THE AIR POLLUTION MODEL (TAPM) VERSION 1: USER MANUAL

Peter Hurley

CSIRO Atmospheric Research
Internal Paper No. 12

October, 1999

CSIRO Atmospheric Research
PB 1, Aspendale, Victoria 3195, Australia
Ph (+61 3) 9239 4400; Fax: (+61 3) 9239 4444

TABLE OF CONTENTS

1 INTRODUCTION

The Air Pollution Model (TAPM) predicts three-dimensional meteorology and air pollution concentrations. Technical details of the model equations, parameterisations, and numerical methods are described in the Technical Paper by Hurley (1999), along with some example simulations. This User Manual describes how to run TAPM from the Graphical User Interface (GUI) provided with the model.

TAPM_GUI is a PC-based interface that allows the user to set up and run TAPM under the Windows 98/NT4 operating system. It connects to databases of terrain, vegetation and soil type, sea surface temperature, and synoptic-scale meteorological analyses for the Australian, New Zealand, Malaysian and Indonesian regions. Model inputs can be selected and saved, the model can be run and model outputs can be analysed through the interface.

Section 2 of this Manual describes how to configure and run the software, lists the databases of input information available to the software, and discusses some limitations of the modelling approach. Section 3 gives details on the use of TAPM_GUI and Section 4 describes the input and output files used by TAPM.

2 GETTING STARTED

2.1 Configuring and Running the Software

TAPM_GUI can be installed by using the Start-Settings-Control Panel-Add/Remove Programs utility in Windows and selecting the setup.exe program in the TAPM_SETUP directory provided. It is recommended that the directory name should be something like C:\TAPM. Note that the directory name must not contain spaces (e.g. 'Program Files'), as spaces in directory or file names can cause problems in some of the command-line batch files used by the GUI, and the name is restricted to be less than or equal to ten characters. After installing the software, the GUI can be started by running the TAPM_GUI.EXE program from the installed directory. Setting-up a Windows short-cut is probably the easiest way to use the software. An alternative version of the software called TAPM_GUI_INV.EXE allows more point sources to be used, but with a shorter emission time cycle (see Section 3.1.2).

The other software optionally needed is the Graphical Information System (GIS), which can be installed through the Start-Settings-Control Panel-Add/Remove Programs utility and the setup.exe program in the GIS_SETUP directory provided. The GIS can either be run separately by running the GIS_PC1.EXE program, or through the Analyse Output – GIS Visualisation menu option from within the TAPM_GUI interface.

To update to a new version of the software, the user needs to uninstall the old version first (using the Add/Remove Programs utility), and then install the new version as described above.

2.2 The Databases

Some databases are provided on CD free of charge to TAPM users, and include gridded terrain height, vegetation and soil type, sea surface temperature, and synoptic-scale meteorology, for regions covering Australia, New Zealand or Malaysia and Indonesia. A condition of the use of these datasets with TAPM is that the data sources must be acknowledged in any publications, and that the data cannot be passed on to or used by unlicenced TAPM users.

The terrain height datasets currently available are:

- Global terrain height data on a longitude/latitude grid at 30 second grid spacing (approximately 1 km) for five tiles covering the regions listed above. It is based on public

domain data available from the US Geological Survey, Earth Resources Observation Systems (EROS) Data Center Distributed Active Archive Center (EDC DAAC). The web site address is http://edcwww.cr.usgs.gov/landdaac/dataproducts.htm.

- Australian terrain height data on a longitude/latitude grid at 9 second grid spacing (approximately 0.3 km). These data are not public domain and so are not provided with the model (except to CSIRO users), but if a non-CSIRO user wishes to purchase a licence to use the data from the Australian Land Information Group (AUSLIG), a program can be provided to read the AUSLIG data and write them into a binary format that is accessible by the software. The web site address for AUSLIG is http://www.auslig.gov.au.

The vegetation and soil type datasets currently available are:

- Australian vegetation and soil type data on a longitude/latitude grid at 3 minute grid spacing (approximately 5 km). These are public domain data provided by CSIRO Wildlife and Ecology.
- Global land cover characterisation data on a longitude/latitude grid at 30 second grid spacing (approximately 1 km) for tiles covering the non-Australian regions listed above. They are based on public domain data available from the US Geological Survey, Earth Resources Observation Systems (EROS) Data Center Distributed Active Archive Center (EDC DAAC). The data are in a preliminary unverified form and should be examined for the application region by the user before running the model. No soil type information is available for non-Australian regions, and a default type (sandy clay loam) has been assumed. The web site address is http://edcwww.cr.usgs.gov/landdaac/dataproducts.htm.

The sea surface temperature datasets currently available are:

- Rand's global long-term monthly mean sea surface temperatures on a longitude/latitude grid at 1 degree grid spacing (approximately 100 km). They are based on public domain information available from the US National Center for Atmospheric Research (NCAR). The web site address is http://www.scd.ucar.edu/dss/catalogs/index.html.

The synoptic scale meteorology datasets currently available are:

- Six-hourly synoptic scale analyses on a longitude/latitude grid at 0.75 degree grid spacing (approximately 75 km). The database is derived from LAPS analysis data from the Bureau of Meteorology (BoM), who have kindly allowed us to provide the data used by TAPM free of charge. See Puri *et al.*, 1998, for a description of LAPS. The web site address for BoM is http://www.bom.gov.au.

2.3 Some Limitations of the Modelling Approach

- TAPM is suitable for horizontal domain sizes below approximately 1000 km by 1000 km. It should not be used for larger domains because of the neglect in the model of the curvature of the earth, the use of a constant coriolis force and the assumption of horizontally invariant synoptic-scale pressure, temperature, and sea surface temperature fields.
- TAPM cannot be used to accurately represent deep atmospheric circulations or extreme weather events, due to the above reasons, the assumption of incompressibility in the model, and the fact that non-hydrostatic effects are not represented above 5000 m. The winds, temperature and humidity are increasingly smoothed from this level up to the model top at 8000 m, in order to minimise reflections of waves from the model top back into the lower part of the model.

- TAPM cannot be used for very steep terrain because of the use of a terrain following coordinate system in the model. This approach cannot represent discontinuities in terrain height (for example, cliffs or bluff bodies).
- TAPM assumes that cloud processes are resolved by the typical inner grid spacings used in the model (i.e. 3 km or less), and that ice and snow processes are not important. Therefore no large-scale cloud convection parameterisation is included in the model.
- The GRS photochemistry option in the model may not be suitable for examining small perturbations in emissions inventories, particularly in VOC emissions, due to the highly lumped approach taken for VOCs in this mechanism. VOC reactivities should also be chosen carefully for each region of application.

3 TAPM_GUI

TAPM_GUI is a graphical user interface that allows TAPM to be configured for a particular application (see Section 2.1 for installation information). The interface allows the user select model inputs, including those from various databases (see Section 2.2), and then run the model. After the model has run, the user can process the data in various ways through the interface and analyse the results. The selections made in TAPM_GUI can be saved in a Default File (*.def) to allow the same settings for a particular model run to be re-loaded at some later time. The **File – Open/Save GUI Default File** menu items are available on the main window menu selection bar (see Figure 1). The **File – Exit** menu item allows the user to exit the GUI; the user is prompted to optionally save the Default File on exit.

3.1 Selecting Inputs, Accessing Databases, and Running the Model

The main inputs can be selected through the controls on the Main Window shown in Figure 1. Below we describe the various sections of this Window.

The **DataBase Directory** section specifies where the database input files (terrain, meteorology, etc) are stored, which may be on a CD drive, a hard disk drive, or a mapped network drive. The appropriate directory needs to be selected before extracting database information.

The **Grid Centre Coordinates** section requires selection of a latitude and longitude coordinate corresponding to the centre of the grids that are set up in the **Grid Parameters** section. This position is used in the database retrieval process to locate the correct position for extraction of input information. A local coordinate system can also be set up; the default system has the origin at the centre latitude and longitude. The local system uses units of metres in the usual (x, y) Cartesian system where the x is positive from west to east and y is positive from south to north, and specifies the centre of the local system with respect to the latitude/longitude grid centre. This local coordinate system is used both in locating pollution sources (see Section 3.1.2), and in the Analyse Output option (Section 3.2) to map selected grid points for output to the local coordinate system.

The **Date Parameters** section sets the start and end dates for the simulation in the form yyyymmdd (e.g. 19970608 corresponds to the 8 June 1997). The value of the Local Solar Time (LST) relative to Greenwich Mean Time (GMT) is also displayed for information (calculated from the selected grid centre longitude). LST is not affected by daylight saving and approximately corresponds to the regional standard time. For example, Eastern Standard Time on the east coast of Australia is EST = GMT + 10 and Western Standard Time in Western Australia is WST = GMT + 8.

The **Extra Surface Parameters** section allows the user to specify the deep soil moisture content in units of m^3 m^{-3} (i.e. volume of water per volume of soil). This value should correspond to the average deep soil moisture content for the duration of the model run. The model may be particularly sensitive to this parameter for short duration runs or case studies, but for longer duration runs an average seasonal/yearly value can be used. Data for this parameter are not generally available, so the default value is set at 0.15, which is a reasonable value to use for most times of the year in regions dominated by sandy clay loam soil (wilting value of 0.175 and saturation value of 0.420). Generally, for sandy soils and dry months a lower value should be used (e.g. 0.05-0.10), while for clay soils and wet months a higher value is appropriate (e.g. 0.20-0.25).

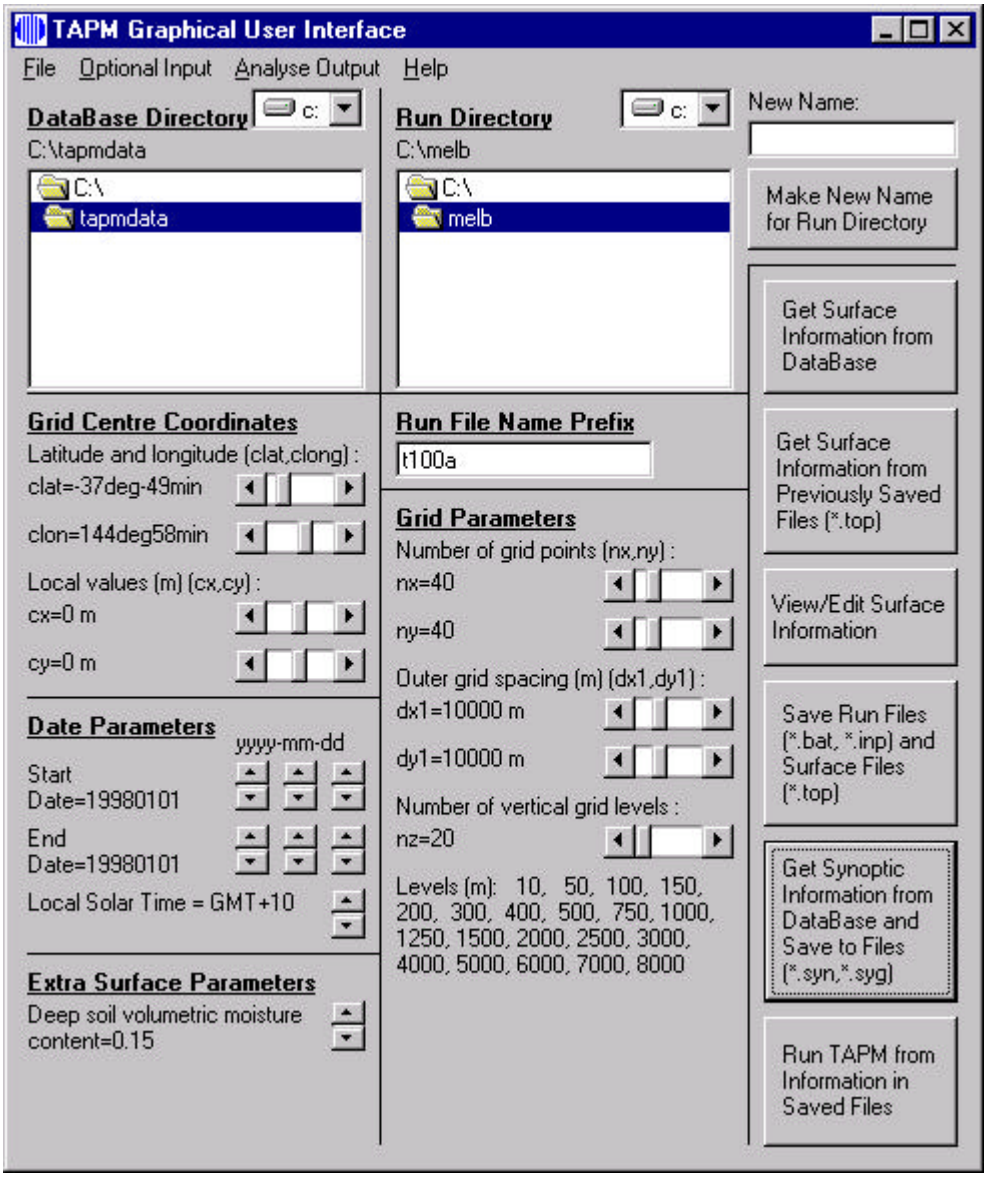

Figure 1. Main Window.

The **Run Directory** section specifies where to store the input files set up by TAPM_GUI operations and the output files from the TAPM run. A new run directory may be created using the text-box and button provided.

The **Run File Name Prefix** section specifies the file name prefix used for the outer grid that is used for various input and output files. For example, if the run file name prefix is set to t100a, then the model run files will be named t100a.bat, t100a.inp, t100a.lis, t100a.top, t100a.syn, t100a.syg, t100a.pse, t100a.gse, t100a.bse. See Section 4 for a description of file name types used by the model.

The **Grid Parameters** section specifies the number of west-east (x) and south-north (y) grid points, the outer grid spacing (m) in these directions, and the number of pre-set staggered vertical grid points. The same number of grid points and the same vertical model levels are used for each grid nest (see Section 3.1.1). It is recommended that a minimum number of grid points for a realistic outer grid simulation should be 20x20x20 points in the x, y, z directions respectively with an outer grid spacing of 10000 m. Preferably more grid points should be used in order to remove the boundary regions as far away as possible from the central region of interest; however, this will increase the model run time.

The buttons on the right of the main window shown in Figure 1 are used once the model simulation has been configured. They extract surface information and synoptic analyses from the databases, save model input files and allow the model to be run. The **Get Surface Information from Database** button extracts terrain height, vegetation type and soil type from files contained in the currently selected database directory. Alternatively, this information may be recalled using the **Get Surface Information from Previously Saved Files (*.top)** button if files (*.top and *.inp) exist in the current Run Directory with the Run File Name Prefix that matches the current GUI selections. Once this information has been obtained, it can be viewed and edited in the **Surface Window** (see below) by pressing the **View/Edit Surface** button. The **Save Run Files (*.bat, *.inp) and Surface Files (*.top)** button saves these files in the Run File Directory using the Run File Name Prefixes for each grid. For example, if there are three grids selected with file name prefixes for the outer to inner grids of t100a, t30a and t10a respectively, then the files t100a.bat, t100a.inp, t100a.top, t30a.top and t10a.top will be saved. The **Get Synoptic Analyses from Database and Save to Files (*.syn, *.syg)** button accesses synoptic meteorological and sea surface temperature analyses from the currently selected Database Directory and outputs this information to files using the Run File Name Prefix (e.g. t100a.syn and t100a.syg). Once the **Save Run Files (*.bat, *.inp) and Surface Files (*.top)** button and the **Get Synoptic Analyses from Database and Save to Files (*.syn, .syg)** button have been pressed, then the model can be run using the saved files by pressing the **Run TAPM from Information in Saved Files** button. When the **Run TAPM from Information in Saved Files** button is pressed, the user is also prompted to save the GUI Default File (*.def). Note that optional input, which includes nested domains and pollution calculations, should be set up before saving files and running the model (see Sections 3.1.1 and 3.1.2).

The **Surface Window** shown in Figure 2, allows the user to view and optionally edit the terrain height, vegetation type and soil type. This information is stored in the (*.top) files for each grid when the **Save Run Files (*.bat, *.inp) and Surface Files (*.top)** button is pressed. The grid domain (e.g. t100a, t30a, t10a) and surface type can be selected and viewed, and the information at any grid point in the domain can be selected by using the left-mouse button in the picture region. If edit mode is turned on, then the Grid Value can be modified using the slider control. A range of grid points can be selected and changed by holding down the left-mouse button, dragging out a rectangular region, and then changing the slider control. Any changes to the gridded information will be saved in the *.top files when the **Save Run Files (*.bat, *.inp) and Surface Files (*.top)** button on the **Main Window** is pressed, but changes made on one grid nest will not affect the other grid nests. The Show Grid Lines check box controls whether grid lines are overlayed on the picture. The Show Network Sites and Show Site Names check boxes control

whether site positions and labels are shown in the picture respectively, while the Edit Network Sites button allows the user to update the network.

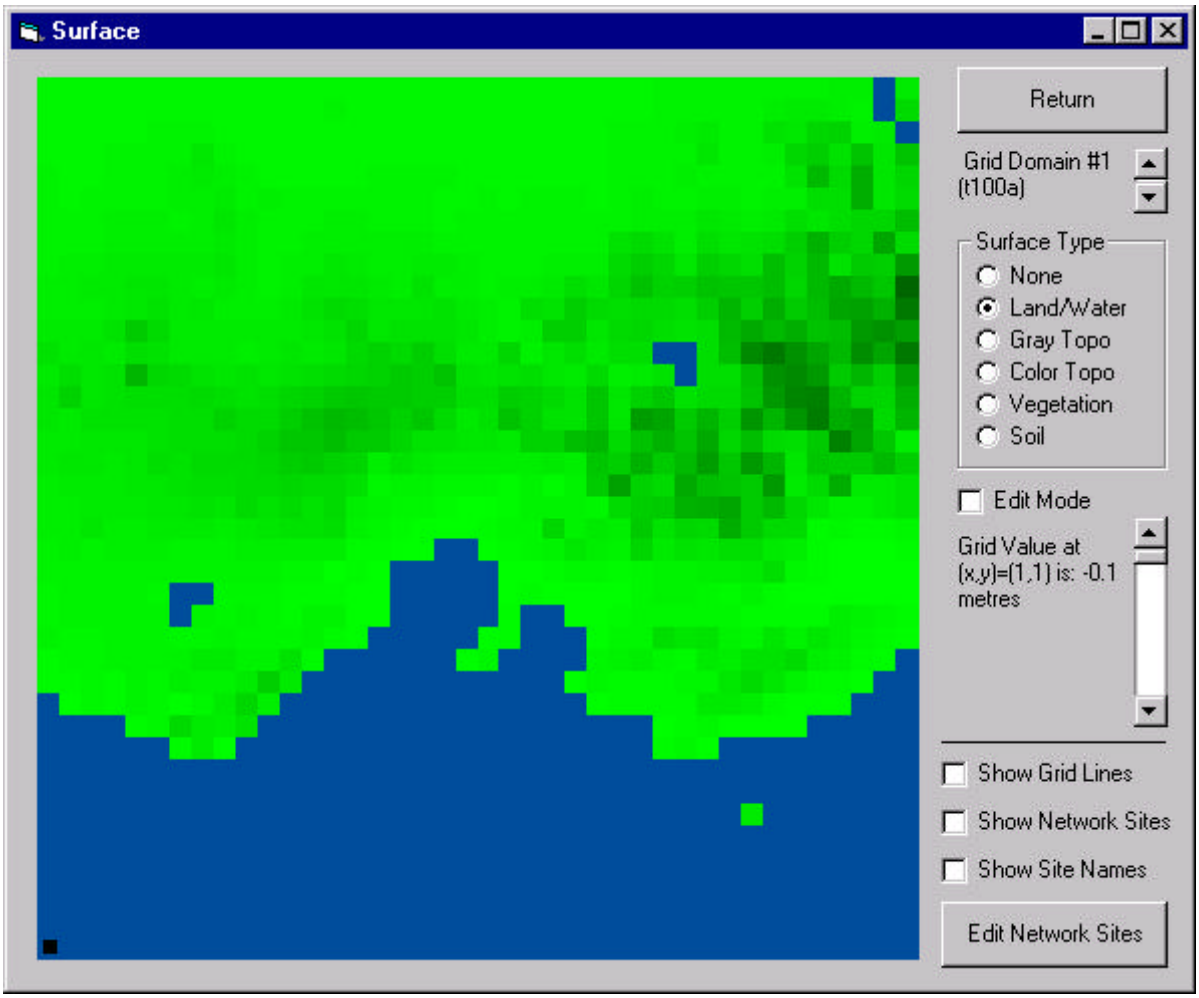

Figure 2. View/Edit Surface Information Window.

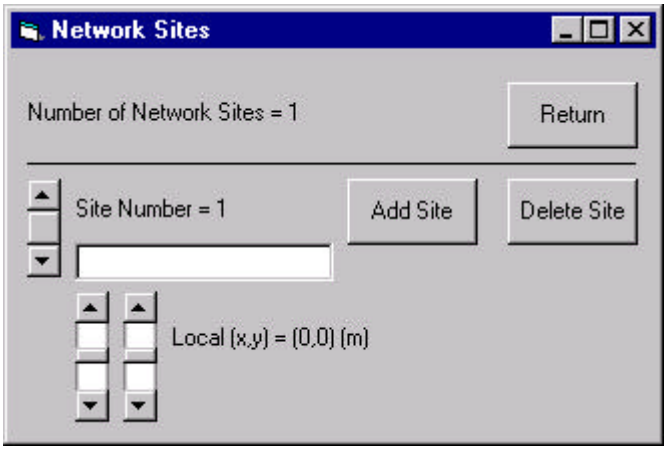

Figure 3. Network Sites Window.

Figure 3 shows the **Network Sites Window**, which allows a network of site positions and descriptions to be input and used as overlays in pictures of grid characteristics in various parts of TAPM_GUI. The sites describe where monitoring sites, towns, or local points of interest are

located on each of the grids. This information is useful when setting up grid characteristics or processing model outputs, but is not used by the model directly. The number of sites is displayed at the top of the window. A particular site can be selected using the slider control on the left of the window, and a text description can be entered in the text box next to the slider. Sites can be added or deleted using the Add Site and Delete Sites buttons respectively. The horizontal site (x,y) coordinates in local units (m) can be selected using the slider controls provided, using the same coordinate system as for the **Grid Centre Coordinates** (local values) on the **Main Window** shown in Figure 1. The network site information is stored in the GUI Default File.

3.1.1 Optional Inputs – Meteorology

Figure 4 shows the **Optional Inputs – Meteorology Window**, which can be selected from the main window menu bar at the top of Figure 1. This window consists of two sections.

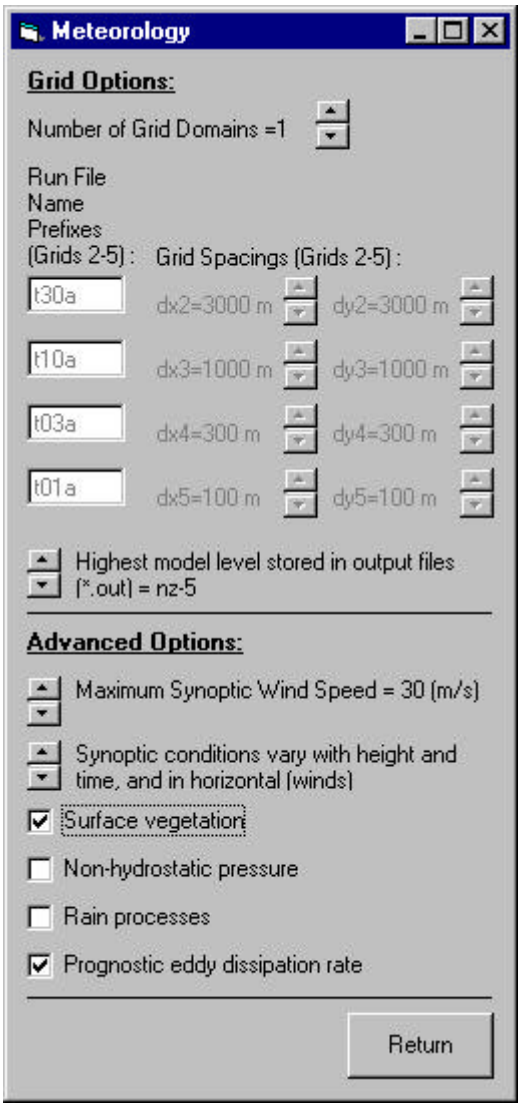

Figure 4. Optional Inputs – Meteorology Window.

The **Grid Options** section controls the grid nesting options for a multiply nested simulation and specifies the number of model levels to be stored in the output file. The number of nested grid domains can be selected, and for each nest, the file name prefix and x, y grid spacings (m) should be chosen. It is recommended that the ratio of grid spacings from one nest to another be in the

range 2 to 4, as this has been found to optimise both model run time and numerical noise generated in the nesting regions. For example, the default grid spacings of 10000, 3000, 1000 m have nesting ratios of 3.33 and 3.0. The maximum number of grid domains is five. If a smaller number is selected, the grid spacings set for the unused grids are ignored by the model. The number of model levels to be stored in the meteorological output file (*.out) is specified with reference to nz, the number of grid levels in the model. If predictions from only the first few model levels will be analysed, the size of the output file can be decreased by reducing the number of levels stored.

The **Advanced Options** section controls some extra model features. The first one is the maximum synoptic wind speed, which is set by default to 30 m s^{-1}, and which should only be changed (increased) if synoptic wind speeds higher than this are important for a particular simulation. This value also controls the advection time step used in the model, and increasing this value will slow the model down. The next option is the synoptic variability option, of which there are three choices:

- synoptic winds, temperature and humidity vary with height;
- synoptic winds, temperature and humidity vary with height and time;
- synoptic winds, temperature and humidity vary with height and time, and synoptic winds also vary with horizontal position.

All three choices use the central grid column synoptic analyses contained in the file *.syn, while only the third choice uses the gridded synoptic analyses contained in the file *.syg.

The next option controls whether soil or both soil and vegetation are used in the surface scheme; the default is to use both soil and vegetation. The non-hydrostatic option only needs to be turned on when the grid spacing is less than about 2000 m and then only if the terrain is steep and the winds are strong, otherwise this option will have little effect other than to increase the run time. Rain processes are only potentially important if rainfall or wet deposition processes are important. The default is to turn them off. Note that rainfall in the model affects the near surface soil moisture content and evaporative fluxes, but has no effect on the deep soil moisture content, as this is constant for a particular run. The option to use a prognostic eddy dissipation rate equation in the turbulence scheme is the default option, but may be switched off if a diagnostic length scale alternative is preferred. The diagnostic scheme is faster to run, but the prognostic scheme generally provides better turbulence predictions.

3.1.2 Optional Inputs – Pollution

Figure 5 shows the **Optional Inputs – Pollution Window**, which can be selected from the **Main Window** menu bar at the top of Figure 1.

The top control on the left of the window controls which (if any) pollutants will be simulated by the model. The options are no pollution calculations (None), one tracer mode (TR1), two tracer mode (TR1, TR2), three tracer mode (TR1, TR2, TR3), a chemistry mode without sulfur chemistry (APM, NO_X, NO_2, O_3) and a chemistry mode with sulfur chemistry (APM, NO_X, NO_2, O_3, SO_2). Emission characteristics for each pollutant are set up though the emission files (see Section 4). The tracer modes do not include chemistry or deposition processes and predicted pollutant concentrations are output in units of μg m^{-3}. Each tracer can represent either specific groups of sources or emissions for the same pollutant, different pollutants, or a combination of both of these. In the chemistry and deposition modes specific pollutants and their interaction with each other are represented. The pollutant APM usually represents particles with aerodynamic diameter less than 10 microns (PM10), but it can be used to represent other airborne size fractions such as PM2.5 if the emissions for APM reflect the variable of interest, and if it can be assumed that all secondary aerosol formation goes into this size fraction.

The next set of controls allows the selection of the pollution grid. The grid may be the same as the meteorology grid – same grid dimensions and grid spacing (the default), or it may be a subset of the meteorology grid with different grid dimensions and/or grid spacing. There is a set of controls for the x and the y directions, and these allow the user to decrease the pollution grid width (centred on the meteorology grid) and the grid spacing (by pre-set factors). These changes will be reflected in the picture of the region in the right of the window by changes in the white grid lines that represent the pollution grid. Underneath these grid controls, the number of grid points in the pollution grid is displayed for information. If a large number of grid points is used, this can become the dominant factor controlling the run time.

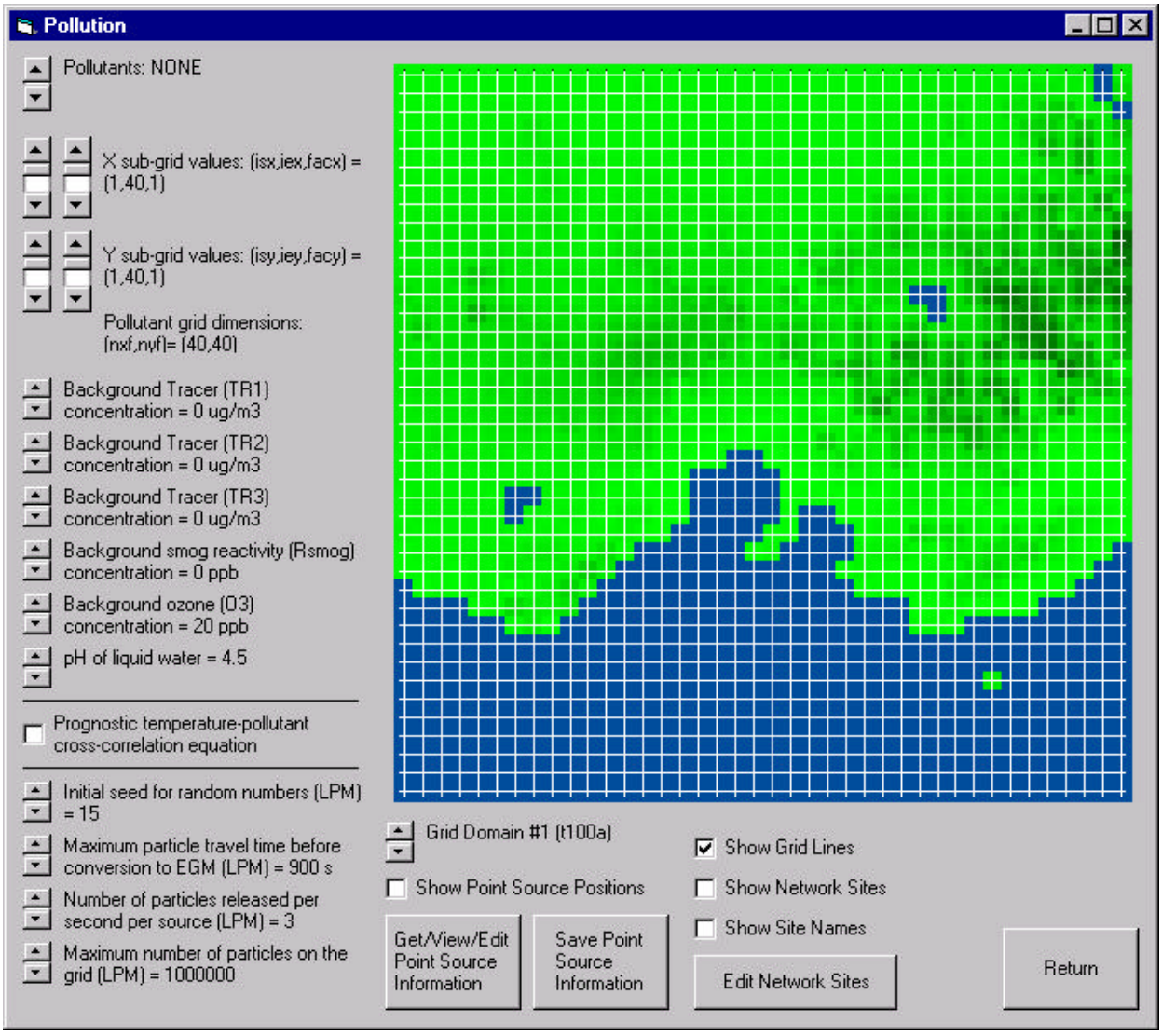

Figure 5. Optional Inputs – Pollution Window.

The next group of controls allows background pollution concentrations and the background pH of any liquid water present to be set. The background concentrations are used to initialise pollutant concentrations and are also used as inflow boundary conditions on the outer grid. The pH value is used in the aqueous chemistry and wet deposition calculations.

The next option controls whether an equation for the cross-correlation of potential virtual temperature and pollutant concentration is used for each primary pollutant. This option is only needed if the user is interested in near-source maximum ground level concentrations from point

sources and if the pollution grid spacing is adequate to resolve these maximum concentrations. If this option is on, then it will be used on those grid nests that have a pollutant grid spacing less than 1000 m, and the computational time for the pollution calculations will approximately double.

The next group of four controls are only relevant if one or more point sources are selected to be represented by a near-source Lagrangian approach (LPM mode). This mode for individual sources can be selected in the point source emission file described later, and is only used on the inner-most grid. This option is computationally intensive, and so it should be used selectively. The first control allows selection of the initial seed for the random number generator. Each different seed will produce a different set of random numbers that are used by the simulation. The next control allows selection of the travel time after which Lagrangian particles are converted to grid concentration and from then on represented by the Eulerian transport equation (EGM mode). The default of 900 s is long enough to allow the maximum ground level concentration from elevated point sources to be represented by the LPM, while keeping the run time to a minimum. The next control specifies the number of particles released per second per source for sources in LPM mode. The default of three is large enough to restrict the statistical sampling error in LPM mode to less than 5–10 % for hourly average concentration; larger values can significantly increase the run time. The last control in this group specifies the maximum number of particles that may be on the grid at any one time. If this number is exceeded, the simulation will stop and print out (in the *.lis file) that the maximum number of particles has been exceeded. This number must then be increased and the simulation repeated.

Beneath the picture shown in Figure 5, are various sliders and check boxes that allow the user to control aspects of the picture display including which grid domain is shown, display of grid lines, display of point source positions, and display of network sites. The network site information can also be edited through the **Edit Network Sites** command button described previously.

There are two command buttons for input and saving point source emission files. Pressing the **Get/View/Edit Point Source Information** command button allows point source information to be loaded from an existing (*.pse) file, or if none is available, start a new one. This selection then loads another window, which allows the creation/editing/viewing of a point source emissions inventory. TAPM_GUI can handle up to 100 sources with up to one year of time varying emissions (at hourly intervals). An alternative version of the user interface (TAPM_GUI_INV) allows up to 5000 sources with up to one week of time varying emissions. The model itself (TAPM) has no such limitations, as the number of sources is a dynamic variable. The file can be saved using the **Save Point Source Information** button, and uses the **Run File Name Prefix** for the outer grid nest and the .pse extension (e.g. t100a.pse). Input of significant numbers of sources or times using the GUI can be time consuming, and it may be better to create the file by other methods, particularly if digital information already exists (Section 4.1 lists the file format).

The **Point Source Emissions Window** shown in Figure 6 allows the creation of a point source emissions inventory. The top left-hand side of the window shows the number of sources. Sources can be added or deleted from the inventory using the Add Source and Delete Source command buttons. A particular source can be selected using the source selection control, and a source text description can be entered. The list of source descriptions is stored in a (*.pst) file that is only used by the GUI. The user can select whether a particular source in the inventory is turned off, is represented by the EGM, or is represented by the LPM near to the source and the EGM far from the source. Note that the LPM option should be used selectively for the most important sources to be modelled. Although this is theoretically the most accurate approach for predicting near-source maximum ground level concentrations, it is also the slowest option to use and, unlike the EGM, the run time increases with each source using this mode.

The local source location should be entered using the same local coordinate system as specified for the Grid Centre Coordinates on the Main Window. Other non-time varying source information

is the stack height (m), the stack radius (m), the buoyancy enhancement factor (used when emissions from adjacent stacks influence each other's plume rise), and the fraction of NO_X emitted as NO.

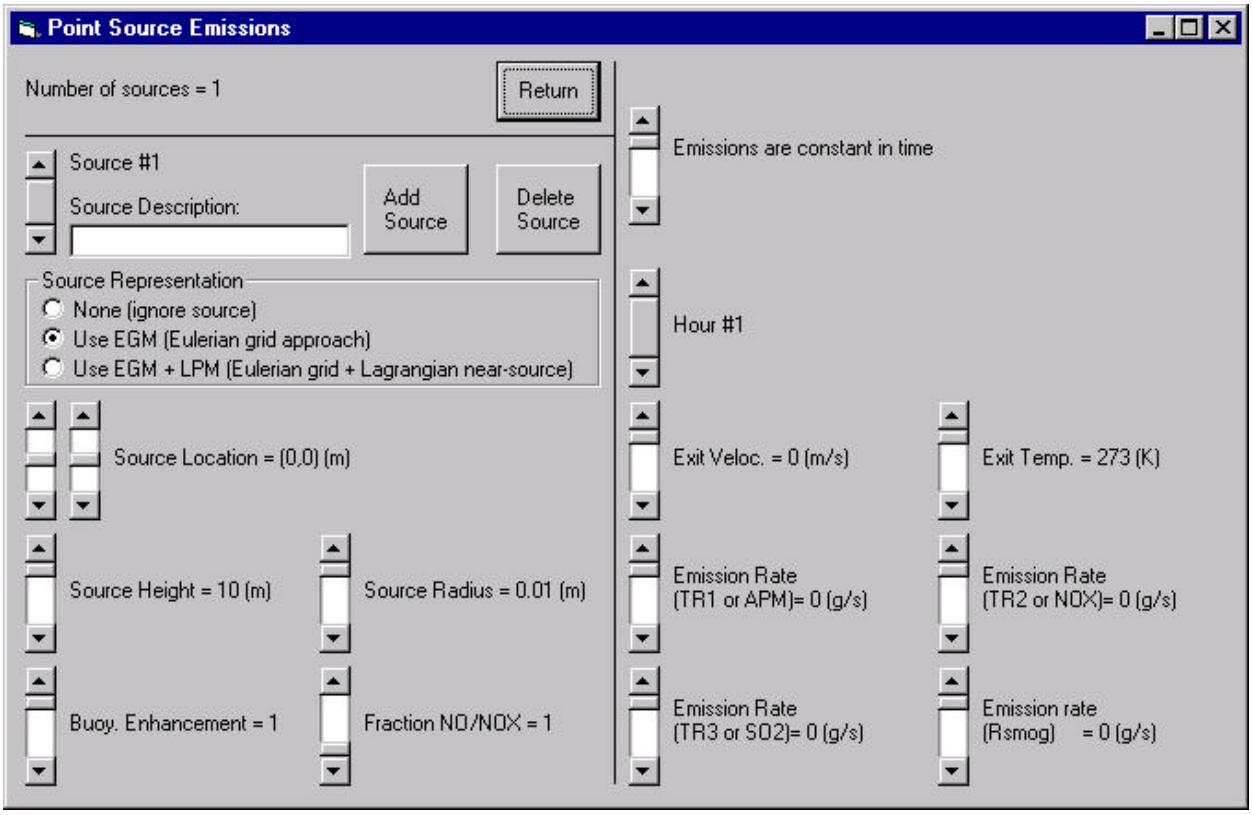

Figure 6. Point Source Emissions Window.

On the right-hand side of the window, time varying emissions can be entered. The top slider controls the number of hours of emission information. The default is that emissions are constant in time, but the slider can be moved to allow time varying emissions. For example, if the slider is set at 24, then emissions vary over a 24 hour period and then the same emissions are reused by the model for every 24 hour period during the model run (i.e. emissions cycle daily). For each hour, emission characteristics need to be input using the sliders provided. The variables include stack exit velocity (m s^{-1}), stack exit temperature (K), and up to four stack emission rates (g s^{-1}). The pollutant emissions needed depends on which pollution mode is being used, as specified in the **Optional Inputs – Pollution Window** (Figure 5). If any of the three tracer modes was selected, the emission rates need to be input using the sliders for those source or pollutant groups that correspond to the appropriate variables TR1, TR2 and TR3 (i.e. if in 1-tracer mode, then only TR1 needs to be specified; if in 3-tracer mode, then TR1, TR2 and TR3 need to be specified). If one of the two chemistry modes was selected, then the corresponding pollutants to those used in the selected mode need to be specified (i.e. APM, NO_X, SO_2 and Rsmog).

Gridded and biogenic surface emissions can also be used by the model (see Section 4.1), but there is no facility to enter these emissions through the GUI.

3.2 Analysing Outputs

Three menu options for analysing outputs can be selected from the **Main Window** menu bar at the top of Figure 1. Output files that can be processed are described in Section 4.

3.2.1 GIS Visualisation

Figure 7 shows the **Analyse Outputs – GIS Visualisation Window**, which allows visualisation of two-dimensional fields of meteorology and pollution using the Graphical Information System (GIS) provided with the TAPM software.

Figure 7. Analyse Output – GIS Visualisation Window.

At the top of the window is a slider control that allows a particular grid to be chosen. The next slider controls how many meteorological levels will be processed for potential viewing by the GIS. Model levels can then be selected with the series of sliders down the left side of the window for the number of levels chosen. Each level selected allows extraction of meteorological information in an x-y plane (plan view) for variables at that model level. For example, if level 1 (klev1 = 1) is chosen, the first model level (approximately 10 m above the ground) will be used. Optionally associated with each level is an x-z or y-z plane (vertical cross-section), which is extracted and stored in the file with the model level information. The vertical cross-section information is along a particular i or j grid line, where i is the grid unit value in the west-east (x) direction (range 1 to nx), and j is the grid unit value in the south-north (y) direction (range 1 to ny). For example, if x-z and cross1 = 20 are chosen, then an x-z cross-section is extracted along the grid line j = 20. If cross1 = 0 is chosen, then no cross-section information is extracted. The file name used for the meteorological information uses the Run File Name Prefix for the particular

grid chosen, and appends the level number and a file name extension corresponding to a particular variable. For example, the processed file for the above selections would be t100a1.wnd for winds (m s^{-1}). Other three-dimensional meteorological variables can be viewed in the GIS as scalars. For example:

- potential temperature (K) t100a1.pt,
- temperature (K) t100a1.tt,
- specific humidity of water vapour (kg kg^{-1}) t100a1.qv,
- specific humidity of cloud water (kg kg^{-1}) t100a1.qc,
- turbulent kinetic energy (m^2 s^{-2}) t100a.tke.

If at least one level of information is selected with the slider near the top of the window, then some two-dimensional variables are also output (with a zero level value in the file name) and can be loaded into the GIS as scalars. For example:

- surface net radiation (W m^{-2}) t100a0.nfx,
- surface sensible heat flux (W m^{-2}) t100a0.hfx,
- surface evaporative heat flux (W m^{-2}) t100a0.efx,
- surface temperature (K) t100a0.ts,
- mixing height (m) t100a0.zi,
- convective velocity scale (m s^{-1}) t100a0.wst.

All available pollution concentration output files (*.glc) are processed for use by the GIS, using the selected averaging period. Optionally, the locations (without labels) of the network sites previously entered in the Network Sites Window can be output for use by the GIS when a region is loaded. The name of the directory where the GIS has been installed can also be entered, if it differs from the default installation directory (C:\GIS).

Once the desired options for processing TAPM output files for the GIS have been chosen, the user then presses the button to **Process TAPM Output and Run the GIS**. The processing is done through a batch file called tapm2gis.bat with input file tapm2gis.inp. This can take some time to run, depending on the characteristics of the model run.

After the processing has finished, the GIS software is loaded, and then the user needs to load a Region file from the TAPM Run Directory using the **File – Region** menu item in the GIS. Other files can then be loaded including winds (**File – Winds**), meteorological scalars (**File – Scalars**) and concentrations (**File – Concentrations**). Visualisation over time can be viewed by pressing the F-key or K-key, and stopped using the space-bar key. A clock window can be displayed using the **View – Clock** menu item. See the **Help** menu item in the GIS for more detail, and many other viewing options.

3.2.2 Meteorology

Figure 8 shows the **Analyse Outputs – Meteorology Window**, which allows extraction of time series, profiles and summary statistics of meteorology.

At the top of the window is a slider control that allows a particular grid to be chosen. Information on the particular grid dimensions and grid spacing are also displayed for information. Below this are three sliders that control an (i, j, k) grid point selection in (x, y, z) coordinates, which is used for the model output extraction process. The horizontal grid point (i, j) can also be selected graphically with the left-mouse button in the picture at the right of the window. The local coordinate (m) of the selected horizontal grid point is displayed under these sliders for information. The **Process TAPM Output and View** button at the bottom of the window processes the TAPM meteorological output files and generates various forms of processed output files as described below. The processing is done through a batch file tapm2ts.bat.

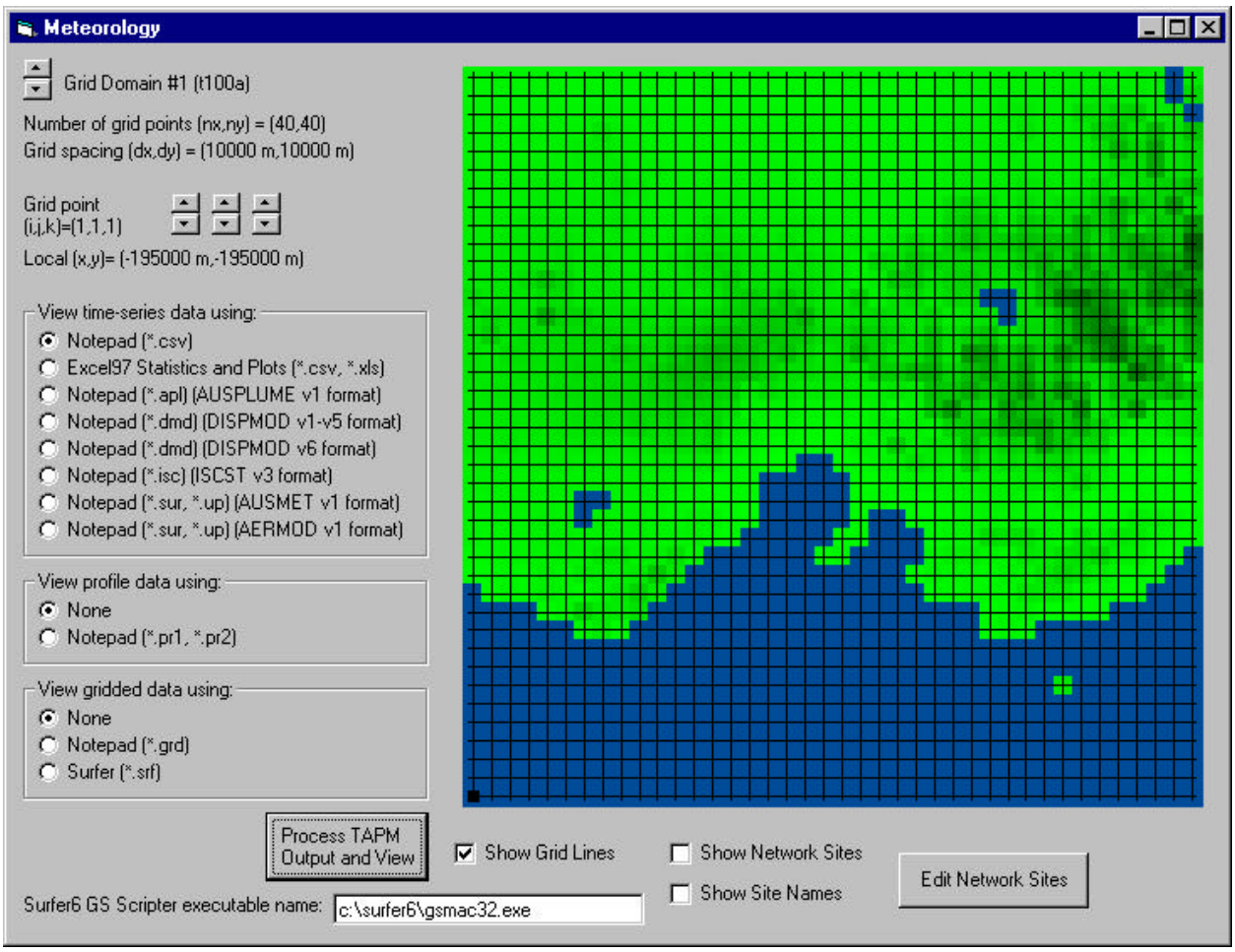

Figure 8. Analyse Output – Meteorology Window.

Time series of meteorological variables are output in various formats including a comma delimited form (*.csv), which can be viewed with the operating system software Notepad, or with Excel (Microsoft software), which includes various statistics and plots of the information. The file name uses the Run File Name Prefix for the grid selected (e.g. t100a), then '_m' for 'meteorology', with the (i, j, k) grid coordinate and the file name extension. For example, if the grid point (3, 2, 1) was chosen along with the *.csv option, the file name would be t100a_m00300201.csv. The first three columns of information in the file are the:

- date,
- hour of the day (e.g. hour 1 is the hourly average from hours 0-1),
- model run time (elapsed hour from the start of the model run).

The next four columns are the:

- vector wind speed WSPD (m s^{-1}),
- vector wind direction WDIR (°),
- temperature TEMP (°C),
- relative humidity RHUM (%),

at the model (i, j, k) grid point selected. The next two columns are:

- screen level temperature TEMPSCR (°C),
- relative humidity RHUMSCR (%),

at the horizontal (i, j) grid point. The remaining columns are surface based variables including the:

- net radiation NETR (W m^{-2}),
- sensible heat flux SENS (W m^{-2}),
- evaporative heat flux EVAP (W m^{-2}),

- convective velocity scale WSTAR (m s^{-1}),
- mixing height ZMIX (m),
- friction velocity scale USTAR (m s^{-1}),
- Obukhov length scale LSTAR (m),
- rainfall rate RAINR (mm hr^{-1}),

all at the selected horizontal (i, j) grid point.

Other forms of time series information can be output (and viewed with Notepad) as basic forms of meteorology input files used by various regulatory air pollution dispersion models including:

- AUSPLUME (*.apl),
- DISPMOD (*.dmd),
- ISCST (*.isc),
- AUSMET (*.sur and *.up) (the diagnostic wind field component of AUSPUFF),
- AERMOD (*.sur and *.up).

The formats of these files are described in the particular model documentation, and it is recommended that the files generated be checked before use to ensure that they conform to the software being used.

The next type of file that can optionally be viewed with Notepad are files of meteorological vertical profiles, either as hourly averaged profiles (*.pr1), or averaged profiles for each hour of the day and for all hours of the day (*.pr2). The file names use the Run File Name Prefix for the grid selected (e.g. t100a), then '_m' for 'meteorology', with the (i, j) grid coordinate and the file name extension. For example, if the grid point (3, 2) was chosen, the file names would be t100a_m003002.pr1 and t100a_m003002.pr2. The files contain comma delimited columns of meteorological variables, including the:

- vector wind speed WSPD (m s^{-1}),
- vector wind direction WDIR (°),
- temperature TEMP (°C),
- relative humidity RHUM (%),
- potential temperature POTTEMP (K),
- specific humidity of water vapour QVAP (kg kg^{-1}),
- specific humidity of cloud water QCLD (kg kg^{-1}),
- turbulence kinetic energy TKE (m^2 s^{-2}),
- scalar wind speed SWSPD (m s^{-1}),
- potential wind power density WPDEN (W m^{-2}),
- height above the ground HEIGHT (m).

The last types of file that can optionally be viewed with Notepad or Surfer (Golden software) are files of gridded meteorological summary statistics at the selected model level. These grids are averaged over the length of the model run for all variables, except for rainfall, which is the total rainfall (mm) for the run. The files are Surfer type grid files (ASCII format) with information for a particular variable contained in each file. The variables are the:

- vector wind speed WSPD (m s^{-1}),
- vector wind direction WDIR (°),
- temperature TEMP (°C),
- relative humidity RHUM (%),
- scalar wind speed SWSPD (m s^{-1}),
- potential wind power density WPDEN (W m^{-2}),
- rainfall total for the run RAINT (mm).

The file names use the Run File Name Prefix for the grid selected (e.g. t100a), then '_' with the variable name, then the model level k and the file name extension. For example, if the grid level

k = 1 was chosen, the file name for vector wind speed would be t100a_wspd01.grd. If the Surfer option is chosen for viewing these files, then a batch file (surfer.bat) and script file (surfer.inp) are used to plot the grids and save them to file (*.srf). The network positions and labels are also included on the plots as overlays, obtained from a separately saved file (e.g. t100a_netw.csv). Note that the x, y coordinates are in meteorological grid units, not local coordinates (m), but this can be changed by editing the header in the grid files, or within Surfer.

The controls along the bottom of the window allow selection of the executable name for GS Scripter for use with the Surfer option, the display of grid lines in the picture, and the display and editing of network sites as described in Section 3.1.

3.2.3 Pollution

Figure 9 shows the **Analyse Outputs – Pollution Window**, which allows extraction of time series, profiles and summary statistics of pollution.

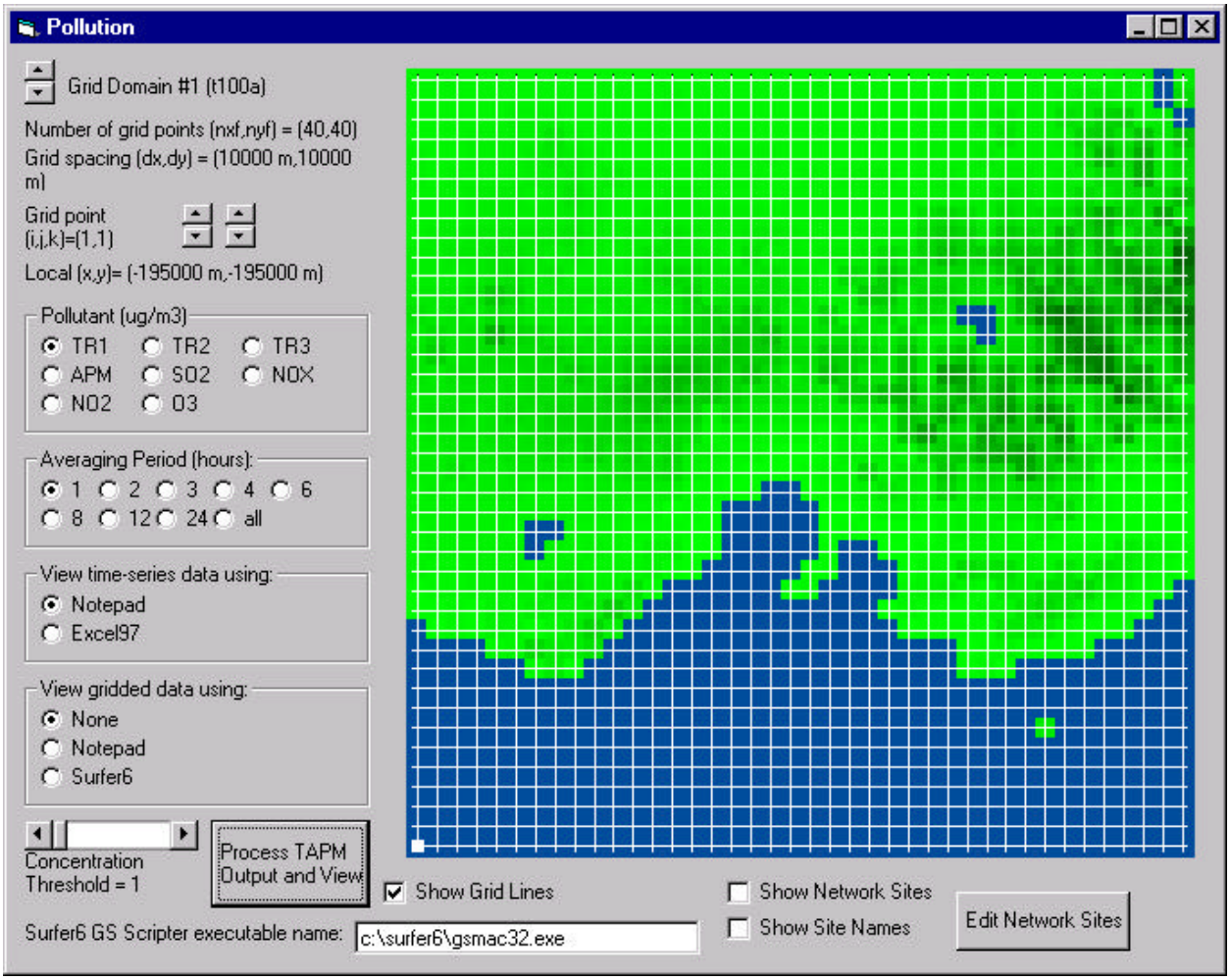

Figure 9. Analyse Output – Pollution Window.

At the top of the window is a slider control that allows a particular grid to be chosen. Information on the particular grid dimensions and grid spacing is also displayed for information. Below this are two sliders that control an (i, j) horizontal grid point selection, which is used for the model output extraction process. The horizontal grid point (i, j) can also be selected graphically with the left-mouse button in the picture at the right of the window. The local coordinate (m) of the selected horizontal grid point is displayed under these sliders for information. A particular pollutant needs

to be chosen for processing, along with an averaging period, using the controls provided. The **Process TAPM Output and View** button at the bottom of the window processes the TAPM pollution output files and generates various forms of processed output files as described below. The processing is done through a batch file glc2ts.bat.

Time series of pollution variables are output in various formats including a comma delimited form (*.csv), which can be viewed with the operating system software Notepad, or with Microsoft Excel97, that includes various statistics and plots of the information. The file name uses the Run File Name Prefix for the grid and pollutant selected (e.g. t100aso2), then '_c' for 'concentration', with the (i, j) grid coordinate and the .csv extension. For example, if the grid point (3, 2) was chosen, the file name would be t100aso2_c003002.csv. The first three columns of information in the file are the date, the hour of the day (e.g. hour 1 is the hourly average from hours 0–1) and the model run time (elapsed hour from the start of the model run). The next three columns are the concentration at the selected grid point CONC (μg m^{-3} or ppb), maximum concentration on the grid CMAX (μg m^{-3} or ppb) and the local maximum concentration for a 5x5 sub-grid region surrounding the selected grid point CLOC (μg m^{-3} or ppb). The concentration units depend on the pollutant mode being used, and are μg m^{-3} in tracer mode, or μg m^{-3} for APM and ppb for the other pollutants in chemistry mode.

Gridded summary statistics can optionally be viewed with Notepad or Surfer. The files are Surfer type grid files with information for a particular variable contained in each file. The variables are the average concentration AVG (μg m^{-3} or ppb), the 9^{th} highest concentration 9TH (μg m^{-3} or ppb), the 2^{nd} highest concentration 2ND (μg m^{-3} or ppb), the highest or maximum concentration MAX (μg m^{-3} or ppb), the number of averaging periods when the concentration is greater than some threshold value NPG, and the number of days that the concentration is greater than some threshold value NDG. The threshold value can be set using the slider provided. The file names use the Run File Name Prefix for the grid selected and the pollutant (e.g. t100aso2), then '_c' with the variable name and optionally the threshold value and the extension .grd. For example, for the pollutant SO_2 with a threshold of value of 1, two sample file names would be t100aso2_cavg.grd and t100aso2_cnpg001.grd. If the Surfer option is chosen for viewing these files, then a batch file (surfer.bat) and script file (surfer.inp) are used to plot the grids and save them to file (*.srf). The network positions and labels are also included on the plots as overlays, obtained from a separately saved file (e.g. t100a_netw.csv). Similarly, the gridded option allows viewing of dry and wet deposition files directly output from the model when in chemistry mode.

The controls along the bottom of the window allow selection of the executable name for GS Scripter for use with the Surfer option, the display of grid lines in the picture, and the display and editing of network sites as described in Section 3.1.

4 TAPM FILES

The GUI (TAPM_GUI.EXE) sets up all of the TAPM input files needed by the model (TAPM.EXE). These include:

- a batch file to run the model (*.bat),
- a general input file read by the model (*.inp),
- surface information for each grid nest (*.top),
- synoptic profile information at the grid centre (*.syn)
- three-dimensional gridded synoptic analyses (*.syg),

and emissions information (see Section 4.1):

- point sources (*.pse),
- gridded surface sources (*.gse),
- biogenic surface sources (*.bse).

When the model is run, two types of nesting files are generated to provide boundary conditions for the inner nested grids:

- meteorology (*.nes),
- pollution (*.trb).

These nesting files can be deleted by the user when the model run is finished.

The model output files:

- general meteorology (*.out),
- rainfall (*.rfl),
- pollution concentration (*.glc),
- deposition (*_dry.grd and *_wet.grd),

are used by the GUI when analysing the output. General meteorology is output as hourly averages, rainfall is output as hourly total (mm hr^{-1}), ground level concentrations are output as hourly averages in either μg m^{-3} (tracer mode or for APM) or ppb (chemistry mode except for APM), and deposition (dry and wet in chemistry mode) is output as μg m^{-2} for the model run period (i.e. an example file name for so2 would be t100aso2_dry.grd, and if the run length was for one month, then the units would be μg m^{-2} $month^{-1}$).

A listing file (*.lis) is also produced when the model is run, and contains information that reflects the options chosen by the user and other diagnostic information for the model run. This file should be checked by the user to ensure that the model has run as expected.

4.1 Emission Files (*.pse, *.gse and *.bse)

Emission files optionally read by the model are the Point Source Emissions file (*.pse), the Gridded Surface Emissions file (*.gse), and the Biogenic Surface Emissions file (*.bse). The model assumes that the file names of these files for a particular run use the Run File Name Prefix of the outer grid with the corresponding extension (e.g. t100a.pse, t100a.gse and t100a.bse). The model can be run with any combination of these files, or no emissions files at all. The listing file (*.lis) will indicate what types of emission information is being used by the model. If generated by the user, these files should be in ASCII, free format, and should be placed in the Run Directory. If the end of file is reached before the end of the TAPM run, the file is rewound and read again as described for the *.pse file in Section 3.1.2 (i.e. the emissions cycle over time).

The required format for the *.pse file can be seen from the following read do-loops (also see Section 3.1.2 for GUI entry of emissions):

```
READ: nsource, nhour
do i = 1, nsource
  READ: mode, x_pse, y_pse, h_pse, r_pse, e_pse, f_pse
enddo
do k = 1, nhour
  do i = 1, nsource
    IF TRACER MODE THEN READ: w_pse, t_pse, tr1_pse, tr2_pse, tr3_pse, dum_pse
    IF CHEMISTRY MODE THEN READ: w_pse, t_pse, apm_pse, nox_pse, so2_pse, rs_pse
  enddo
enddo
```

where:

nsource	number point sources,
nhour	number of hours over which the time-varying emissions cycle (1=constant),
mode	controls the source mode (-1=OFF, 0=EGM, 1=EGM+LPM),
x_pse	west-east (x) local coordinate of the stack (m),
y_pse	south-north (y) local coordinate of the stack (m),
h_pse	stack height above the ground (m),
r_pse	internal stack radius (m),
e_pse	buoyancy enhancement factor (1=no enhancement due to near-by stacks),
f_pse	fraction of the NO_X emission that is NO (0=no NO, 1=all NO),
w_pse	stack exit velocity (m/s),
t_pse	stack exit temperature (K),
tr1_pse	emission rate of TR1 (g/s),
tr2_pse	emission rate of TR2 (g/s),
tr3_pse	emission rate of TR3 (g/s),
dum_pse	not used,
apm_pse	emission rate of APM (g/s),
nox_pse	emission rate of NOX (g/s) (expressed as NO2),
so2_pse	emission rate of SO2 (g/s),
rs_pse	emission rate of Rsmog (g/s).

The required format for the *.gse file can be seen from the structure of the following read-do loops:

```
READ: nx_gse, ny_gse, dx_gse, dy_gse, cx_gse, cy_gse
do k = 1, nhour
  do j = 1, ny_gse
    do i = 1, nx_gse
      IF TRACER MODE THEN READ: tr1_gse, tr2_gse, tr3_gse, dum_gse, dum_gse
      IF CHEMISTRY MODE THEN READ: apm_gse, nox_gse, so2_gse, rs_gse, no2_gse
    enddo
  enddo
enddo
```

where:

nx_gse	number of west-east (x) grid points for the emissions grid,
ny_gse	number of south-north (y) grid points for the emissions grid,
dx_gse	west-east (x) grid spacing (m) for the emissions grid,
dy_gse	south-north (y) grid spacing (m) for the emissions grid,
cx_gse	west-east (x) grid centre local coordinate value (m) for the emissions grid,
cy_gse	south-north (y) grid centre local coordinate value (m) for the emissions grid,
nhour	number of hours over which the time-varying emissions cycle (1=constant),
tr1_gse	emission rate of TR1 (g/s),
tr2_gse	emission rate of TR2 (g/s),
tr3_gse	emission rate of TR3 (g/s),
dum_gse	not used,
apm_gse	emission rate of APM (g/s),
nox_gse	emission rate of NOX (g/s) (expressed as NO2),
so2_gse	emission rate of SO2 (g/s),
rs_gse	emission rate of Rsmog (g/s),
no2_gse	emission rate of NO2 (g/s).

The format for the *.bse file is the same as for the *.gse file, but the grid characteristics do not necessarily have to match the *.gse emission grid characteristics. The Rsmog emissions in the *.bse file are modified in the model by a radiation and temperature correction function, with the values of *rs_bse* in the file assumed to be at a temperature of 30 °C and for a photo-synthetically active radiation (PAR) level of 1000 μmol m^{-2} s^{-1}.

5 ACKNOWLEDGEMENTS

Thanks to the US Geological Survey, Earth Resources Observation Systems (EROS) Data Center Distributed Active Archive Center (EDC DAAC) for the public domain global datasets of terrain height and land-use. Thanks to Dean Graetz and CSIRO Wildlife and Ecology for providing the Australian vegetation and soil type database. Thanks to Paul Stewart, Terry Hart and the Bureau of Meteorology for allowing the LAPS analyses to be distributed with the model. Thanks to the US National Center for Atmospheric Research (NCAR) for the public domain dataset of long-term monthly average sea surface temperatures.

Thanks to Bill Physick for providing code to read the global terrain dataset and to Julie Noonan for extracting the global land-use dataset and converting it to a latitude/longitude grid suitable for use by the TAPM GUI.

The encouragement and feedback from users of pre-release versions of the software (Bill Physick, Julie Noonan, Melita Keywood, Mark Hibberd, Ashok Luhar, Michelle Cox, Bronwyn Dunse, Merched Azzi and Reinout Boers) is appreciated.

Thanks also to Mark Hibberd for providing helpful comments and improvements on the first draft of this manual.

6 REFERENCES

Hurley, P.J. (1999). 'The Air Pollution Model (TAPM) Version 1: Technical Description and Examples', *CAR Technical Paper No. 43*. 42 p.

Puri, K., Dietachmayer, G.S., Mills, G.A., Davidson, N.E., Bowen, R.A., and Logan L.W.: 1998, 'The BMRC Limited Area Prediction System, LAPS', *Aust. Met. Mag.*, **47**, 203-223.

CSIRO Atmospheric Research Internal Papers

1. Dilley, A. C. Interactive display of images written in DISIMP file format using the IDL program DISPLAY. 1996.
2. Daubney, I. E. GPS Receiving Barometric Pressure Monitoring System (RBPMS). 1996.
3. Holper, P. N. Evaluation of the 1996 CSIRO Division of Atmospheric Research Open Day. 1996.
4. Dilley, A. C. glenlnIDL – a graphical user interface for the GENLN2 package. 1997.
5. Davy, E. A. DAR Publications: maintenance of collections and databases of the CSIRO Division of Atmospheric Research. 1998.
6. Dilley, A. C. ; Edwards, M. The automatic processing of ASDA format NOAA HRPT data at CSIRO DAR. 1998.
7. Enting, I. G. Green's function methods of tracer inversion: invited lecture for Workshop on Inverse Methods in Global Biogeochemical Cycles. Heraklion, Crete, 16-20 March 1998. 1998.
8. Bennett., J. W. CSIRO Single Channel Infrared Radiometer model DAR011. 1998.
9. Elliott, T. I.; Rotstayn, L. D.; Dix, M. R. Using the CSIRO GCM Mark 2. 1998.
10. Cechet, R. P. CAR-SONDE : portable, stand-alone atmospheric sounding system. 1999.
11. Davy, E. A. Science Citation Index analysis of CAR post 1983 publications during citation years 1992-1997. 1999.