releases

ADMS 5 can model releases that may have cross-wind and upstream or downstream components of emission velocity as well as a vertical velocity component (which may be positive or zero). A jet is a subset of the more general directional release in that it is assumed to be largely composed of a substance other than air. By contrast a directional release will include typical industrial releases, which are largely air. Both of these release types are defined as 'jet' sources in ADMS 5.

Defining the release

The user enters the total exit velocity w_{tot} together with two angles θ_1 and θ_2, as shown in **Figure 9.5**. θ_1 is the angle of elevation and θ_2 is the angle in the horizontal, measured anti-clockwise from an axis pointing west-east.

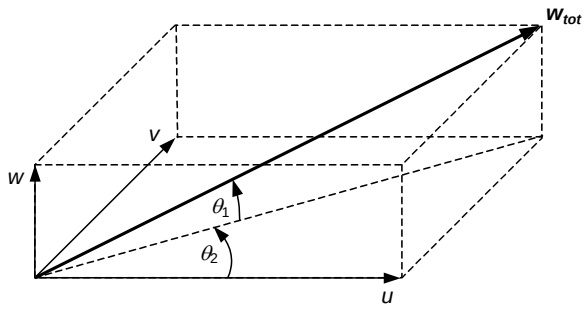


Figure 9.5 – Definition of release angle θ_1 and θ_2.

The streamwise, transverse and vertical components of the emission velocity (u, v and w) respectively, are then given by

$$u = w_{tot} \cos\theta_1 \cos\theta_2$$
$$v = w_{tot} \cos\theta_1 \sin\theta_2 \quad (9.14)$$
$$w = w_{tot} \sin\theta_1$$

Calculation of concentration

The ADMS 5 plume rise module was originally written for a release that was allowed to have non-zero longitudinal and transverse emission velocities. It is supplied with the calculated values of u, v and w.

In calculating concentrations, the transverse position of the plume centreline (y_p) is included in the Gaussian expressions. If the release has an initial upstream component of velocity the plume will turn back on itself as its velocity tends towards ambient. The plume centreline is double-valued for a range of upstream distances and in this case the concentrations are simply summed. No account is taken of any interaction between the two parts of the plume, i.e. the part with $u < 0$ and the part with $u > 0$.

Limitations

It is assumed that the plume trajectory differs from the mean wind direction by a small angle, e.g. less than 20°. It then follows that the plume cross-section may be approximated as perpendicular to the mean wind direction. This assumption will be valid in the far field where the cross-stream velocity v has become small, but not close

to the source if the initial value of v is large compared with the mean wind velocity.

9.6 Multiple sources

The user is allowed to enter data for up to 300 sources. Within that limit, up to 30 line sources, 30 area sources and 30 volume sources can be modelled.

A single source may emit up to 10 pollutants and a pollutant, if particulate, may consist of up to 10 different particle sizes.

9.6.1 Groups

Up to 20 user-defined groups can be entered in the user interface, and in addition a group containing all of the sources can be modelled. Each user-defined group may contain any of the sources, so that a source may be a member of more than one group, but a source can only appear once in any group. If **Radioactive decay** or **Fluctuations** are selected, only one group is permitted.

9.6.2 Sources

A source may be continuous (plume) or discrete (puff), and may be a point, area, volume, line or jet source if continuous, and a point or jet source if discrete. Each source may emit up to 10 pollutants. If the **Puff** option is selected, only one source may be modelled. If the **Fluctuations** option is used up to 50 sources may be modelled, of which 20 may be line, area or volume sources.

9.6.3 Pollutants

A pollutant is defined by a mass emission rate (plume) or total mass of the emission, start time and duration (puff), together with a units conversion factor. If the pollutant is particulate, up to 10 different particle sizes may be defined, but if gaseous, only one species may be defined.

9.6.4 Particle sizes and gases

For each particle size or gaseous pollutant the user is asked to define dry and wet deposition parameters, but these are only used if the **Dry deposition** and **Wet deposition** options on the **Setup** screen are selected.

9.6.5 Combining multiple sources into stacks

If multiple point sources are close together then they can behave as a single stack, for instance multiple flues in a single chimney. ADMS 5 contains the ability to enter multiple point sources and for the model to combine them into a single stack. The parameters of this combined stack are worked out from the parameters of the individual flues as follows:

- **Height**

  The heights of all the flues in the stack should be identical; the stack height is set to this height.

- **Position**

  The flues in a particular stack should be located very close together (a warning is issued if the stacks are greater than 20 m apart, and the model will stop with an error message if the stacks are greater than 50 m apart), and the stack location is taken to be the mean position of the flues.

- **Efflux**

  The total volume flow rate (at normal temperature and pressure, or 'NTP') is conserved. The individual volume flow rates at NTP are calculated for each flue, and the results summed to produce the volume flow rate at NTP for the combined stack.

- **Diameter**

  The total stack area is conserved. Hence the combined diameter is given by

  (9.15) $$D = \sqrt{\frac{4}{\pi}\sum_i A_i}$$

  where A_i are the areas of the individual flues.

- **Temperature**

  A weighted mean temperature is calculated, with the individual flue temperatures weighted by the individual mass fluxes.

- **Water content**

  The total mass emission rate of water is conserved. In ADMS 5, the plume water content is specified in terms of a mixing ratio R (i.e. kg of water per kg of dry release). The mass fraction of the release that is water is therefore $R/(1+R)$.

  The mass emission rate of water for each individual plume (Q_i) can be therefore be calculated as follows:

  (9.16) $$Q_i = \frac{R_i}{1+R_i} m_i$$

  where m_i is the mass flux of each individual plume (kg/s) and R_i is the mixing ratio of each individual plume.

  The mass fraction F of the total release that is water is therefore

(9.17) $$F = \frac{\sum_i Q_i}{\sum_i m_i}$$

and the mixing ratio for the combined plume is $F/(1-F)$.

- **Molecular mass**

  If the individual plume molecular masses are M_i, then it may be shown using the ideal gas equation that the molecular mass of the total plume M is given by

(9.18) $$M = \frac{\sum_i m_i}{\sum_i \frac{m_i}{M_i}}$$

  where m_i is the mass flux of each individual plume (kg/s).

- **Specific heat capacity**

  If the individual plume specific heat capacities are c_{pi}, the specific heat capacity of the total plume c_p is given by

(9.19) $$c_p = \frac{\sum_i c_{pi} m_i}{\sum_i m_i}$$

- **Emission rates**

  Pollutant emission rates are summed.

9.7 Output grids and points

'Gridded' output points defined in a horizontal plane can be specified in four different ways:

1. Cartesian grid with regularly spaced grid lines (shown in **Figure 9.6**);
2. Cartesian grid with variable spacing between the grid lines (shown in **Figure 9.6**);
3. cylindrical polar grid with regularly spaced values (shown in **Figure 9.7**); and
4. cylindrical polar grid with variably spaced values (shown in **Figure 9.7**).

With all grid types multiple output heights can be specified. Specified points can also be defined with respect to a Cartesian or cylindrical polar coordinate system. Up to 100 specified points can be defined in the interface, and additional points can be included in a model run by the use of an additional specified points (*.asp*) file.

The grid options apply to all types of model runs, including short-term average, long-term average, complex terrain, etc.

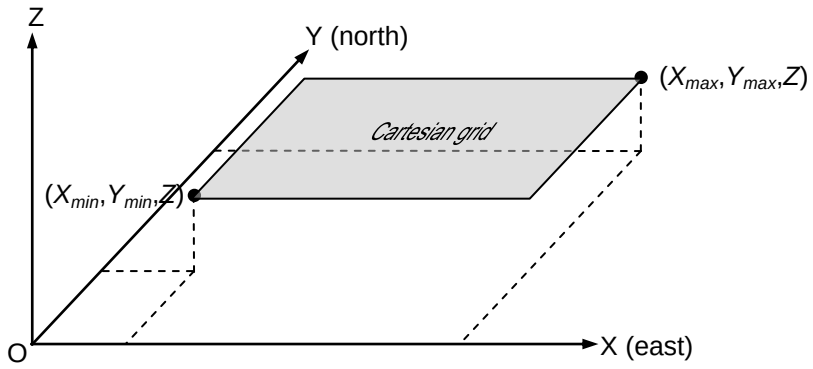


Figure 9.6 – Cartesian grids (cases 1 and 2) where (X,Y,Z) constitutes the coordinates of a point. OX is aligned due east and OY due north. Up to 2001 grid lines are allowed in the X and Y directions and up to 501 in the Z direction.

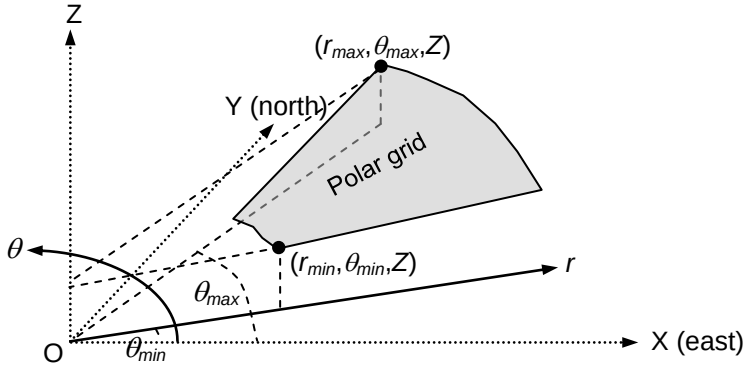


Figure 9.7 – Polar grids (cases 3 and 4) where r is the radial distance from the origin (0,0,0) (located in a Cartesian coordinate system) and θ is the angle between OX and r. θ = 0° points due east, θ = 90° due north. Up to 2001 grid lines are allowed in the r and θ directions and up to 501 in the Z direction.

9.8 Averaging times and statistics

9.8.1 Averaging times of one hour or longer

The mean concentration averaging time or sampling time (t_s) is entered for each pollutant in the **Output** screen.

If an averaging time greater than one hour is specified, it must be a whole number of hours and hourly sequential data must be supplied. The program will sum successive hourly averages to calculate the mean concentration and other statistics for the required averaging time.

The program can calculate rolling or running averages. For example, if concentrations are calculated for 24 hours of data, $\{C_i, i = 1,2,\ldots,24\}$:

- A **short-term**, 8-hour (non-rolling) average would be calculated as

  (9.20) after 8 hours of meteorological data: $C_{t_s=8} = \frac{1}{8}\sum_{i=1}^{8} C_i$

  (9.21) after 16 hours of meteorological data: $C_{t_s=8} = \frac{1}{8}\sum_{i=9}^{16} C_i$

  (9.22) after 24 hours of meteorological data: $C_{t_s=8} = \frac{1}{8}\sum_{i=17}^{24} C_i$

- The **long-term**, 8-hour (non-rolling) average over this 24-h period would be

  (9.23) $$\frac{\sum_{i=1}^{8} C_i + \sum_{i=9}^{16} C_i + \sum_{i=17}^{24} C_i}{8 \times 3}$$

- A **short-term** 8-hour **rolling** average would be:

  (9.24) before 8 hours of meteorological data: $C_{t_s=8}$ = no value

  (9.25) after 8 hours of meteorological data: $C_{t_s=8} = \frac{1}{8}\sum_{i=1}^{8} C_i$

  (9.26) after 9 hours of meteorological data: $C_{t_s=8} = \frac{1}{8}\sum_{i=2}^{9} C_i$

  (9.27) after 10 hours of meteorological data: $C_{t_s=8} = \frac{1}{8}\sum_{i=3}^{10} C_i$

  etc.

- The **long-term**, 8-hour **rolling** average would be the average of the 17 calculated short-term values, i.e. the short-term rolling averages from hour 8 to hour 24.

9.8.2 Averaging times shorter than one hour

The mean concentration averaging time or sampling time (t_s) is entered for each pollutant in the **Output** screen.

Averaging times up to and including one hour are used to calculate the component of lateral plume spread due to changes in the mean wind direction, unless σ_θ (standard deviation of the horizontal wind direction) is specified as a meteorological variable (described in Section 3.3). It might be appropriate to use an averaging time shorter than one hour as a first estimate to compare with an air quality standard that has an averaging time shorter than one hour. A better way to compare with an air quality standard shorter than one hour is to use an averaging time of one hour in the **Output** screen and to use the fluctuations module with a fluctuations averaging time shorter than one hour.

For averaging times greater than one hour, unless data are given by the user for σ_θ, the changes in mean wind direction within each hour of meteorological data are calculated using one hour.

9.8.3 Maximum daily output

Maximum daily output can be calculated for any averaging time. If maximum daily output is selected, for each day the maximum value of the averaging periods which end on that day is calculated.

9.8.4 Finite duration releases

Finite duration releases, or puffs, have a release duration of t_r. For an instantaneous release $t_r = 0$. Output from puff releases can be time dependent, i.e. instantaneous values, or time-integrated over the lifetime of the puff.

For time dependent output an appropriate averaging time is $t_s = 0$, although $t_s = 1$ hour is often used. If fluctuations are modelled the fluctuations averaging time is set to 0 seconds.

For time-integrated output a suitable averaging time is $t_s = t_r$ or $t_s = 1$ hour, whichever is longer. If fluctuations are modelled the fluctuations averaging time is automatically set by the program to be very large ($\rightarrow \infty$).

9.8.5 Limitations

The longest period for which a rolling average can be calculated is 72 hours (3 days). The longest period for which a non-rolling average can be calculated is 168 hours (1 week).

9.8.6 Long-term statistics

Long-term averages of hourly means and percentiles can be calculated using meteorological data which have been statistically analysed and therefore have a frequency or weighting attached to each meteorological condition, or from raw data which have not been statistically analysed. The latter data may be sequential or a number of unrelated meteorological conditions (although in this case the long-term average is unlikely to be useful), so an equal frequency or weighting is attributed to each hour's data. The model outputs long-term average *mean concentrations*, long-term average *mean deposition fluxes*, long-term average *mean activity (radioactivity)*, long-term average *percentiles of concentration*, and long-term average *exceedences per annum*.

For each set of meteorological data and associated frequency *f* (normalised by dividing the sum of the frequencies so that *f* lies between 0 and 1), the concentration at each point is calculated, say $C(x,y,z)$. Then the long-term *mean concentration* at that point, $C_{LT}(x,y,z)$, is given by the sum of Cf over all the data sets. Note that if statistical data are used, when only the wind direction or the precipitation rate (if wet deposition is not being calculated) changes between successive lines of the meteorological data file, the calculation on the local grid is not repeated. In these cases, the numbers output to the screen corresponding to the hour's meteorological data change by more than one between each calculation of concentration.

To calculate the concentration at each point corresponding to a specified *percentile p* at ground level, say $C_p(x,y,z)$, the values of $C(x,y,z)$ at one point for each combination of meteorological variables are considered along with their frequency of occurrence *f*. First of all the concentration values are arranged in descending order (i.e. highest at the beginning, lowest at the end) and the values of *f* rearranged accordingly. Then, starting at the highest concentration, the frequencies are summed until their cumulative value is $(100 - p)/100$. Percentile calculations may require a large amount of computing time and memory space, which is largely a function of the number of meteorological data sets.

Exceedences can also be calculated. In the same way as described above for percentiles, concentrations are ordered, along with their cumulative frequency. The number of concentrations above the exceedence value under consideration are calculated and output as an equivalent value of 'exceedences per annum'.

For example, consider a run of statistical data with 120 meteorological data lines. Take one output point where concentrations are greater than the exceedence value with a frequency of 12 times. The value output at this point for this exceedence is:

(9.28)
$$12 \times \frac{8760}{120} = 876$$

9.9 Marine boundary layer

ADMS 5 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 all sources and receptors should be over the sea; it is not suitable for coastal modelling.

9.9.1 Wind profile

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

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

where u_* (m/s) is the friction velocity, ν (m^2/s) is the kinematic viscosity of air, g is the acceleration due to gravity (m/s^2), α_m = 0.11 and α_{Ch} is the Charnock parameter, which has a default value of 0.018 but may be entered by the user, using the *.aai* file.

The velocity profiles used are the same as those used over the land, described in the ADMS 5 Technical Specification (CERC, 2015). The equations are solved by iteration.

9.9.2 Heat fluxes

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

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

where α_H = 0.4 and α_q = 0.62.

Then the sensible heat flux $F_{\theta 0}$ is given by

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

where c_p is the specific heat capacity of air (J/kg/K), ρ is the density of air (kg/m^3), 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 5 Technical Specification (CERC, 2015)), and for convective conditions ($1/L_{MO} < 0$) ψ_H is given by

(9.32) $$\psi_H = \ln\left[\frac{(1+y)^2}{(1+y_{surface})^2}\right]$$

where

(9.33) $$y = \sqrt{1-16\left(\frac{z+z_{0_H}}{L_{MO}}\right)} \quad \text{and} \quad y_{surface} = \sqrt{1-16\left(\frac{z_{0_H}}{L_{MO}}\right)}$$

The latent heat flux λE is given similarly by

(9.34) $$\lambda E = \frac{-\lambda\rho\kappa^2(q(z)-q_{sat0})u(z)}{\left[\ln\left(\frac{z+z_{0q}}{z_{0q}}\right)-\psi_q\left(\frac{z+z_{0q}}{L_{MO}}\right)\right]\left[\ln\left(\frac{z+z_0}{z_0}\right)-\psi\left(\frac{z+z_0}{L_{MO}}\right)\right]}$$

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 $\psi_q = \psi_H$. Again the equations are solved by iteration (Panofsky and Dutton, 1984).

9.9.3 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:

- wind speed and direction;
- temperature (°C);
- sea surface temperature (°C);

  Users are strongly recommended to include temperature data in runs using the marine boundary layer option. *However, if no temperature data are available, then users can select an option to assume that the marine boundary layer is neutral, in which case no heat fluxes are calculated.*

- 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 5 for regions not altogether remote from land (e.g. North Sea).

- height above sea level of temperature measurements (in metres) (if temperature data are included in run); and
- height above sea level of humidity measurements (in metres) (if humidity data are included in run).

9.10 Dry deposition

Dry deposition modifies the airborne concentration in two ways:

1. a reduction in plume strength (integrated flux of pollutant) *Q* with distance as material is removed from the plume at the surface, and
2. adjustment of the vertical profile because removal of material occurs at the surface. This is illustrated in **Figure 9.8**.

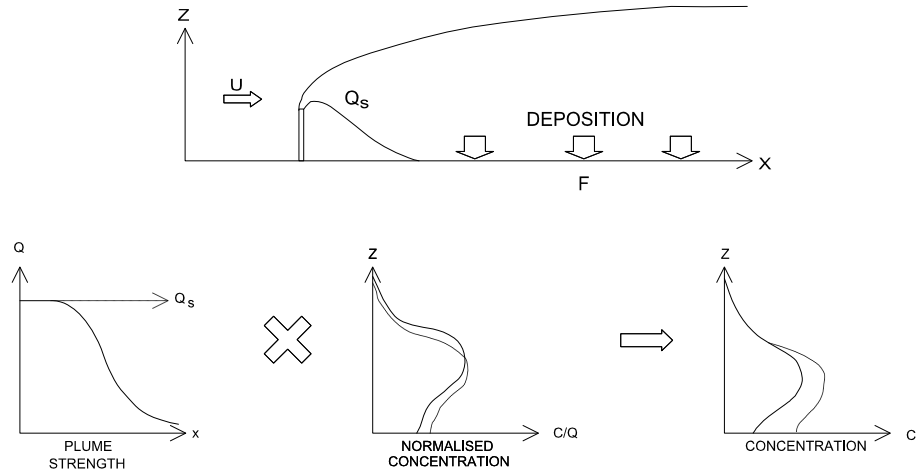


Figure 9.8 – Dry deposition modifies the vertical profile of concentration. Profiles of concentration and normalised concentration are shown.

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

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

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

(9.36) $$v_d = \frac{v_s}{1 - \exp(-v_s / v'_d)}$$

When v_s is zero, $v_d = v'_d$ and when v'_d is zero $v_d = v_s$. (These limits can be derived from equation (9.36), as well as making physical sense.)

One or both of v'_d and v_s may be known by the user and input directly to the model, or they can be estimated by the model on the basis of either gas type or particle size and density.

If v'_d is estimated by the model it is expressed as the reciprocal of the sum of three resistances:

(9.37) $$\frac{1}{v_d'} = r_a + r_b + r_s$$

where r_a is the aerodynamic resistance, r_b is the sub-layer resistance and r_s the surface layer resistance, which can either be calculated by the model or input by the user.

These resistances depend on the pollutant species, nature of the surface and the wind speed. For particles $r_s = 0$, while for gases r_s is either input by the user or estimated by the model as 30 s/m for reactive gases (e.g. HCl), 500 s/m for unreactive gases (e.g. SO_2) and infinite for inert gases.

The terminal velocity v_s is always zero for gaseous pollutants, while for particles it is estimated from the properties of the particle.

9.11 Wet deposition

The uptake of gases in clouds and rain, and their subsequent deposition at the ground in solution, is a complex kinetic process that must be simplified for application in practical models of wet deposition. The wet deposition module in ADMS 5 allows users not only to model the amount of pollutant being deposited at the ground in solution, but also the corresponding depletion of the plume strength, as indicated in **Figure 9.9**.

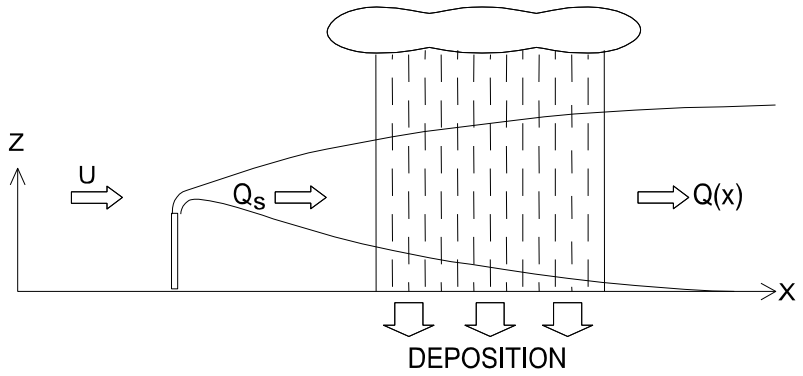


Figure 9.9 – Wet deposition model.

The type of wet deposition that can be modelled in ADMS 5 depends on the source and pollutant type:

- wet deposition of all pollutants from all source types can be modelled using a washout coefficient;
- wet deposition of SO_2 and CO_2 from all source types calculated using a washout coefficient is pH-limited;
- wet deposition of SO_2 and HCl from point sources can be modelled using the 'falling drop' method.

Using a washout coefficient to model wet deposition is a significant simplification of the processes involved, but the predicted values are relatively accurate when the uptake of pollutants is irreversible. However, when a pollutant is subject to significant 'out-gassing' (i.e. where the pollutant passes back from the droplet to the atmosphere due to relatively low air concentrations of the pollutant in the local vicinity), wet deposition predicted by the washout coefficient methodology may be significantly over-estimated. In such cases, where possible, it is advisable to use the 'falling drop' method.

The washout coefficient methodology is discussed in Section 9.11.1, and the 'falling drop' methodology is briefly described in Section 9.11.2; for full details of both methods, please refer to the ADMS 5 Technical Specification (CERC, 2015).

9.11.1 Washout coefficient method

When wet deposition is modelled through use of a *washout coefficient* Λ, the amount of material incorporated into any falling rain or precipitation is ΛC per unit horizontal area per unit vertical distance per unit time, where C is the local airborne

concentration.

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

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

The pollutant remaining in the plume, or plume strength, Q, therefore decreases with downwind distance.

The following simplifications are made:

- uptake of pollutants is irreversible; uptake in rain does not lead to a *redistribution* of material in the plume;
- all plume material lies in or below the rain cloud; furthermore no distinction is made between in-cloud scavenging (*rainout*) or below-cloud scavenging (*washout*);
- solution in raindrops does not lead to saturation except for SO_2 and CO_2 where a pH-limited value is calculated;
- the rainfall rate is constant and uniform over the area of calculation.

The washout coefficient Λ is dependent on a large number of parameters, including the nature of the pollutant, the rainfall rate, the droplet size distribution and the pollutant concentrations in the air and in the raindrops. A value for Λ may be entered by the user or estimated by the system, in one of the following ways:

- by specifying a constant value Λ, which is independent of the precipitation rate input to the meteorological input module, i.e. wet deposition will be calculated even when the precipitation rate is zero or if no precipitation is specified; or
- by specifying constants A and B which give a washout coefficient dependent on precipitation rate of the form

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

  The default values of A and B are $A = 10^{-4}$ and $B = 0.64$.

For some species, the amount of material removed by wet deposition may be limited by its solubility in rainwater and the pH of the raindrops. For acidic species such as SO_2, this has a significant effect near the source, where high gaseous concentrations and the low pH of the raindrops limit the uptake of material into the raindrops. In ADMS 5, a set of solubility coefficients and a chemical reaction scheme are used to calculate acidity within the raindrops and thus the effective solubility of SO_2 and CO_2. An effective washout coefficient (Λ_{eff}) is then determined by estimating the amount of material absorbed by the raindrops.

If this pH-limited washout coefficient Λ_{eff} is less than the value of Λ specified in the interface, the wet deposition flux is assumed to be pH-limited and Λ_{eff} is used. Otherwise the usual washout coefficient Λ is used.

9.11.2 'Falling drop' method

After passing through the plume centreline, raindrops often fall through some relatively 'clean' air before reaching the ground. If the pollutant in question undergoes out-gassing (i.e. where the concentration gradient between the drop and the air causes the pollutant to pass back into gaseous form), the pollutant concentration within the drop at the ground will be reduced relative to the maximum concentration that was achieved at, or in fact often just below, the plume centreline. The gas of most interest that undergoes significant out-gassing before the drop reaches the ground is SO_2. **Figure 9.10** gives an example vertical concentration profile as the raindrop travels vertically through the plume.

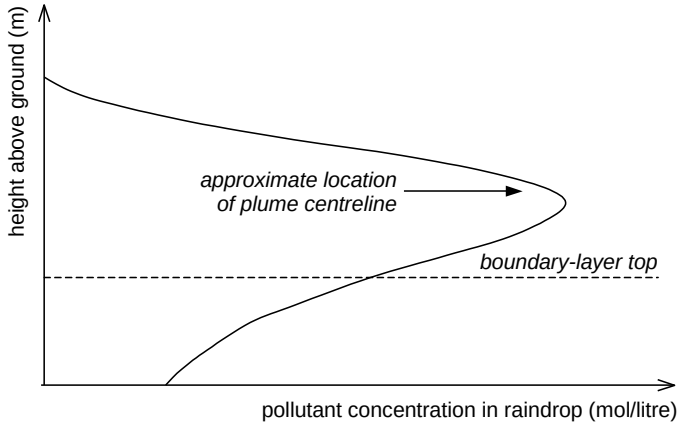


Figure 9.10 – Pollutant concentration within a raindrop falling vertically through the plume, when significant out-gassing occurs below the plume centreline.

The major features of this process may be examined by using a simple model of a rain drop falling through a vertical profile of gas with the assumption of kinetically controlled gas uptake and rapid solution equilibration.

As mentioned in Section 9.11.1, the uptake of acidic pollutants such as SO_2 is limited at high concentrations by the limiting effect of the raindrop pH. In practice, the pollutant that affects SO_2 concentrations most significantly is HCl. In addition, any dissolved CO_2 and other pollutants in the 'clean' rain entering the top of the plume affects the initial pH, and consequently the initial uptake of both SO_2 and HCl. Therefore, the 'falling drop' method has been implemented in ADMS 5 in the following way:

- the kinetic and thermodynamic processes of SO_2 and HCl are modelled as a coupled system; and
- the user-entered pH of the rain entering the top of the plume is used to calculate a constant value of [H+] in solution that represents the dissolved CO_2 and other pollutants in 'clean' rain.

9.12 Radioactive decay

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

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

where N_i is the number density (mol/kg) of isotope i in the emission, λ_i is the decay constant of isotope i ($\lambda_i = (\ln 2)/(T_{1/2,i})$ where $T_{1/2,\,i}$ is the half-life of isotope i), and f_{ij} is the fraction of isotope j decaying into isotope i.

The user may enter up to 10 parent isotopes in any one run, and up to 50 isotopes, including decay products, will be output. Half-lives (in seconds) of 825 isotopes are contained in the file *Halflife.dat* and decay information in *Decay.dat* provided with the model (Lederer *et al.*, 1968; Gray, 1972 ; Nuffield Science Foundation, 1984; Dzhelepov and Peker, 1961). Both of these files are required by the module. The data are presented in a readable format in the file *Isotopes.pdf*.

9.12.1 Limitations

Calculations are restricted to the isotopes listed in the file *Isotopes.pdf*.

9.13 Gamma ray dose rate

Gamma dose rate due to both airborne and deposited radioactive material may be calculated at all ground level output points.

9.13.1 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

$$\Phi_i(r) = \iiint \frac{f(E_i)C(E_i,r)\,B(E_i,\mu_i|r-r'|)\,e^{-\mu_i|r-r'|}}{4\pi(r-r')^2}\,d^3r' \tag{9.41}$$

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

$$B(E_i,\mu_i r) = 1 + a(E_i)\mu r\, e^{b(E_i)\mu r}\,. \tag{9.42}$$

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

The total dose rate is obtained by summing over all energy groups, and the accumulated gamma dose rate in space is obtained by summing the contributions from all isotopes. 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

$$D_{\text{air}} = \sum_{i=1}^{n} C_{bi} M_{ai} E_i \Phi_i\,. \tag{9.43}$$

The input data are the concentration fields for the different isotopes, which are obtained from the radioactive decay calculations, and the gamma dose database file *gdatbase.dat* which, for each isotope emitting gamma rays, lists the decay constant (days^{-1}), the number of energy levels, the energy of each branch (MeV) and the branching ratios.

In evaluating the integral for $\Phi(r)$, two approximations can be made. The result for the gamma ray dose rate from a semi-infinite cloud is used when the cloud is large compared to the mean free path of the gamma rays; and if the plume is narrow and elevated, the integral is evaluated as a line source.

9.13.2 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:

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

where for each energy level

(9.45) $$F_i = \frac{1}{4\pi} \int_0^{2\pi} \int_0^{\infty} \frac{f(E_i) \times F_{dr}(E_i, r) \times B(E_i, \mu_i r) e^{-\mu_i r}}{r} dr d\theta$$

and F_{dr} is the rate of activity of deposited material (Bq/m^{2}s).

9.13.3 Limitations

Strictly speaking, the calculation of the gamma ray dose rate due to deposited radioactive material should take into account the decay of material *after* it has been deposited. However, in the ADMS calculations, this effect has been ignored for simplicity. This approach is satisfactory if the isotopes of interest have half-lives much longer than the period of interest.

9.14 Plume visibility

The plume visibility module uses the initial water content of the release and the humidity of the ambient air to determine whether the plume will be visible at each downstream distance. In addition, when plume visibility is chosen as a model option, the effect of water on the plume density and the heating and cooling effects of condensation and evaporation are taken into account in the plume rise module.

As input, the module requires:

- surface humidity (as specific humidity q_0 or relative humidity RH_0) to be entered as a meteorological variable;
- surface temperature to be entered as a meteorological variable, since the moisture properties of the atmosphere depend strongly on temperature; and
- the initial water vapour mixing ratio of the plume in kg/kg (i.e. the mass of water vapour per unit mass of dry release at the source).

Output includes the length and maximum liquid water content of each visible plume and, for long-term calculations, the frequency of visible plumes and visible plume 'groundings'.

9.14.1 Visibility criteria

The plume visibility is calculated based on the liquid water content of the plume. The visibility of the plume is calculated based on two criteria:

- whether the liquid water is above a critical threshold (0.002 kg/kg), in which case it is visible; otherwise,
- how the plume size compares to the optical visible length for that water content.

For the second criterion, the optical visible length in metres is calculated using the relation given by Gultepe *et al.*:

(9.46) $$OptVisLen = \frac{1002}{\left(LWC.N_d\right)^{0.6473}}$$

where LWC is the liquid water content and N_d is the number of droplets in the plume, which is set to 2000 cm^{-3}, based on typical cloud condensation nuclei concentrations.

The plume is counted as visible if the plume thickness directly above a receptor is greater than the optical visible length, i.e.:

(9.47) $$2\sigma_z \, fg\left(y\right) > OptVisLen$$

where $2\sigma_z$ is the plume depth at the plume centreline, f is an opacity factor (for which a value of 1 has been found to produce the best results) and $g\left(y\right)$ is a Gaussian function to take account of the variation in plume depth away from the plume

centreline.

The liquid water threshold, number of droplets in the plume and opacity factor can be altered using an *.aai* file, refer to Section 4.5.2 for more details.

In addition two criteria based on the ambient relative humidity are applied:

- if the ground level ambient relative humidity is 100% the conditions are taken to be foggy and the plume is not visible,
- if the ambient humidity at the plume height is greater than 98% then the plume is taken to be indistinguishable from cloud and therefore not visible.

The 98% ambient relative humidity threshold used to indicate that the plume is indistinguishable from cloud can be altered using an *.aai* file, refer to Section 4.5.1 for more details.

The plume is defined as 'grounded' if the vertical plume spread parameter σ_z is larger than the plume centreline height z_p.

9.14.2 Temperature and humidity output

If the **Plume visibility** option is selected, ADMS 5 can output the plume-affected temperature, specific humidity and/or relative humidity at each output point. Exceedences of user-input thresholds of these parameters can also be output, as well as simultaneous exceedences of pairs of threshold values of temperature and specific humidity and/or temperature and relative humidity.

This option may be of particular interest to users interested in the dispersion of live micro-organisms such as Legionella. These micro-organisms are sensitive to the conditions within the plume. For example, they may only survive if the temperature and/or relative humidity are above or below certain threshold values.

9.14.3 Limitations

Plume visibility calculations are currently restricted to non-passive releases (i.e. sources with some plume rise).

If both the **Plume visibility** and **Buildings** options are selected, the effect of the building(s) is not taken into account in the plume visibility calculations.

ADMS 5 may not be suitable for modelling plumes from large cooling towers, due to the large size of the source, and the effect which the cooling tower structure itself may have on dispersion.

9.15 Chemistry

The chemical scheme in ADMS 5 uses the reaction rates from the Generic Reaction Set of equations (GRS) (Azzi and Johnson, 1992; Venkatram *et al*., 1994), a semi-empirical photo-chemical model which reduces the complicated series of reactions involving NO (nitric oxide), NO_2 (nitroden dioxide), O_3 (ozone) and many hydrocarbons to seven reactions.

In ADMS 5, hydrocarbons are not included in the chemical scheme, which then reduces to just two equations, namely

$$\begin{aligned} NO_2 + h\nu &\rightarrow NO + O_3 \\ NO + O_3 &\rightarrow NO_2 \end{aligned} \tag{9.48}$$

where hν is ultra-violet radiation. Equations (9.48) represent exact chemical reactions, which happen very quickly.

The model uses background concentrations of ozone, NO_X (nitrogen oxides) and NO_2. These values should be rural background values, i.e. values that do not take into account the effect of the sources being modelled, to avoid "double-counting". During the day, these background values will be adjusted in order to ensure that they are in equilibrium. This is done by assuming:

- conservation of mass for NOx;
- conservation of the total mass of NO_2 and O_3; and
- photo-stationary equilibrium for NO_2, NO and O_3.

During the night no adjustments are made to the background values. A local night-time chemistry option is available, refer to Section 4.8.6. With this option, the chemistry scheme is applied to the NO and NO_2 values before the background values of these pollutants are added on. This allows the effects of chemistry to be taken into account, without incorrectly affecting the background concentrations.

At each receptor the NO_X may have originated from more than one source. A weighted mean age of pollutant is calculated for each downwind receptor. At each receptor the minimum non-zero pollutant age is used in the chemistry as the time over which the reactions occur.

9.15.1 Limitations

The chemistry scheme does not include a reaction between NO and O_2 and so may not be applicable for near ground level plumes with high concentrations (ppm) persisting over several hundreds of metres downstream.

9.16 Buildings

The building effects module is used to calculate the dispersion of pollution from sources near large structures, up to a distance of about 60 building heights. The ADMS 5 model of building effects has the following features.

- Up to 25 cuboidal or cylindrical buildings may be defined by the user in terms of their height, length, width and orientation (the latter two parameters are disregarded for cylindrical buildings). A main building is defined for each source (refer to Section 4.9 for advice on the choice of main building.) Then, for each wind direction the buildings are reduced to a single cuboidal effective wind-aligned building whose height is a function of the height of the main building (see Section 9.16.1).
- The disturbed flow field consists of a recirculating flow region or cavity in the lee of the building, with a diminishing turbulent wake downwind.
- Concentrations within the well-mixed recirculating flow region are uniform and based upon the fraction of the release that is entrained.
- The concentration at a point further downwind is the sum of two contributions: a ground-level plume from the recirculating flow region and an elevated plume from the non-entrained remainder. The turbulent wake reduces plume height and increases turbulent spread.
- The concentration and deposition are set to zero within the user-defined buildings.

The building effects module interacts with the rest of ADMS 5, using the standard concentration profiles, but with modified plume height and plume spread. The stages in the analysis of building effects are illustrated in **Figure 9.11**, while **Figure 9.12** shows how a complex of buildings is treated.

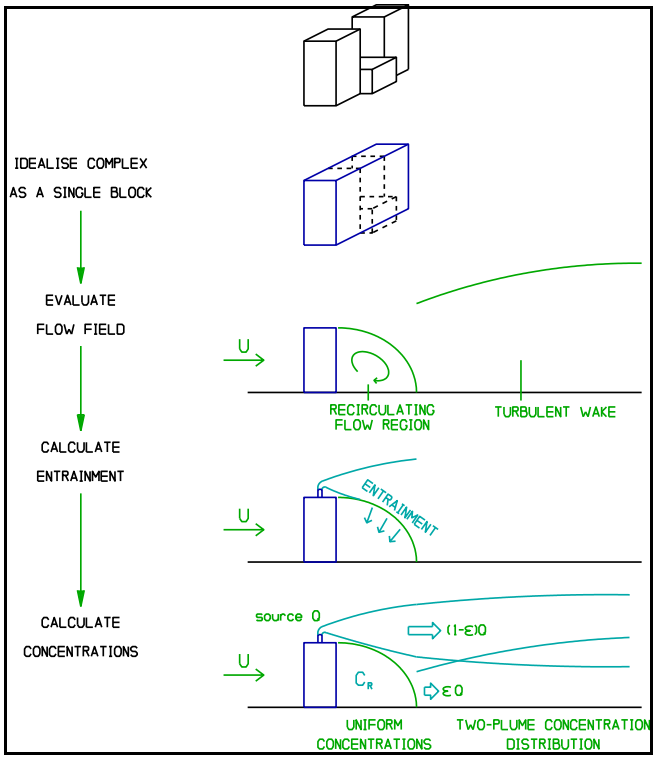


Figure 9.11 – Stages in the analysis of building effects.

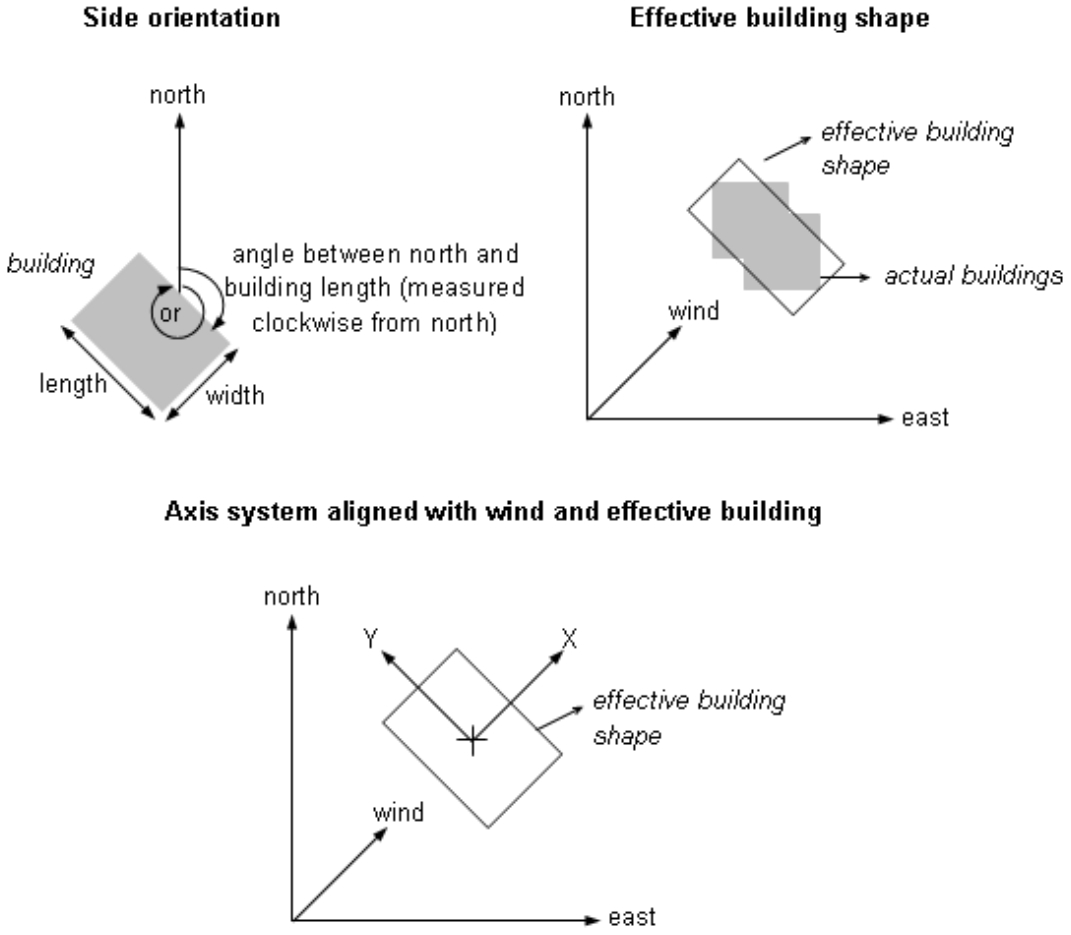


Figure 9.12 – Building effects module definitions.

9.16.1 Determination of the 'effective building'

The effective building is derived by the following algorithm:

1. Circular buildings are converted to 'equivalent' square blocks, with the same centre as the input circular building and side length $D_i/\sqrt{2}$, oriented such that the wind is normal to the building face.
2. Any buildings of height, H_i, less than a fraction $1/\alpha$ of the source height are ignored, where

   (9.49) $$\alpha = 1 + 2\,\min\left(1, \frac{W_i}{H_i}\right)$$

   where W_i is the crosswind width of building i.
3. Any buildings that are greater than a certain distance from the plume centreline in the crosswind direction are ignored.

   Specifically, a building will be ignored if all its vertices are greater than $0.5\,\sigma_y(|x|)$ from the plume centreline in the crosswind direction, where x is distance from the source in the alongwind direction, and $\sigma_y(x)$ is the horizontal

plume spread (not including building effects) at distance *x* downwind of the source (see example in **Figure 9.13**).

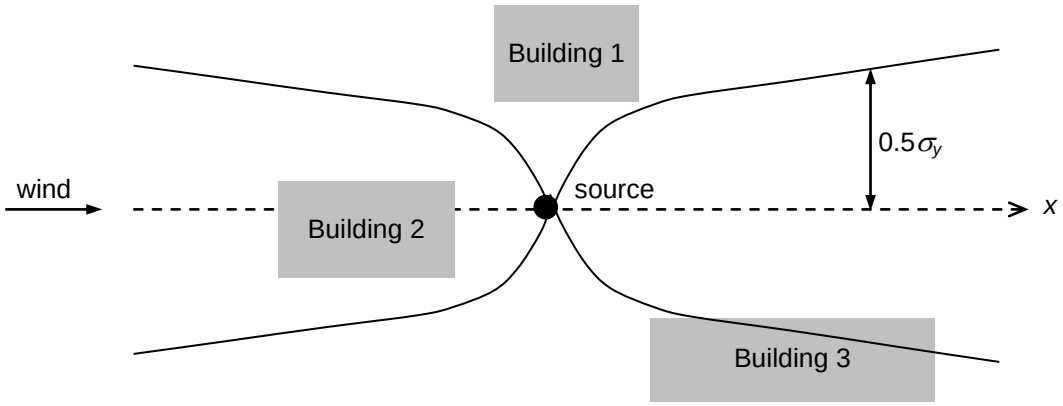


Figure 9.13 – Example source and building configuration. Buildings 2 and 3 will be included in the effective building, but building 1 will not.

4. The user specifies which is the main building; H_B and θ_B are the height and orientation of this unit.

   Multiples of 90° are added or subtracted until $-45° < \theta_B \leq 45°$. A different main building may be selected for each source. If the main building is too far from the plume centreline, according to 3 above, then an alternative main building is automatically selected. The new main building will be that with its centre closest to the source, of those that are able to be modelled according to 3. If the main building is not tall enough to be considered, according to 2 above, then no buildings are modelled for that source.

5. A subset Σ is then defined by the main building plus all other buildings (a) that are at least $0.5\,H_B$ high and (b) whose projected crosswind and along-wind separations from another subset member do not exceed half the projected crosswind width of the main building.

6. In the general case where Σ includes more than one building, W_E is the projected crosswind width and L_E is the along-wind projection from the furthest upwind mid-face to the furthest downwind mid-face (see **Figure 9.14**).

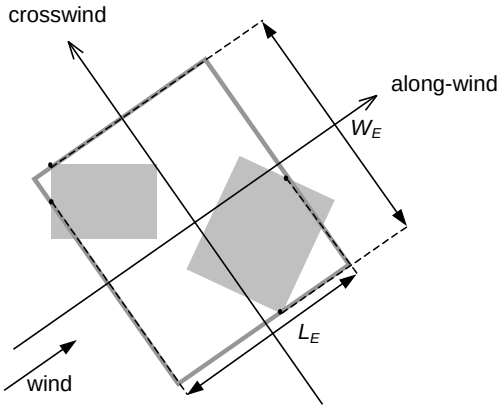


Figure 9.14 – Effective building for case where Σ includes two buildings. The grey shaded rectangles represent the user-input buildings and the grey open rectangle represents the effective building.

7. In the special case where Σ only includes one building, $L_E = min(L_F, L_D)$ where L_D is the along-wind length of the building, as seen when travelling along the wind direction, and L_F is the along-wind projection from the furthest upwind mid-face to the furthest downwind mid-face as described in 6 (see **Figure 9.15**).

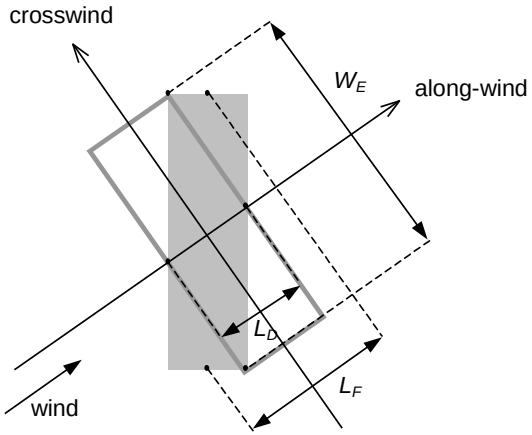


Figure 9.15 – Effective building for case where Σ only includes one building. The grey shaded rectangle represents the user-input building and the grey open rectangle the effective building.

9.16.2 Limitations

The buildings module is based on experiments in which there was one dominant site building and several smaller surrounding buildings less important for dispersion.

9.17 Complex terrain

The complex terrain module applies a three-dimensional flow and turbulence field to the dispersion modelling calculations. When variable terrain height is modelled there are two possible flow field solutions. The flow field solution used depends on the meteorological conditions. The critical parameter is the maximum height of the dividing surface, H_c, described in Section 9.17.2. The critical dividing surface height is 20 m.

Table 9.5 below summarises the dependence of the model behaviour on the maximum dividing surface height. Note that when modelling variable terrain height for very stable flows, the stable flow solution does not account for any spatial variation of surface roughness.

Complex terrain modelling option	$H_c > 20$m	$H_c \leq 20$m
Variable terrain height, with or without variable surface roughness	Very stable flow solution (assuming a constant roughness value) described in Section 9.17.2	FLOWSTAR-D solution
Variable surface roughness only	FLOWSTAR-D solution	FLOWSTAR-D solution

Table 9.5 – Summary of model behaviour when running the complex terrain option.

9.17.1 FLOWSTAR-D solution

In this case the flow field and turbulence values used in the dispersion modelling calculations are those output from the FLOWSTAR-D (F*) model. FLOWSTAR models air flow and dispersion over hills with roughness changes, including the effects of stratification. The model is described in some detail in Carruthers *et al*. (1988) while the basis of the theory is presented in Hunt *et al*. (1988b, 1988c) and Hunt (1985).

The airflow model uses linearised analytical solutions of the momentum and continuity equations, and includes the effects of stratification on the flow. If modelled, ideally hills should have moderate slopes (say less than 1 in 3) but the model is useful even when this criterion is not met. The terrain height is specified at up to 66,000 points which are interpolated by the model onto a regular grid of up to 256×256 points. The best results are achieved if the points at which the elevation and/or surface roughness are specified are regularly spaced. The effects of roughness and terrain can be modelled separately. The airflow model has been extensively tested with laboratory and field data.

9.17.2 Very stable flow solution

When very stable flows approach an isolated hill, the flow may divide, with the air above a certain height H_c (the *dividing surface*) flowing over the hill in a terrain-following manner and the air below the dividing surface flowing around the hill in a 2-dimensional flow. This is shown in **Figure 9.16**.

The maximum height of the dividing surface, H_c, is defined by an energy balance

equation, which locates the lowest height at which the kinetic energy of an air parcel in the flow approaching the hill is equal to the potential energy attained by elevating an equivalent fluid parcel from this height to the top of the hill:

$$\frac{1}{2}U^2(H_c) = \int_{h_c}^{h_{\max}} N^2(z) \times (h_{max} - z)\, dz \,. \tag{9.50}$$

The flow below the dividing surface is two-dimensional potential flow apart from upstream flow in a thin layer close to the terrain. Above the dividing surface, the FLOWSTAR-D solution is used except close to the ground where a weighted average of the FLOWSTAR-D and terrain-following flow fields is used.

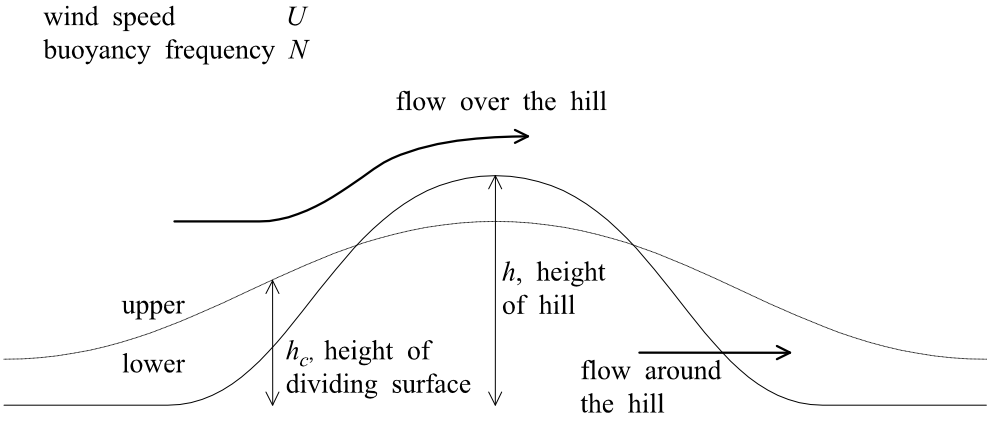


Figure 9.16 – Effect of a hill on the flow in very stable conditions. H_c is the maximum height of the dividing surface.

9.17.3 Regions of reverse flow

ADMS 5 models plume dispersion from sources located in a reverse flow region. Regions of reverse flow occur when there is a component of the local velocity in the opposite direction to the upstream mean wind velocity. Regions of reverse flow sometimes occur in flows over complex terrain either upstream or downstream of hills, depending on the meteorological conditions and the terrain gradients. The flow field output from FLOWSTAR shows these regions of reverse flow.

If the source is in a region of reverse flow, then the pollutant becomes ‘well mixed’ within that region, i.e. it has a constant concentration, and the plume flows out of the region from the downstream edge.

9.17.4 Running the complex terrain module

During a model run the following calculations take place:

- interpolation of terrain and roughness data (if variable) onto the local calculation grid;
- calculation of mean air flow;
- calculation of shear stress and turbulence;
- calculation of mean streamline through source; and

- calculation of concentration distribution.

It is not recommended that the model be used unless hill slopes are greater than about 1 in 10 or there are large surface roughness changes.

The model can be used for very large changes in surface roughness. An order of magnitude variation in the surface roughness is allowable with the approximations made in the calculation.

9.18 User-input 3D flow field

For this model option, the flow field and turbulence velocity values are used exactly as entered by the user. However, it was found that the turbulent kinetic energy (TKE) values in the example mesoscale flow field datasets, which were used during validation of this module, were significantly lower than those usually used in ADMS 5. This may be because ADMS 5 TKE includes some flow field effects such as convective eddies, which in the mesoscale data are shown in the vertical wind component rather than the turbulence.

For this reason, prior to use in the ADMS 5 dispersion algorithms, the local user-input values of TKE ($TKE_{user}(x,y,z)$) are adjusted to make them a similar order of magnitude to the ADMS 5 TKE values ($TKE_{adjusted}(x,y,z)$). This is done in the following way:

$$TKE_{adjusted}(x,y,z) = \frac{\overline{U}_{ADMS}(z)}{\overline{U}_{user}(z)} \frac{\overline{TKE}_{ADMS}(z)}{\overline{TKE}_{user}(z)} TKE_{user}(x,y,z) \quad (9.51)$$

where $\overline{U}_{ADMS}(z)$ and $\overline{TKE}_{ADMS}(z)$ are the mean ADMS 5 wind speed and TKE values at a particular height z, assuming neutral conditions, and $\overline{U}_{user}(z)$ and $\overline{TKE}_{user}(z)$ are the corresponding values calculated from the user-input data.

The ADMS 5 turbulent velocities σ_u, σ_v and σ_w, and a local value of the frictional velocity u_{*local} are then derived from the adjusted TKE values. The way this is done depends on the stability of the upwind meteorological conditions. That is:

- for **stable/neutral conditions** ($h/L_{MO} \geq -0.3$):

$$(\sigma_u, \sigma_v, \sigma_w) = (2.5, 2.0, 1.3)\sqrt{\frac{TKE_{adjusted}}{11.94}} \quad (9.52)$$

and

$$u_{*local} = \begin{cases} \sqrt{\dfrac{TKE_{adjusted}}{11.94 T_{wN}^2}} & \text{for} \quad -0.3 \leq \dfrac{h}{L_{MO}} < 1 \\ \sqrt{\dfrac{TKE_{adjusted}}{11.94\left(1-\alpha_s \dfrac{z_{min}}{h}\right)^{3/2}}} & \text{for} \quad \dfrac{h}{L_{MO}} \geq 1 \end{cases} \quad (9.53)$$

where z_{min} is the lowest input height and the parameter α_s is used to represent the changing characteristics depending on whether conditions are ideal (i.e. no waves or instabilities) or disturbed (as defined in the Technical Specification document P09/01, CERC, 2015).

- for **convective conditions** ($h/L_{MO} < -0.3$), it has been assumed that the ratio (R) of the convective velocity scale w_* to the frictional velocity u_* for the user-input data is the

same as that for the ADMS 5 upwind meteorological data. Then, if

$$R_{upwind} = \frac{w_*}{u_*} \tag{9.54}$$

a local value of u_* can be defined in the following way:

$$u_{*local} = \sqrt{\frac{TKE_{adjusted}}{R_{upwind}^2\left(0.6 + 0.4T_{wC}^2\right) + 11.94T_{wN}^2}} \tag{9.55}$$

where this expression is evaluated at the minimum user-input vertical height z_{min} and

$$T_{wC}(z_{\min}) = 2.1\left(\frac{z_{\min}}{h}\right)^{1/3}\left(1 - 0.8\frac{z_{\min}}{h}\right) \tag{9.56}$$

$$T_{wN}(z_{\min}) = 1 - 0.8\frac{z_{\min}}{h} \tag{9.57}$$

Then, the turbulent velocities σ_u, σ_v and σ_w can be defined in the following way:

$$\begin{aligned} \sigma_u^2 &= u_{*local}^2\left(0.3R_{upwind}^2 + 6.25T_{wN}^2\right) \\ \sigma_v^2 &= u_{*local}^2\left(0.3R_{upwind}^2 + 4.0T_{wN}^2\right) \\ \sigma_w^2 &= u_{*local}^2\left(0.4R_{upwind}^2T_{wC}^2 + 1.69T_{wN}^2\right) \end{aligned} \tag{9.58}$$

Finally, note that the derived turbulent velocities σ_u, σ_v and σ_w are subject to a minimum value of turbulence that ranges between 0.01 and 0.2 m/s, depending on the user input minimum value of L_{MO}. This accounts for the fact that in urban areas (high minimum L_{MO}), where conditions never become very stable, there will always be some turbulence. Alternatively. the user may set a constant value of minimum turbulence, as described in Section 4.22.

9.19 Coastline

The coastline module models the case where the presence of a coastline or other thermal change has a large impact on surface concentrations over the land. This may occur during the day when there is advection of stably stratified air from the sea on to the heated land surface, with development of an internal convective layer (Venkatram, 1977) – see **Figure 9.17**. The module is invoked if the following conditions are all satisfied:

1. the sea is colder than the land,
2. meteorological conditions on land are specified as convective,
3. the wind is blowing onshore (i.e. from the sea to the land).

A release into the stable layer mixes only slowly but eventually it is advected into the convective layer and then is rapidly mixed down to the surface where large surface concentrations may occur.

The input required when modelling a coastline includes the usual meteorological data over the land and, in addition, the same parameters over the sea. These are estimated from the parameters over the land and the sea surface temperature (or sea-land temperature difference), which must be one of the variables in the data file. The rate of growth of the internal boundary layer is assumed to be independent of wind direction provided the air flows from sea to land.

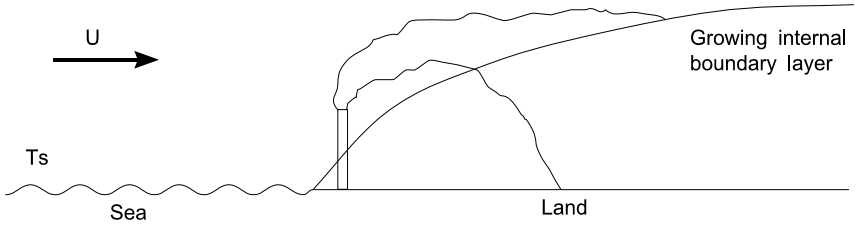


Figure 9.17 − Effect of coastline on plume.

9.19.1 Limitations

It is assumed that the coastline is straight near the location of interest.

The module has not been validated against field data.

9.20 Puffs

For instantaneous or finite duration releases the puff model is used, which models the release in terms of three separate regions: front end, plateau (centre) and rear end.

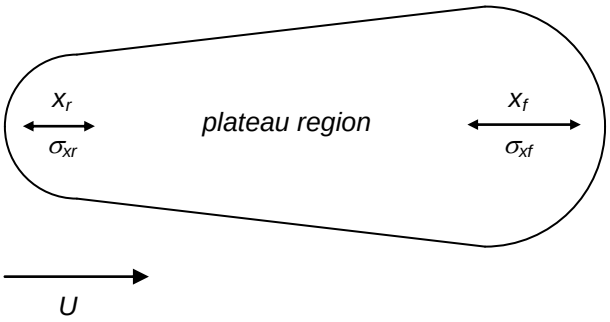


Figure 9.18 – Schematic diagram of puff.

1. The **front end** longitudinal spread is given by

(9.59) $$\sigma_{xf}^2 = \sigma_u^2 (t - t_f)^2 + \left(\frac{1}{2} \sigma_z (t - t_f) \frac{\partial U}{\partial z} \right)^2$$

where t_f is the time at which the release commenced. σ_u and $\partial U/\partial z$ are evaluated at $z_m(x_f)$, the mean height of the puff at the front end $x = x_f$.

2. In the **plateau (centre region)** of the puff $\sigma_x = 0$ and longitudinal gradients of the mean concentration are caused only by diffusion in the vertical and the transverse directions. This region is eventually completely eroded by longitudinal dispersion from the front and rear as σ_{xf} and σ_{xr} increase.

   In the plateau region the dispersion parameters σ_y, σ_z and the centreline height vary with distance downwind. These parameters are calculated by the same algorithms used for the plume model.

3. The **rear end** longitudinal spread is given by

(9.60) $$\sigma_{xr}^2 = \sigma_u^2 (t - t_r)^2 + \left\{ \frac{1}{2} \sigma_z (t - t_r) \frac{\partial U}{\partial z} \right\}^2$$

where t_r is the time that the release ends, and σ_u and $\partial U/\partial z$ are evaluated at the mean plume height $z_m(x_r)$, where x_r is the distance downwind that the rear of the puff has travelled.

Both turbulence and shear contribute to longitudinal spread σ_x in regions (1) and (3).

For an instantaneous release the start and end times of the release are identical, $t_f = t_r$, and therefore there is no plateau region.

9.21 Fluctuations

The fluctuations module produces statistics of fluctuations in concentration. Both turbulence and changes in meteorology cause variations in concentration from the mean value, i.e. fluctuations. The fluctuations module takes both sources of variation into account, although the only change in meteorology considered is the meandering of the mean wind. For short sampling times of an hour or less, the boundary layer turbulence is usually the dominant cause of fluctuations, while for longer times, changes in mean wind direction can become important. Meteorological parameters other than the wind direction are assumed to remain constant over the sampling time.

For further discussion of fluctuations see CERC Technical Note on turbulence, fluctuations and averaging times (CERC, 1998).

9.21.1 Calculation procedure

The user enters an averaging time or sampling time t_s for mean concentration calculations, and a fluctuations averaging time t_{avf}. The sampling time t_s determines the lateral spread of the release due to changes in mean wind direction, unless meandering σ_θ is entered as a meteorological variable. It should therefore reflect the frequency of the meteorological observations, usually 1 hour. Results from the fluctuations module depend on t_s, t_{avf} and the type of release, i.e. plume, time-integrated puff or time dependent puff.

1. Plume (continuous release)

   The fluctuations module estimates the probability that the "concentration averaged over the period t_{avf}" exceeds a particular value c, so that the output is a function of (x, y, z, c), which can be output in the variety of ways explained in Section 9.21.2. The results are equivalent to those obtained by making many measurements over the period t_{avf}, during the sampling time t_s, i.e. over a period where the meteorology remains constant but the boundary layer turbulence varies.

   If the user selects the calculation of toxic response with a power p_{dose}, the module calculates the ensemble average of the $(p_{dose})^{th}$ power of the concentration. This quantity is often used in calculating the effect of exposure to concentrations of toxic substances (Griffiths, 1991).

2. Time-integrated puff

   The fluctuations module calculates the probability that the time-integrated concentration exceeds a particular value $\hat{c}$, so the output is a function of $(x, y, z, \hat{c})$. The results are equivalent to those that would be obtained by making time-integrated measurements for a large number of identical puff releases, under identical meteorological conditions, but with different boundary layer turbulence. Note that in this case t_{avf} is not required since time-integrated concentrations are being considered.

   Toxic dose calculations are not meaningful for time-integrated puffs.

3. Time dependent puff

   The fluctuations module calculates the probability that the *instantaneous* concentration exceeds a particular value of $\hat{c}$, i.e. $t_{avf} = 0$. Output is therefore a function of $(x, y, z, t, \hat{c})$. The results are equivalent to those that would be obtained by making instantaneous concentration measurements for a large number of identical puffs released under identical meteorological conditions but with different boundary layer turbulence.

   Toxic response is the ensemble average of the $(p_{dose})^{th}$ power of the instantaneous concentration.

Note that the predictions of fluctuations for case (3) are likely to be subject to greater errors than in cases (1) and (2), as a result of the very limited experimental data available with which to test the theory.

9.21.2 Short-term output

For short-term runs the results are output in three ways:

1. As concentration standard deviation σ_c.
2. As a probability distribution function (see **Figure 9.19**), where up to 10 concentrations are selected either automatically, by the program, in which case they are different for each receptor point, or specified by the user, in which case they are constant for all receptors. The output is the probability of the concentration *exceeding* these concentrations.

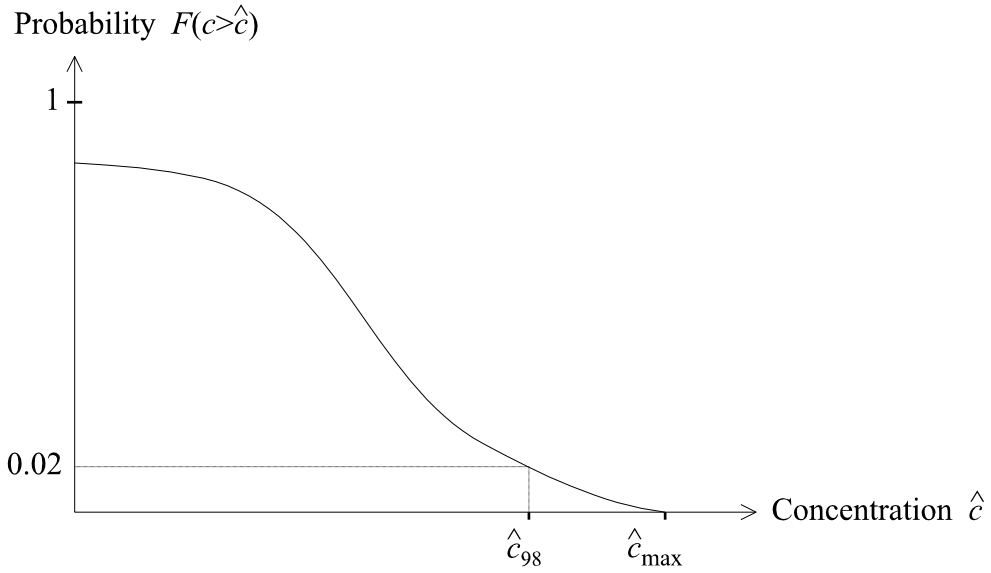


Figure 9.19 – Probability density function at (x, y, z) or (x, y, z, t).

3. As percentiles, e.g. the 98th percentile gives the concentration which is exceeded 2% of the time (see **Figure 9.20**).

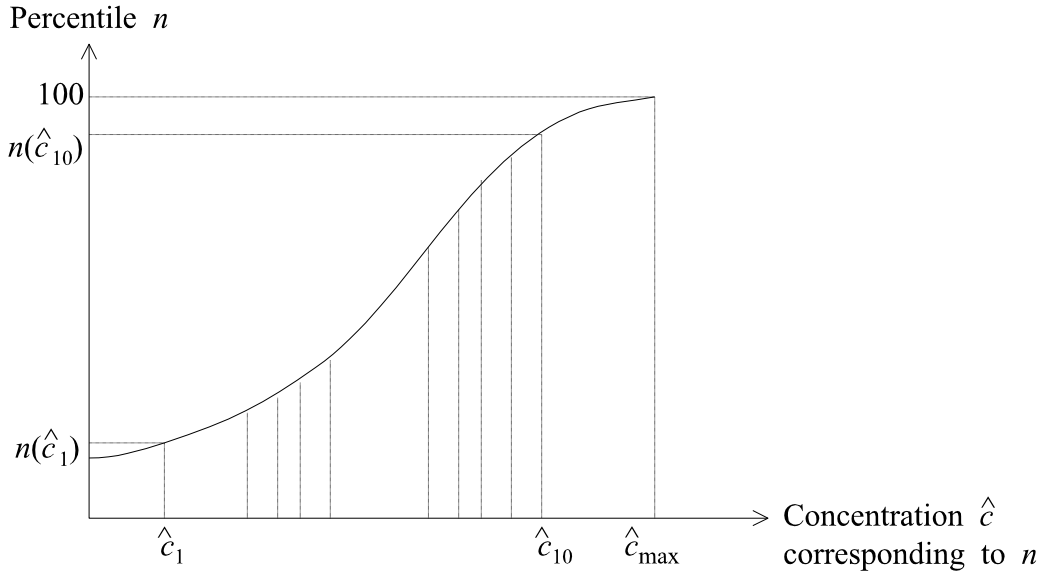


Figure 9.20 – Percentiles of concentration.

9.21.3 Long-term output

For long-term runs the user specifies up to 10 concentrations $\hat{c}$ and the probability of exceeding each concentration during the long-term averaging period is calculated. Output is given as a probability of exceedence plus the expected number of exceedences, i.e.:

- the probability of exceeding $\hat{c}$,
- the number of times per year it is predicted that $\hat{c}$ would be exceeded, taking into account the effect of short-term fluctuations.

For example, for a fluctuations averaging time t_{avf} of 15 minutes and a long-term averaging period of 1 year, the output gives the probability that the 15-minute average concentration exceeds $\hat{c}$, and the number of 15-minute periods during the year when $\hat{c}$ is exceeded.

Fluctuations percentiles and toxic response may not be calculated for long-term runs.

9.21.4 Limitations

Calculations using the **Fluctuations** option are limited to a maximum of 50 sources, of which up to 20 may be line, area or volume sources.

It should be noted that estimates of concentration fluctuations are likely to exhibit larger errors than estimates of mean concentration.

9.22 Fires and flares

ADMS 5 allows the user to enter as source efflux parameters the momentum flux F_m (m^4/s^2) and the heat release (or heat flux) F_b (MW) as alternatives to the velocity, volume flow rate or mass flux with temperature or density.

The definitions of the momentum flux F_m and the heat release F_b are those used in the HMIP D1 document (HMIP, 1993), namely

(9.61) $$F_m = Vw\frac{\rho}{\rho_a}$$

and

(9.62) $$F_b = \frac{V}{2.9}\left(\frac{\rho_a - \rho}{\rho_a}\right)$$

where the volume flux V, exit velocity w and density ρ are defined at exit conditions, i.e. the top of the fire.

If the user specifies F_m and F_b the model calculates the corresponding efflux parameters V (or w) and T (or ρ). Therefore, if V (or w) and T (or ρ) are known it may be simpler to enter them directly.

In dealing with fires, users who have some specialist knowledge of fires may find it useful to use F_m and F_b as efflux parameters since these variables may be known from experimental data in certain cases. However, users should be aware that many of the relevant factors needed to estimate F_b and F_m, such as combustion efficiency and fraction of heat radiated, are very case-specific and great care is needed when using data from a case different to the user's particular scenario.

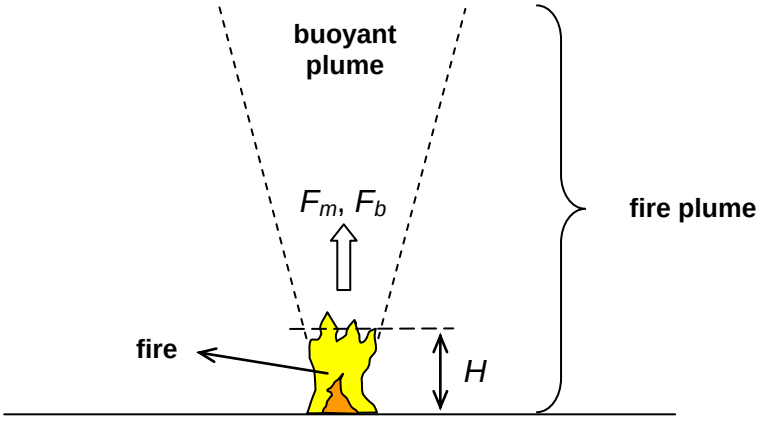


Figure 9.21 – Fire with buoyant plume. The efflux parameters may be specified in terms of pairs of parameters, such as the momentum flux F_m and the heat release F_b.

In general terms, a *fire* of the type shown schematically in **Figure 9.21**, consists of a flaming region directly above the fuel source. The fuel source may be a pool of hydrocarbon, a

wooden structure, etc. Above the flaming region is a hot region, which is referred to as the *buoyant plume*. In the buoyant plume there is essentially no flame or the flame is very intermittent. Here the combustion products from the fire are carried upwards in the strong buoyant flow generated by the heat released. The combination of the fire and associated buoyant plume is sometimes referred to as the *fire plume*.

Fires may vary considerably in their configuration and a broad distinction is usually drawn between high and low momentum situations. In high momentum situations the fuel has significant vertical momentum of its own, and gives rise to a jet flame or flare, e.g. venting of gas at a refinery. In low momentum situations the fuel has no significant vertical momentum of its own e.g. it is at rest on the ground, so that the vertical momentum of the rising gases is a result of the buoyant forces acting on the hot gases, e.g. a pool fire. Note that we use the term "hot plume" even when the momentum is high and this region is initially jet-like.

ADMS 5 cannot be applied to the flaming region itself, but may be used to model the dispersion of combustion products from the start of the buoyant plume if the data are known. The following three sections give advice on estimating the input data: source height, heat release and momentum flux.

Users should be aware that the range of behaviour exhibited by fires is very wide and so results are highly dependent on the precise details. References on fires such as Drysdale (1999) and DiNenno *et al*. (1988) may be useful. Users may also wish to refer to a pollution modelling guidance document published by the Ontario Ministry of the Environment (Lakes Environmental Consultants Inc., 2003), which describes an alternative method for deriving the effective stack height and diameter of a flare.

9.22.1 Source height

Since the hot plume begins at the top of the flame, at height H above the fuel source, the source height used in ADMS 5 should be the sum of the flame height and the height of the fuel source. This means that an estimate of the *flame height* is required.

There are various correlations for flame height as a function of fire parameters in the literature, with no single expression applicable to all fires. However, a useful expression that is applicable for many situations (DiNenno *et al*. (1988), Section 1, Chapter 18) is

$$\frac{H}{D} = -1.02 + 3.7 Q_*^{2/5} \tag{9.63}$$

where D is the fire diameter and the dimensionless parameter Q_* is essentially a Froude number. Q_* is defined by

$$Q_* = \frac{Q_{tot}}{\rho_a c_p T_a \sqrt{gD} D^2} \tag{9.64}$$

where Q_{tot} is the heat release rate of the fire in W (see below), ρ_a, c_p and T_a are the density (kg/m^3), specific heat capacity (J/kg/K) and temperature (K), respectively, of ambient air, and g is the acceleration due to gravity.

The expression in equation (9.63) is valid for $0.12 < Q_* < 1.2\text{x}10^4$. Outside of this range, different dependences on Q_* apply (see Drysdale, 1999). In particular, for fully-developed turbulent jet flames, the length is independent of exit velocity, and a useful approximation to flame length (DiNenno *et al*., 1988, Section 2, Chapter 4) is

(9.65) $$\frac{H}{d} = \frac{15}{C_T}\sqrt{\frac{M_a}{M_f}}$$

where d is the exit diameter, M_a and M_f are the molecular weights of air and fuel respectively, and C_T is the fuel concentration in a stoichiometric fuel-air mixture (e.g. $C_T \approx 0.1$ for methane).

In both the above expressions for flame length, the effects of wind are not taken into account – this aspect is less well understood, and users should refer to the references for more information.

9.22.2 Heat release Fb

This parameter is of particular importance for low momentum fires since it also governs the development of the momentum flux above the fire. However, in a general situation, the heat flux entering the hot plume (F_b) may be expressed as

(9.66) $$F_b = 10^{-6}(1-\alpha_r)Q_{tot}$$

with $Q_{tot} = \varepsilon_c \dot{m}_f H_c$.

In these formulae,

- α_r is the *fraction of heat release radiated*.

  The fraction of heat radiated varies considerably from one fire to another. It depends on the radiative characteristics of the flame, e.g. its temperature and soot content, which in turn depend on the combustion processes taking place, as well as its geometry. A value of $\alpha_r = 0.3$ is sometimes taken.

- ε_c is the *combustion efficiency*.

  This is likely to depend on the fuel, but is also influenced by the availability of air (e.g. restricted air supply). DiNenno *et al*. (1988, Section 1, Chapter 6) quotes a value of near unity for pools of methanol and heptane, but values of 0.45 and 0.63 for a polystyrene fire and a stack of wood pallets, respectively. The production of significant quantities of smoke by a fire indicates that inefficient combustion is taking place.

- $\dot{m}_f$ is the *fuel supply rate* in kg/s.

  This parameter will be straightforward to estimate in some circumstances, for example in the case of gaseous fuel issuing from a pipe supplied at a known rate. However, in situations where the fuel is liquid or solid, and therefore needs to be converted to the gas phase, the rate at which fuel is supplied itself

depends on the intensity of the fire, whose radiant heat acts to vaporise the volatile combustible material in the fuel and so sustain the combustion. There are many data available on the topic of burning rate for different materials and configurations (e.g. DiNenno *et al*. (1988), Section 2, Chapter 1), and the user is advised to refer to these in a given situation.

- H_c is the *heat of combustion* in J/kg.

  This is a standard physical property of the fuel, although it may be less straightforward to estimate if the fuel is a mixture of substances, several of which contribute significantly to the combustion.

9.22.3 Momentum flux Fm

F_m must be the momentum flux at the top of the flame. In many cases it will be possible to make a good estimate of volume flow rate or velocity, and the user will simply use the value found for F_b from the section above, with the definition of F_m in equation (9.61), to calculate the density of the plume to enter into ADMS 5.

If no information on velocity or volume flow rate is available, it is possible to estimate the momentum flux at the base of the plume by means of standard plume and jet theory – see, for example, DiNenno *et al*. (1988, Section 1, Chapter 6), Fischer *et al*. (1979, Chapter 9) or Fannelöp (1994).

9.23 Wind Turbine Effects

The effect of wind turbines on plume dispersion can be modelled in ADMS 5. The scientific methodology behind the model is described in this section. In particular, the ADMS model of a wind turbine wake is discussed. The wake is a region downstream of the turbine where the flow field is affected by the turbine. The plume dispersion is affected by the wake as a result of the changes to the wind speed and turbulence levels. These changes to the flow field affect the calculation of the location of the plume centreline and the plume spread parameters.

If the turbine effects are being modelled then a calculation grid is created with a spatial extent that covers the usual modelling region together with the turbine locations. This domain is then extended by 10% in each direction, so that the flow field can properly be resolved over the full modelling region. The resolution of the calculation grid is based on the minimum turbine diameter of the turbines in the wind farm.

For details about how to include wind turbine effects in a model run please refer to Section 4.23.

An overview of the ADMS model of a wind turbine wake is as follows. The wind speed downstream of a wind turbine in operation is reduced owing to the action of the wind turbine on the flow field and so a wake exists downwind of the wind turbine. Similarities between the decay of the wake behind a wind turbine and the dispersion of a plume of passive gas emitted from an elevated source are exploited in the model of wind turbine wakes. For example, a plume is characterised by an amount of gas emitted continuously, which is advected downwind whilst mixing with the ambient air, at a rate determined by the amount of turbulence in the atmosphere. A wind turbine wake is characterised by a wind speed deficit produced by the turbine, which is also advected downwind and gradually mixes with the ambient air, in a similar manner to a plume, but with some special characteristics.

The basic method is that the fully-expanded wind turbine wake immediately downwind of a wind turbine is characterised as a thin volume source, with source strength and dimensions calculated to give a maximum wind speed deficit determined by conservation of momentum according to the properties of the wind turbine. The wind speed deficit is the quantity which is then dispersed downwind.

The dispersion of the wind speed deficit, or wake, is modelled using the standard ADMS dispersion model used for pollutants, with some modifications to account for the special dispersion characteristics of wind turbine wakes. These modifications include the additional shear-induced turbulence in the wake and the lack of large-scale wind direction meandering in wind farm environments.

Wind farms are modelled by considering the wake effects from individual wind turbines in downstream order, so that wakes from upstream wind turbines affect the flow field used when characterising the effective sources for downstream wind turbines and when dispersing their wakes. The wake model has been validated against measured data from Tjæreborg Enge, Nysted wind farm and Noordzee wind farm.

Section 9.23.1 describes how each wind turbine is characterised as an effective volume source; section 9.23.2 describes the treatment of the additional shear-induced turbulence in a wind turbine wake; and section 9.23.3 describes the treatment of wind turbine wake interaction.

9.23.1 Representing a wind turbine as an effective volume source

As described above, the fully expanded wind turbine wake immediately downstream of the wind turbine is characterised as an effective volume source, with a source strength that gives a maximum wind speed deficit determined by conservation of momentum. The wake effects are modelled by calculating the dispersion of this initial wind speed deficit downstream of the wind turbine.

Characterisation of the effective source

Classical 1-D momentum theory for a wind turbine states that the rate of change of momentum across a wind turbine rotor disc (i.e. the thrust) is caused solely by the pressure gradients across the disc, and that since the pressure recovers at some distance downstream, the momentum in the 'stream-tube' upstream of this pressure gradient is equal to the momentum in the stream-tube downstream of the pressure gradient, i.e. the stream-tube expands as the flow speed reduces.

The quantity used to describe the strength of the wind speed deficit in the wake relative to the upstream flow is called the axial induction factor a, which is defined as follows:

(9.67) $$U_d = (1-a)U$$

where U_d is the axial flow speed through the rotor disc (m/s) and U is the upstream wind speed (m/s).

The 1-D momentum theory together with (9.67) leads to a relationship between U and the axial flow speed in the fully expanded wake U_w, as given by Hansen (2008):

(9.68) $$U_w = (1-2a)U$$

which yields a maximum wind speed deficit in the wake $\Delta U_{\max}$ of $2aU$. This is the initial value of the wind speed deficit immediately downwind of the turbine.

The wake is modelled in ADMS as an effective volume source with square cross-section representing the fully-expanded wake where the axial flow speed is U_w. The cross-sectional area of the source is equal to the cross-sectional area of the expanded wake. This type of source has an initial 'top-hat' vertical and horizontal profile, which decays to a Gaussian-shape profile as it evolves downstream. **Figure 9.22** shows a schematic of the expanding stream-tube and the effective volume source.

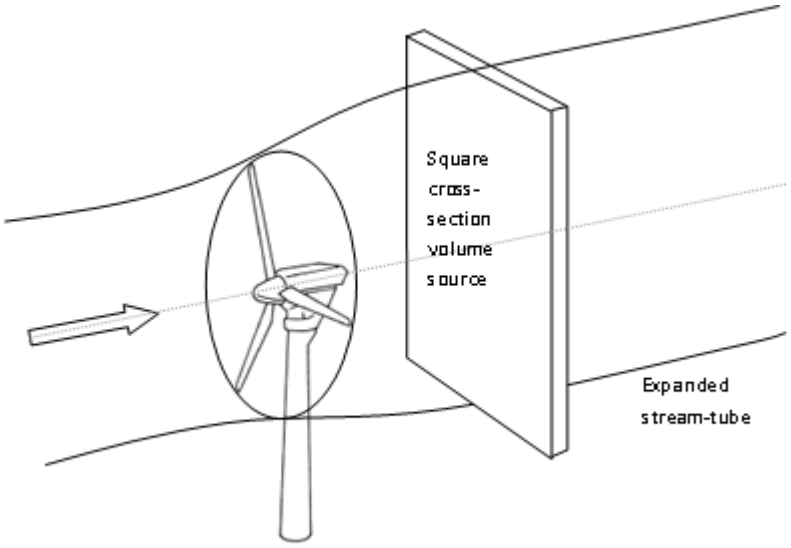


Figure 9.22 – Schematic of the expanding stream-tube of air flowing through the turbine rotor, and the effective volume source representing that turbine.

The justification for using a square cross-section volume source rather than a circular one is mainly pragmatic; a square cross-section volume source is simpler to model than a circular volume source and a circular volume source is unlikely to yield much benefit due to the evolution of the profile to a Gaussian shape within a few wind turbine diameters.

To calculate the dimensions of the volume source we use the classical 1-D momentum theory to find:

$$dy = dz = \frac{D}{2}\sqrt{\frac{\pi(1-a)}{(1-2a)}} \tag{9.69}$$

where dy is the crosswind extent of the volume source, dz is the depth of the volume source, and D is the diameter of the rotor. As most modern wind turbines have efficient mechanisms to bring the wind turbine rotor disc perpendicular to the inflow wind as quickly as possible the model makes an assumption of zero yaw, i.e. it assumes that the wind turbine rotor disc is always perpendicular to the upstream wind and does not take account of any rotations. If the wind direction changes between meteorological conditions then the model assumes that there is an instantaneous re-orientation of the wind turbine. The volume source is therefore always perpendicular to the upstream flow. The along-wind extent of the volume source is set to ten percent of the crosswind extent *dy*.

The wind turbine effective source strength is calculated inversely from the known maximum wind speed deficit $\Delta U_{\max} = 2aU_{\infty}$ where U_{∞} is the upstream wind speed. In model terms, with concentration as a surrogate for ΔU, $\Delta U_{\max}$ can be expressed as

$$\Delta U_{\max} = \frac{QV_{src}}{\dot{V}} \tag{9.70}$$

Q is the source strength (in units of ms^{-2}), V_{src} is the source volume, and $\dot{V}$ is the volume flow rate through the source. Therefore

(9.71) $$Q = 2aU^2$$

These parameters are all functions of the axial induction factor a, which must in turn be calculated from information about the turbine itself.

Calculation of the Axial Induction Factor

ADMS calculates the axial induction factor from the thrust coefficient C_T (input to the model as a function of wind speed using the *.wtd* file described in Section 4.23.1).

The relationship between thrust coefficient C_T and axial induction factor a is, following Hansen (2008),

(9.72) $$C_T = \begin{cases} 4a(1-a)F, & a \le a_c \\ 4\left(a_c^{\,2} + (1-2a_c)a\right)F, & a > a_c \end{cases}$$

where $a_c \approx 0.2$.

This becomes an equation for a in terms of C_T:

(9.73) $$a = \begin{cases} \frac{1}{2}\left(1-\sqrt{1-C_T}\right) & C_T \le 0.64 \\ \frac{C_T - 4a_c^{\,2}}{4(1-2a_c)} & C_T > 0.64 \end{cases}$$

9.23.2 Shear-induced turbulence

At the edge of a wind turbine wake, there is a gradient in the speed of the air flow, which generates extra turbulence. An extra component of turbulence, σ_{shear}, has been added to account for this; σ_{shear} is calculated as

(9.74) $$\sigma_{shear}[i+1] = \begin{cases} 0.4|\Delta U|\frac{x}{X_{crit}}, & x \le X_{crit} \\ \sigma_{shear}[i] \times e^{-dt/t} & x > X_{crit} \end{cases}$$

based on Bevilaqua and Lykoudis (1978) where x is the downwind distance from the (effective) source and $|\Delta U|$ is the local wind speed deficit.

Here, *i* and *i*+1 represent consecutive points in a downstream direction along the wake centreline; *dt* represents the time taken by the wake centreline to travel from the *i*th to the (*i*+1)th point. *t* represents the time taken by the plume to spread the width (depth) of the wake:

(9.75) $$t = \frac{2R}{\sigma_{tot}}$$

R is the wake radius, defined as the effective volume source half-width plus the local plume spread, calculated at a given point as a weighted average over all upstream wakes; σ_{tot} is the combination of the upstream turbulent velocity and the local shear-induced turbulence:

(9.76) $$\sigma_{tot}[i+1] = \sqrt{\sigma_{shear}[i]^2 + \sigma[i+1]^2}$$

X_{crit} is a critical distance, which is dependent on the inflow turbulence and is the distance downstream from the effective source at which σ_{shear} starts to decay. For inflows with low turbulence, the shear-induced turbulence increases downstream from zero at the effective source to X_{crit}, in proportion with the local wind speed deficit, and then decays; for turbulent inflows, the initial value of σ_{shear} is $0.4|\Delta U|$ at the wind turbine and it decays immediately. X_{crit} is defined as

(9.77) $$X_{crit} = \begin{cases} 4D & TI \le TI_{lower} \\ 4D\left[1 - \dfrac{(TI - TI_{lower})}{(TI_{upper} - TI_{lower})}\right] & TI_{lower} < TI < TI_{upper} \\ 0 & TI \ge TI_{upper} \end{cases}$$

Where D is the diameter of the wind turbine. The inflow turbulence is characterised by the turbulence intensity TI, which is expressed as a percentage and represents the ratio of the horizontal turbulence to the horizontal mean flow. TI is defined as

(9.78) $$TI = 100 \times \frac{\sqrt{{\sigma_u}^2 + {\sigma_v}^2}}{\sqrt{u^2 + v^2}}$$

TI_{lower} and TI_{upper} are threshold values determined during validation of the model; these are set to 12% and 18% respectively.

9.23.3 Treatment of wind turbine interaction

In standard dispersion, in the absence of plume chemistry, the plume due to one source does not affect any other source or plume, and the concentration field due to multiple sources is the sum of the concentration field due to each source.

In the case of wind turbine wake modelling, the wake downwind of a turbine affects all downwind turbines. For this reason, developments have been made to account for the effect of the change in the wind field due to upstream turbines on downstream turbines. This section describes these changes.

Ordering of wind turbines

Before the wake calculations are carried out for each input meteorological condition all the input wind turbines are re-ordered according to their downwind position; the most upstream wind turbine is modelled first, the most downstream wind turbine is modelled last. The assumption is made that wind turbines are only affected by the wakes from upstream wind turbines.

Modification to flow field

The flow field is modified by each wind turbine wake and includes the effect of the wakes from all upstream wind turbines. This is made possible by the ordering of wind turbines described above.

The flow field is perturbed by each wind turbine by adjusting the values of the velocity components from the current flow field to include the wind speed deficit. This is done over a grid of points covering the modelling region and the wind turbine locations at a range of heights within the atmospheric boundary layer. This process is repeated iteratively covering the effect of each wind turbine in downstream order. This change in the flow field for each wind turbine in a wind farm affects the characterisation of the effective volume source and the dispersion of the wake.

9.24 Urban canopy flow

It is generally accepted that as wind approaches a built-up urban area, the profile is displaced vertically by a height related to the mean height of the buildings, while the flow within the building canopy is slowed by the buildings (Belcher *et al.* 2013), as depicted in **Figure 9.23**. The ADMS 5 spatially-varying urban canopy flow option incorporates above-canopy displaced flow and turbulence profiles linked to in-canopy profiles, allowing the flow field within urban areas to be characterised on a neighbourhood-by-neighbourhood basis.

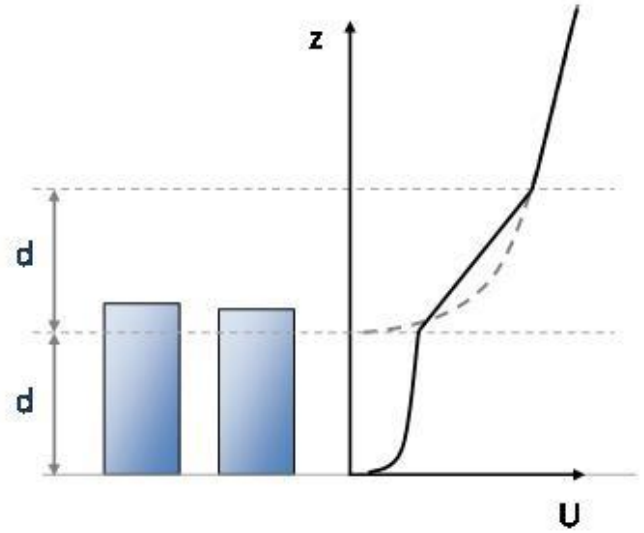


Figure 9.23 – Diagram of urban canopy vertical profile of velocity relative to displacement height *d*. The dashed line represents the standard velocity profile displaced upwards by *d*.

9.24.1 Input data

The urban area is characterised by gridded values of the average building height, the average street canyon width ($\hat{g}$), the ratio of the plan area of buildings to urban canopy grid cell area (λ_P) and, for a user-specified set of wind direction sectors, the ratio of the frontal area of buildings to urban canopy grid cell area (λ_F). Urban canopy grid cell dimensions are set so that these parameters are able to represent the features of distinct urban neighbourhoods (typically 500 m to 2 km). The urban canopy module follows MacDonald *et al.* (1998) in calculating an effective roughness z_{0b} and a displacement height d from these input data. Please refer to Section 4.24.1 for details of the input file format for urban canopy flow data.

9.24.2 Velocity profile

The full urban canopy vertical velocity profile consists of three sections as shown in **Figure 9.23**: above twice the displacement height ('above-building'), below the displacement height ('below-building') and a transition region between these two regions. For the 'above-building' section, the standard ADMS velocity profile as described in the ADMS technical specification documents (CERC, 2015) is displaced upwards by d and a modified friction velocity is used, which is calculated using the local roughness z_{0b}. Below the displacement height a logarithmic velocity profile is

applied, with a 'below-building' roughness z_{0s} of 0.1 m, which represents the effect on the flow of small obstacles which may be present at street level. The effective friction velocity for the below-building flow is calculated by matching the below-building velocity at the displacement height to a fraction $(1 - \lambda_P)^2$ of the above-building velocity at twice the displacement height. A linear interpolation is performed in the transition region between the below-building velocity at the displacement height and the above-building velocity at twice the displacement height.

9.24.3 Turbulence profile

The full urban canopy vertical profile of turbulence consists of two sections: above and below the displacement height. Above the displacement height, the standard ADMS stability-dependent turbulence profiles are displaced upwards by d and the modified friction velocity related to the local roughness z_{0b} is used. Below the displacement height, the turbulence velocities decay towards the ground according to $\exp(-(d - z)/2\hat{g})$, where z is the height above ground level.

9.24.4 Flow regimes

The effects of the urban canopy module are divided into four regimes according to the values of z_{0b} and d. For the lowest values of d, below 1 mm, no urban canopy flow calculations are performed, while for the highest values of d, greater than the maximum of 2 m and half the average building height, full urban canopy flow calculations are performed as described above. Two intermediate regimes are defined: for low values of d, less than the maximum of 1 m and a tenth of the average building height, a 'standard' ADMS flow profile with the local roughness z_{0b} is used, known as the 'no displacement' solution; while for moderate values of d the flow is interpolated between the no displacement and full urban canopy profiles to give a 'low displacement' solution.

9.25 Running AERMOD

ADMS 5 includes the capability to run some of the AERMOD options from the ADMS 5 interface. This section outlines the methodology used in running AERMOD (version 15181) using an ADMS 5 input (*.apl*) file.

When AERMOD is run from ADMS 5 the *.apl* file is converted to an AERMOD input file. The user is given the choice of several meteorological processing options, as described in Sections 2.5.7 and 9.25.1. Once AERMOD has run, the output from AERMOD is processed into ADMS 5 format output files, see Section 6.2).

If the complex terrain module is used AERMAP is run, as described in Section 9.25.3, and if the buildings module is used BPIPPRIME is run, as described in Section 9.25.4.

9.25.1 Meteorological data

When AERMOD is run from ADMS 5 there are three choices for the meteorological data:

- the ADMS 5 meteorological data (*.met*) file can be processed using the ADMS meteorological processor;
- the ADMS 5 meteorological data can be processed using AERMET; or
- AERMOD format meteorological data files can be used.

If AERMOD format meteorological data files are used, the information entered into the **Meteorological** screen of the ADMS 5 interface is not used.

If the ADMS meteorological processor is used with an ADMS 5 meteorological data file then the output from the ADMS meteorological processor is converted into the format required for input into AERMOD. If a vertical profile (*.prf*) file is specified then the data in this file is also used in creating the files for input into AERMOD.

If AERMET (version 15181) is used the ADMS 5 meteorological data (*.met*) and vertical profile (*.prf*) files are converted into a surface file for input into stage 2 of AERMET and, if required, an onsite file for input into stage 1 of AERMET. The specified upper air file is also input into stage 1 of AERMET. The three stages of AERMET are then run and the met files created used in AERMOD.

When running AERMET the midday Bowen ratio (B) is required, this is either specified by the user or calculated from the modified Priestley –Taylor parameter (α) as:

$$B = \begin{cases} B_{dry} & \alpha < 0.45 \\ B_{dry} + \dfrac{\alpha - 0.45}{1 - 0.45}\left(B_{wet} - B_{dry}\right) & 0.4 \le \alpha \le 1 \\ B_{wet} & \alpha > 1 \end{cases}$$

where B_{dry} and B_{wet} are the values for dry and wet grassland given in **Table 9.6**. Values of the midday Bowen ratio are calculated for each season.

	Spring	Summer	Autumn	Winter
Dry	1	2	2	2
Wet	0.3	0.4	0.5	0.5

Table 9.6 – Midday Bowen ratio values for dry and wet grassland.

PROFBASE, the base elevation above mean sea level for the potential temperature profile is always set to 0 when running AERMOD from ADMS 5.

9.25.2 Output

When AERMOD is run from ADMS 5, AERMOD is set to produce a binary output file containing the concentrations at each output point for every line of meteorological data. A post-processor is then run to convert this file into ADMS 5 format output files. This allows the long-term statistics available in ADMS 5 to be calculated when AERMOD is being used.

9.25.3 Complex terrain

If the **Complex terrain** module is being used then AERMAP (version 11103) is run before AERMOD. In order to run AERMAP, the spatially varying terrain (*.ter*) file specified is converted into a *.dem* format terrain file for input into AERMAP. The output from AERMAP is then used by AERMOD.

9.25.4 Buildings

If the **Buildings** module is being used then BPIPPRIME (version 04274) is run before AERMOD. The information on the **Buildings** and **Source** screen is converted into the format required for BPIPPRIME which is then run. The output from BPIPPRIME is then converted into the format required for input into AERMOD.

APPENDIX A Model Limits

This appendix gives guidance on the availability and compatibility of the model options. Section A.1 summarises the restrictions on the number of sources, pollutants and source groups, and outlines which options can be used with each type of source. Details of types of concentration output available with each option are given in Section A.2, while Section A.3 gives guidance on using multiple model options in the same run. Section A.4 gives guidance on running AERMOD from an ADMS 5 parameter file.

A.1 Sources, groups and pollutants

The types of source available (Point, Line, Area, Volume and Jet) and associated emissions are described in Section 3.2. Overall restrictions are as follows.

- The maximum number of sources is 300.
- Within the overall limit, up to 30 of each of line, area and volume sources may be modelled.
- The maximum number of user-defined groups that can be created and run in one *.apl* file is 20. Additionally, a group containing all of the sources can be modelled.
- The maximum number of pollutants or parent isotopes that can be entered into the pollutant palette is 30. The number of pollutants or parent isotopes that may be emitted by any source, or output from the model, is 10.
- The maximum number of parent and daughter isotopes output is 50.

Table A.1 shows which model options can be used with each type of source, and with multiple sources.

Model option	Point source	Line, area or volume source	Jet source	Multiple sources
Deposition	✓	✓[1]	✓[1]	✓
Radioactive decay	✓	✓	×	✓
Gamma dose	✓	×	×	×
Odours	✓	✓	✓	✓
Plume visibility	✓	×	×	✓
Chemistry	✓	✓	✓	✓
Complex terrain	✓	✓	×	✓
Buildings	✓	[2]	[2]	✓
Coastline	✓	×	×	✓
Puff	✓	×	✓	×
Fluctuations[3]	✓	✓	×	✓
Marine boundary layer	✓	✓	✓	✓
Temperature and humidity output	✓	×	[4]	×
User-entered 3D flow field	✓	✓	×	✓
Calm conditions	✓	✓	×	✓
Time-varying sources	✓	✓	✓	✓
Wind turbine effects	✓	✓	×	✓
Urban canopy flow	✓	✓	×	✓

Table A.1 – Allowed combinations of model options with different source types.

1. The 'falling drop' wet deposition method can only be modelled for point sources.
2. Line, area, volume and jet sources and buildings can be in the same run but the effect of the buildings will be ignored for these sources.
3. A maximum of 50 point sources and 20 line, area and volume sources are allowed with fluctuations.
4. Temperature output can be calculated for a downstream jet source and short term runs only. No humidity output can be calculated for jet sources.

A.2 Concentration output

Table A.2 summarises the types of concentration output available with each model option. Many of the model options also produce other output, details of which are given elsewhere.

Model option	Short-term average	Long-term average	Rolling average	Percentiles	Short-term line plotting output[1]
Deposition	✓	✓	✓	✓	✓
Radioactive decay	✓	✓	✓	✓	✓
Gamma dose	✓	✓	[2]	[3]	×
Odours	✓	✓	✓	✓	✓
Plume visibility	✓	✓	[4]	✓	✓
Chemistry	✓	✓	✓	✓	✓
Complex terrain	✓	✓	✓	✓	✓
Buildings	✓	✓	✓	✓	✓
Coastline	✓	✓	✓	✓	✓
Puff	✓	×	×	×	✓
Fluctuations	✓	✓	[2]	✓	✓
Marine boundary layer	✓	✓	✓	✓	✓
Temperature and humidity output	✓	✓	✓	✓	✓
User-entered 3D flow field[5]	✓	×	×	×	✓
Calm conditions	✓	✓	✓	✓	✓
Time-varying sources	✓	✓	✓	✓	✓
Wind turbine effects	✓	✓	✓	✓	✓
Urban canopy flow	✓	✓	✓	✓	✓

Table A.2 – Concentration output available with each model option.

1. Short-term line plotting output is available for single point or jet sources only.
2. No gamma dose or fluctuations output is given for averaging times greater than 1 hour.
3. No percentiles output is given for gamma dose levels.
4. Plume visibility available for 1-hour averaging times only.
5. User entered 3D flow field data can only be used with a single line of meteorological data.

A.3 Using multiple model options in the same run

Table A.3 shows which model options can be used together.

Model options	Deposition	Radioactive decay	Gamma dose	Odours	Plume visibility	Chemistry	Complex terrain	Buildings	Coastline	Puff	Fluctuations	Marine boundary layer	Temperature and humidity output	User-entered 3D flow field data	Calm conditions	Background	Time-varying sources	Wind turbine effects	Urban canopy flow
Deposition	-	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Radioactive decay	✓	-	✓	×	✓	×	✓	✓	✓	✓	×	✓	×	✓	×	×	✓	✓	✓
Gamma dose	✓	✓	-	×	✓	×	×	×	×	×	×	✓	×	×	×	×	✓	×	×
Odours	✓	×	×	-	✓	×	✓	✓	✓	×	✓	✓	✓	✓	✓	✓	✓	✓	✓
Plume visibility	✓	✓	✓	✓	-	✓	✓	[1]	✓	×	✓	✓	✓	✓	✓	✓	✓	✓	×
Chemistry	✓	×	×	×	✓	-	✓	✓	✓	×	×	✓	✓	✓	✓	✓	✓	✓	✓
Complex terrain	✓	✓	×	✓	✓	✓	-	✓	×	×	✓	×	✓	[2]	✓	✓	✓	✓	×
Buildings	✓	✓	×	✓	[1]	✓	✓	-	×	×	×	✓	✓	✓	✓	✓	✓	✓	×
Coastline	✓	✓	×	✓	✓	✓	×	×	-	×	×	×	×	×	×	✓	✓	×	×
Puff	✓	✓	×	×	×	×	×	×	×	-	✓	✓	[3]	×	×	[4]	✓	×	×
Fluctuations	✓	×	×	✓	✓	×	✓	×	×	✓	-	✓	✓	✓	×	×	✓	✓	×
Marine boundary layer	✓	✓	✓	✓	✓	✓	×	✓	×	✓	✓	-	✓	×	✓	✓	✓	✓	×
Temperature and humidity output	✓	×	×	✓	✓	✓	✓	✓	×	[3]	✓	✓	-	✓	✓	✓	✓	✓	×
User-entered 3D flow field data	✓	✓	×	✓	✓	✓	[2]	✓	×	×	✓	×	✓	-	×	✓	✓	×	×
Calm conditions	✓	×	×	✓	✓	✓	✓	✓	×	×	×	✓	✓	×	-	✓	✓	×	✓
Background	✓	×	×	✓	✓	✓	✓	✓	✓	[4]	×	✓	✓	✓	✓	-	✓	✓	✓
Time-varying sources	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	-	✓	✓
Wind turbine effects	✓	✓	×	✓	✓	✓	✓	✓	×	×	✓	✓	✓	×	×	✓	✓	-	×
Urban canopy flow	✓	✓	×	✓	×	✓	×	×	×	×	×	×	×	×	✓	✓	✓	×	-

Table A.3 − Model options that may be used together.

1. Plume visibility and buildings can be modelled in the same run but the buildings will have no effect on the plume visibility calculations. However, additionally modelling plume visibility when modelling buildings may affect predicted model concentrations, and temperature and humidity output; this is because the water content of the plume is accounted for in the plume rise calculations when plume visibility is modelled.
2. The complex terrain option must be selected when modelling user-entered 3D flow field data but the terrain and/or roughness files specified are not used.
3. Only temperature output can be specified with puffs.
4. Background concentrations can only be entered for time dependent puffs.

A.4 Restrictions on running AERMOD

This section outlines the restrictions on running AERMOD using an ADMS 5 parameter file. Refer to Section 2.5.7 for more details about running AERMOD with an ADMS 5 parameter file. Section A.4.1 gives details on the types of source available in ADMS 5 which can be run with AERMOD, Section A.4.2 gives details on the output types and locations which can be used and Section A.4.3 gives the details of which of the ADMS 5 model options can be used and what restrictions apply to their use.

A.4.1 Sources and groups

The following restrictions apply to the types of sources available in ADMS 5 which can be modelled with AERMOD.

- Line and area sources are modelled as passive.
- Jet sources cannot be modelled.
- Efflux type **Fm** and **Fb** cannot be used.
- The **Ambient** temperature option cannot be used in conjunction with the **Mass** efflux type.

In order to run AERMOD, group output must be selected with only one group selected for output. The **All sources** option counts as one group, so if this option is selected, no user-defined groups can be selected for output.

A.4.2 Output types and locations

The following restrictions apply to the types of output available in ADMS 5 when running AERMOD.

- Only one pollutant can be modelled. If more than one pollutant is selected for output the first pollutant will be used.
- Averaging times must be at least one hour.
- A comprehensive output file cannot be created from AERMOD.

The following restrictions apply to the types of output location available in ADMS 5 when running AERMOD.

- A polar coordinate system cannot be used.
- The output grid can only contain one height.

A.4.3 Model options

The following ADMS 5 model options have restrictions applied to them when run with AERMOD.

- **Dry deposition**:
  - For gaseous pollutants a dry deposition velocity must be specified, this deposition velocity must be greater than zero.
  - For particulate pollutants particle diameter, density and mass fraction must be specified.
- **Buildings**: there are no restrictions on running buildings, as with ADMS 5 the effects of buildings are only considered for point sources but other sources can be contained in the run.
- **Complex terrain**:
  - A spatially varying roughness file cannot be modelled
  - Flow field output cannot be created
- **Meteorological** data must contain year, day and hour.
- **Time varying emissions**:
  - **Var file**: line and area sources will be modelled as passive
  - **Var file**: density, mass flux, Fm, Fb, source coordinates, source height, source width and source height cannot be specified using a *.var* file
  - **Var file**: the diameter must be constant in the *.var* file
  - **Fac file**: the *.fac* file cannot contain wind direction dependent operation.

The following ADMS 5 model options cannot be used when running AERMOD.

- The **Isotopes** palette cannot be used, as a result of this the **Radioactive decay** and **Gamma dose** options also cannot be modelled.
- **Wet deposition** is not modelled, the run will continue but without including the effects of wet deposition.
- **Plume visibility** cannot be modelled.
- **Odours** cannot be modelled.

  *Odours can be modelled by using g/s as a surrogate for ou_e and selecting g/m^3 on the **Output** screen. However, care has to be taken to avoid large numbers – this can be avoided by dividing the emission rate by a factor (for example, 1000) and then multiplying the resultant concentration by the same factor.*

- **Chemistry** will not be modelled, the run will continue but without including the effects of chemistry.
- The **Coastline** option cannot be used.
- The **Puff** option cannot be used.

- The **Fluctuations** option cannot be used.
- An **Additional Input file** (*.aai*) cannot be specified, this means that no options which are specified using the additional input file can be used.
- The **Background** value for the pollutant being modelled must be zero.

In addition if AERMET is chosen as the meteorological option:

- the wind must either not be in sectors or in sectors of 10^{o}; or
- if the wind is in sectors of 10^{o} and a *.prf* file is used, then the *.prf* file must contain wind speed values otherwise AERMET will generate invalid data.

APPENDIX B NOx in ADMS 5

In ADMS 5, all NO_X data (emissions, background concentrations and output concentrations) in mass units are 'NO_X as NO_2'. This Appendix explains what this means and describes the implications for model users.

B.1 What is NOx?

NO_X is the name given to a mixture of two pollutants, nitrogen dioxide (NO_2) and nitrogen oxide (NO). Care must be exercised in specifying NO_X concentrations or emission rates in mass units (e.g. ng/m^3, $\mu g/m^3$, mg/m^3, g/s), since a specified volumetric concentration (e.g. in ppb) has a range of mass concentrations, depending on the relative proportions of NO and NO_2.

The conversion factor from ppb to $\mu g/m^3$ for any gaseous pollutant is M/24.06 (derived from the Ideal Gas Equation with an ambient temperature of 20°C and pressure of 1013 mb), where M is the molecular mass of the pollutant in grams. For example, the molecular mass of NO_2 is 46 g, so the conversion factor for NO_2 is 1.91. Similarly, the molecular mass of NO is 30 g, so the conversion factor for NO is 1.25. Therefore, for instance, 10 ppb NO_X could be 19.1 $\mu g/m^3$ if it is all NO_2, or 12.5 $\mu g/m^3$ if it is all NO, or somewhere in between for a mixture of NO and NO_2.

The following sections discuss how NO_X is treated in ADMS 5. This is especially important, as it is standard practice to express concentrations of NO_X in mass units as 'NO_X as NO_2'.

B.2 What does 'NOx as NO2' mean?

Pollutant levels can be measured in terms of volumetric concentration, and converted to a mass concentration using the Ideal Gas Equation, which is dependent on the molecular mass of the pollutant. Since NO_X is a composite pollutant, its effective molecular mass depends on the relative proportions of its constituents. This is often not known, so the assumption often made in the conversion is that NO_X has the molecular mass of NO_2; the resulting mass concentration is termed a '**NO_X as NO_2**' concentration.

More generally, the term 'NO_X as NO_2' refers to any mass emission rate or mass concentration of NO_X that has been calculated from a volumetric emission rate or concentration assuming that 100% of the NO_X is NO_2. Note that 'NO_X as NO_2' mass concentrations (and emission rates) generally overestimate the true situation, since the molecular mass of NO_2 (46 g/mol) is greater than the molecular mass of NO (30 g/mol).

When actual masses of each of NO and NO_2 are used to convert from volumetric to mass units, the NO_X concentration is referred to as '**True NO_X**' in this *User Guide*.

B.3 Treatment of NOx in ADMS 5

In ADMS 5, all NO_X data in mass units are 'NO_X as NO_2'.

- NO_X emission rates must be input as 'NO_X as NO_2' values. If you have 'true NO_X' emission rate data, these must be converted to 'NO_X as NO_2' before being input to ADMS 5. Instructions for this conversion are given in Section B.5.
- Background concentrations of NO_X entered into the model in mass units must be 'NO_X as NO_2'. If background data have been obtained in the form of 'True NO_X', these must be converted to 'NO_X as NO_2', before being entered into the model.
- NO_X concentrations output from ADMS 5 in mass units are also 'NO_X as NO_2'. When comparing ADMS 5 results with, for example, measured concentrations, care should be taken that both sets of data are in the same form. ADMS 5 output concentrations can be converted to 'True NO_X', if required, using the methods described in Section B.5 below.

B.4 NO2 emissions in ADMS 5

If the **Chemistry** option is used in ADMS 5, the relative proportions of NO and NO_2 in the emitted NO_X are required in order to calculate the effects of the chemical reactions. The user may input NO_2 emission rates in addition to NO_X emission rates. Alternatively, if NO_X emission rates are input to ADMS 5 but no NO_2 emission rates are input, it is necessary for the model to make an assumption regarding the split between NO and NO_2. In this situation it is assumed by default that 5% of the emitted NO_X is NO_2, i.e.

$$NO_2 \text{ emission rate} = 0.05 \text{ '}NO_X \text{ as } NO_2\text{' emission rate} \quad (1)$$

B.5 Converting between 'NOx as NO2' and 'True NOx'

This section describes how NO_X emission rates or concentrations in mass units can be converted from 'True NO_X' to 'NO_X as NO_2', and vice versa. In each case, it is just necessary to convert the NO part of the NO_X concentration. Therefore,

To convert from 'True NO_X' to 'NO_X as NO_2':

$$\text{'}NO_X \text{ as } NO_2\text{'} = NO_2 + \frac{M_{NO_2}}{M_{NO}} \text{ ('True } NO_X\text{'} - NO_2) \quad (2)$$

where M_{NO_2} is the molecular mass of NO_2 (46 g/mol) and M_{NO} is the molecular mass of NO (30 g/mol). For example, an emission rate of 10 g/s 'True NO_X', of which 2 g/s is NO_2, is equivalent to 14.3 g/s 'NO_X as NO_2'.

And to convert from 'NO_X as NO_2' to 'True NO_X':

$$\text{'True } NO_X\text{'} = NO_2 + \frac{M_{NO}}{M_{NO_2}} \text{ ('}NO_X \text{ as } NO_2\text{'} - NO_2) \quad (3)$$

For example, a concentration of 10 μg/m^3 'NO_X as NO_2', of which 5 μg/m^3 is NO_2, is equivalent to 8.26 μg/m^3 'True NO_X'.

Note that if you are converting an emission rate, and the emission rate of NO_2 is unknown, it will be necessary to make an assumption regarding the fraction of the NO_X emission rate that is NO_2. As noted above, if NO_2 emissions are not input and chemical reactions are modelled, ADMS 5 assumes by default that 5% of the 'NO_X as NO_2' emission rate is NO_2. Therefore, in order to be consistent with ADMS 5, the same assumption should be used for the conversion. In this case, the conversion equation becomes

$$\text{'NO}_X\text{ as NO}_2\text{'} = \frac{M_{NO_2}}{\left(0.95M_{NO} + 0.05M_{NO_2}\right)}\ \text{'True NO}_X\text{'} \qquad (4)$$

i.e.

$$\text{'NO}_X\text{ as NO}_2\text{'} = 1.494\ \text{'True NO}_X\text{'} \qquad (5)$$

Note also that to convert 'NO_X as NO_2' concentrations output from ADMS 5 to 'True NO_X', the NO_2 concentration is required. To obtain this,

- If **Chemistry** is modelled, or if NO_2 emissions have been input, then select NO_2 on the **Output** screen in ADMS 5 to calculate the NO_2 concentrations (selecting the same output units as for NO_X).
- If **Chemistry** is not modelled and NO_2 emissions are not input, then it will be necessary to make an assumption regarding the split between NO and NO_2.

APPENDIX C Air Quality Limits and Guidelines

This section summarises air quality limits, guidelines and objectives. The limits are subject to change; it is therefore important to ensure that you are referring to current limits.

The air quality limits, guidelines and objectives for the main pollutants are listed below as follows.

- Section C.1: UK Air Quality Strategy and Regulations.
- Section C.2: EU limit values.
- Section C.3: Lithuanian limit values
- Section C.4: French Limit Values
- Section C.5: US National Ambient Air Quality Standards.
- Section C.6: World Health Organisation guidelines.

Relationship between percentiles and exceedences

Many of the air quality limits are stated in terms of *"not more than N exceedences of a threshold value per year"*, where *N* is an integer. The limits can be restated in terms of percentiles by calculating the appropriate percentile from the maximum number of exceedences *N*, as described below. In ADMS 5, results can be calculated in terms of exceedences (in which case the threshold values are specified by the user on the **Output** screen) and/or percentiles (in which case the percentiles to be calculated are specified on the **Output** screen).

For example, the 2004 UK AQS particulates (PM_{10}) limit is *"not more than 35 exceedences per year of 50 mg/m³ as a 24-hour average"*. This means that 35 exceedences are acceptable, whereas 36 are not. The corresponding percentile to be calculated is therefore

$$\frac{365-35}{365}\times 100 = 90.41^{th}\ \text{percentile}$$

where 365 is the number of 24 hour periods per year. If the limit is stated in terms of the hourly average, the percentile is calculated using

$$\frac{8760-N}{8760}\times 100$$

where 8760 is the number of 1-hour periods per year. Similarly for a 15-minute average, the number of 15-minute intervals per year is 35040, so the percentile to be calculated is

$$\frac{35040-N}{35040}\times 100\,.$$

C.1 UK Air Quality Strategy and Regulations

The Air Quality Strategy for England, Scotland, Wales and Northern Ireland[1] (AQS), published in 2007 by the Department for Environment, Food and Rural Affairs (Defra), defines a reviewed set of limits and targets on pollutant concentrations. Some of them already have a legal status[2,3,4,5,6]. The main differences with respect to the previous AQS[7] are the modification of the dates to be achieved by and the introduction of limits on polycyclic aromatic hydrocarbons (PAHs) and $PM_{2.5}$.

Table C.1 lists UK air quality objectives for England, Wales, Scotland and Northern Ireland. The Environmental Protection UK website[8] is a useful source of information as well as the UK legislation website[9] for the air quality regulations.

Pollutant	Region	Measured as	Limit, objective or target	Concentration[9]	Maximum no. of exceedences allowed	To be achieved by[11]	Exceedence expressed as percentile[12]
Benzene	UK	Running annual mean	Objective[1]	16.25 μg/m^3 (5 ppb)	-	31 Dec 2003	n/a
	UK	Annual mean	Limit[2-5]	5 μg/m^3 (1.54 ppb)	-	1 Jan 2010	n/a
	Scotland and Northern Ireland	Running annual mean	Objective[1]	3.25 μg/m^3 (1 ppb)	-	31 Dec 2010	n/a
1,3-Butadiene	UK	Running annual mean	Objective[1]	2.25 μg/m^3 (1 ppb)	-	31 Dec 2003	n/a
Carbon monoxide (CO)	UK	Maximum daily running 8-hour mean	Limit[2-5]	10 mg/m^3 (8.6 ppm)	None	31 Dec 2003	100th percentile
	Scotland	Running 8-hour mean	Objective[1]	10 mg/m^3 (8.6 ppm)	None	31 Dec 2003	100th percentile
Lead (Pb)	UK	Annual mean	Limit[2-5]	0.5 μg/m^3	-	31 Dec 2004	n/a
	UK	Annual mean	Objective[1]	0.25 μg/m^3	-	31 Dec 2008	n/a
Nitrogen dioxide (NO_2)	UK	1-hour mean	Limit[2-5]	200 μg/m^3 (105 ppb)	18 times a year	31 Dec 2005	99.79th percentile
	UK	Annual mean	Limit[2-5]	40 μg/m^3 (21 ppb)	-	31 Dec 2005	n/a
Nitrogen oxides (NO_X)	UK	Annual mean	Critical level[2-5]	30 μg/m^3 (16 ppb) [13]	-	31 Dec 2000	n/a

Table C.1 – UK air quality regulations.

Pollutant	Region	Measured as	Limit, objective or target	Concentration[9]	Maximum no. of exceedences allowed	To be achieved by[11]	Exceedence expressed as percentile[12]
Ozone (O_3)	UK	Maximum daily running 8-hour mean	Target[2-5]	120 μg/m^3	25 days per year (averaged over 3 years)	target for 2010	-
	UK	Maximum daily running 8-hour mean	Long-term objective[2-5]	120 μg/m^3	-	-	-
	UK	Maximum daily running 8-hour mean	Objective[1]	100 μg/m^3	10 times a year	31 Dec 2005	97.26th percentile
	UK	AOT40 calculated from 1 hour values from May to July, average over 5 years	Target[2-5]	18,000μg/m^3 × h [13]	-	1 Jan 2010	n/a
	UK	AOT40 calculated from 1 hour values from May to July	Long-term objective[2-5]	6,000μg/m^3 × h [13]	-	-	n/a
Particulate matter (PM_{10})	UK	24-hour mean	Limit[2-5]	50 μg/m^3	35 times a year	31 Dec 2004	90.41th percentile
	UK	Annual mean	Limit[2-5]	40 μg/m^3	-	31 Dec 2004	n/a
	Scotland	24-hour mean	Objective[1]	50 μg/m^3	7 times a year	31 Dec 2010	98.08th percentile
	Scotland	Annual mean	Objective[1]	18 μg/m^3	-	31 Dec 2010	n/a
Particulate matter ($PM_{2.5}$)	UK	Annual mean	Limit[2-5]	25 μg/m^3 [14]	-	1 Jan 2015	n/a
	Scotland	Annual mean	Objective[6]	10 μg/m^3	-	31 Dec 2020	n/a
Sulphur dioxide (SO_2)	UK	15-minute mean	Objective[1]	266 μg/m^3 (100 ppb)	35 times a year	31 Dec 2005	99.90th percentile
	UK	1-hour mean	Limit[2-5]	350 μg/m^3 (132 ppb)	24 times a year	31 Dec 2004	99.73th percentile
	UK	24-hour mean	Limit[2-5]	125 μg/m^3 (47 ppb)	3 times a year	31 Dec 2004	99.18th percentile
	UK	Annual mean	Critical level[2-5]	20 μg/m^3 (8 ppb) [13]	-	31 Dec 2000	n/a
	UK	Winter mean (1st October – 31st March)	Critical level[2-5]	20 μg/m^3 (8 ppb) [13]	-	31 Dec 2000	n/a

Table C.1 – UK air quality regulations. *(continued)*

Pollutant	Region	Measured as	Limit, objective or target	Concentration[9]	Maximum no. of exceedences allowed	To be achieved by[11]	Exceedence expressed as percentile[12]
PAHs (benzo[a]pyrene)	UK	Annual mean	Target[2-5]	1 ng/m^3	-	31 Dec 2012	n/a
	UK	Annual mean	Objective[1]	0.25 ng/m^3	-	31 Dec 2010	n/a
Arsenic	UK	Annual mean	Target[2-5]	6 ng/m^3	-	31 Dec 2012	n/a
Cadmium	UK	Annual mean	Target[2-5]	5 ng/m^3	-	31 Dec 2012	n/a
Nickel	UK	Annual mean	Target[2-5]	20 ng/m^3	-	31 Dec 2012	n/a

Table C.1 – UK air quality regulations. *(continued)*

1. Department for Environment, Food and Rural Affairs, 2007: The Air Quality Strategy for England, Scotland, Wales and Northern Ireland: volume 1. Published in partnership with the Scottish Executive, the National Assembly for Wales and the Department of the Environment for Northern Ireland. Available on https://www.gov.uk/government/policies/protecting-and-enhancing-our-urban-and-natural-environment-to-improve-public-health-and-wellbeing/supporting-pages/international-european-and-national-standards-for-air-quality.
2. The Air Quality Standards Regulations 2010, UK Statutory Instruments 2010 no. 1001.
3. The Air Quality Standards (Wales) Regulations 2010, Wales Statutory Instruments, 2010 no. 1433 (W.126).
4. The Air Quality Standards (Scotland) Regulations 2010, Scottish Statutory Instruments, 2010 no. 204.
5. The Air Quality Standards Regulations (Northern Ireland) 2010, Northern Ireland Statutory Rules, 2010 no. 188.
6. The Air Quality (Scotland) Amendment Regulations 2016, Scottish Statutory Instruments, 2016 no. 162.
7. Department for Environment, Food and Rural Affairs, 2003: The Air Quality Strategy for England, Scotland, Wales and Northern Ireland: Addendum. Published in partnership with the Scottish Executive, the National Assembly for Wales and the Department of the Environment for Northern Ireland.
8. www.environmental-protection.org.uk/air-quality-and-climate/air-quality/.
9. http://www.legislation.gov.uk/
10. Conversions of ppb and ppm to μg/m^3 and mg/m^3 at 20°C and 1013 mb.
11. Date to be achieved by and maintained thereafter.
12. Percentile values are given for reference and are not defined explicitly in the air quality standards.
13. Objective for protection of vegetation and ecosystems.
14. 25 μg/m^3 is a cap to be seen in conjunction with 15% reduction.

C.2 EU limit values

The EU limit values[1,2] are listed in **Table C.2**.

Pollutant	Measured as	Limit concentration[3]	Maximum no. of exceedences allowed	To be achieved by	Exceedence expressed as percentile[4]
Benzene[1]	Annual mean	5 μg/m^3	-	1 Jan 2010[5]	n/a
Carbon monoxide[1] (CO)	Maximum daily running 8-hour mean	10 mg/m^3	None	1 Jan 2005[5]	100th percentile
Lead[1] (Pb)	Annual mean	0.5 μg/m^3	-	1 Jan 2005[5]	n/a
Nitrogen dioxide[1] (NO_2)	1-hour mean	200 μg/m^3 (104 ppb)	18 times a year	1 Jan 2010[5]	99.79th percentile
	Annual mean	40 μg/m^3 (21 ppb)	-	1 Jan 2010[5]	n/a
Nitrogen oxides[1] (NO_X)	Annual mean	30 μg/m^3	-	19 Jul 2001[6]	n/a
Ozone[1] (O_3)	Maximum daily running 8-hour mean	120 μg/m^3	25 days per year (averaged over 3 years)	target for 1 Jan 2010[5]	-
	AOT40[10], calculated from 1-h values from May to July and averaged over 5 years	18000 μg/m^3 × h	-	target for 1 Jan 2010[6]	n/a
	Maximum daily running 8-hour mean within one year	120 μg/m^3	None	long-term objective[5]	100th percentile
	AOT40[10], calculated from 1-h values from May to July	6000 μg/m^3 × h	-	long-term objective[6]	n/a
Particulate matter[1] (PM_{10})	24-hour mean	50 μg/m^3	35 times a year	1 Jan 2005[5]	90.41th percentile
	Annual mean	40 μg/m^3	-	1 Jan 2005[5]	n/a
Particulate matter[1] ($PM_{2.5}$)	Annual mean	25 μg/m^3	-	1 Jan 2010[8]	n/a

Table C.2 – EU limit values.

Pollutant	Measured as	Limit concentration[3]	Maximum no. of exceedences allowed	To be achieved by	Exceedence expressed as percentile[4]
Sulphur dioxide[1] (SO_2)	1-hour mean	350 μg/m^3 (130 ppb)	24 times a year	1 Jan 2005[5]	99.73th percentile
	24-hour mean	125 μg/m^3 (46 ppb)	3 times a year	1 Jan 2005[5]	99.18th percentile
	Annual mean (calendar) and winter mean (1 October to 31 March)	20 μg/m^3 (7.4 ppb)	-	19 Jul 2001[6]	n/a
Arsenic[2]	Annual mean	6 ng/m^3	-	31 Dec 2012	n/a
Cadmium[2]	Annual mean	5 ng/m^3	-	31 Dec 2012	n/a
Nickel[2]	Annual mean	20 ng/m^3	-	31 Dec 2012	n/a
PAHs (benzo[a]pyrene) [2]	Annual mean	1 ng/m^3	-	31 Dec 2012	n/a

Table C.2 – EU limit values. *(continued)*

1. Directive 2008/50/EC of the European Parliament and Council of 21 May 2008 on ambient air quality and cleaner air for Europe.
2. Directive 2004/107/EC of the European Parliament and of the Council of 15 December 2004 relating to arsenic, cadmium, mercury, nickel and polycyclic aromatic hydrocarbons in ambient air.
3. Conversions of ppb and ppm to μg/m^3 and mg/m^3 at 20°C and 1013 mb.
4. Percentile values are given for reference and are not defined explicitly in the air quality standards.
5. Limit value for the protection of human health (see corresponding directive).
6. Critical level for the protection of vegetation/ecosystems (see corresponding directive).
7. AOT40 (see corresponding directive) means the sum of the difference between hourly concentrations greater than 40 ppb (80 μg/m^3) and 80 μg/m^3 over a given period (e.g. growing season) using only the hourly values measured between 8:00 and 20:00 Central European Time each day. It is expressed in μg/m^3 × hours.
8. Target value for 1 Jan 2010, Limit value 1 Jan 2015.

C.3 Lithuanian limit values

The Lithuanian limits are identical to the EU limits (see Section C.2) [1].

1. http://oras.gamta.lt/cms/index?rubricId=260ccbe8-5401-4f3b-adb1-b4ab1b9aa2b5

C.4 French Limit Values

The limit values for France[1] are listed in **Table C.3**.

Pollutant	Measured as	Limit concentration	Maximum no. of exceedences allowed	To be achieved by	Exceedence expressed as percentile[2]
Benzene[3]	Annual mean	5 μg/m^3	-	1 Jan 2010[4]	n/a
Carbon monoxide (CO)[3]	Maximum daily running 8-hour mean	10 mg/m^3	None	1 Jan 2010	100th percentile
Lead (Pb)	Annual mean	0.5 μg/m^3	-	1 Jan 2002[5]	n/a
Nitrogen dioxide (NO_2)[3, 6]	1-hour mean	200 μg/m^3 (104 ppb)	18 times a year	1 Jan 2010[7]	99.79th percentile
	Annual mean	40 μg/m^3 (21 ppb)	-	1 Jan 2010	n/a
Nitrogen oxides	Annual mean	30 μg/m^3 (16 ppb)	-	19 Jul 2001	n/a
Ozone (O_3)[3, 8]	Maximum daily 8-hour mean within one year	120 μg/m^3	25 times a year (average over three years)	1 Jan 2010[9]	100th percentile
	AOT40[9], calculated from 1-h values from May to July	18000 μg/m^3 × h	-	1 Jan 2010[11]	n/a
Particulate matter (PM_{10})	24-hour mean	50 μg/m^3	35 times a year	1 Jan 2005	90.41th percentile
	Annual mean	40 μg/m^3	-	1 Jan 2005[12]	n/a
Particulate matter ($PM_{2.5}$)	Annual mean	25 μg/m^3	-	1 Jan 2015[13]	n/a

Table C.3 – French limit values.

Pollutant	Measured as	Limit concentration	Maximum no. of exceedences allowed	To be achieved by	Exceedence expressed as percentile[2]
Sulphur dioxide (SO_2)[3, 14, 15]	1-hour mean	350 µg/m^3 (130 ppb)	24 times a year	1 Jan 2005	99.73th percentile
	24-hour mean	125 µg/m^3 (46 ppb)	3 times a year	1 Jan 2005	99.18th percentile
	Annual mean (calendar) and winter mean (1 October to 31 March)	20 µg/m^3 (7.4 ppb)	-		n/a
Arsenic	Annual mean	6 ng/m^3	-	31 Dec 2012	n/a
Cadmium	Annual mean	5 ng/m^3	-	31 Dec 2012	n/a
Nickel	Annual mean	20 ng/m^3	-	31 Dec 2012	n/a
PAHs (benzo[a]pyrene)	Annual mean	1 ng/m^3	-	31 Dec 2012	n/a

Table C.3 – French limit values. (*continued*)

1. Code de l'environnement Article R221-1, Modifié par Décret n°2010-1250 du 21 octobre 2010 - art. 1 accessible via www.legifrance.gouv.fr.
2. Percentile values are given for reference and are not defined explicitly in the air quality standards.
3. The volume must be reduced to conditions of temperature and pressure: 293 K and 101.3 kPa.
4. Objective value of 2 µg/m^3.
5. Objective value of 0.25 µg/m^3.
6. Alert concentration limit set to 200 or 400 µg/m^3 hourly mean (see corresponding directive).
7. Up to December 2009 the number of exceedences allowed was 175 (98th percentile).
8. Alert concentration threshold for progressive implementation of emergency measures:

First level: 240 $\mu g/m^3$ average hourly exceeded for three consecutive hours;
Second level: 300 $\mu g/m^3$ average hourly exceeded for three consecutive hours;
Third level: 360 $\mu g/m^3$ averaged hourly.

9. Long-term objective of no exceedences.
10. AOT40 (see corresponding directive) means the sum of the difference between hourly concentrations greater than 40 ppb (80 $\mu g/m^3$) and 80 $\mu g/m^3$ over a given period (e.g. growing season) using only the hourly values measured between 8:00 and 20:00 Central European Time each day. It is expressed in $\mu g/m^3 \times$ hours.
11. Long-term objective value of 6,000 $\mu g/m^3 \times h$
12. Objective value of 30 $\mu g/m^3$ annual average.
13. Objective value of 10 $\mu g/m^3$ annual average.
14. Objective of 50 $\mu g/m^3$ annual average.
15. Alert concentration limit set to 500 $\mu g/m^3$ hourly average exceeded for three consecutive hours (see corresponding directive).

C.5 US National Ambient Air Quality Standards

Table C.4 lists the US National Ambient Air Quality Standards (NAAQS) as given by the U.S. Environmental Protection Agency[1].

Pollutant	Primary or Secondary[2]	Measured as	Limit concentration[3]	Maximum no. of exceedences allowed	Exceedence expressed as percentile[4]
Carbon monoxide (CO)	Primary	1-hour mean	35ppm (40 mg/m^3)	Once a year	99.99th percentile
	Primary	Running 8-hour mean	9 ppm (10 mg/m^3)	Once a year	99.99th percentile
Lead (Pb)	Both	Running 3-month mean	0.15 μg/m^3	-	n/a
Nitrogen dioxide (NO_2)	Both	Annual mean	53 ppb (28 μg/m^3)	-	n/a
	Primary	3-year average of the 98th percentile of the daily maximum 1-hour mean	100 ppb (52 μg/m^3)	-	n/a
Ozone	Both	3-year average of the fourth-highest daily maximum running 8-hour mean over each year	0.070 ppm (137 μg/m^3)	-	n/a
Particulate matter (PM_{10})	Both	24-hour mean	150 μg/m^3	Once a year (on average over 3 years)	99.73th percentile
Particulate matter ($PM_{2.5}$)	Both	3-year average of the 98th percentile of the 24-hour mean	35 μg/m^3	-	n/a
	Secondary	3-year average of the annual mean	15.0 μg/m^3	-	n/a
	Primary	3-year average of the annual mean	12.0 μg/m^3	-	n/a
Sulphur dioxide (SO_2)	Secondary	3-hour mean	0.5 ppm (1300 μg/m^3)	Once a year	99.97th percentile
	Primary	3-year average of the 99th percentile of the daily maximum 1-hour mean	75 ppb (195 μg/m^3)	-	n/a

Table C.4 – US National Ambient Air Quality Standards.

1. National Ambient Air Quality Standards (40 CFR part 50), U.S. Environmental Protection Agency, 2010. Available on https://www.epa.gov/criteria-air-pollutants/naaqs-table (see website for more detail).
2. Primary standards are defined for the protection of public health, whereas secondary standards are defined for the protection of public welfare, for example to minimise effects on visibility and vegetation.
3. mg/m^3 and $\mu g/m^3$ concentrations are approximately equivalent values.
4. Percentile values are given for reference and are not defined explicitly in the air quality standards.

C.6 World Health Organisation guidelines

The World Health Organisation (WHO) guidelines[1,2] are listed in **Table C.5**. These guidelines are currently under review[3].

Pollutant	Measured as	Limit concentration	Maximum no. of exceedences allowed	Exceedence expressed as percentile
Carbon monoxide[1] (CO)	15-minute mean	100 mg/m^3	None	100^{th} percentile
	30-minute mean	60 mg/m^3	None	100^{th} percentile
	1-hour mean	30 mg/m^3	None	100^{th} percentile
	Running 8-hour mean	10 mg/m^3	None	100^{th} percentile
Lead[1] (Pb)	Annual mean	0.5 $\mu g/m^3$	-	n/a
Nitrogen dioxide[2] (NO_2)	1-hour mean	200 $\mu g/m^3$	None	100^{th} percentile
	Annual mean	40 $\mu g/m^3$	-	n/a
Particulate matter[2] (PM_{10})	24-hour mean	50 $\mu g/m^3$	3 times a year	99^{th} percentile
	Annual mean	20 $\mu g/m^3$	-	n/a
Particulate matter[2] ($PM_{2.5}$)	24-hour mean	25 $\mu g/m^3$	3 times a year	99^{th} percentile
	Annual mean	10 $\mu g/m^3$	-	n/a
Ozone[2] (O_3)	Daily maximum running 8-hour mean	100 $\mu g/m^3$	None	100^{th} percentile
Sulphur dioxide[2] (SO_2)	10-minute mean	500 $\mu g/m^3$	None	100^{th} percentile
	24-hour mean	20 $\mu g/m^3$	None	100^{th} percentile
Cadmium[1] (Cd)	Annual mean	5 ng/m^3	-	n/a

Table C.5 – World Health Organisation guidelines.

1. WHO Regional Office for Europe, 2000: Air Quality Guidelines for Europe, Second Edition. WHO Regional Publications, European Series, No. 91. Available from www.euro.who.int.
2. WHO Regional Office for Europe, 2006: Air Quality Guidelines, Global Update 2005 − Particulate Matter, Ozone, Nitrogen Dioxide and Sulfur Dioxide. Available from www.euro.who.int.
3. WHO Expert Consultation: Available evidence for the future update of the WHO Global Air Quality Guidelines (AQGs), 2015: www.euro.who.int/__data/assets/pdf_file/0013/301720/Evidence-future-update-AQGs-mtg-report-Bonn-sept-oct-15.pdf?ua=1

APPENDIX D Surfer Tips

This appendix is intended only as a quick guide to some of the more frequently used features of Surfer, that is:

- how to create contour maps (2D);
- how to create surface maps (3D);
- how to overlay contours on digital map tiles or surface maps;
- how to customise a map; and
- how to save or print surface maps or maps with transparency properties.

This is not a comprehensive guide to all the features of the Surfer software package and it is recommended that users refer to the online help or to the Surfer user manual for further information. The information given in this section is based on Surfer 10, other versions of Surfer may differ.

D.1 Contour maps (2-dimensional)

Contour maps are useful for visualising gridded data such as concentrations, terrain data, etc.

Contour maps of concentrations should be produced using the **2-D Output Plotter (Plotting in Surfer)** utility (refer to Section 6.3 for details). Contour maps of other quantities can be produced as explained below, as long as the data format is XYZ ASCII, i.e. X, Y and Z coordinates in a text file.

There are two steps to creating a contour map:

1. create a grid (*.grd*) file from the original data,
2. make a contour map from the grid file.

Any contour map in Surfer is based on a grid file (.grd*) that contains the data to be plotted interpolated onto a regular grid.*

Open Surfer and create a new plot by clicking on the menu, **File, New** and choosing **Plot**. This example shows how to create a contour map of terrain data.

Step 1 Choose **Grid, Data**... and open the terrain file.

You will need to choose **Files of type: All Files (*.*)** to be able to see *.ter* files and select the terrain file you want to use.

Step 2 On the **Data Import Options** screen (**Figure D.1**), verify that the data will be read in as comma-delimited format and click on **OK**.

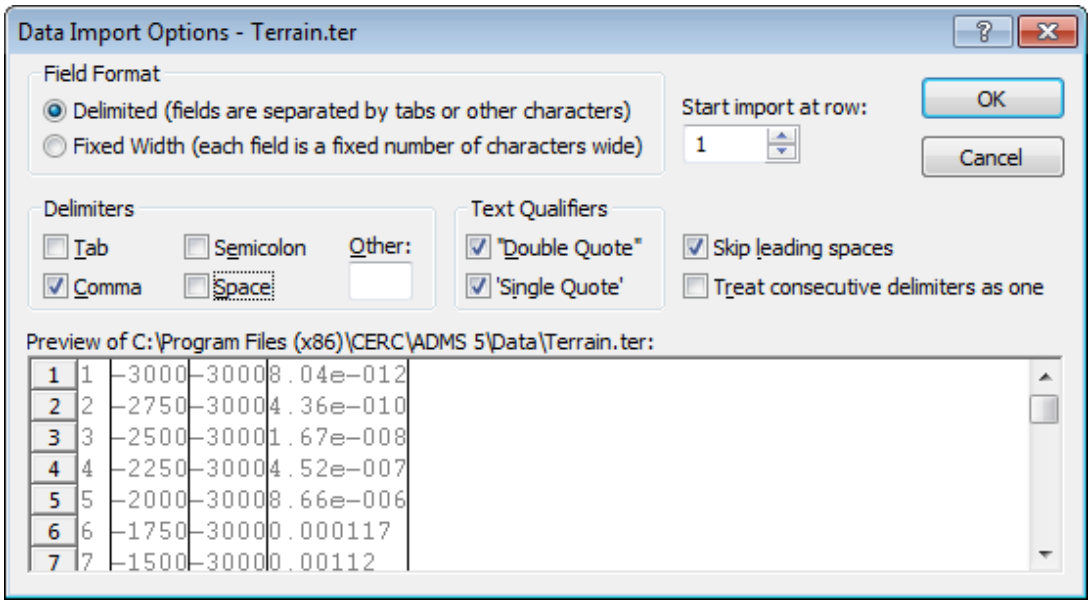


Figure D.1 – Data import options for the terrain file.

Step 3 On the **Grid Data** screen (**Figure D.2**), select columns B, C and D corresponding to X, Y and Z respectively. The first column in the terrain file, column A, contains a counter that is not needed in the plotting.

In this screen you can change the grid file name or location

Click on **OK** to create the grid terrain file.

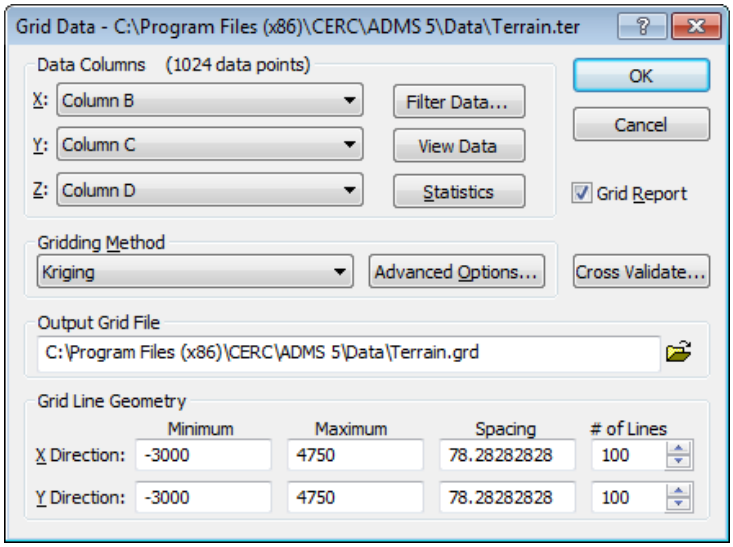


Figure D.2 – Settings for gridding the terrain file.

Step 4 From the menu select **Map**, **New**, **Contour Map**... and then select the grid file just created.

The contour map is created with a default layout. Section D.4 describes how to customise the colours, labels and other features of a map.

An example of a final contour map is shown in **Figure D.3**.

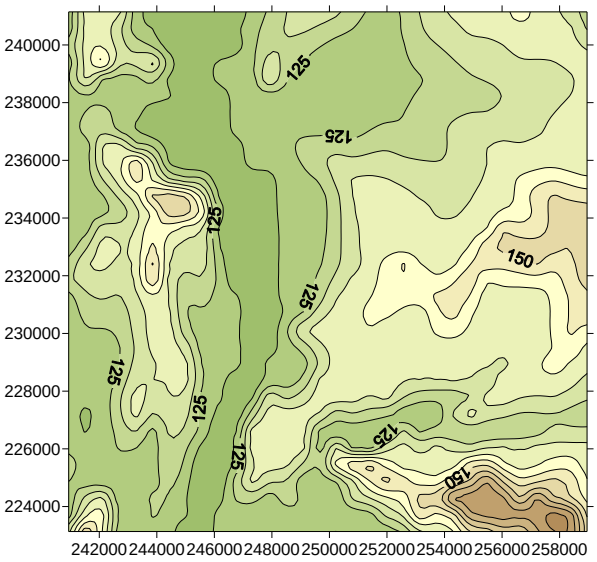


Figure D.3 – Example of a contour map of terrain data.

D.2 Surface maps (3-dimensional)

Creating a 3-D plot, for instance from an ADMS 5 terrain file (*.ter*), is an alternative way of visualising the terrain that can provide useful information as to which wind directions are likely to be most important in terms of dispersion modelling.

Open Surfer and create a new plot by clicking on the menu, **File, New** and choosing **Plot**.

As with the creation of contour maps, you first need to create a grid file (*.grd*) from your data. Create a *.grd* file as described in Section D.1. Click **Map, New, 3D Surface**... to create a 3-D surface map.

The remainder of this section describes how to create a contour map with contour lines (isopleths) from which a 3-D map with isopleths is made.

*Any surface map in Surfer is based on a grid file (.*grd*) which contains the data to be plotted interpolated onto a regular grid.*

D.2.1 Create the grid file

In order to create both 2-D and 3-D maps, you will need a grid file of the data to be plotted. Refer to Section D.1 for instructions on how to create a grid file.

D.2.2 Create the 2-D contour map

Once the grid file has been created, produce the contour map of the terrain data, as explained in Section D.1. Customise the map (see Section D.4).

The appearance of the final 3-D surface map, in terms of contour levels, colours, etc. must first be set up at this stage in the 2-D contour map.

For this example the final contour map of the terrain file is the same as that shown in **Figure D.3**.

D.2.3 Create the 3-D surface map

Once the 2-D contour map has been created and customised, open a new Surfer plot document (**File, New** and choose **Plot**) and create the 3-D surface map, from the same data, as follows.

Step 1 Open the terrain grid file with the **Map, New, 3D Surface**... menu command.

A default 3D view of the terrain file is produced by Surfer as illustrated in **Figure D.4**.

This step could be achieved in the same Surfer plot as that of the 2-D contour map. In such case, go directly to Step 3.

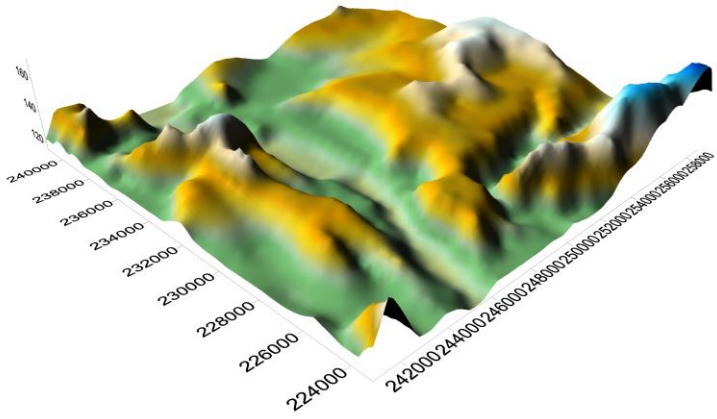


Figure D.4 – Default 3-D view of the terrain file.

Step 2 To overlay the contour plot on the surface plot, go back to the contour plot, copy it (**Edit, Select All** and **Edit, Copy** commands) and paste it onto the surface plot document (**Edit, Paste** command).

Step 3 To set both plots on the same spatial scales, select and overlay them (**Edit, Select All** followed by **Map, Overlay Maps**).

Step 4 In order to use the colours of the contour plot only, click on **3D surface** in the **Object Manger** window. Then click on the **Overlays** tab in the **Property Manager** window. Click on the cell next to **Color modulation** and from the drop down menu select **Use Overlay color only**.

An example of a final 3-D map of a terrain file is shown in **Figure D.5**.

Many elements of the plot can be modified. Please refer to Section D.4 for details of how to do this.

It is often useful to change the perspective of the map, add more details (level intervals), look at the coordinates to check that the required area is covered, etc. Click on **3D Surface** in the **Object Manager** window to access the **Property Manger** window for the 3-D surface properties.

The colours of the surface map can be modified in the **General** tab by clicking on the colour ramp. X and Y grid lines can be overlaid (**Mesh** tab). Light and shadow effects are available in the **Lighting** tab. The **Overlays** tab controls how plots are to be overlaid. The 3-D parameters of the plot can be changed by clicking on **Map** in the **Object manager** window and then clicking on the **View** tab in the **Property Manager** window. The spatial scale and map extents are gathered in the **Scale** and **Limits** tabs.

---

*If you are creating a plot to be printed from a report rather than printing directly from Surfer, you should export the plot as an image (**File, Export...** command) and import the image into your report. Refer to Section D.5 for more details.*

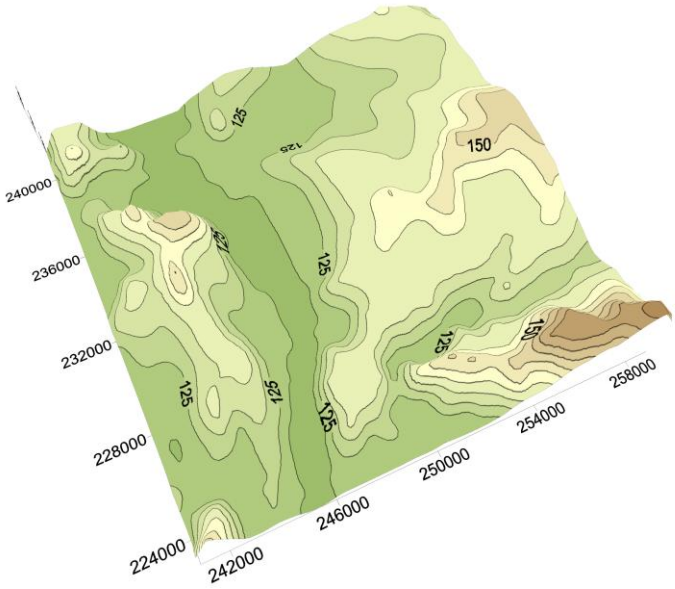
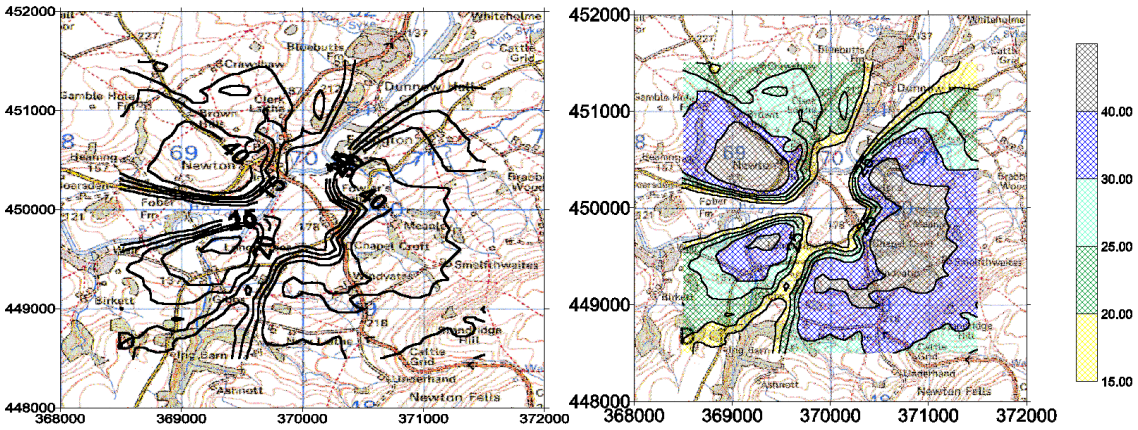


Figure D.5 – Contour plot overlaid on the surface plot.

D.3 Overlay on a digital or surface map

Displaying standard contour maps of concentration, deposition, etc., on top of digital base maps or 3-D surface maps allows attractive layouts for reports to be created without the need to purchase a Geographical Information System (GIS). **Figure D.6** shows examples of a digital map with either overlaid contour lines or colour-filled transparent contours. **Figure D.7** gives an example of site characteristics, such as the locations of the stacks and/or receptors, overlaid on a surface map.

The next section explains how to overlay an ADMS 5 item (contour map of concentrations or site characteristics) on such maps. The same methodology can be used to overlay both types of data.



Reproduced from the Ordnance Survey 1:50,000 colour raster map
with the permission of The Controller of Her Majesty's Stationary Office © Crown copyright.

Figure D.6 – Digital map with contour lines overlaid (left) or colour-filled contours overlaid (right) produced with Surfer.

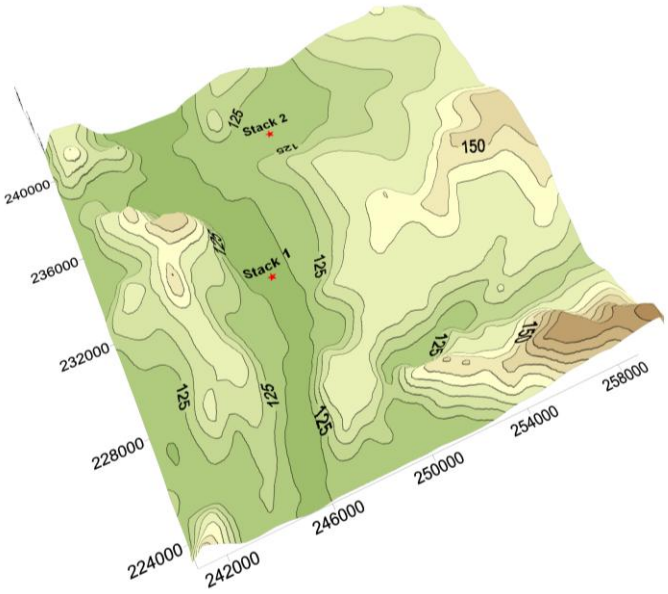


Figure D.7 – Surface map with overlaid site characteristics produced with Surfer.

To overlay an ADMS 5 item on a map there are three steps:

1. prepare the map (digital base map or terrain surface map),
2. prepare the item to overlay (contour map of concentration, stack location, etc.),
3. merge both together to produce the final map.

The map and the item(s) to overlay should use the same coordinate system to ensure that the overlaid item is located at the correct geographical reference point on the map.

D.3.1 Prepare the map

The map can be any image, as long as it has appropriate geospatial information (e.g. uses a world file). Typically the base map will be a digital map (of the region of interest) or a surface map (showing the topography of the region).

Digital base map

Digital map data are available in a large number of graphical formats such as *.tif*, *.dxf*, etc. The tile position can be geographically referenced. For more information about appropriate formats of digital map data, please consult the Surfer user guide.

Step 1 Choose **Map, New, Base Map**... and locate the base map you want to use.

The map is loaded into Surfer as illustrated in **Figure D.8**. In this example, it covers approximately a 5 km × 5 km area. Depending on the extent and the resolution of your map, you might need to zoom into a small section to view the data in more detail and check that the tile is for the area you want.

Note that many different formats of base maps can be loaded, including AutoCAD drawings, bitmaps, Windows metafiles, etc.

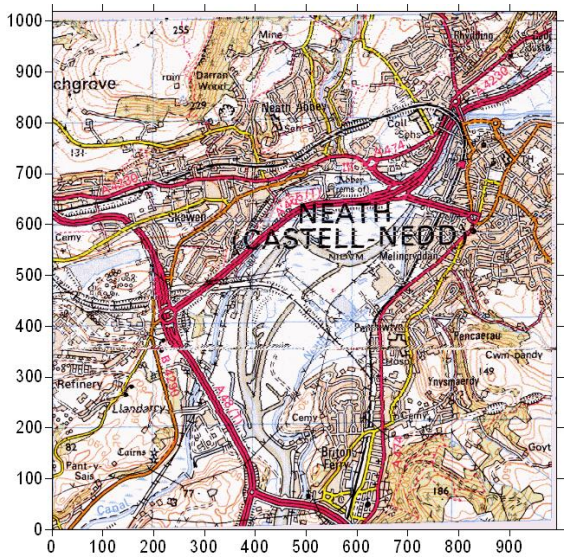


Figure D.8 – Base map loaded into Surfer.

Step 2 If the input base map does not contain geographical information, for instance through a world file, then the X and Y axes are automatically labelled relative to an origin (0,0) at the bottom left-hand corner. As previously explained, it may be necessary to reference the base map geographically to ensure the coordinate system matches that used by ADMS 5. To do this, select **Base** in the **Object Manager window** and then select the **Base Map** tab in the **Property Manager** window. Set the minimum and maximum X and Y coordinates of the map. Click on **OK** to apply the new coordinates to the map as shown in **Figure D.9**.

Step 3 Save this Surfer file.

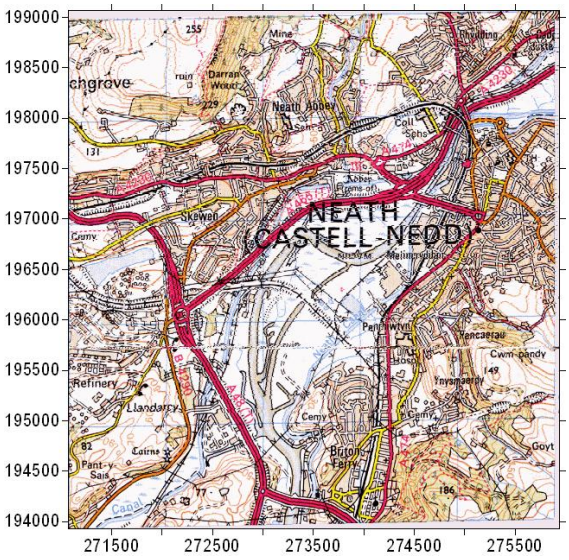


Figure D.9 – Map coordinates synchronised with ADMS 5 coordinates.

Terrain surface map

A surface map of the terrain file can be created in the usual way (refer to Section D.2). When you are happy with its appearance, save it as a Surfer file.

It is always possible to modify the map appearance after an item has been overlaid on it. Just double-click on the surface map to access its appearance properties.

D.3.2 Prepare the item to overlay

Many different items can be overlaid on the map. The most commonly used are a contour map of concentrations and the locations of the stacks and receptors (if any). There is no restriction on the number of items that can be overlaid.

Each item to be overlaid should be prepared as explained below.

Contour map of concentration

Create the Surfer contour map in the usual way (refer to Section D.1), editing the levels, labels, fonts, etc., as detailed previously. Choose whether you require filled contours or just a series of contour lines, although this could be modified once the

contour map has been overlaid on the map. When you are happy with the contour map save it as a Surfer file (*.srf*).

Site characteristics

In this example, we only consider two potential locations of the stack of an industrial site. The stack positions can be imported into Surfer by creating a *Post Map*, as follows.

Step 1 Go to **File**, **New** and select **Worksheet**, then enter the X and Y coordinates of the stack locations and, optionally, the stack names (see **Figure D.10**).

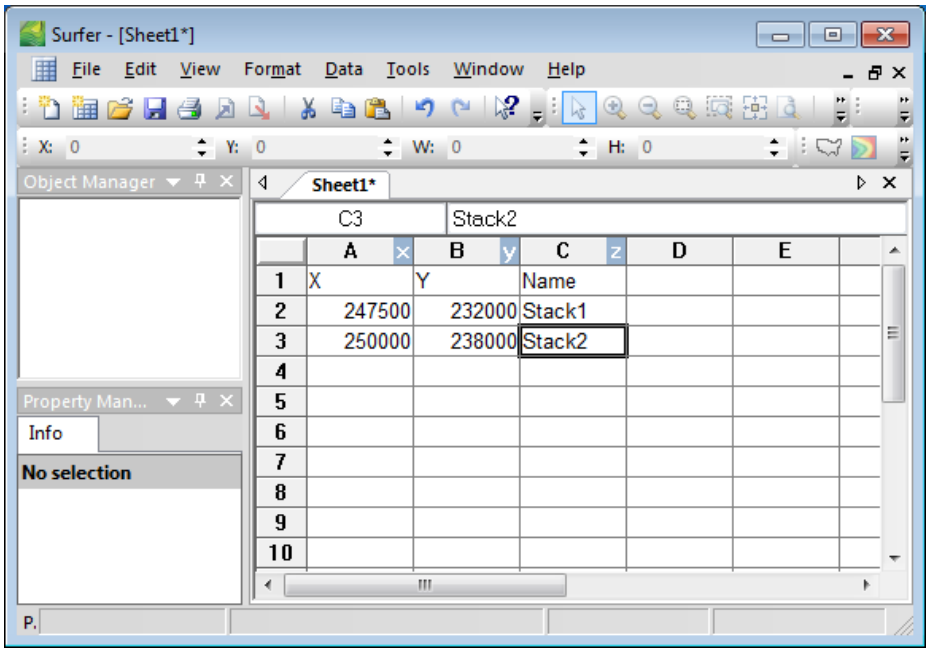


Figure D.10 – Stack locations.

Step 2 Save the file as **Text Data** (*.txt*). A **Data Export Options** window will appear. You can use this to select the **Delimiter** and the **Text Qualifier** before saving the file. Close the worksheet.

D.3.3 Overlay the item on the map

The items can be overlaid one at a time or with several items together.

Most of the appearance properties of the map and the item can be edited after the item has been overlaid on the map. The only exception to this is the coordinate system, which must be defined prior to the overlay step.

Overlay site characteristics

To overlay the stack locations on a map, proceed as follows.

Step 1 Open the digital base map or the terrain surface map previously saved.

Step 2 Click on **Map**, **New**, **Post Map**... and select the text file previously created. This adds the stack locations to the map.

Step 3 To put the two images on the same spatial scale, click on **Edit, Select All** and then on **Map, Overlay Maps**. The stack locations are overlaid on the map.

Figure D.7 shows an example of the final 3D view of the terrain file with two possible locations of the stack of an industrial site.

You can edit the post map properties by clicking on **Post** in the **Object Manage**r window. Several options are then available in the **Property Manager** window. The **General** tab allows you to modify the stack symbol. In the **Labels** tab, you can choose to display the name of the stacks (if they are in the post map file) and set the font, colour and size to use to display the labels.

Overlay a contour map of concentration

To overlay the contour map of concentrations on a map, proceed as follows.

Step 1 Open the digital base map or the terrain surface map previously saved.

Step 2 Open the contour map you wish to overlay.

Step 3 Click on the contour map, choose **Edit, Select All** and then **Edit, Copy**. Click on the map, choose **Edit, Paste**.

Step 4 To put the two images on the same spatial scale, click on **Edit, Select All** and then on **Map, Overlay Maps**.

*If the contour map does not appear, it may lie below the map image. Right-click on **Base** in the **Object Manager** window and select **Order Objects**, **Move to Back** to set it as the bottom layer or select the **Arrange**, **Order Objects**, **Move to Back** menu command. The contour map will then be located on top of the base map.*

Figure D.11 shows an example of a contour map overlaid on a digital base map.

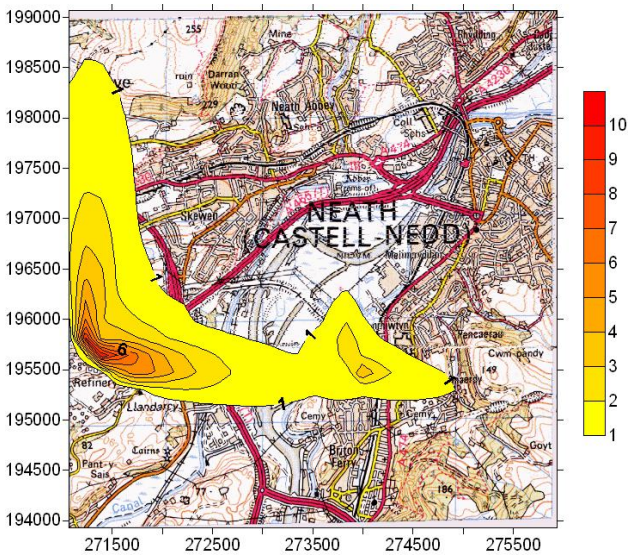


Figure D.11 – Contour plot of concentration overlaid on a base map.

Once the plot has been produced, you may wish to modify the map extent, change its scale, or modify the layout (colour-filled contours). If colour-filled contours are chosen, they should be made transparent so that the map underneath is visible. Refer to

Section D.4 for an explanation of how to do this.

D.4 Customise a map

The easiest way to customise a feature present on a contour, surface or post map is to click on that feature in the **Object Manager** window (**View, Managers, Object Manager** menu command) to access the relevant **Property Manager** window (**View**, **Managers**, **Property Manager** menu command). Modifications can be applied to the map by editing the information in the **Property Manager** window and further modifications can be made by moving through the various tabs.

This section contains explanations of how to edit and modify the most common features of the map (contour levels, colour, pattern and labels; line colour, thickness and pattern; colour scale; axes and grid lines; map scale and extent). The examples will use the contour map shown in **Figure D.12** and the contour map overlaid on a surface map shown in **Figure D.11**.

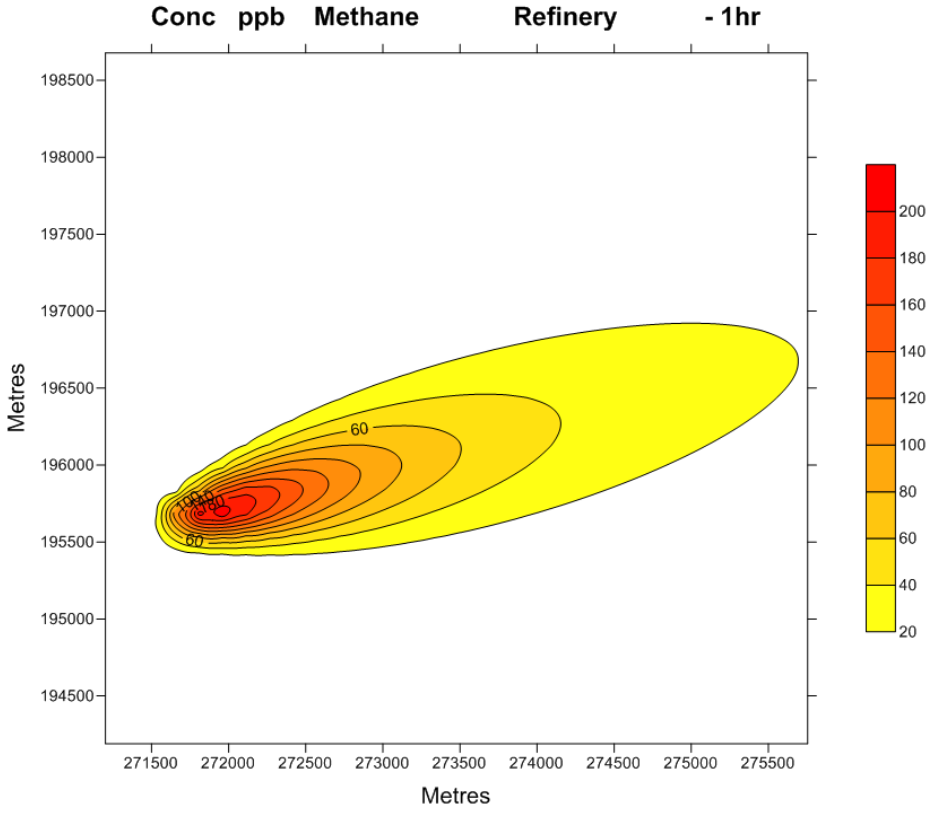


Figure D.12 – Example of contour plot of concentrations.

To access the properties of the contour map, click on **Contours** in the **Object Manager** window.

D.4.1 Levels

Levels can be edited by selecting the **Levels** tab in the **Property Manager** window for the contours. Simple modifications can be made by editing the values shown, but more advanced manipulation can be performed by clicking on **Simple** next to the **Level Method** item and selecting **Advanced** from the drop down menu. Clicking on **Edit Levels**... brings up a dialogue box for editing the levels properties, as shown in **Figure D.13**.

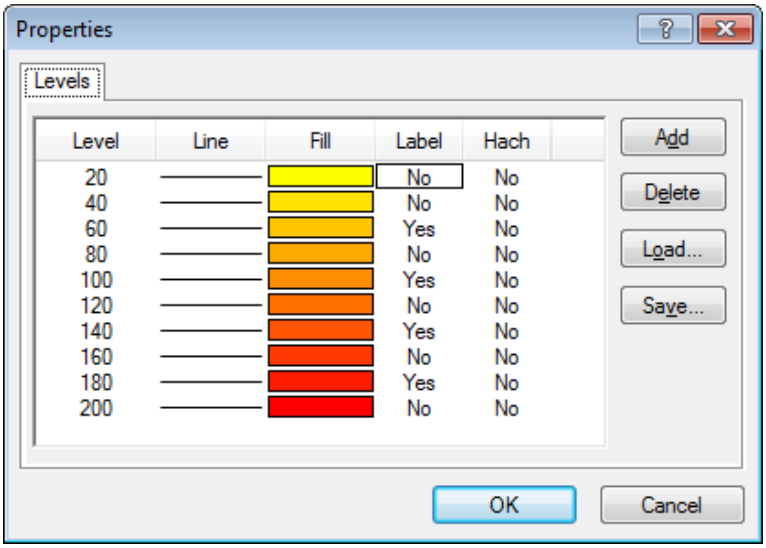


Figure D.13 – **Levels** editor screen.

The contour levels can be edited in three ways:

- *All existing levels together*: click on the **Level** button on top of the level column to specify regular intervals. This may change the colours of the filled contours to a grey/black scheme but colour scheme and pattern can easily be changed.
- *One existing level at a time*: double-click on the value of the level to edit and change the value to the one you want.
- *Add a level between two existing ones*: to add a new contour level (e.g. 190 in **Figure D.13**), click on the next highest level (200 in this case) and then on the **Add** button. The new contour will be added, i.e. the halfway value between the one you clicked on (200) and the one below it (180).

Once you are happy with the contour levels that are shown in the contour plot, you can save them in a level file (*.lvl*) in order to re-use the configuration for other plots, for example plots you want to compare to each other. Click on **Save**... and save the level file with a unique name. This will not only save the contour levels but also the choice of colours, pattern of the filled contours, labels, line thickness and colour, label fonts, etc. (i.e. most of the parameters that can be edited in **Figure D.13**).

To apply an existing level file to a contour plot, select **Advanced** from **Level method** and click on **Edit Levels**... as described above to bring up the properties dialogue box. Click on the **Levels** tab and then on the **Load**... button, and select the particular *.lvl* file required.

D.4.2 Contour colour and pattern

To edit the colour and pattern of contour maps, select **Edit Levels**... as described before to access the screen shown in **Figure D.13**. Click on the **Fill** button at the top of the Fill column to open the screen shown in **Figure D.14**.

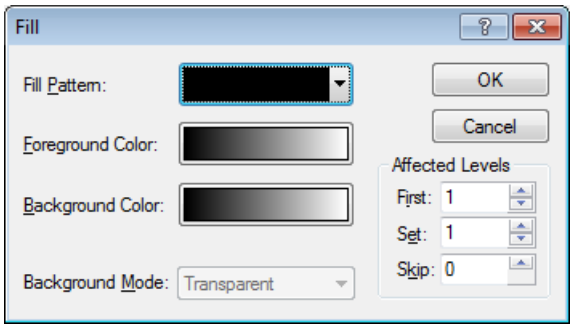


Figure D.14 – Setting the spectrum of colours and patterns for the contour plot.

By default, the pattern style is solid but other patterns such as lines, checks, etc., are available. For some patterns the choice will become available to make the pattern either **Opaque** or **Transparent**.

Click on the **Foreground Color** coloured box to access the screen shown in **Figure D.15**. In this **Colormap** screen, you can choose the colours of the minimum and maximum contour levels as well as those for any intermediate level. The result will be a ramp of colours ranging from the minimum to the maximum colour (passing through intermediate level colours if specified).

---

*You must make sure that the **Fill Contours** box is checked in the **General** tab of the **Property Manager** for the **Contours** layer in order for the colours to appear on the contour plot. If you just want a contour plot with labelled contour lines then leave this box unchecked.*

Alternatively, the user can specify a unique colour for every contour level. Similarly to the level values, each individual colour can be edited by double-clicking on the coloured box.

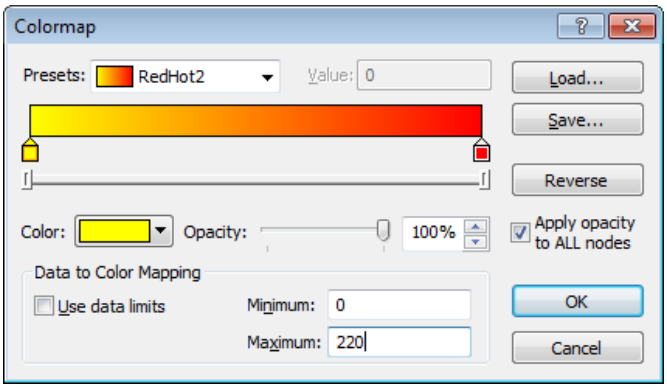


Figure D.15 – Choosing colours for filling contours.

Transparent colour-filled contours

Transparency is needed when you overlay a contour map of concentrations on a base map. Solid colour is used by default for the contours and the map underneath is not visible.

To set the transparency of colour-filled contours, proceed as follows.

Step 1 Select **Advanced** for the **Level method** and click on **Edit Levels**... as described

above to open the window shown in **Figure D.16**.

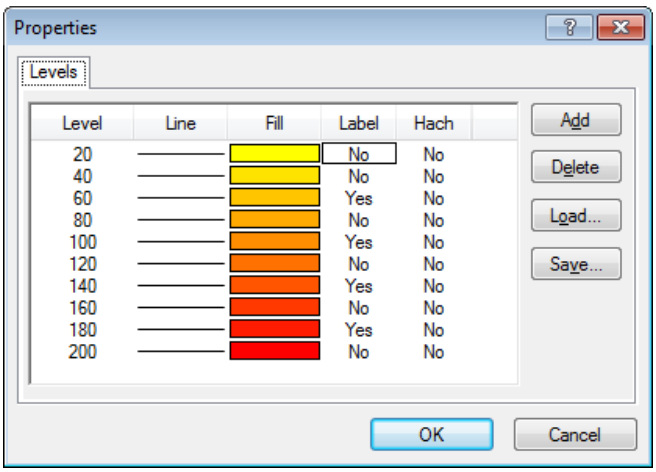


Figure D.16 – Contour properties.

Step 2 To set the transparency, click on the **Fill** button (at the top of the column of coloured boxes). This opens the screen shown in **Figure D.17**.

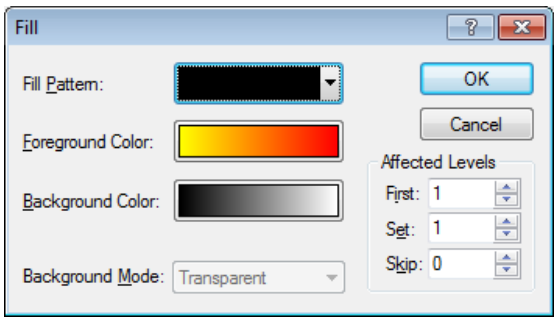


Figure D.17 – Fill pattern options of contour plots.

A transparent background only works with vector pattern fills such as diagonal cross, crosshatch, forward slash, backward slash, vertical, and horizontal.

Select the diagonal cross pattern, for example, and set the **Background Mode** to **Transparent** as shown in **Figure D.18**.

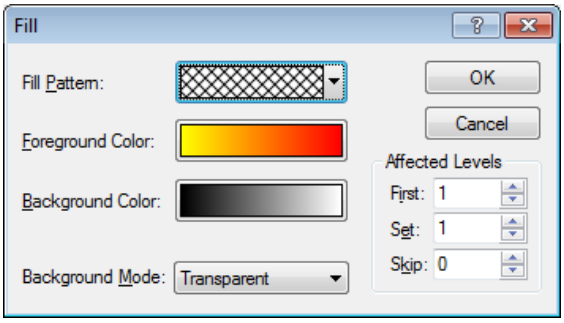


Figure D.18 – Setting the transparency of colour-filled contours.

Figure D.19 gives an example of a transparent filled-colour contour plot of concentration overlaid on a digital base map.

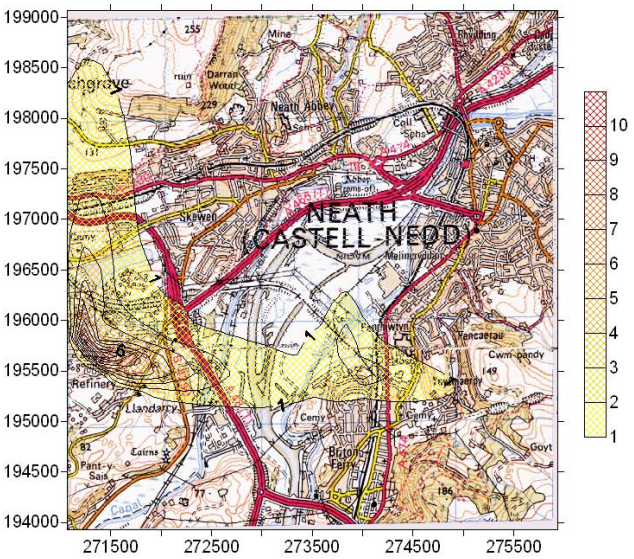


Figure D.19 – Example of transparent filled-colour contour plot of concentration overlaid on a digital base map.

*Copying and pasting a plot with transparency properties into another document (for example a Microsoft Word document) maintains the transparency properties in the image you see on the screen, but not when the document is printed. When you print, the transparency and/or the colours are lost. To avoid this, export the plot as an image (**File, Export...** command). See Section D.5 for more detail.*

D.4.3 Line colour, thickness and pattern

Colour, thickness and pattern of some or all the contour lines in the Surfer plot can be modified. To do so, in the levels editor screen shown in **Figure D.13**, click on the **Line** button to access the line properties. By default the lines are solid black lines but their colour, pattern and thickness can be modified. The lines can all be the same colour and thickness, they can ramp from minimum to maximum attributes or each have a unique colour and thickness.

D.4.4 Contour labels

By default the label format is usually set to label contours using numbers with up to two decimal places. If the numbers in an ADMS 5 output file are very small, then the contours will all be labelled 0.00 on the map. It is a common mistake to think that this is due to a fault in the software. The solution is to change the labelling format: in the levels editor screen shown in **Figure D.13** click on the **Label** button to access the properties of the contour labels.

The labelling frequency can be edited so as to label every line, every two lines, etc. The format and font of labels can also be adjusted: the size or colour changed, decimal digits added, exponential format selected (for example, for very small numbers) etc.

D.4.5 Colour scale

When a contour plot has filled-colour contours, it is useful to include a colour scale on the plot. This is included by default when contour plots are generated from the **2-D Output Plotter (Plotting in Surfer)** utility. As for the contour labels, the colour scale labels are also numbers with up to two decimal places as the default. To change the format of these labels click on the colour scale bar on the contour plot to access the **Property Manager** window. The label format and font can be changed in the same way as for the contour line labels.

D.4.6 Axes and grid lines

In order to access the properties of an axis, click on the axis to display the setting in the **Property Manager** window. A number of features can be edited here, such as the label format and angle, the ticks, the scale of the plot, and the grid lines.

It can be very useful to have grid lines at specific intervals on top of the contour map, and an example is shown in **Figure D.20**.

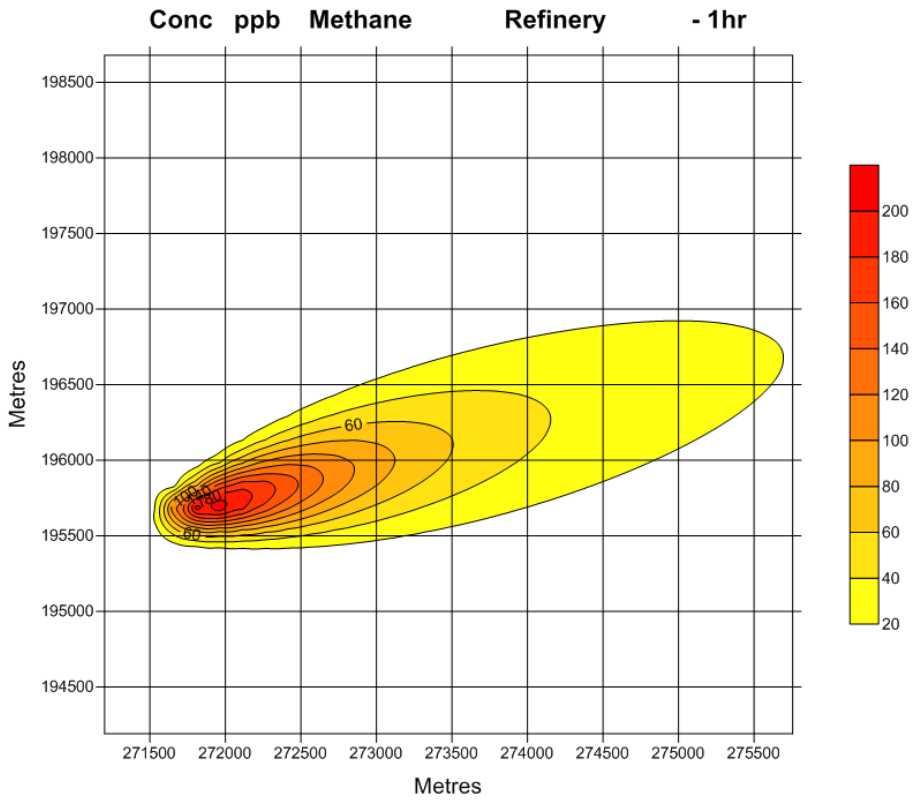


Figure D.20 – Use of grid lines superimposed on contour plot.

In the **Property Manager** window, click on the **Grid Lines** tab and check the **Show** check box of the **Major Grid Lines** (and **Minor Grid Lines** if you wish the minor grid lines to be shown too). The grid lines' properties (style, colour, width) can be modified by clicking on the button showing the line. Major and minor grid line intervals are defined by the major interval of the axis label (**Scaling** tab) and the minor ticks options (**Ticks** tab).

D.4.7 Scale and extent

To modify the scale of a map, click on **Map** in the **Object Manager** window to view the relevant **Property Manager** window and select the **Scale** tab. If you want the map to occupy more or less of the page then you can change the scale or stretch the map by dragging one of the corners. The **Proportional XY Scaling** option is useful to maintain the same scales in both X and Y directions.

*When you open the **Property Manager** for the scale and extent, the units for the X and Y scales may be in map units per inch. To change this to centimetres, go to **Tools**, **Options** and change the **Page Units** under the **Environment**, **Drawing** menu.*

Extent of a base map

It is often useful to make the base map extent a little larger than the extent of the contour map so that the surrounding land use can be viewed, or to zoom in on one particular area of the map.

To do this, click on **Map** in the **Object Manager** window to view the **Property Manager** window for it, select the **Limits** tab and indicate the new limits. Note that the scale is automatically at the scale of the original base map.

D.5 Save or print a map with transparency properties

Maps with transparency properties may not print the image as you see it on the screen. Copying and pasting a map from Surfer into another application is not enough to keep the map appearance characteristics. The colours and the transparency are usually lost through the process.

The solution is to export the map as an image using the **File, Export**... command and save it as an image (**Figure D.21**). Suitable formats to maintain both perspective and transparency for printing are *Windows Bitmap (.bmp)*, *CompuServe Bitmap (.gif)*, *JPEG Compressed Bitmap (.jpg)*, *Portable Network Graphics Bitmap (.png)*, and *Tagged Image (.tif)*. The *Windows Picture (Enhanced) (.emf)* does not keep the transparency or the colour ramp.

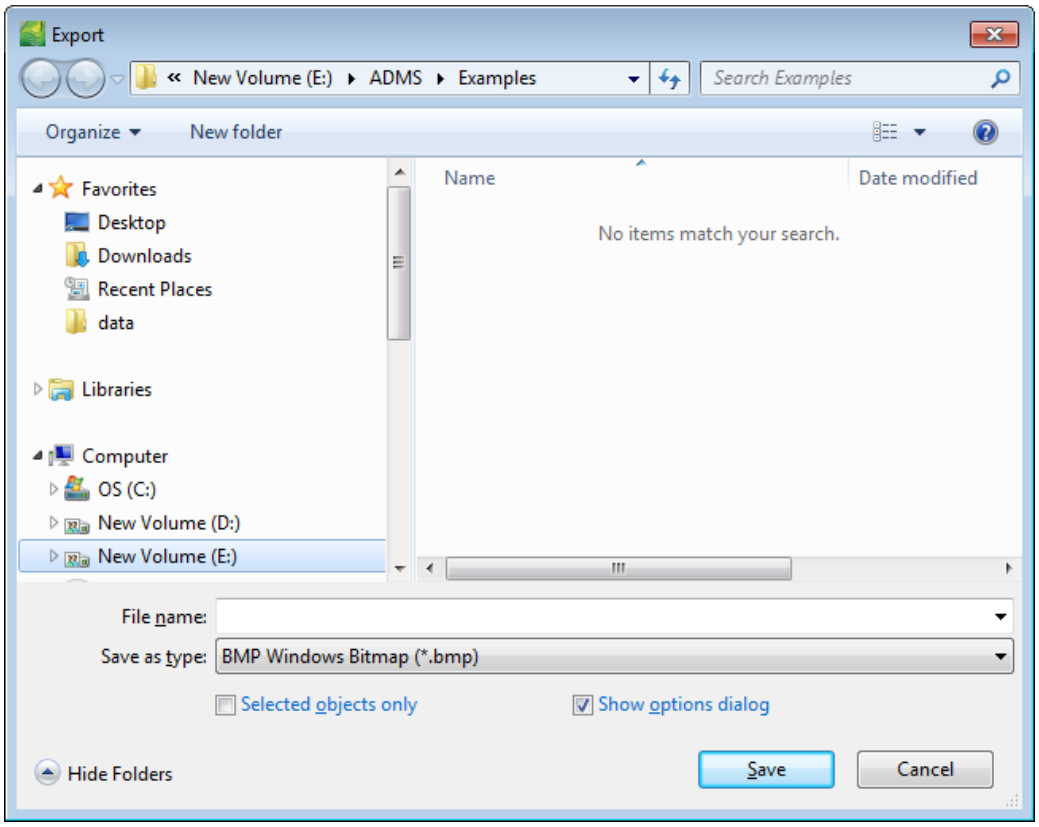


Figure D.21 – Save the map as an image.

APPENDIX E Useful Contacts

This appendix contains information on:

1. ADMS 5: contact, Helpdesk, support contracts;
2. where to obtain meteorological and terrain data for ADMS 5;
3. visualisation tools for ADMS 5: Surfer, ArcGIS, MapInfo; and
4. official organisations.

E.1 ADMS 5 contact information

ADMS Helpdesk

Tel: (01223) 357 773 – ask for ADMS Helpdesk
Fax: (01223) 357 492 marking your fax "FAO ADMS Helpdesk"
Email: help@cerc.co.uk Website: cerc.co.uk

This service is available to those with valid support contracts (see below) between the hours of 9.30am and 5.00pm (GMT in winter, BST in summer), Monday to Friday (excluding UK public holiday periods and the period 25th December to 1st January).

Support contract

A valid support contract entitles the user to

- use of the ADMS Helpdesk;
- model upgrades;
- access to further technical advice and downloads on the User Area of the CERC website cerc.co.uk/user-area/overview.html
- attendance at the annual ADMS User Group meeting; and
- regular newsletters.

All annual licence holders are entitled to support during the period of the licence. Permanent licence holders receive free support for one year, after which an annual support contract should be purchased.

Scope of the ADMS Helpdesk service

The scope of the ADMS Helpdesk Service is to provide answers to specific questions about using ADMS 5, such as *"How do I incorporate time-varying emission factors?"*, to respond to any reported error messages that occur while running the model, and to record and report on any issues found. Where appropriate, CERC staff can also provide advice on setting up particular modelling scenarios and advice on interpreting the results.

Surfer, ArcGIS and MapInfo problems should be directed to Golden Software, ESRI and MapInfo, respectively.

E.2 Contact details for ADMS 5 input data

Meteorological data

Meteorological data should be ordered in a format suitable for input into ADMS 5. Most ADMS 5 users will require hourly sequential data for one or more years.

For the UK, meteorological data in ADMS format are available from a variety of suppliers, including the UK Met Office (metoffice.gov.uk). Hybrid datasets combining, for example, wind speed and direction from one location with cloud cover data from another may be appropriate; users are advised to discuss such options with their provider.

For France, meteorological data are available from Météo France (meteofrance.com). Supplied data include wind speed and direction measured at 10 m in height (for the last 10 minutes), temperature, cloud cover (note that it is usually measured only at the main station of each département, and that very often numerous night data are missing), and rainfall.

Terrain data

In order to model the effect of terrain upon dispersion, you will need to obtain terrain data. Many raster formats, e.g. OS terrain 50 or SRTM data, can be converted into the correct format for ADMS 5 using the ADMS Mapper, refer to Section 5.6 of the ADMS Mapper User Guide for more details. For some older formats the ADMS Terrain Converter can be used, refer to Section 7.4 of this document for details. The **ADMS Terrain Converter** utility can convert the following formats:

- For sites in Great Britain, Landform PANORAMA or PROFILE data in NTF format.
- For sites in Northern Ireland, OS-NI (Ordnance Survey-Northern Ireland) data.
- For sites in France, BD ALTI 50, 75 or 100 (50, 75 or 100 m spatial resolution) data in XYZ ASCII format, provided by the Institut Géographique National (IGN).

Background data

Background concentration data for a wide range of sites in the UK are available from the UK National Air Quality Information Archive website

uk-air.defra.gov.uk/data/data_selector

Background data may also be available from local authorities

E.3 Output visualisation tools

Surfer

Golden Software, Inc.
809 14th Street
Golden
CO 80401-1866
U.S.A.

Tel: +1 (303) 279 1021 Email: info@goldensoftware.com or surfersupport@goldensoftware.com
Fax: +1 (303) 279 0909 Web: goldensoftware.com

ArcGIS

For queries about ArcGIS, contact the local ESRI distributor for your country (details may be obtained from esri.com).

For the UK, the local distributor is:

ESRI (UK) Ltd.
Millennium House
65 Walton Street
Aylesbury
HP21 7QG
United Kingdom

Tel: +44 (0)1296 745 500 Email: sales@esriuk.com
Fax: +44 (0)1296 745 544 Web: esriuk.com

MapInfo

For queries about MapInfo, contact the local MapInfo distributor for your country

mapinfo.com/product/mapinfo-professional/

E.4 Official organisations

Department for Environment, Food and Rural Affairs (Defra)
Defra is responsible for the UK air quality strategy.

Web: gov.uk/government/organisations/department-for-environment-food-rural-affairs

Environment Agency (EA)
The Environment Agency regulates air quality in England.

Web: gov.uk/government/organisations/environment-agency

Scottish Environment Protection Agency (SEPA)
SEPA regulates air quality in Scotland.

Web: sepa.org.uk

Natural Resources Wales (NRW)
NRW regulates air quality in Wales.

Web: naturalresourceswales.gov.uk

Northern Ireland Environment Agency (NIEA)
The NIEA regulates air quality in Northern Ireland.

Web: daera-ni.gov.uk/northern-ireland-environment-agency

European Environment Agency (EEA)
Web: eea.europa.eu

World Health Organization (WHO)
Web: who.int

APPENDIX F References

This is the main reference list for this User Guide, in particular Section 9.

Validation studies

ADMS 5 has been validated against a number of data sets in order to assess various configurations of the model such as flat or complex terrain, line/area/volume sources, buildings, dry deposition and visible plumes. The model results have been compared to observational data or other model results if available. Documents or presentations containing results of ADMS 5 validation can be found on the CERC website cerc.co.uk.

The full ADMS 5 Technical Specification is available from the CERC website at the address cerc.co.uk/environmental-software/technical-specifications.html.

Published technical papers

The following publications provide additional technical information on the ADMS 5 model, its validation and application.

Carruthers, D.J., Edmunds, H.A., Lester, A.E., McHugh, C.A. and Singles, R.J., 2000: Use and validation of ADMS-Urban in contrasting urban and industrial locations. In *Int. J. Environment and Pollution*, **14**, pp. 364-374.

Carruthers, D.J., Dixon, P., McHugh, C.A., Nixon, S.G. and Oates, W., 2001: Determination of Compliance with UK and EU Air Quality Objectives From High-Resolution Pollutant Concentration Maps Calculated Using ADMS-Urban. In *Int. J. of Environment and Pollution*, **16**, pp. 460-471.

Carruthers, D.J., Dyster, S. and McHugh, C.A., 2000: Contrasting methods for validating ADMS using the Indianapolis dataset. In *Int. J. Environment and Pollution*, **14**, pp. 115-121.

Carruthers, D.J., Dyster, S. and McHugh, C.A., 2003: Factors affecting interannual variability of NO_X and NO_2 concentrations from single point sources. In *Clean Air and Environmental Protection*, **33**, 1, pp. 15-20.

Carruthers, D.J., Edmunds, H.A., Bennett, M., Woods, P.T., Milton, M.J.T., Robinson, R., Underwood, B.Y. and Franklyn, C.J., 1995: Validation of the UK-ADMS Dispersion Model and Assessment of its Performance Relative to R91 and ISC using Archived LIDAR Data. Study commissioned by Her Majesty's Inspectorate of Pollution, published by DoE. DoE/HMP/RR/95/022.

Carruthers, D.J., Holroyd, R.J., Hunt, J.C.R., Weng, W-S., Robins, A.G., Apsley, D.D., Thomson, D.J. and Smith, F.B., 1994: UK-ADMS: a new approach to modelling dispersion in the Earth's atmospheric boundary layer. In *J. of Wind Engineering and Industrial Aerodynamics*, **52**, pp. 139-153.

Carruthers, D.J., Holroyd, R.J., Hunt, J.C.R., Weng, W-S., Robins, A.G., Apsley, D.D., Smith, F.B., Thomson, D.J. and Hudson, B., 1991: UK Atmospheric Dispersion Modelling System. In *Proceedings of the 19th NATO/CCMS International Technical Meeting on Air Pollution Modelling and its Application, September 1991, Crete, Greece*. Eds. Han van

Dop and George Kallos, Plenum Publishing Corporation, New York.

Carruthers, D.J., McKeown, A.M. and McHugh, C.A., 1997: Air dispersion study of options for abatement of SO_2 emissions at Castle Cement, Clitheroe. In *Environment Agency Report NW-9/97-25/B-BADQ.*

Carruthers, D.J., McKeown, A.M., Ellis, K.L. and McHugh, C.A., 1997: Air dispersion climate from the Castle Cement Works at Ribblesdale. In *Environment Agency Report NW-7/97-20-BAZSQ.*

Carruthers, D.J., McKeown, A.M., Hall, D.J. and Porter, S., 1999: Validation of ADMS against wind tunnel data of dispersion from chemical warehouse fires. In *Atmospheric Environment*, **33**, pp. 1937-1953.

CERC, 1998: Turbulence, Fluctuations and Averaging Times. CERC Technical Note, 22 December 1998.

CERC, 2015: ADMS 5 Technical Specification.

Dyster, S.J., Thomson, D.J., McHugh, C.A. and Carruthers, D.J., 2001: Turbulent Fluctuations and Their Use in Estimating Compliance Standards and in Model Evaluation. In *Int. J. of Environment and Pollution*, **16**, pp. 57-68.

Fackerell, J.E. and Robins, A.G., 1982: Concentration fluctuations and fluxes in plumes from point sources in a turbulent boundary layer. In *J. Fluid Mech.*, **117**, pp. 1-26.

Hunt, J.C.R., Holroyd, R.J., Carruthers, D.J., Robins, A.G., Apsley, D.D., Smith, F.B. and Thomson, D.J., 1990: Developments in Modelling Air Pollution for Regulatory Purposes. In *Proceedings of the 18th NATO/CCMS International Technical Meeting on Air Pollution Modelling and its Application*, Vancouver, Canada.

Irwin, J.S., Carruthers, D.J., Stocker, J., and Paumier, J., 2003: Application of ASTM D6589 to evaluate dispersion model performance. In *Int. J. Environment and Pollution*, **20**, pp. 4-10.

McHugh, C.A., Carruthers, D.J., Higson, H. and Dyster, S.J., 1999: Comparison of Model Evaluation Methodologies with Application to ADMS 3 and US Models. Paper 58 in *Proceedings of the 6th International Conference on Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes, Rouen, 11-14 October 1999.*

Pasquill, F. and Smith, F.B., 1983: Atmospheric Diffusion - Study of the dispersion of windborne material from industrial and other sources. Third edition. Ellis Horwood Ltd, Chichester.

Robins, A. and McHugh, C.A., 2001: Development and Evaluation of the ADMS Buildings Effects Module. In *Int. J. of Environment and Pollution*, **16**, pp. 161-174.

Thomson, D.J., 1990: A stochastic model for the motion of particle pairs in isotropic high-Reynolds number turbulence and its application to the problem of concentration variance. In *J. Fluid Mech.*, **210**, pp. 113-153.

Thomson, D.J., 1997: The 'inertial-meander' sub-range of a dispersing plume. In *J. Applied Meteorology*, **36,** pp. 1046-1049.

Timmis, R., Wilkinson, S., Carruthers, D.J. and McHugh, C.A., 2000: Recent Studies to Validate and Compare Atmospheric Dispersion Models for Regulatory Purposes in the UK. In *Int. J. of Environment and Pollution*, **14**, pp. 431-442.

Others

The following publications are references to papers and reports on dispersion relevant to the technical content of ADMS 5.

Azzi, M., Johnson, G. and Cope, M.E., 1992: An Introduction to the Generic Reaction Set Photochemical Smog Mechanism. In *Proceedings of the 11th International Conference of the Clean Air Society of Australia and New Zealand (Brisbane, Australia, July 1992*), **2**, pp. 451-462. Eds. P. Best, N. Bofinger, D. Cliff.

Belcher S.E, Coceal, O., Hunt, J.C.R., Carruthers, D.J., and Robins, A.G., 2013: A review of urban dispersion modelling. ADMLC report, available at http://webarchive.nationalarchives.gov.uk/20131103234051/http://www.admlc.org.uk/ADMLCReport7.htm (accessed November 2016)

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.

Bevilaqua, P.M. and Lykoudis, P.S., 1978: Turbulence memory in self-preserving wakes. In *J. Fluid Mech.*, **89**, pp. 589-606.

Briggs, G.A., 1969: *Plume Rise*. U.S. Atomic Energy Commission Div. Tech. Inf.

Briggs, G.A., 1985: Analytical parameterizations of diffusion - the convective boundary layer. In *J. Clim. Appl. Met.*, **14**, pp. 1167-1186.

Carruthers, D.J., Hunt, J.C.R. and Weng, W.-S., 1988: A computational model of stratified turbulent airflow over hills - FLOWSTAR I. In *Computer Techniques in Environmental Studies*, pp. 481-492. Ed. P. Zanetti, Springer-Verlag.

Caughey, S.J. and Palmer, S.G., 1979: Some aspects of turbulence structure through the depth of the convective boundary layer. In *Quart. J. R. Met. Soc.*, **105**, pp. 811-827.

Cimorelli, A.J, Perry, S.G., Venkatram, A., Weil, J.C., Paine, R.J., Wilson, R.B., Lee, R.F., Peters, W.D., Brode, R.W. and Paumier, J.O., 2004: AERMOD Description of Model Formulation. United States Environmental Protection Agency, Office of Air Quality Planning and Standards, Emissions Monitoring and Analysis Division, Research Triangle Park, EPA-454/R-03-004.

CLAG, 1994: Critical Load of Acidity in the United Kingdom. Summary Report of the Critical Loads Advisory Group, Institute of Terrestrial Ecology, Penicuik, Midlothian.

Clarke, R.H., 1979: A model for short and medium range dispersion of radionuclides released into the atmosphere. NRPB Report NRPB-R91.

Davies, B.M., Jones, C.D., Manning, A.J. and Thomson, D.J., 2000: Some field experiments on the interaction of plumes from two sources. In *Quart. J. R. Met. Soc.*, **126**, pp. 1343-1366.

DiNenno, P.J., Beyler, C.L., Walton, W.D., Custer, R.L.P. and Watts, J.M. (ed.), 1988: SFPE Handbook of Fire Protection Engineering. Society of Fire Protection Engineers, Boston, MA.

Drysdale. D., 1999: An Introduction to Fire Dynamics. Second edition. John Wiley & Sons.

Dzhelepov, B.S. and Peker, L.K., 1961: Decay schemes of radioactive nuclei. Pergamon.

EN 13725:2003: Air Quality - Determination of odour concentration by dynamic olfactometry.

Fannelöp, T.K., 1994: Fluid Mechanics for Industrial Safety and Environmental Protection.

Elsevier, Industrial Safety Series, Vol. 3.

Fischer, H.B., List, E.J., Koh, R.C.Y., Imberger, J. and Brooks, N.H., 1979: Mixing in Inland and Coastal Waters. Academic Press.

Gray, D.E. (ed.), 1972: American Institute of Physics Handbook (third edition). McGraw-Hill, New York.

Griffiths, R.F., 1991: The use of probit expressions in the assessment of acute population impact of toxic releases. In *J. Loss Prev. Process Ind.*, **4**, pp. 49-57.

Gultepe I., Milbrandt J., Belair S., (2006) Visibility paramaterisation from microphysical observations for warm fog conditions and its application to the Canadian MC2 model, AMS meeting, Atlanta, January 2006

Hanna, S.R., Briggs, G.A. and Hosker, R.P., 1982: Handbook of Atmospheric Diffusion. US Dept of Energy office of Scientific and Technical Information Publication DOE/TIC-22800.

Hanna, S.R. and Paine, R.J., 1989: Hybrid plume dispersion model (HPDM) development and evaluation. In *J. Appl. Meteorol.*, **28**, pp. 206-224.

Hanna, S.R., Egan, B.A., Purdum, J. and Wagler, J., 1999: Evaluation of ISC3, AERMOD and ADMS dispersion models with observations from five field sites. Hanna Consultants Report P020. Fairfax, VA.

Hansen, M.O.L., 2008: Aerodynamics of wind turbines. Second edition. Earthscan.

HMIP, 1993: Guidelines on Discharge Stack Heights for Polluting Emissions. Environmental Protection Act 1990, Technical Guidance Note (Dispersion) D1, June 1993. HMSO.

Holtslag, A.A.M. and van Ulden, A.P., 1983: A simple scheme for daytime estimates of the surface fluxes from routine weather data. In *J. Clim. Appl. Met.*, **22**, pp. 517-529.

Hunt, J.C.R., 1985: Turbulent diffusion from sources in complex flows. In *Ann. Rev. Fluid Mech.*, **17**, pp. 447-458.

Hunt, J.C.R., Holroyd, R.J and Carruthers, D.J., 1988a: Preparatory studies for a complex dispersion model. CERC Report HB9/88.

Hunt, J.C.R., Leibovich, S. and Richards, K.J., 1988b: Turbulent shear flow over hills. In *Quart. J. R. Met. Soc.*, **114**, pp. 1435-1470.

Hunt, J.C.R., Richards, K.J. and Brighton, P.W.M., 1988c: Stably stratified flow over low hills. *Quart. J. R. Met. Soc.*, **114**, pp. 859-886.

Hunt, J.C.R., Stretch, D.D. and Britter, R.E., 1988d: Length scales in stably stratified turbulent flows and their use in turbulence models. In *Proceedings IMA Conference on Stably Stratified Flow and Dense Gas Dispersion*, pp. 285-322. Ed. J.S. Puttock, Clarendon Press.

Lakes Environmental Consultants Inc., 2003: Flare sources. In *Proposed guidance for air dispersion modelling*, p. 22. Ontario Ministry of the environment.

Lamb, R.G., 1982: Diffusion in the convective boundary layer. In *Atmospheric Turbulence and Air Pollution Modelling*, pp. 159-230. Eds. F.T.M. Nieuwstadt and H. van Dop, D. Reidel, Dordrecht.

Lawson, R.E., Snyder, W.H. and Thompson, R.S., 1989: Estimation of maximum surface concentrations from sources near complex terrain in neutral flow. In *Atmospheric Environment*, **23**, pp. 321-331.

Lederer, C.M., Hollander, J.M. and Perlman, I., 1968: Table of isotopes. Sixth edition. New York, Wiley.

Macdonald, R.W, Griffiths, R.F. and Hall, D.J., 1998: An improved method for estimation of surface roughness of obstacle arrays. In *Atmospheric Environment*, **32**, pp. 1857–1864

Moore, D.J. and Lee, B.Y., 1982: An asymmetrical Gaussian plume model. Central Electricity Generating Board Report RD/L/2224N81.

Nuffield Science Foundation, 1984: Nuffield Advanced Science - Book of Data. First revised edition. Longman.

Oke, T.R., 1987: Boundary layer climates. Second edition. Methuen.

Panofsky, H.A. and J.A. Dutton, 1984: Atmospheric Turbulence, Models and Methods for Engineers and Scientists. John Wiley and Sons, New York.

Perry, S.G., 1991: CTDMPLUS: A dispersion model for sources near complex topography. Part I: Technical formulations. In *J. Appl. Meteor.*, **31**, pp. 633-645.

Sehmel, G.A. 1980: Particle and gas dry deposition: a review. In *Atmospheric Environment*, **14**, pp. 983-1011.

Snyder, W.H. and Lawson, R.E., Jr., 1991: Fluid modelling simulation of stack-tip downwash for neutrally buoyant plumes. In *Atmospheric Environment*, **25A**, pp. 2837-2850.

U.S. E.P.A., 1995: User's Guide for the Industrial Source Complex (ISC3) Models. Volume I: User Instructions (Report EPA-454/B-95-003a) ; Volume II: Description of model algorithms (Report EPA-454/B-95-003b), September 1995. U.S. E.P.A., Office of Air Quality Planning and Standards, Emissions Monitoring and Analysis Division, Research Triangle Park, NC 27711.

van Ulden, A.P. and Holtslag, A.A.M., 1985: Estimation of atmospheric boundary layer parameters for diffusion applications. In *J. Clim. Appl. Met.*, **24**, pp. 1194-1207.

Venkatram, A., 1977: Internal boundary-layer development and fumigation. In *Atmospheric Environment*, **11**, pp. 479-482.

Venkatram, A., Karamchandani, P., Pai, P. and Goldstein, R., 1994: The Development and Application of a Simplified Ozone Modelling System (SOMS). In *Atmospheric Environment*, **28**, pp. 3665-3678.

Weil, J.C., 1985: Updating applied diffusion models. In *J. Clim. Appl. Met.*, **24**, pp. 1111-1130.

CERC

Cambridge Environmental Research Consultants Ltd
3 King's Parade, Cambridge, CB2 1SJ, UK
Tel: +44 (0)1223 357 773, Fax: +44 (0)1223 357 492
Email: help@cerc.co.uk
Website: www.cerc.co.uk