EAMS
(Erosion Assessment and Modelling System)
User Manual

Version 2.00

Assoc. Prof. Garry Willgoose
Centre of Environmental Dynamics,
University of Newcastle,
Callaghan, NSW, 2308
AUSTRALIA.
E-mail: garry.willgoose@newcastle.edu.au

(1) Introduction

The EAMS (Erosion Assessment and Modelling System) software is designed for the rapid assessment of erosion on disturbed sites. In particular it is designed predict erosion many years in future after the landform has changed its shape in response to the erosion that has occurred on it. It does by integrating a widely known landform evolution model, SIBERIA, with data input and manipulation software, an erosion database, and data visualisation software with lite-GIS capabilities.

The EAMS suite consists of two main packages. The first component is EAMS-MOSCOW (simply referred to as Moscow below) which does the erosion modelling. Moscow essentially sets up all the commands that SIBERIA uses when it runs (thus its name). The second component is EAMS-Viewer (simply referred to as Viewer below) which is used for data visualisation and provides a simple GIS capability to assist in setting up files for the more advanced capabilities of SIBERIA (e.g. spatial variation of erosion and runoff properties). In both cases the software is capable of handling extremely large data sets (e.g. the software has been run with datasets of more 500,000 data points).

Broadly Moscow does:

- Data conversion of files containing irregular points of elevation in DXF (i.e. AutoCAD), XYZ (i.e. triples of x,y,z, coordinates) and MINDRAFT (i.e. RAW) formats into the internal format of EAMS.
- Gridding of this data and automatic correction of artefacts that are generated in the gridding.
- Selection and manipulation of erosion, runoff, geology and other parameters from a database of parameters. Assess to the advanced capabilities of SIBERIA (e.g. spatial, temporal and/or depth varying runoff and erosion properties) through the input of command files that control these capabilities.
- Running of SIBERIA with output of landform elevations, slopes, gully locations and other indicators of erosion/deposition at various (user-controlled) times during the design lifetime requested. Additionally, as an option, output of spatially distributed estimates of instantaneous and cumulative erosion over the design lifetime, as well as indicators of gully erosion can be output. For a full compendium of SIBERIA’s capabilities the user should refer to the SIBERIA manual provided with the EAMS software.

Broadly Viewer does:

- Visualisation of SIBERIA output in 2D (i.e. plan view), 3D (i.e. a lit surface) and draped (i.e. output data projected onto the 3D landform) format.
- A flexible and powerful means of selection of parts of the landform (called regions), and saving those regions so that they may be able to be used in SIBERIA to control the spatial variability capabilities of SIBERIA.
- Virtual Earthworks. This provides tools for manipulation of the landform so that problems arising from the design of the landform (e.g. poor erosion performance in parts of the

landform) can be easily corrected. Output of this redesigned landform back to Moscow for revised erosion assessment.

Hardware requirements

Minimal Requirements: EAMS will run on Windows 95 or later with at least 128 megabytes of memory with a Pentium 1/2/3/4/Pro or compatible Processor. The Monitor must be set to High Colour (16 Bit) or better (in the Monitor Control Panel). Installation requires 50 Mbytes of free disk space. A CD drive is required for installation (for installation from physical media, not needed for electronic download). With 128Mbytes you will not be able to open all software at the same time, but need to close one component of the suite to use the next part.
Preferred requirements: For optimal performance Windows NT4/2000/XP and 256 Mbytes of memory (512 Mbytes if large data sets, more than 100,000 data points, will be regularly used) and 1 Gbyte free space on the System Disk (typically C: drive) are recommended. Performance bottlenecks (*and where they typically occur*) that may limit the speed of the calculations are the disk speed (*data extraction from DXF and XYZ files, and their gridding*), processor speed (*the SIBERIA erosion simulations*) and Microsoft OpenGL compatible 3D graphics board (*data visualisation*). If these parts of the analysis become limiting then it is suggested that consideration be given to upgrading these components.

In addition, a considerable amount of disk space can be consumed by the output files. For instance, a 500 x 500 computational domain (the biggest currently supported by the PC implementation of EAMS; at 10m spacing this would cover an area 5km by 5km) with (optional) erosion output data will consume about 20 Mbyte for each time at which data output is requested. Thus for a simulation over a 100 year design life time with data output every 10 years, the total disk space consumed by one simulation will be about 220 Mbyte (this includes the time=0 data file with the initial landform). Disk space required is linear with the number of nodes (i.e. a 250 x 250 simulation uses 25% the space of a 500 x 500 simulation). The monte-carlo risk assessment capability (currently available only on the UNIX version of EAMS) can easily consume several Gbytes of disk space in a single simulation.

2 EAMS Tutorial

2.1 Introduction

EAMS consists of two components.

- The first part is MOSCOW, which calculates the erosion and hydrology properties. To start MOSCOW you double click on the EAMS-Moscow icon in the PROGRAMS menu under START.
- The second part is VIEWER, which is used to view the results of the simulations. To start VIEWER double click on the EAMS-Viewer icon icon in the PROGRAMS menu under START.

The software has been written in openGL to allow portability across platforms. EAMS currently runs on Wintel machines and Silicon Graphics Irix V6.0 or later. One of the unfortunate quirks is that in dialogs you must have the mouse inside a text box when you are typing **at all times**. It is not sufficient to click in the window and type away but the mouse must be inside the window at all times.

Depending on how much memory and free disk space you have you may not be able to have both applications open at the same time. Typically if the amount of memory in your machines is less than 128 Mbytes you will only be able to open either Moscow or Viewer, but not both.

2.2 Erosion Simulation using EAMS-Moscow

To start EAMS-Moscow double click on the EAMS-Moscow icon under the Programs menu item on the Start menu bar.

On starting Moscow two windows will appear. The main window is a small blue window appears with a menu bar across the top (with menu bar headings FILE, IMPORT, SIBERIA, PARAMETER, HELP). The second background window is a larger black window which will appear in the background. Many diagnostics will appear in this background window. The menus in the main window are to perform the work and are selected using the left button on the mouse. To quit MOSCOW pull down the FILE menu and select EXIT.

2.2.1 Importing Elevation Data

Data can be imported in a number of forms from a number of sources. The main data source of interest here is AutoDesk's .dxf format which is used by a number of packages as an interchange format (e. Vulcan, Datamine). We will demonstrate the importation of a .dxf file.

Two example .dxf file are included. They are `test1.dxf` and `test2.dxf`. The remainder of this section assumes that you wish to import file `test2.dxf`. To import this file

- Select the menu item IMPORT DXF from the IMPORT menu.

- **File name input:** You will be prompted for the file name of the data file to import. To select this file name click with the left mouse button in the OK button. To tell the dialog to select `test2.`dxf place the mouse to the right of the *****, delete the star and type test2 so that the dialog now says **test2.dxf**. Click OK.
- **Region for gridding:** You will be prompted for the domain to used for the simulation. This dialog is asking for the east, west, north and south boundaries of the domain to be gridded. Initially the dialog will have –1 in all four of the boxes. This indicates to the program that it should grid all the data. If you wish to grid a subset of the data you should input here the appropriate values. If any one of the four values is –1 then the program will grid all the data (ie. To grid a subset of the data you must input valid values for **all** four coordinates). After inputting these numbers you should click on the OK button.
- **Grid Spacing:** You will be prompted for the grid spacing to used by this analysis. This grid spacing is in metres. You should input an appropriate value and click OK. Typically a grid spacing of about 20m is a good starting value. The value can then be refined in following runs if necessary. Computation times and disk space limit how fine a resolution you should adopt. A good rule of thumb is not to have more than 100-200 grid points in either direction initially as it will take too long to do the calculations (i.e. a 2 km by 4 km square domain with 20m spacing will have 100 by 200 grid points in the appropriate directions). After inputting a value press OK.
- At this point there will be a large amount of text printed out to the black background window behind the main window. Most of this can be ignored. When the text message 'Finished writing the raw file' appears the data importation and gridding of the data is complete. You have successfully created the digital terrain map to be used for the erosion calculation in the next step. The name of the file in which the gridded digital terrain map is stored is called 'test2.grid.raw'. There will also be another file called 'test2.raw'. This is an intermediate file that can be useful for more advanced applications. It can be ignored for the moment. These file names are generated automatically and are based on the name of the input DXF file. For instance if the input file was called 'example.dxf' then the output gridded digital terrain map file would be called 'example.grid.raw'. *Warning*: *If a file called 'test2.grid.raw' already exists the gridding step will fail. The moral is always rename or delete the file before doing a new run. Most of the codes in EAMS will fail if the file already exists This is a safety measure to ensure you don't delete important preexisting files by accident.*

It in conclusion at the end of this example there will be three files. The first is 'test2.dxf', the original file; 'test2.raw' an intermediate file you don't need to worry about at this stage; and 'test2.grid.raw' the final file that has the gridded elevation data in it.

2.2.1 The Erosion Calculations

This is a simple three stage process. The first stage identifies the input digital terrain map data to be used, what data needs to be output and the erosion database to use. The second stage sets run characteristics. The third runs the erosion assessment

Stage 1: Selecting the Data and Erosion Parameters

- Select the INPUT FROM RAW menu item from the SIBERIA menu
- Input the grided digital terrain map data to be used for the simulation. In this case the file is ‘test2.grid.raw’. Click OK to continue
- The next dialog identifies the data to be output from the simulation.
  - The ‘Start Time’ gives the year at which the simulation should start. Simply input **0** here. This is used by the erosion code to automatically generate file names for the output files. See the next dot points for how this is used.
  - For the ‘Output File Name’ you input the file name to be used as the based for the output file name. You may ignore the selection of **raw** and **rst2** output in the top left corner of the dialog (it should always have **rst2** selected). The default name will be based on the input file name selected.
  - The ‘Number of Outputs’ is the number of times after the start of the simulation at which output data are wanted. eg. if data were wanted for years 1,5,50 and 100 years into the future you would input 4 here (because there are 4 times for output; the actual timed will be input in a moment). See the next dot point for how this is used.
  - The ‘Output File Name’ and ‘Start Time’, ‘Number of Outputs’ are used for constructing the output file names. The ‘Output File Name’ is used as the first part of the file name and the time at which an output has been requested is added to ‘Start Time’ and this is appended to this file name. This way a unique file name is created that identifies what time the file represents. This can be used in two ways; (1) say we are interested in years after the construction of a landform (and the input landform is this initial landform) then if we input ‘Output File Name’=junk, ‘Start Time’=0 and we request an output at 50 years then the file name of the results at 50 years will be ‘junk-0000050.rst2’. If we request it at 100 years then the corresponding file name will be ‘junk-0000100.rst2’; (2) On the other hand say we are interested in the results at some year in the future and we know what the year is the simulation starts at then if we input ‘Output File Name’=junk2, ‘Start Time’=1990 and we request outputs at 1,5, 50 and 100 years then the corresponding file names will be ‘junk2-0001991.rst2’, ‘junk2-0001995.rst2’, ‘junk2-0002040.rst2’ and ‘junk2-0002090.rst2’.
  - After inputting this data you should click on the OK button
- The next dialog asks for the years at which data is to be output (these are the 1, 5, 50, 100 years previously referred to above). The number of years required here is given by the

‘Number of Outputs’ specified in the previous dialog. Since we input 4 to the previous dialog there are 4 text fields. You should input 1 in the first text box, 5 in the second, 50 in the third and 100 in the fourth. This requests output to be generated at times 1, 5, 50 and 100 years after the start of the run. Click OK when done.

· The next dialog requests the name of the erosion parameter database to be used for this simulation. With the examples provided with the code is a database called qca-v1.1.sdb. Enter this name in the dialog and click OK. This selects a database developed for the Bowen basin coal mines and shows a range of parameter sets for different types of materials. The exact values are not important for this example, only that there are range of ready made range of parameter sets for use in the example.
· Immediately after you press OK for selecting the database you are prompted to select a data set out of the database so as to set default parameters for the run. You can always change this choice later but you must make a choice at this stage. You can scroll up and down the list by clicking on the buttons with ‘<’ ‘>’ signs. Select the topmost data set by clicking on ‘OakyCreek_TopSoil_SandyLoam’ and click on the RETRIEVE button. You have now selected the default parameter values for the simulation.

You have now selected the input data file, the output required and the default SIBERIA parameters to be output.

Stage 2: Setting the Run Parameters

This stage modifies the default parameters to determine the parameters used for the simulation. You do this by modifying individual values in the default parameter set that were input in the previous stage (i.e. Stage 1).

· To select parameters for modification select the EDIT PARAMETERS option from the PARAMETERS menu. This menu item has further submenus with the options RUN PARAMETERS, EROSION PARAMETERS, HYDROLOGY PARAMETERS, DTM PARAMETERS, CHANNEL PARAMETERS, TECTONIC PARAMETERS, DEPENDENT MODEL PARAMETERS, ADVANCED PARAMETERS #1 and ADVANCED PARAMETERS #2. These menu items group together all the parameters of the model into functional groupings. For the moment we will ignore most of these and only modify the ones that control the direction of the simulation. Accordingly select the RUN PARAMETERS menu item from the sub menu.
· The RUN PARAMETERS dialog appears with three text items. The topmost of these text items is the number of years that the simulation will be carried out for. Since we want to get output for 1, 5, 50 and 100 years we must run the model for 100 years. Input 100 into this text box. (Aside: If you input 50 here instead of 100 then the model would only run for 50 year and only output the 1,5, and 50 year files … it never gets to 100 years to output the 100 year data). Click OK.

You have now set all the run parameters that are necessary. You can change hydrology and erosion parameters at this stage if it is desired. For instance, if a cover factor is required (eg. vegetation) that can be modified in the EROSION PARAMETERS menu item.

Stage 3: The simulation

To perform the simulation select the menu item RUN SIBERIA from the SIBERIA menu. This performs the calculations. A large amount of information will appear on the background window behind the main MOSCOW window (this can be saved, see Section 3 for details). When STOP appears on the screen the simulation is finished.

You should EXIT (on the FILE menu) from MOSCOW.

When you look in the directory after this example you will find the following additional files ‘test2-0000001.rst2’, ‘test2-0000005.rst2’, ‘test2-0000050.rst2’ and ‘test2-0000100.rst2’. Depending on the settings in the file ‘siberia.setup’ you may also find the following files ‘test2-0000001.rsu’, ‘test2-0000005.rsu’, ‘test2-0000050.rsu’ and ‘test2-0000100.rsu’.

2.3 Demonstration Simulations

A series of demonstration tutorials are provided in the demonstrations folder (see under the START menu bar) together with results from the demonstrations. These are provided as tutorials on how to use the EAMS software together with the answers against which the user can check their answers to ensure they have the same answers. The general format is that the required files for the simulation are provided in the ‘demonstration’ folder while the answers are provided in an appropriately named folder within the demonstration folder.

Before doing any of the demos below you should ensure that SIBERIA will output the data that you require. Select ‘siberia.setup’ from the START\EAMS menu and open the file with ‘wordpad’ and uncomment (remove the # in the first column) the command lines that are

```
OUTPUT ZCHANGE
OUTPUT AVEZCHANGE
```

Then save the file with the save menu (not the ‘save as’ menu).

By uncommenting these lines this means that SIBERIA will output the elevation change at every point (ZCHANGE) and average change of elevation for the catchment upstream of that point (AVEZCHANGE).

2.3.1 Demo 1

Demo 1 is a small DXF input file example. The .dxf file itself is quite small and the area covered by the domain is also quite small. This example consists of a flat-topped spoil dump surrounded on all sides by steep batters.

- Step 1: Start Moscow (from the START Bar).
- Step 2: Since you are inputting a .dxf file select the ‘input DXF’ from the INPUT menu in Moscow. You will be prompted for the DXF file name. The demonstration folder may be a bit difficult to find initially. The directory is on the disk that the SYSTEM is installed on (typically the C: drive) and the directory path is ‘C:\program files\landtech\EAMS\demos’. If you have trouble finding open the ‘demos’ folder on the EAMS START menu item and see where this directory is located. After finding the correct directory you will find a file ‘demo1.dxf’, which you should select.
- Step 3: After selecting the correct dxf file you will prompted in a dialog for what portion of the dxf file you wish to select. Initially it has –1 for all the coordinates for the bottom left corner and top right hand corner, which indicates that all the data should be input. Click OK to this dialog so that the program inputs all the data.
- Step 4: After selecting the potion of the domain to be input you will be prompted for the resolution that the domain will be gridded at (SIBERIA works on a grid). At this stage you need to know something about the size of the domain being input. For the current limitation of EAMS you should ensure that the dimensions of the grid are less than 500 x 500 grid points (i.e. a 10m grid would be able to cover 5km x 5km max, if you want to cover a greater are you would need to select a lower resolution). For this demonstration if you select a resolution of 10m then this will give a domain about 150 x150 (as a first step its always worthwhile doing a coarse resolution run first because it will be fast, then worry about better resolution if you need to later). Click OK after entering 10m for the resolution.
- Step 5: At this stage EAMS will go off and extract the dxf file and grid the data onto the domain requested. When the gridding is done there will be a message in the black DOS window saying that the gridding is complete. At the end of this stage the ‘demos’ folder will have two additional files in it. The first is ‘demo2.raw’ which is the data extracted from the dxf file in RAW format, and ‘demo2.grid.raw’ which is the data gridded onto the grid requested for the boundaries requested, in RAW format.
- Step 6: At this stage you need to set up the data for the SIBERIA run. To do this you must input the gridded RAW file, select runoff/erosion/run parameter for the SIBERIA run and tell EAMS what data output is required – actually running SIBERIA is an additional step we will do later. To do this you select the menu item ‘Input RAW’ under SIBERIA menu item. The first dialog you will request a generic name for the output files, the number of output files (the time of output will requested in the next step … we just need to know how many output are required at this stage), and start date for the run. At this dialog input (1) 0 for the

Start Time, (2) 2 for the number of output files, and (3) leave the generic name as it is in the dialog (i.e. C:\program files\landtech\EAMS\demos\demo1).

- Step 7: The next dialog asks for the times at which output is required. Since you input 2 at the last dialog there are two values to be input. Input 10 in the first item and 100 in the 2nd item, then click OK.
- Step 8: You are asked to input a SIBERIA database (i.e. a .sdb file). The location of standard databases is at 'C:\program files\landtech\EAMS'. Input the file 'siberia-parameters.sdb' and click OK.
- Step 9: After opening the database EAMS asks you select a database. For this demo click on the item described as 'OakyCreek_Topsoil_SandyLoam' (after clicking the name will appear in the text entry part of the dialog at the bottom. Once this parameter set is elected click on the RETRIEVE button at the bottom of the dialog.

You have now successfully input the model parameters, and the data on what files to be output by the model. The next step is to modify these parameters, and then run SIBERIA.

- Step 10: To modify SIBERIA under DATABASE there are a series of submenus of parameters. Under the RUN parameters menu item change the run time to 100 years (the default value is 1 year). Under the EROSION parameters can the fluvial erosion rate (the 3rd item down) to 0.3e-2 (for the OakyCreek data set the default value is 0.3e-3). You have now modified the parameters for this run so that the run will go for 100 years (instead of 1 year) and you have increased the erosion rate by a factor of data over the default value in the database.

You have now modified the parameters and are ready to run SIBERIA.

- Step 11: To run SIBERIA select the menu item for 'Run SIBERIA' under the 'SIBERIA menu'. A large amount of information will output to the black DOIS screen. When the run is complete (a few 10's of seconds on a typical PC) an appropriate message will output on this screen.

You will now find 4 new files in the demos folder. They are 'demo1-000010.rst2' and 'demo1-000010.rsu' which are the output files for 10 years, and are 'demo1-000100.rst2' and 'demo1-000100.rsu' which are the output files for 100 years.

2.3.1 Demo 2

Demo 2 is a larger version of Demo 1. It is a near natural landform with a rolling hill with a drainage line down the centre.

The main difference with demo 1 is that the .dxf file has a much greater number of x,y,z data values, and it covers a much larger area. Because it covers a larger area you cannot grid it at the same resolution at 10m.

If you run demo2 putting in all the same information as in demo1 EXCEPT that you should use a grid resolution of 100m so that a grid of about 60 x 60 is generated. In addition you should run it with a resolution of 20m. One of the most useful ways to visualise the data form this run is to use Viewer to DRAPE the elevation change over the landform. In this case because the slopes are so low there is very little erosion apparent in the elevations, however, the plot of Z_change shows the changes quite obviously. If you have trouble seeing the difference adjust the range of the colour bar and turn the lighting model on. The difference between the 100m and 20m data sets clearly show the effect of increasing data resolution.

After each run you will now find 4 new files in the demos folder. They are 'demo2-000010.rst2' and 'demo2-000010.rsu' which are the output files for 10 years, and are 'demo2-000100.rst2' and 'demo2-000100.rsu' which are the output files for 100 years.

2.3.1 Demo 3

Demo 3 shows how to use SIBERIA's capability to have spatially variable erosion rates, and how to input .rst2 files (instead of .raw files). The RST2 files are the native format of SIBERIA so they can be input directly into EAMS.

- Step 1: To input the file select then menu item ' Input RST' under the SIBERIA menu. You will be prompted fro a.rst2 file input. Input the file 'demo3.rst2'. You will then prompted for a boundary file and should input the file 'demo3.bnd'. You have now inputted the gridded elevation data and the parameters fro the run (the parameters of the run are included in the header of the RST2 file, so you do not need to input a SIBERIA parameter database.
- Step 2: The next dialog you will request a generic name for the output files, the number of output files (the time of output will requested in the next step … we just need to know how many output are required at this stage), and start date for the run. At this dialog input (1) 0 for the Start Time, (2) 2 for the number of output files, and (3) leave the generic name as it is in the dialog (i.e. C:\program files\landtech\EAMS\demos\demo3).
- Step 3: The next dialog asks for the times at which output is required. Since you input 2 at the last dialog there are two values to be input. Input 100 in the first item and 1000 in the 2nd item, then click OK.

You have now successfully input the model parameters, and the data on what files to be output by the model. The next step is to modify these parameters, and then run SIBERIA.

- Step 4: To modify SIBERIA under DATABASE there are a series of submenus of parameters. Under the RUN parameters menu item change the run time to 1000 years (the default value is 1000 years. Note that even though you have changed the value for the length of run you must at least open up this dialog, otherwise EAMS will not allow you to continue.

You have now modified the parameters and are ready to run SIBERIA.

- Step 5: To run SIBERIA select the menu item for 'Run SIBERIA' under the 'SIBERIA menu'. A large amount of information will output to the black DOIS screen. When the run is complete (a few 10's of seconds on a typical PC) an appropriate message will output on this screen.

You will now find 4 new files in the demos folder. They are 'demo3-000100.rst2' and 'demo3-000100.rsu' which are the output files for 100 years, and are 'demo3-001000.rst2' and 'demo3-001000.rsu' which are the output files for 1000 years. These four files are for the condition where the material erodibility is the same over the whole domain.

You should store these four files above in another directory because we are going to rerun this run and include spatial variable erodibility and we will then compare the results with Viewer.

You should repeat the steps 1-5 just as above with one important difference. When changing parameters you should open the EROSION parameters dialog. You should change the 'erosion model' to 3 (the first item in the dialog, the default value is 0) and the erosion file (the second item in the dialog) you should enter 'demos/demo3.erode' (the default value is blank). This tells the model to input spatial variable erodibility (the value of 3) and the read this data from 'demos/demo3.erode'. Run SIBERIA after this.

If you input both the RST2 files from the with and without spatial variability in Viewer you will see that the difference between the two files in the elevations is very small … in fact its very hard to see from the elevations alone. However, if you plot the ZCHANGE instead of the elevations you will see in the spatial variability data set that there is a large region in the left hand of the data set that has near zero erosion. In demo3 this region has an erosion rate set to a factor of 10 less than the other part of the domain. This region is defined in 'demo3.rgn' and if you look in the file demo3.erode (use wordpad) you will see a command

EROSION RELATIVE 0.1 0.7 1.4 'demo\demo3.rgn'

This line specifies that the erosion rate in the area specified by demo3.rgn is 0.1 of the value elsewhere.

3 Technical Background.

3.1 EAMS

EAMS consists of two linked software packages. The first is EAMS-Moscow (hereafter simply called Moscow) and the second is EAMS-Viewer (hereafter simply called Viewer). Moscow basically controls the simulation of the erosion results. Viewer is used to visualise and analyse the results and prepare data files for Moscow.

Moscow can be considered a menu driven shell that interfaces with a series of other packages as shown in Figure 3.1. The purpose of the four components is:

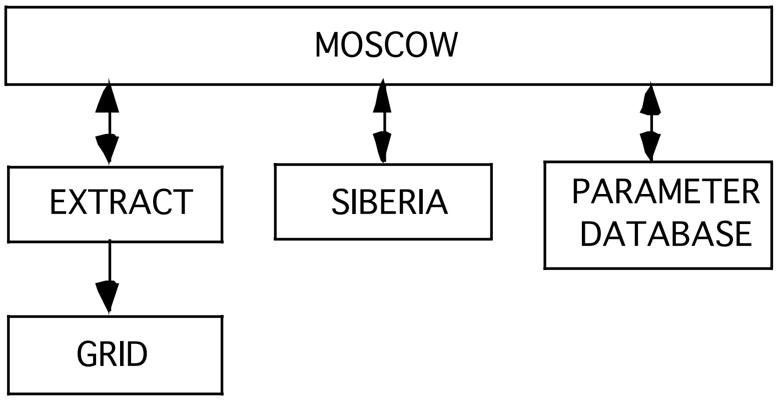


Figure 3.1: Schematic of Moscow

- EXTRACT: This package extracts x,y,z data from a range of different file formats and converts them into a common file format for TIN data (triangulated irregularly spaced nodes) called .raw format (used by the MinDraft mine management package). Current supported file formats are (1) AutoCAD's ASCII dxf format (commonly used by many mine management packages as an exchange format) and (2) a simple text file format that consists of x,y,z triples with each triple on each line (again this is a common data file exchange format). If you are considering relying solely on the dxf file format please ensure that you read the section below on the dxf file format extraction because not all dxf file formats (there are many depending on the version of AutoCAD) are currently supported. We do our best but we can't keep up with all the changes, documented and undocumented, in the dxf file format.
- GRID: This package takes the .raw file output from EXTRACT and grids it onto a grid using a DeLaunay triangulation given the domain characteristics specified by the user. The data is output into a gridded version of the raw format (called .grid.raw).

- SIBERIA: This is the long-term erosion model, or landform evolution model. It operates on a gridded set of elevations. See the next section, and the SIBERIA User Manual provided with EAMS, for a compendium of its capabilities.
- DATABASE: This is the database of erosion and hydrology parameters. A variety of parameters are provided by us as a result of our experience. The user can also extend this database to include their own parameters based on their own data.

Comprehensive details of how these components work and how to control are provided in the sections below.

3.2 SIBERIA

What follows is a description of the philosophy and methodology used by SIBERIA. Greater detail can be found in Willgoose, et al. (1989, 1991a-d, 1994), Willgoose (1993, 1994a,b) and Willgoose and Riley (1993, 1998a,b). For full documentation of the capabilities the user is referred to the SIBERIA User Manual.

The flood response of a catchment to rainfall is dependent on the geomorphic form of the catchment. But the catchment runoff not only responds to catchment form, it also shapes it through the erosion processes that act during runoff events. Over geologic time the catchment form, shaped by the range of erosion events, reflects the runoff processes that occur within it. The channel network form and extent reflect the characteristics of both the hillslope and channel processes. Hydrologists have long parameterised the influence of the geomorphology on flood response (e.g., Rodriguez-Iturbe and Valdes, 1979). Geomorphologists have largely fitted statistics to the landscape ignoring the historic processes that created the landscape (Strahler, 1964; Shreve, 1966) though there have been some notable exceptions to this generalisation (Gilbert, 1909; Horton, 1945). The difficulty of the problem is such that the number of researchers that have attempted to unify the geomorphology and the hydrology is small (Kirkby, 1971; Dunne, 1989; Huggett, 1988), even though the importance of both specialisations has long been recognised by geomorphologists:

> "to look upon the landscape ... without any recognition of the labor expended in producing it, or of the extraordinary adjustments of streams to structures and of waste to weather, is like visiting Rome in the ignorant belief that the Romans of today have no ancestors." (page 268, Davis, 1954)

The main stumbling blocks to the fulfilment of the promise of this scientific paradigm have been the range of temporal scales (geologic versus flood event timescales) and spatial scales (catchment and channel length scales) that are important in the problem; the heterogeneity in both space and time of the dominant processes; and the problem of the unification and observation of the processes acting at these disparate scales. Physically based computer models of catchment development (e.g., Ahnert, 1976; Kirkby, 1987) are important tools in the understanding of the interactions between hydrologic process and response, primarily because of their ability to explore the sensitivity of the system to changes in the physical conditions, without many of the difficulties of identification and generalisation associated with the heterogeneity encountered in field studies.

The ultimate goal is to develop a quantitative understanding of how channel networks and hillslopes evolve with time using a computer model of landscape evolution. Catchment form and hydrologic response will then be seen in the context of the complete history of erosion development of the catchment.

A large scale model of catchment evolution (SIBERIA) involving channel network growth and elevation evolution is documented below. This model integrates a model of erosion processes, theoretically and experimentally verified at small scales, with a physically based conceptualisation of the channel growth process. Neither the properties of the channel network nor the properties of the hillslopes can be viewed in isolation. They must be viewed as components of a complicated large scale non linear system: the drainage basin. The basic tenet of this work is that it is necessary to understand the physics of the catchment processes to be able to fully understand the catchment form and that it is necessary to " ... identify linked process equations and so define geomorphic systems in such a way that an analytical, predictive approach can be used ..." (p. 48, Huggett, 1988). It is not claimed, nor is it intended, that the model presented below account for all the processes occurring in the catchment. Rather a general model framework is presented which is both physically realistic and incorporates the dominant physical processes and which provides a useful tool for the study of the important interactions within the catchment. It is, however, believed to model the dominant processes occurring in fluvial landforms.

A crucial component of this model is that it explicitly incorporates the interaction between the hillslopes and the growing channel network based on physically observable mechanisms. An important, and explicit, differentiation between the processes that act on the hillslopes and in the channels is made. A point is defined to be a channel when selected flow and transport processes exceed a threshold value. If a function (called the channel initiation function) is greater than some predetermined threshold at a point, then the channel head advances to that point. The channel initiation function is primarily dependent on the discharge and the slope at that point, and the channel initiation threshold is dependent on the resistance of the catchment to channelisation. Channel growth is thus governed by the hillslope form and processes that occur upstream of the channel head. The channel initiation function is independent of Smith and Bretherton's (1972) definition of channels as points of instability in the flow equations. Nevertheless, these concepts are not necessarily contradictory. This is particularly true given the recent realisation that Smith and Bretherton's analysis would lead to a system of rills spaced at an "infinitesimal distance apart" unless a basic scale is built into the equations (Loewenherz-Lawrence, 1990). Introducing this scale of separation is conceptually consistent with the threshold analogy. The elevations on the hillslopes and the growing channels interact through the different transport processes in each regime and the preferred drainage to the channels that result. The interaction of these processes produces the long–term form of catchments. The preferential erosion in the channels results in the familiar pattern of hills and valleys with hillslope flow being towards the channel network in the bottoms of the valleys.

The model below simulates the growth and evolution of the channel networks and the contributing hillslopes. Two variables are solved for in the plane: the catchment elevation and an indicator variable that identifies where channels exist in space. In the computer implementation (computer code SIBERIA) a drainage direction is assigned to each node in the discretized space on the basis of the direction of steepest slope from node to node. These drainage directions are used to determine the area contributing to (i.e., flowing through) each node. From these areas, and thus discharge, and the steepest slopes at the nodes, continuity equations for flow and sediment transport are written. These areas and steepest slopes are also used to evaluate the channel initiation function (which may be, for example, overland flow velocity) which is then used in the channelisation function to determine regions of active channel network extension.

The governing differential equations for elevation and channel indicator functions are:

$$\frac{\partial z}{\partial t} = U + \frac{1}{\rho_b}\left(\frac{\partial q_{sx}}{\partial x} + \frac{\partial q_{sy}}{\partial y}\right) + D\left(\frac{\partial^2 z}{\partial x^2} + \frac{\partial^2 z}{\partial y^2}\right) \quad (3.1)$$

$$\frac{\partial Y}{\partial t} = f\left(d_t,\ Y,\ \frac{a}{a_t}\right) \quad (3.2)$$

where in Equation (3.1) z is the elevation (positive upwards); x and y the horizontal directions; t time; U the tectonic uplift per unit time; ρ_b the bulk density of the soil (where the sediment flux is in units of mass/time); q_{sx} and q_{sy} the sediment transport in x and y directions; and D is the diffusivity. Equation (3.1) is simply a continuity equation for sediment transport. In Equation (3.2) Y is the variable describing whether that point in the catchment is a channel ($Y \approx 1$) or a hillslope ($Y \approx 0$), d_t is the rate of channel growth at a point, and a and a_t are the channel initiation function and its threshold respectively. Equation (3.2) describes the transition of a point in the catchment from hillslope to channel on the basis of a threshold in the channel initiation function a. The constitutive equations that are used for channel initiation function, a, and sediment transport, q_s, are those that apply in the region being modelled. Here we use (the details of these equations may be slightly different from the implementation depending on the input parameters for the computer simulations, see later sections for these details)

$$a = \beta_5 q^{m_5} S^{n_5} \quad (3.3)$$

$$q_s = K q^{m_1} S^{n_1} \quad (3.4)$$

$$K = G \quad f(Y) \quad (3.5)$$

$$f(Y) = \begin{cases} \beta_1 & Y = 1 \ (channel) \\ \beta_1 O_t & Y = 0 \ (hillslope) \end{cases} \tag{3.6}$$

where q is the discharge per unit width, β_1 and β_5 are rate constants that may be variable on space and m_1, n_1, m_5, n_5 are typically constants with respect to space and time (though that restriction can be loosened in various ways within the model). K is the erodibility. The coefficient O_t represents a reduction factor, in a reduction exists, in sediment transport rate in the hillslopes compared to that in channels. The factor G is related to the runoff processes to be modelled and assumes particular importance when the subsurface saturation runoff mechanism is simulated. For Hortonian runoff G=1. Methods for determining the other coefficients from the governing physics will discussed later in this chapter. The discharge in a channel, Q_c, and the discharge per unit width, q, again may be parameterised in any way suitable. We choose to represent them here as

$$Q_c = \beta_3 A^{m_3} \tag{3.7}$$

$$q = \frac{Q_c}{w} \tag{3.8}$$

$$w = \beta_4 Q_c^{\ m_4} \tag{3.9}$$

where A is the catchment area draining to that point in the channel, w is the width of the channel at that point, β_3 and β_4 are constants that may be variable on space and m_3 and m_4 are constants with respect to time and space. The equations are solved on a spatial domain with boundary conditions

$$\frac{\partial z}{\partial p} = 0 \tag{3.10}$$

where p is the direction perpendicular to the catchment boundary.

The governing equations are non linear partial differential equations of two states; these two states are elevation, z, and an indicator variable for channelisation, Y. The most important qualitative characteristic of a catchment, the branched network of channels that form the backbone of the drainage system of a basin, is thus explicitly modelled. There are five important variables distributed in space, that are derived from these two states. They are the steepest downhill slope, the contributing area, the discharge, and the distribution of channel initiation function and sediment transport in space. The channel initiation function and sediment transport feed back into, as inputs, the two state equations for elevation and channelisation. Thus there is a non linear interaction between the elevation and channelisation, and the channel initiation function and sediment transport in space. This interaction is the central feature of the model that drives the drainage network growth.

3.3 Extended Output from SIBERIA

The level of detail output of the erosion model is controlled by a file called 'siberia.setup'. All of the extended capabilities of the erosion model SIBERIA are controlled by this file. A sample file is provided with MOSCOW with more complete documentation of how to use it written into the file. Below is simply a summary of its capabilities. Comments in this file are denoted by a '#' character in the first column. All of the commands discussed below are written into this file, all the user has to do is uncomment them out (using Wordpad or some other similar text editor) to make them active.

3.3.1 Storing Diagnostic Output from SIBERIA

Diagnostic output is all the output that is written out to the text window behind MOSCOW as the code is running. This data is sometimes useful to check that a run has been performed correctly or that to check on some more detailed characteristics of the simulation. The command to output diagnostic output to a file is

ECHO FILENAME

where file name is the name of the file that you want the information go to.

3.3.2 More Detailed Data Output

By default the erosion model SIBERIA outputs 8 internal variables from the model. These are output in the file whose name ends with '.rst2'. These are for each point on the grid (1) slope, (2) a spatial variation variable, (2) a variable that indicates whether a channel or gully exists at that point, (4) elevation, (5) area draining through that point, (6) the direction that water drains at that point (7) the depth of any channel at that point and (8) the depth of the soil at that point. Not all of this output is necessarily meaningful as it depends upon what components of the model are active and what parameters are meaningful. Some of these states (eg. channel depth and soil depth) are still in research stage and will not be reliable until a great deal more research is done on channel and soil model.

SIBERIA is also able of putting out other data from the simulation and the user can control which, if any, they want. These are controlled by the file siberia.setup. The generic form of the command to output a particular characteristic is

OUTPUT characteristic

Where 'characteristic' is one of the data types that SIBERIA can output. They are documented in the siberia.setup file but include (this is not a compete list … see the file for more recent options)

YIELD: This is the rate of loss of elevation per year (in mm/year) at that point in the grid at the time at which the file was output.

AVEYIELD: This is the rate of loss of elevation per year (in mm/year) averaged over the catchment draining though that point in the grid at the time at which the file was output. The difference with YIELD is that yield is for that point in the grid while AVEYIELD is the average for the catchment draining through that point.

ZCHANGE: The is the elevation change at the point since the start of the simulation.

AVEZCHANGE: As for AVEYIELD this is the elevation change for the catchment draining through that point since the start of the simulation (note that if the drainage pattern has changed during the simulation the beginning and end catchment will not be the same).

GULLYPOT: Is a plot of the Channel Initiation Function (CIF, see the channel model) throughout the catchment. Even if the channel model is not activate this may be a useful indicator of regions of high risk of gully erosion.

LOGGULLYPOT: Is the log of GULLYPOT. The log may be useful to plot because the typical range of GULLYPOT is normally extremely large and typically ranges over several orders of magnitude.

TONNESHECTARE: This is YIELD with the units of tonnes/hectare rather than mm elevation change. This is simply a conversion based on bulk density.

AVETONNESHECTARE: This is AVEYIELD with the units of tonnes/hectare rather than mm elevation change. This is simply a conversion based on bulk density.

3.3.3 Inputting Regions of Spatially Variable Runoff/Erosion

SIBERIA has the capability to inputs regions that have different runoff and erosion properties from surrounding areas. The general principle is that a default set of hydrology and erosion parameters are specified exactly as we have done with the database above. These are assumed to apply everywhere in the region, except where we say other parameters are to apply. We specify where these other parameters apply and what values these parameters have in another file. An example file called test.erode is included to show the principle. This file must be constructed with an editor separately from the MOSCOW parameter dialogs. Comments in this file are indicated by ‘#’ in the first column.

The principle is that we define a region over which the new parameters will apply (using the region capability of VIEWER, see Section 2.4), we specify which of both or either of the runoff and erosion properties are to be changed in that region, and how those values are to be changed. A sample command to modify the erosion parameters is erosion absolute 0.01 0.6 0.7 'test1.rgn'

The general form of this command is that

- the first word is erosion to indicate that this is to modify the erosion parameters
- the second work is either ‘relative’ or ‘absolute’ to indicate how the parameters are to modified. ‘absolute’ means in the region over which parameters are to changed that the default parameters are to be replaced by the values specified here. ‘relative’ means that the default parameters are to be multiplied by the values here.
- The next 3 numbers are the new parameters in the order, coefficient on the fluvial sediment transport equation, exponent on discharge and exponent on slope.
- Finally the file that specifies the region (called a region file) over which the parameters are to apply. This file name should be surrounded by apostrophes.

The general form of the runoff command looks similar to this with some slight modifications

runoff absolute 1.0 0.8 'test0.rgn'

where

- The first word is runoff to indicate that we wish to modify the runoff parameters
- The second word is ‘relative’ or ‘absolute’ as for the erosion command
- The next two parameters are the runoff parameters in the order coefficient and exponent on area in the discharge equation respectively.
- Finally the region file for the new parameters.

The recommended method for constructing this file is to copy the example ‘test.erode’ and edit and copy individual lines in the file.

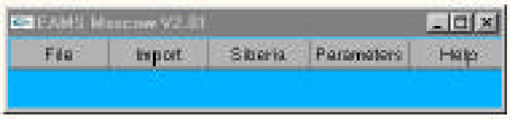
To input the file to MOSCOW the following process must be followed

RUNOFF: The Runoff model in the RUNOFF PARAMETERS dialog must be set to 3 and the name of the file that you have just constructed above must be input into the Runoff Model file box directly below in the dialog.

EROSION: The Erosion model in the EROSION PARAMETERS dialog must be set to 3 and the name of the file that you have just constructed above must be input into the Erosion Model file box directly below in the dialog.

Finally, if you wish to modify both runoff and erosion properties in the one run you can mix erosion and runoff commands in the one file and input the same file name to the two different file boxes above.

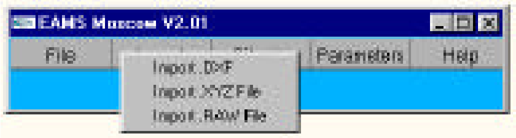
(2) EAMS Moscow Menu Options



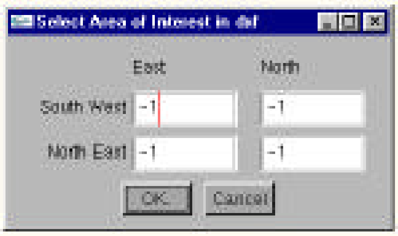
Moscow has 5 pull down menus. The are operated by using the left-hand mouse button. The FILE and HELP menus are self-explanatory. HELP brings up help files in Internet Explorer and you can then explore the capabilities of Moscow in much the same way as you explore an Internet web site.

4.1 IMPORT Menu

Importing a digital elevation map and gridding these elevations onto a grid is a single stage process where you specify the input file name, the area to be extracted and the resolution of the grid to be generated. It is slected by choosing one of the menu options 'Import>Import DXF File', 'Import>Import XYZ File' or 'Import>Import RAW File'.



The menu controls the import of non-native file types into Moscow and their gridding. Three file types are supported. For all three file types the data that is requested next is the same so we will not distinguish between the three in the discussion below. The details of the format of the files are outlined in the section 4.1.1. After triggering the menu you will be prompted for the file name of the to be input. Subsequently you will be prompted for the X and Y coordinates of the edges of the area to be gridded and for the resolution of the grid to be used for gridding the data.



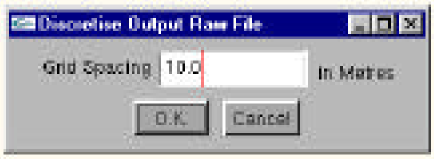
Immediately after selecting the Import menu the user is asked for the name of the file to be imported. Only files with the correct extension are shown (i.e. .dxf for DXF files, .xyz for XYZ files and .raw for RAW files).

Next the eastings and northings of the domain to be gridded are requested. This dialog allows you to extract a small portion from a file that covers a much larger area.

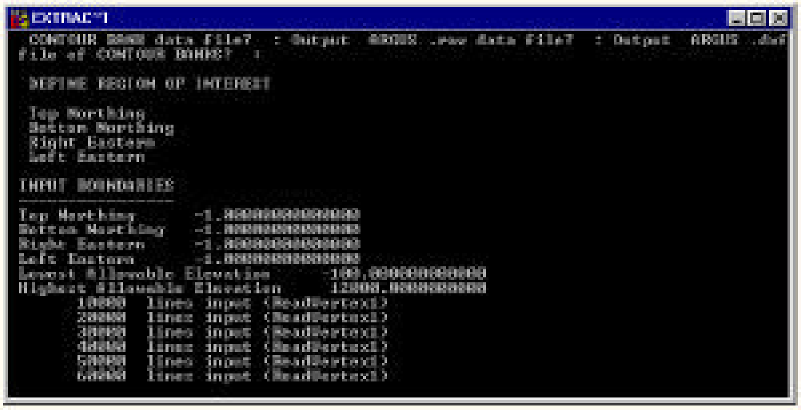
The initial information in the dialog is all –1's as in the figure to the left. If you leave these –1s and simply click OK to this dialog the code will grid all the data in the file. To do this it analyses the input data and makes the west-most point the western edge of the domain and the southern-most point the southern edge of the domain. The eastern edge of the domain is the nearest grid point that encompasses all the data so that the eastern most point is just inside the eastern most edge of the grid. The procedure is similar for the northern edge of the domain.

If you wish to grid only a portion of the file then you must enter the eastings and northings of the portion to be extracted from the file. The top line of the dialog is the easting and northing of the south-west corner of the domain (i.e. bottom left corner of the map) and the bottom line is the easting and northing of the north-east corner (i.e. top right corner of the map). If the distance from the western to eastern boundary is an even multiple of the grid resolution then the eastern edge of the grid coincides with the eastern boundary entered (likewise for the northings). On the other hand if the distance is not an integer multiple of the grid resolution then the eastern boundary of the grid is the nearest integer multiple of the grid resolution that is just to the west of the eastern boundary. Thus the eastern edge of the grid is just to the west of the input value for the north-east corner. The process is the same for the northings.

Finally, when the boundaries are specified only those points inside the boundaries are used in the gridding. Points outside the boundary are ignored.



After inputting the information about the boundaries of the domain to be extracted and gridded the next dialog asks for the resolution of the grid onto to which the input elevation data is to be gridded. Enter the appropriate value and continue.



Immediately after you respond with the resolution of the grid the extraction will start. During this time a series of information lines will be output to the command window. The window to the left shows the start of a typical dxf file extraction. Most of this will not be of interest to users so you may skip to reading the next section at this stage. However, sometimes this data will be useful for diagnosing a problem in the input files. The 6 lines immediately after “INPUT BOUNDARIES” indicate the range of values that will be extracted from the file. Here the –1s against the northings and easting indicate that all data will be extracted. The next two lines show the maximum and minimum elevations that will accepted. These latter two values cannot be changed but reflect the highest and lowest elevations on the Earth, in metres. Be aware that if you have data outside this range it will be skipped. We have occasionally seen dxf files where the elevations have been in cm’s or mm’s and their values lay outside this range. In this case the data cannot extracted until the original files are converted in metres. It is possible the same may occur if the elevations are expressed in feet.

For every 10,000 lines in a dxf file a progress line is written to the screen. Likewise for every 10,000 valid (x,y,z) triples extracted a progress line is output. If you are inputting a large file (several 100,000 data points or more) you may want to take a coffee break because a dxf file can be VERY large and is consequently very slow to input (much slower than an equivalent XYZ file).

4.1.1 The File Types

4.1.1.1 DXF Import

A DXF file is a simple text file (there is binary version of the DXF format but is not widely supported so we do not support it in EAMS) that includes all information describing an object. It is a much more general format than simply the elevations of a surface since it is used for all sorts of engineering design applications. This means it is capable of representing complex objects ranging from mechanical parts like components of an engine to cartographic information including everything that is shown on a topographic map, including land use, houses, roads, names as well as the actual elevations. In addition Autodesk (the developers of AutoCAD, the originator of the DXF format) are continually upgrading the format as they upgrade AutoCAD. Each new revision of AutoCAD results in major increases in the capabilities (and thus the complexity of reading) of the DXF format. On the other hand the DXF format is a common exchange format used by Mine Management packages.

However, writing a general DXF reader is considered a very difficult task. In general any DXF reader can only read a subset of the DXF files presented to it, and this is true even for any given version of the DXF format. The DXF reader in EAMS generally conforms with V11 of AutoCAD (i.e. it can read anything output by V11 of AutoCAD, with some limitations listed below) but is also capable of reading some data generated by later versions of the file format.

The known limitations in the DXF Import software:

- Layers are ignored. The reader reads all recognised x,y,z triples, ignoring any layer information. Thus you should only output from your CAD package those layers which include surface elevation information.

If you have a file that you cannot be read by EAMS (a good indication of this is that it extracts many less valid data points than you think are in the file) please send us the file and we will attempt to incorporate the missing reading capabilities in the next revision of EAMS. This is true even if you are able to work around this problem by generating an earlier version of the DXF file with your CAD software.

4.1.1.2 XYZ Import

This file is particularly simple and, if need be, can be easily created by just about any program. Many CAD packages can generate this format or something closely approximating it. It simply consists of a series of lines, with one data point per line. Each line has on it the x, y and z coordinates of the data point. The 3 data values should be separated by spaces. There should be no lines of information before the first data point and there should be no information in the file after the last data point.

4.1.1.3 RAW Import

This is the native file format used by the MINDRAFT mine management package. It is also the native format of EAMS. The user may then ask why there is a RAW import option at all? You will soon realise that the import of dxf files in particular is quite slow and there may be times when you wish to grid the same domain with a variety of resolutions (e.g. 2m, 5m, 10m) to see the effect of grid size, etc on the model predictions, etc. In this case you can extract the dxf file first time through (which creates a raw file of the x,y,z data) and in subsequent gridding you can use the raw file extracted first time through instead of re-extracting the dxf file. This way you can skip the dxf extraction on subsequent time through the gridding. A .raw file is generated as an intermediate step between dxf (and xyz) extraction and before gridding. It has the same name as the original file but with the .raw extension. If the original file was ‘sample1.dxf’ then after dxf extraction there will be file called ‘sample1.raw’ which consists of the (x,y,z) data in MINDRAFT format. There will also be a file called ‘sample1.grid.raw’ but this is the data after gridding in MINDRAFT format. Remember .raw files are the original x,y,z data while .grid.raw are the result of gridding that data.

Thus to skip the extraction of the dxf file use the .raw file from the previous extraction. Note you will still be prompted for the boundaries. The boundaries are used in the gridding process, NOT the extraction so when the original .raw file is created it will have the elevations for the entire domain. So if you enter –1’s you will get the whole domain of the original dxf file.

4.1.2 Gridding Data

The gridding is done by:

- Creating a DeLaunay triangulation of all the points inside the specified domain.
- Interpolating the elevations for the grid off the DeLaunay triangulation.

Since only points inside the domain are using in creating the triangulation then there will a small area along the edges of the boundary that cannot be gridded. This is the region where a triangle would otherwise have been created that would traverse the boundary with some vertices inside the boundary and some outside. If this is a problem then the user should consider using a larger boundary.

Be mindful, however, that even in this case the boundaries may need to be some distance from the edge of the region being studied by SIBERIA, otherwise there may be boundary effects on the erosion simulations. An example of this would be where deposition in the real world is creating an alluvial fan that extends out to the boundary. On the boundary of the domain SIBERIA imposes fixed elevations which will suppress the alluvial fan in the simulation (because the elevations can’t increase at the boundary). This suppression of deposition may propagate upstream, and effect the simulated erosion and deposition some distance from the boundary.

4.2 SIBERIA Menu

Once the elevation data is extracted and gridded simulation with SIBERIA is a two stage process. The first stage is to tell EAMS what the run characteristics are going to be (e.g. model parameters, run time, at what times to output data for analysis). The second stage is to actually run SIBERIA.

We discuss the first stage first. This stage is triggered by selection of one of the two menu options ‘Siberia>Input from RST2’ or ‘Siberia>Input from RAW’.

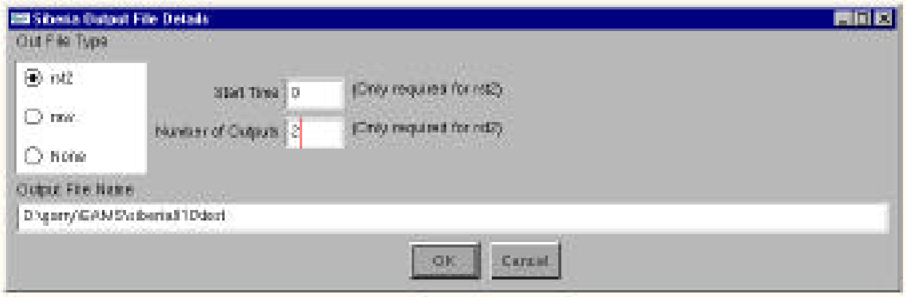
The first dialog presented is a file input dialog for the elevation to be used as the initial conditions for the simulation.

- For RAW input there is a single file dialog requesting the .grid.raw file to be used for input.
- For RST2 input two file dialogs are presented one after the other. The first file dialog requests the RST2 file to be input. This file has in it elevations and parameters, as well as other information that may or may not be used by SIBERIA depending on the run options requested for the SIBERIA run in the dialogs that follow. The second file dialog requests the BND file that for that RST2 file. This file has the computational boundaries in it (i.e. what nodes are used in the calculations and which ones are ignored). Some RST2 files do not need BND files. In this latter case you should just press cancel for the BND file dialog without selecting a file.

The directory that the RST2 or RAW file resides in is then set as the default directory for all the output files.

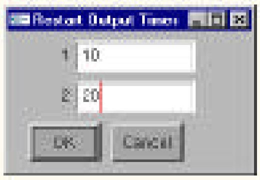
The dialogs that follow are the same for both RST2 and RAW input.

The next dialog inputs information about the data files to be output with the results of the simulation in them. By default, the user should not change the format of the file to be output from the default (RST2, indicated by the radio button in the top right hand corner) RST2 file format. This is the file format expected by Viewer. RAW file format can only be viewed by MINDRAFT and the ACIRL mine rehabilitation package ARGUS.



The next item to fill out in the dialog is the generic filename for the output. The default entry in this field is the directory in which the original RST2 or RAW file is in and the filename is the same file name as this initial filename. The actual filename is made by using this generic filename as a starting point and then appending onto it the time at which the output is made by SIBERIA. The process of construction this file name is fairly straight forward. It is the number of timesteps of the simulation plus the 'Start time' entered in this dialog. Thus if you request that output is to be produced after 12 timesteps/years (the input of this data comes in the dialog) and you input 1970 as the start time then in this example the file name of that output will be test-0001982.rst2.

Finally the last piece of information in this dialog is the 'number of outputs' to be produced. This is the number of times at which you will want output. This of you want outputs at times 10 and 20 you should input 'Number of Outputs'=2. Then press OK to this dialog.



After this another dialog comes up which request the times at which outputs are requested. It has the same number of input boxes as requested in the previous dialog for 'Number of Outputs'. In the case at left because you input 2 at the previous dialog there are 2 boxes. If are more than 10 times the first 10 will be requested and a new dialog will request the next 10, etc until all the times are input.

The times of output should be input in ascending order.

Once these data are input you are ready to set up the characteristics of the SIBERIA run. This process is a little different for RAW file input and RST file input so please continue with appropriate section that follows.

4.2.1 RST2 Input

The next step involves setting the run parameters, erosion parameters, hydrology parameters, etc. Default parameters for the simulation are read from the RST2 file. These parameters are in the first 27 lines of the RST2 file, but unless you know what you are doing we discourage you from messing with this file because SIBERIA expects it in a very specific form. These parameters need to be finetuned for your simulation. EAMS will not allow you to run SIBERIA until you have done this.

Details of the various parameter options see Section 4.3.2. As an absolute minimum you will need to specify the run time in years of the simulation (this is the 'Run Parameters' dialog). Once you have done this you can then go to the menu 'Run Siberia' in Section 4.2.3.

4.2.2 RAW Input

The next step involves setting the run parameters, erosion parameters, hydrology parameters, etc. If you have requested input from a RAW file you will be immediately asked to open a SIBERIA database (go to 'Open Database' section of Section 4.3 to see how this works) and select a parameter data set to be used. This parameter set then becomes the default parameter set for your run. You can always go back and select a new parameter set, and this will overwrite all your current parameters.

One you have input the database you must modify the parameters for the SIBERIA run. The database selection process above for RAW files reads default values from the database. You still need to finetune them for your simulation. EAMS will not allow you to run SIBERIA until you have done this.

Details of the various parameter options see Section 4.3.2. As an absolute minimum you will need to specify the run time in years of the simulation (this is the 'Run Parameters' dialog). Once you have done this you can then go to the menu 'Run Siberia' in Section 4.2.3.

4.2.3 RUN SIBERIA

When you select this menu option two things are done

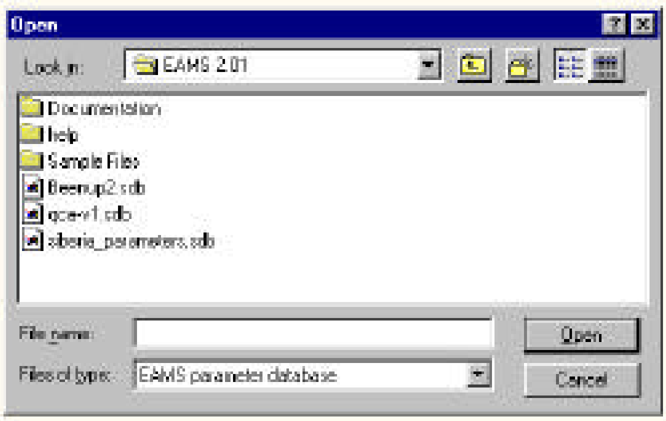
- The data files required by SIBERIA are constructed. The details of this process are not important but sometimes it may be necessary for us to look at these files to diagnose a bug in the software, or a run time problem you are having. The run data is all summarised in the file 'siberiakb.com' in the 'Program Files' directory on the System Disk. In addition for RAW file input the requisite RST2 (i.e. initial elevations and dummy parameters) and BND (i.e. computational boundaries) files expected by SIBERIA will be generated. The names of these files are based on the name of the input RAW file. If the input raw file is call 'junk.raw' then the RST2 file name will be called 'junk-start.rst2' and the boundary file will be called 'junk.bnd'. The RST2 file can be used in EAMS-Viewer just like any other RST2 file (i.e. just checking to make sure the data is OK or perhaps selecting regions that have different erodibilities or runoff, etc).
- SIBERIA is run using the data files just constructed.

4.3 DATABASE Menu

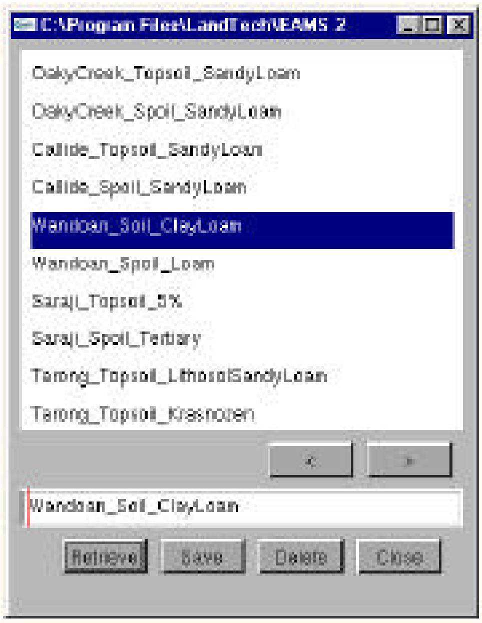
This menu controls the opening and manipulation of the database(s) containing the SIBERIA parameters.

The process that needs to be followed here is to open a SIBERIA parameters database, select a set of parameters that are considered by the user to be appropriate, and then edit these parameters to set up the run.

4.3.1 Open Database



The EAMS distribution installation supplies a number of standard databases. In addition you may create your own for your project (more on that later). The extension for a SIBERIA database is '.sdb'. The standard databases are to be found in the program files directory on the system disk.



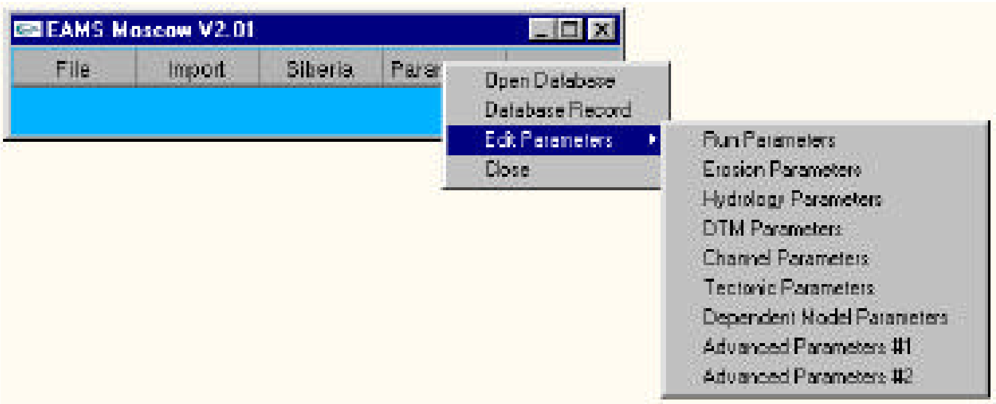
When you have selected a database you will be prompted by a dialog to select a parameter set from this database. A typical database is shown to the left. The large scrolling window (up and down are controlled by the left and right arrows near the bottom right of the window) consists of a series of names for datasets. To select a dataset you simply scroll up and down until you find the set you want and click once on the name to highlight it (it will also appear in the text box at the bottom of the dialog; in this case we show selection of 'Wandoan_Soil_ClayLoam') and then click on the 'Retrieve' button to extract those parameters from the database.

Do not worry if you have selected the wrong dataset because you can always go back and select a new data set with the 'Database Record' menu item in the original Database menu. This menu item simply returns you to parameter set selection dialog above.

4.3.2 Database Record

This menu item simply returns you to parameter set selection dialog above. This allows you to input a new parameter set without having to first select the parameter database.

4.3.3 Set Parameters



Once you get to using this set of menu items you will have extracted the default erosion, hydrology and other parameters from the database. You now need to finetune the SIBERIA run characteristics (mainly how long do you want to run the simulation for, perhaps different hydrology and erosion characteristics). At very least you will wish to check that the default values are in fact appropriate for your problem. The 'Edit Parameters' menu item above provides access to all the current values for the parameters that are required for a SIBERIA run.

EAMS requires you check the parameters before you will be allowed to Run SIBERIA. At very least you will always have to select the 'Run Parameters' to set the duration of the run. If you don't EAMS will refuse to run SIBERIA. You may choose to select other parameters as well but they are not compulsory.

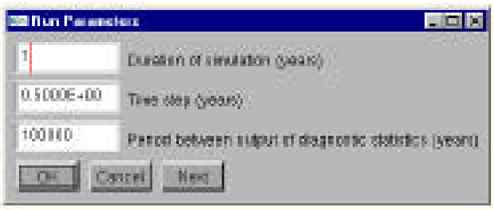
4.3.3.1 Run Parameters

SIBERIA names for Parameters:

1: RunTime (§3.2.1)

2: InitTimeStep (§3.2.2)

3: StatsTime (§3.2.1)



The run time is the duration (timesteps) from beginning to end of the simulation. The units of years imply that the erodibility and tectonics parameters are for that process per year (i.e. one timestep = one year). [ADVANCED: While this is the default option this is not the only way to use this capability. The run time could be interpreted as months provided the rates of the processes are calculated to be per month. For instance you could input that the run time is to be 10 years in which case b1 will be interpreted as the erodibility of the material/year. However, if you divided b1 by 12 then you could run the model for 120 time steps and get the same result. While normally you will be interested in the erosion per year there can be occasions where having better temporal resolution can be useful].

The Time Step (i.e. InitTimeStep) is the resolution used internally within SIBERIA during its calculations. The appropriate size of the time step is determined by the stability and mass balance compliance of the solver. If you input a negative number then SIBERIA will dynamically estimate the time step based on an approximate and empirical criteria based on a maximum 1% mass balance error (use this option with caution for DEM's with large flat areas as it can sometimes set ridiculously small timesteps). In either case the user should assure themself that the resolution is high enough to ensure that the results are correct. For a typical temperate climate, a resolution of the DEM of 10m and moderate slopes (mostly less than 20%), a normal agricultural soil and a domain of size 1km x 1km a good starting point for a time step is about 0.01 when the time step is in years. This is not, however, a golden rule and it is sometimes useful to start with a very large timestep to get a rough, qualitative idea of what the erosion is like without excessive computational times, particularly if you are working with a large domain. If you do this be aware that the results may be grossly in error and it is possible that SIBERIA will crash and burn without warning.

The period between diagnostic statistics is simply the time at which summary statistics are output to the DOS window. This output is particularly useful when the user sets 'siberia.setup' to echo all simulation output to a file. In this case the diagnostic statistics are output to the file specified in 'siberia.setup' as well as to the screen. This allows some simple monitoring of simulation output, such as is sometimes required to correctly set the timestep for difficult problems.

4.3.3.2 Erosion Parameter

SIBERIA names for Parameters:

1: ModeErode (§3.2.1)

2: FileErode (§3.2.3)

3: b1 (§3.2.2)

4: m1 (§3.2.2)

5: n1 (§3.2.2)

6: QsHold (§3.2.2)

7: b12 (§3.2.2)

8: m12 (§3.2.2)

9: Ot (§3.2.2)

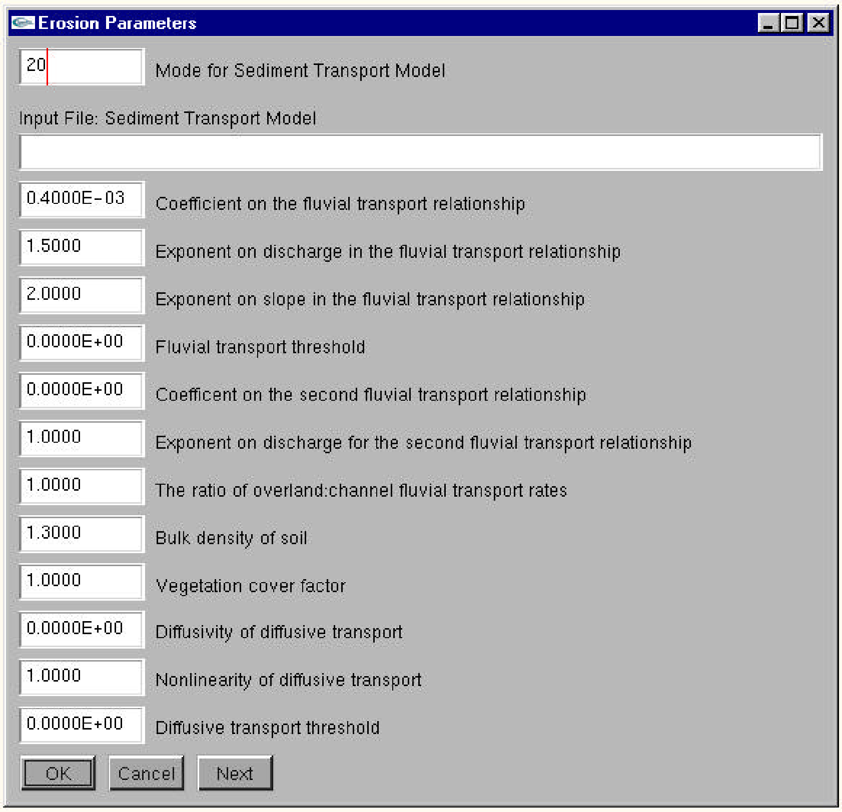
10: Bulk (§3.2.2)

11: Cover (§3.2.2)

12: dZ (§3.2.2)

13: dZn (§3.2.2)

14: dZHold (§3.2.2)



There are a range of erosion models that can be selected. Full details of the capabilities of this component of the model are to be found in the SIBERIA User Manual.

The erosion model selected is determined by the value of ModeErode. The default model is invoked by either ModeErode equal to 0 or 20. When either ModeRunoff (see the runoff parameters) or ModeErode is greater than or equal to 20 the grid resolution input by GridXY in the DEM options parameters is used and all properties are input as per unit width. When both ModeRunoff and ModeErode are less than 20 the grid resolution is assumed by the model to be 1 and GridXY is ignored so that all properties are per node. Normally a user will input ModeErode=20 to select the default erosion model while using GridXY.

For the default erosion mode with fluvial erosion the typical user will input values for b1, m1, n1, Bulk and Cover. If they have the data to calibrate the soil creep they will also input a value for dZ. All other parameters would normally be left as the default values.

Some notes can be made about the units of the processes. It is here that most mistakes are made in setting parameters. The parameter b1 would normally be calibrated such that the erosion is measured in cubic metres of sediment per unit time and per unit width of flow and the bulk density would be input as tonnes per cubic metre. This ensures that the elevation changes calculated by the model are in metres. For the diffusivity dZ the units here are in cubic metres of bulk soil per unit time and per unit width.

The user should also remember that in the absence of channels (see the channel model) the erosion rate for the domain (which is hillslope) is

$$q_s = \beta_1 O_t q^{m_1} S^{n_1}$$

so the presence of the term O_t should not be neglected in the determination of the parameter values. It is suggested that in cases where channels/gullies are not modelled that O_t should be set to be equal to 1.

As a final check that the correct parameters are determined it is worthwhile to check by hand that the correct amount of material is lost per year during the simulation by comparing it with what the calibration data would suggest. This can be easily checked locally by looking at erosion lost/year in Viewer (i.e. the mean elevation over some set portion of the domain for two different times of a simulation). A more accurate estimates can be determined by looking at the change in total mass in the statistics controlled by StatsTime in the ‘Run Parameters’ dialog. Total mass is simply the sum of the elevation of every node in the domain. Thus the erosion loss can easily be calculated by taking the difference between total mass at two times, and dividing it by the number of years between the two times and the number of nodes (given by ‘Total Area’ in the same table of statitsics) to give the elevation change per unit time. The statistics tabled used here can be accessed in the log file specified for output in the file ‘siberia.setup’.

For methods to calibrate b1 the user is referred to the SIBERIA User Manual.

4.3.3.3 Hydrology Parameters

SIBERIA names for Parameters:

1: ModeRunoff (§3.2.1)

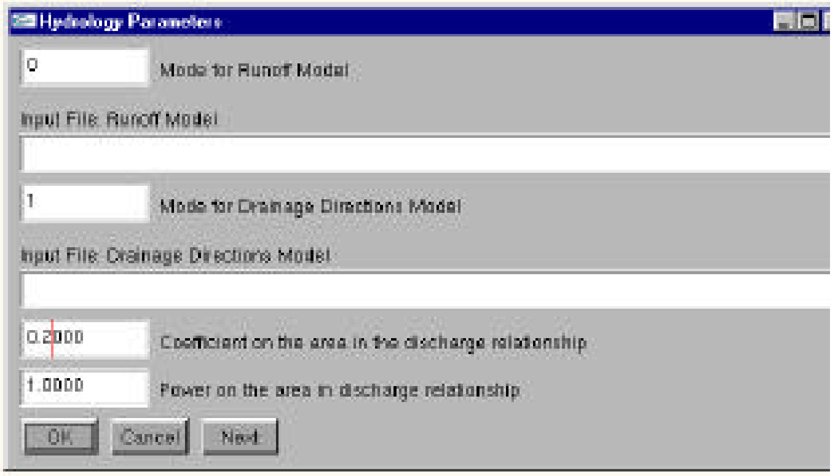
2: FileRunoff (§3.2.3)

3: ModeDir (§3.2.1)

4: FileCtrBank (§3.2.3)

5: b3 (§3.2.2)

6: m3 (§3.2.2)



There are a range of runoff models that can be selected. Full details of the capabilities of this component of the model are to be found in the SIBERIA User Manual.

The runoff model selected is determined by the value of ModeRunoff. The default model is invoked by either ModeRunoff equal to 0 or 20. When either ModeRunoff or ModeErode (see the erosion parameters) is greater than or equal to 20 the grid resolution input by GridXY in the DEM options parameters is used and all properties are input as per unit width. When both ModeRunoff and ModeErode are less than 20 the grid resolution is assumed by the model to be 1 and GridXY is ignored so that all properties are per node. Normally a user will input ModeRunoff=20 to select the default runoff model while using GridXY.

For the default runoff mode the typical user will input values for b3 and m3. All other parameters would normally be left as the default values.

Some notes can be made about the units of the processes. It is here that most mistakes are made in setting parameters. The parameter b3 would normally be calibrated such that the runoff is measured in cubic metres of runoff per second for a 1 in 2 year runoff event (see the SIBERIA User Manual for the reasoning behind this choice) per unit width of flow. This discharge is then used in the calibration of the erosion model.

In the event that the runoff properties of the landform themselves are not of interest to the user then a simpler calibration approach is possible which mean that output runoff values are not meaningful but the erosion predictions are still correct. If the default runoff model is substituted into the erosion equation then the following equation results

$$q_s = \left(K\beta_3^{\,m_1}\right)A^{m_1 m_3} S^{n_1} = K' A^{m'} S^{n_1}$$

and given a known area and slope for a plot to correctly calculate the erosion all that is necessary is to calibrate K' to the data. When inputting this into EAMS it is sufficient to set $\beta_3 = 1$ and $m_3 = 1$, and then set the erosion parameters such that $K = K'$ and $m_1 = m'$.

One simple way to do this calibration is to use another model (e.g. USLE, CREAMS, WEPP) to predict the erosion rate on a range of plots with different lengths and slopes over the range likely to be encountered in your problem (we recommend at least 3 of each). Then use this data to do a multiple regression against area and slope to determine the values of K', m', and n_1.

The parameter ModeDir is most useful for inputting contour banks where flow is constrained in the directions it can flow. The default model without Contour backs is ModeDir=1. Before using this model the user should carefully read the SIBERIA User Manual. We are currently working on extensions to Virtual Earthworks in EAMS-Viewer to automate the construction of the files necessary to model contour banks.

4.3.3.4 DTM Parameters

SIBERIA names for Parameters:

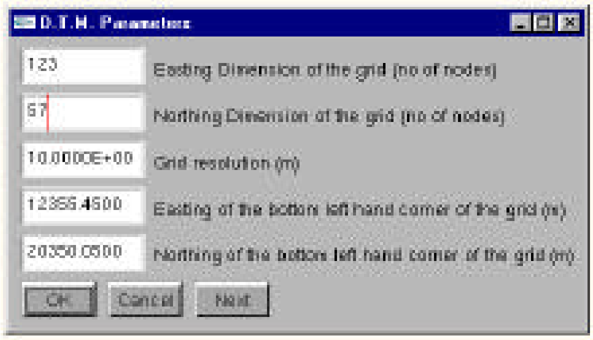
1: kx (§3.2.1)

2: ky (§3.2.1)

3: GridXY (§3.2.2)

4: East (§3.2.2)

5: North (§3.2.2)



These are the characteristics of the elevation data used in the simulation. You should never modify the first two parameters. These tell SIBERIA how big the grid is that being input to the model. If these are changed SIBERIA will be unable to read the RAW or RST2 file input.

The third parameter gives the resolution of the grid. This also should not be changed because it can be used within SIBERIA to convert between discharge/unit width to discharge/node within the model. Thus changing this may lead to incorrect results.

The final two parameters give the south-west corner of the grid. These number are not currently used in SIBERIA but are used in EAMS-Viewer to georeference the digital maps. The final three numbers should agree with what you input when you extracted and the gridded the data earlier.

4.3.3.5 Channel Parameters

SIBERIA names for Parameters:

1: ModeChannel (§3.2.1)

2: 1/at (§3.2.2)

3: b5 (§3.2.2)

4: m5 (§3.2.2)

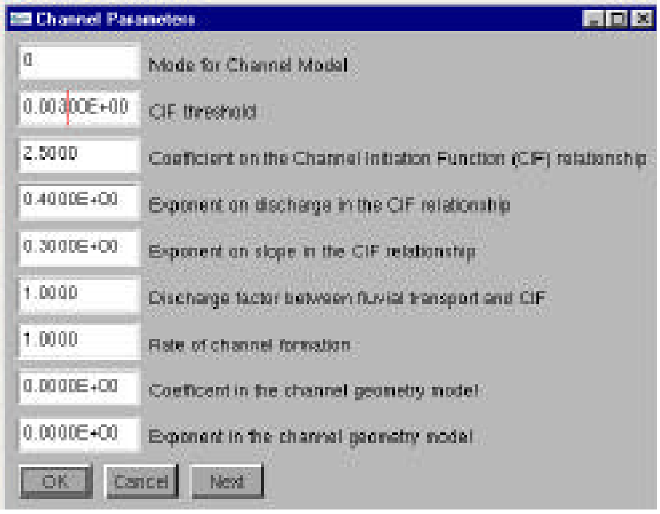
5: n5 (§3.2.2)

6: a1 (§3.2.2)

7: DTime (§3.2.2)

8: b6 (§3.2.2)

9: m6 (§3.2.2)



This component of the model controls the ability of SIBERIA to generate channels. When this model is not used the catchment is considered to be entirely hillslope and erosion rates are calculated as if the catchment is entirely hillslope.

The operation of this component of SIBERIA and the user is referred to the SIBERIA manual for a detailed discussion of how this model works.

4.3.3.6 Tectonic Parameters

SIBERIA names for Parameters:

1: ModeUplift (§3.2.1)

2: FileUplift (§3.2.3)

3: TimeUp (§3.2.1)

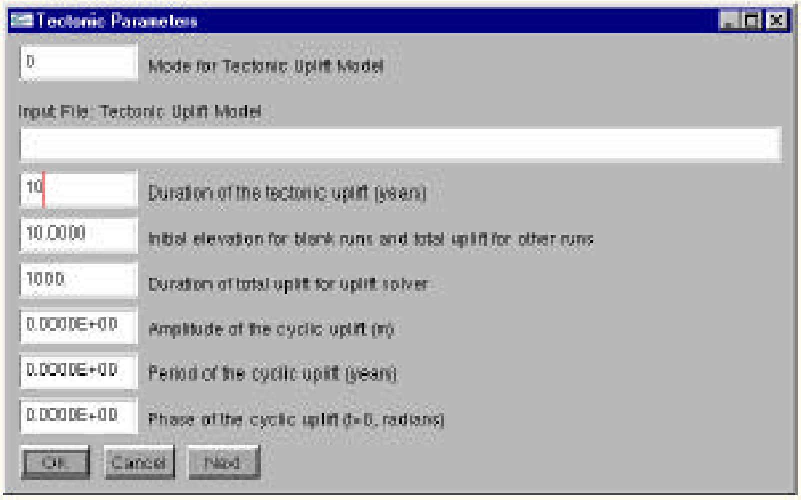
4: ZInit (§3.2.2)

5: Notch (§3.2.2)

6: TAmp (§3.2.2)

7: TPeriod (§3.2.2)

8: TPhase (§3.2.2)



For the vast majority of practical applications most users will not use this capability. For greater detail on how to use this capability see the SIBERIA User Manual.

4.3.3.7 Advanced Parameters #1

SIBERIA names for Parameters:

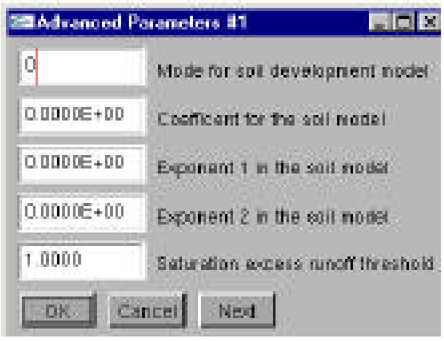
1: ModeSoil (§3.2.1)

2: SoilRate (§3.2.2)

3: SoilExp1 (§3.2.2)

4: SoilExp2 (§3.2.2)

5: SMThreshold (§3.2.2)



This model is part of an ongoing research project into the development of soil profiles using SIBERIA. You should ***never*** modify these parameters. You use this model at your own risk.

4.3.3.8 Advanced Parameters #2

SIBERIA names for Parameters:

1: ModeI (§3.2.1)

2: ModeSolver (§3.2.1)

3: ModeRandom (§3.2.1)

4: ModeMC (§3.2.1)

5: DirReg (§3.2.1)

6: FactMx (§3.2.2)

7: FRanMn (§3.2.2)

8: YFix (§3.2.2)

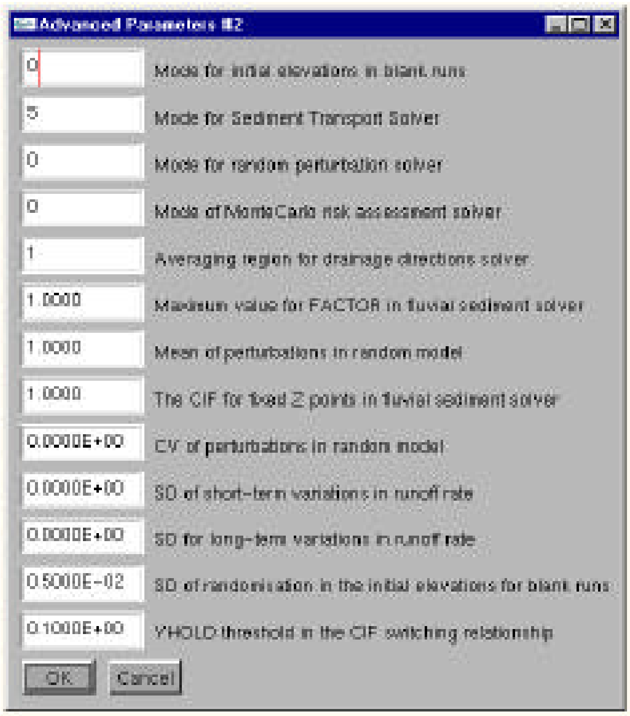
9: FRanCV (§3.2.2)

10: b3SDs (§3.2.2)

11: b3SDl (§3.2.2)

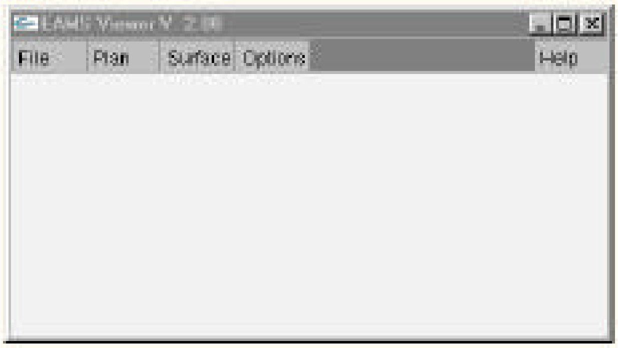
12: FranZ (§3.2.2)

13: YHold (§3.2.2)



Most of these parameters control the internal operation of SIBERIA or control very advanced modes of operation of SIBERIA that should be used with extreme caution. It is ***unlikely*** that you will ever need to modify these parameters in normal day to day applications. Moreover, if you modify them it is possible that SIBERIA will not run or will give erroneous results.

5 EAMS Viewer Menu Options



Viewer Window

When EAMS-Viewer is initially started two windows will appear on the screen.

The 1st window looks like the one at the left with a menu bar across the top. This window is used to control the application. The menu items are triggered with the left mouse button. Below we will refer to this window as the 'viewer window'

The 2nd window is a black DOS command window and opens in the background. Nothing of any importance will appear on this window but do not attempt to close this window as doing this will kill the application. If this black window gets in the road you can shrink it to the window bar at the bottom of the screen.

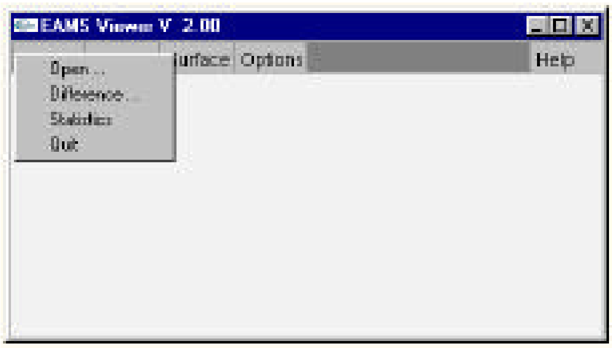
Viewer Window showing content region popup menu

There are two parts to the viewer window (see left).

The first part is the menu bar across the top. These menu items are triggered by use of the left mouse button and the individual items are explained in the sections that follow.

The second part is the content portion of the viewer window which is the large white section below the menu bar. If you do anything that triggers some form of output to the screen then this information appears here. This window can be resized to provide more visible lines of output and the window can be scrolled up and down using a popup menu in the content portion (see above). This popup menu is accessed using the right mouse button.

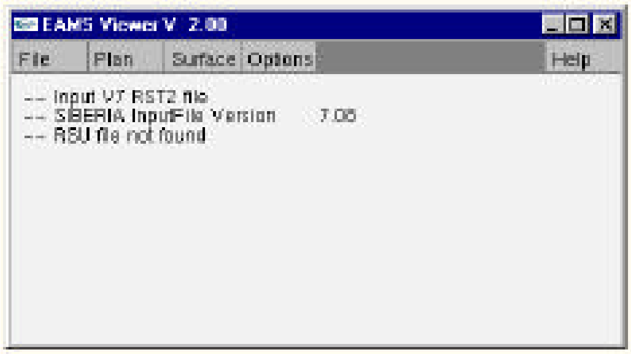
5.1 File Menu



Viewer Window showing the File pulldown menu

The file menu controls all aspects of manipulation of files of data. The individual menu items are explained in the following sections.

5.1.1 Open

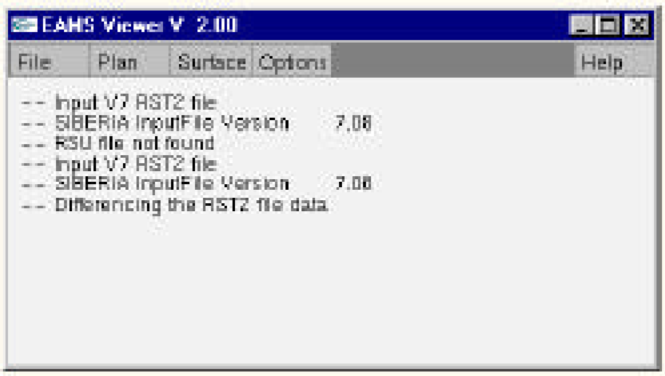


Viewer Window and the content region after file input

The open menu item is used to input the RST file input. If when you ran SIBERIA you also generated a corresponding RSU file (see the SIBERIA manual and the ‘siberia.setup’ file) then Viewer will also input this as well provided it is the same directory as the original RST2 file.

The content window shown here is for a RST2 file (generated in this case by version 7.08 of SIBERIA) and the Viewer has not found a corresponding RSU file.

5.1.2 Difference



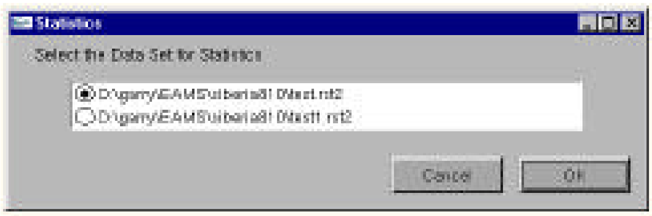
Viewer Window and the content region after file input using differencing

The difference menu item allows you examine the difference between two SIBERIA output files. As for the open menu, if there are corresponding RSU files they are also input and differenced.

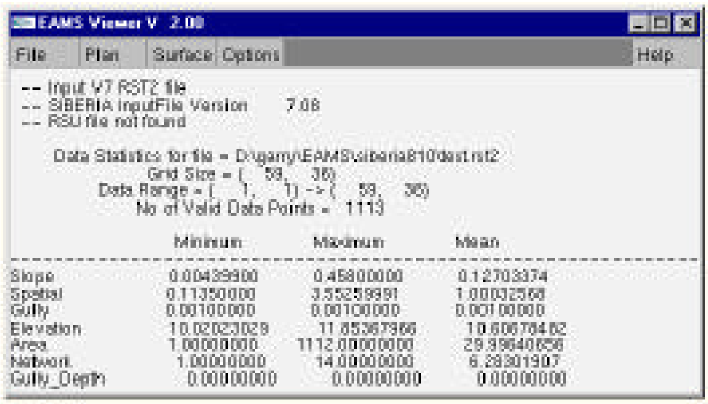
After selecting this menu item you will be presented with two file input dialogs (corresponding to the 1st and 2nd files respectively). After input of the two files the content window will look something like the screen snapshot above.

After inputting the files you will find under the PLAN and SURFACE menu items two additional datasets. The 1st dataset is simply the first data set input. The 2nd data set is the first data set *minus* the second dataset. The menu item for the 2nd data set will have a ‘(-)’ before the name of the 2nd file to indicate that it is a difference data set and that it is the file above it in the menu list *minus* the one in the menu item.

5.1.3 Statistics



This command request statistics for each data file and is slightly different from the statistics command that is part of the popup menu in the PLAN windows. It outputs statistics for all of the states in the nominated file (selected from the dialog on the left that comes up after requesting this option) over the whole domain. If you wish to get statistics for a region of the file then you must select statistics from the popup menu in the PLAN window.



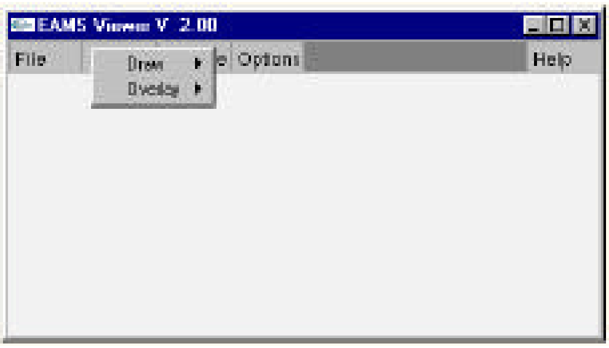
Viewer Window and the content region after statistics output

A typical set of results from a statistics request is shown at left. Note that the statistics are provided for the region within the boundaries of the domain. The ‘no of valid points’ is the number of points identified to be within the boundary. In general the ‘network’ statistics result is meaningless because it simply summarises the integer values for drainage directions (see the SIBERIA manual for an explanation for how this works).

5.1.4 Quit

This command quits the application and closes all currently open windows and files.

5.2 PLAN Menu

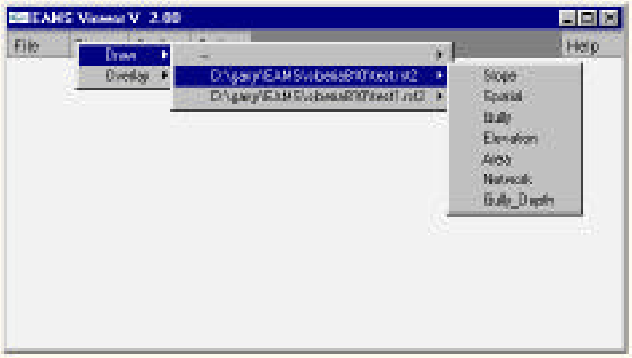


Viewer Window and the Plan pulldown menu

This menu controls the display of a plan of a data set a plan window. The plan window is the main window in which data analysis is done and is the work house of Viewer.

There are two menu items on this menu and they are discussed below.

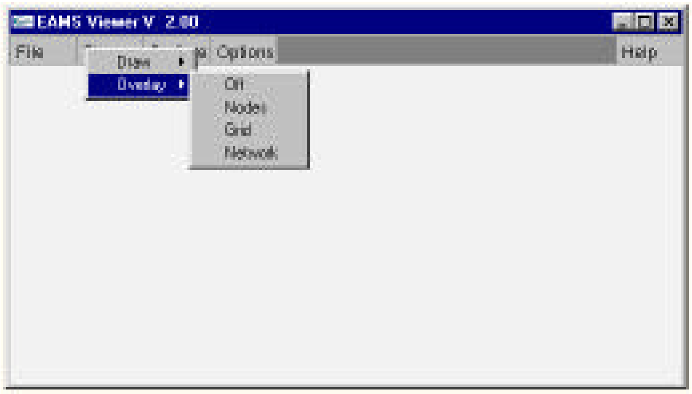
5.2.1 Draw



This menu item allows you to select the data set that you wish to display on the screen. Initially when Viewer is started there are no submenus to this menu item but each time a file is opened its data is added to the menu item as shown at the left.

To select a dataset to be displayed simply select the data set in this drop down menu. Aftre selection of this item the plan window will be drawn on the screen using defaults for the size of the window and the data range. The appearance of this window can then be manipulated in the plan window (see below).

5.2.2 Overlay



The overlay menu item allows you to overlay various pieces of information on top of all existing plan windows. This overlay is always positioned on top of whatever changes occur to the plan picture of the data. When the overlay option is selected it is applied to ALL plan windows.

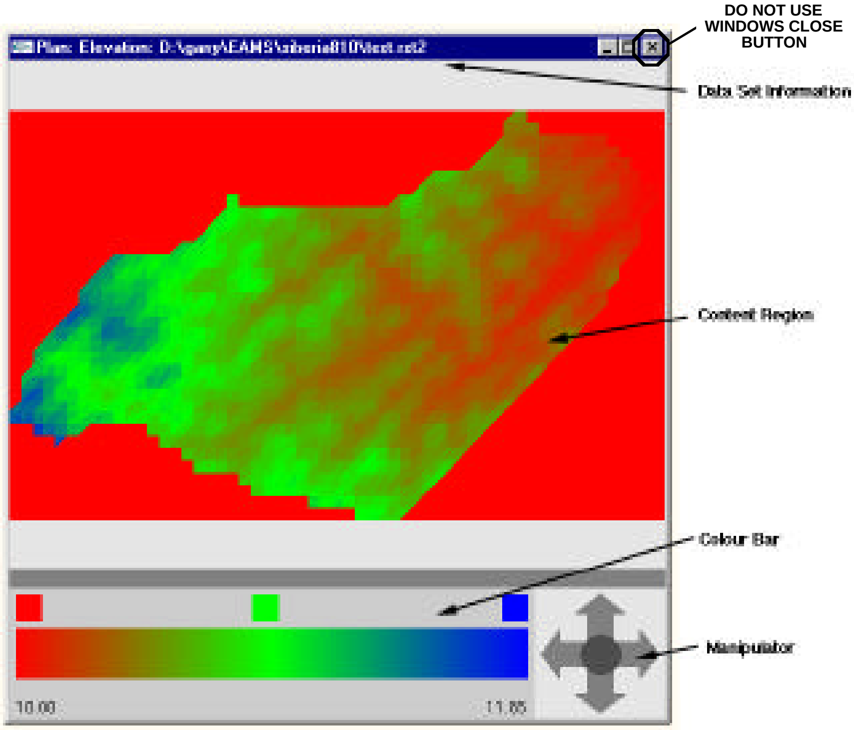
The data than can be overlayed are

(1) the position of each data point in the domain

(2) the grid that the data is displayed on

(3) the drainage network for the domain.

5.2.3 The Plan Window



The Plan Display Window showing its various component

When initially displayed the plan window will look something like the picture above. There are three regions in this window that will be discussed in detail in the following sections. In each of these regions there is a popup menu for modification of the window appearance. These popup menus are accessed with the right mouse button. The three regions of the window are

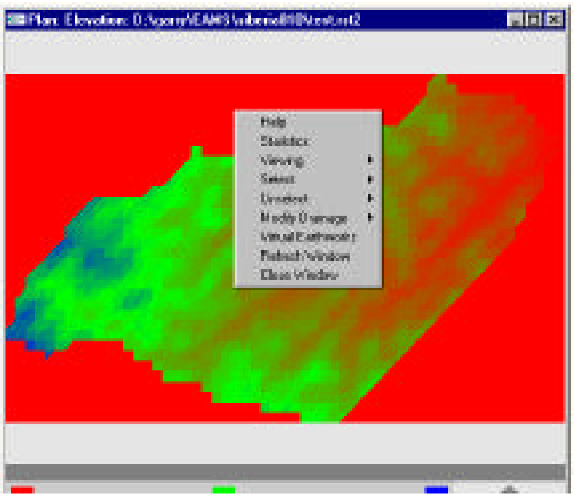
(1) The Content Region: This is the large region in which the data is displayed.

(2) The Colour Bar: The colour is the left hand part of a strip across the bottom of the window and controls the colours used in the content region.

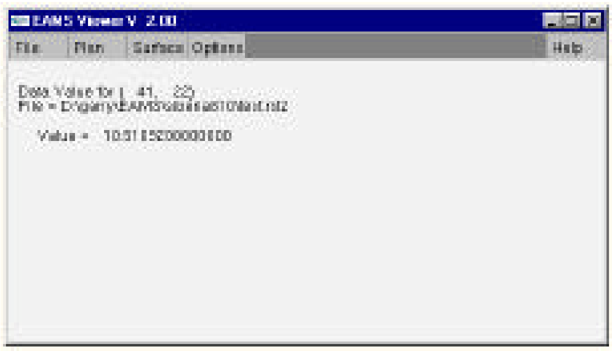
(3) The Manipulator: This is the set of arrows on the right hand side of the strip on the bottom of the window. This controls the position of the data in the content region and allows you move left, right, up and down as well as zoom in and out.

NB: It is important NOT to use the Windows close button (the cross in the top right corner the window as this will totally quit EAMS-Viewer, not just close the window).

5.2.4 Plan Window Components: Content Region



Content Region with main popup menu visible



Viewer Window with results for a Single Point

Viewer Window with results for a Rectangle, showing the left and right boundaries of the selection and the data range.

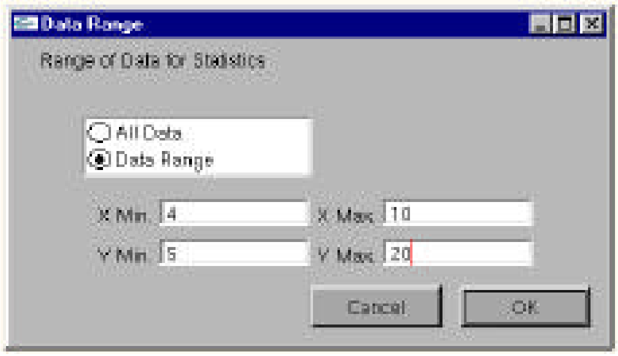
The content region of the window is that region in which the data is displayed. There is a pop-up menu in this region (accessed using the right mouse button) that provides tools for data analysis. The picture to the left shows the main popup menu in this region and we will now discuss the role of the main menu item here.

In addition you can also inquire the value of the variable in the content window by use of the left mouse. This doesn’t require the pulldown menu.

Inquire a Single Point: Click with left mouse button on the node for which you wish to know the value. The position and value of that point are output in the Viewer Window (see left).

Inquire a Rectangle of Points: You select a rectangle of points by rubber banding a box. Simply left mouse click at one corner of the desired rectangle and with the mouse held down drag to the other corner, then release the mouse. The results for the rectangle are output in the Viewer Window (see left). If you wish to have more precise control of the rectangle position use the pulldown menu in the window, which will allow you to type in the rectangle coordinates.

Statistics>



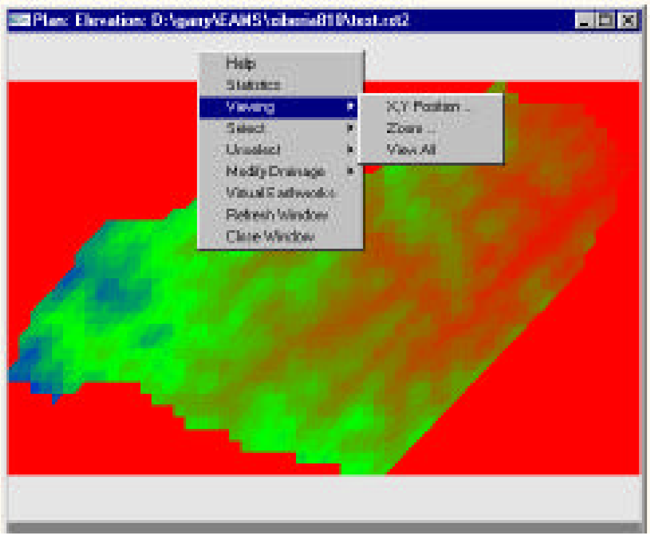
The Data Range dialog

The Viewer Window with the results of Statistics request. Note that the data range and the number of data points are also output. Note that the number of 'valid data points' will be less than the number of points you would calculate from the dimensions of the rectangle if the rectangle partly lies outside of the boundary of the landform.

The statistics menu item allows you more precisely control the region for which you want to data results than is possible by using the rubber banding in the content region.

When you request this option you are presented with a dialog as on the left. You may request to analysis the whole of the region (i.e. click in the radio button that says 'All Data') of a rectangular subset of a region (i.e. click in the radio button that says 'Data range"). If you want to analyse a rectangular subset you then need to specify the minimum and maximum values for the x and y coordinates (the units are nodes starting at (1,1) at the bottom left hand corner). When you click OK the data will be output to the Viewer Window (as shown left). Wether you get the statsistics for all data sets in the appropriate file or just those for the window you are examining depends on what mode is set in the Options menu in the Viewer Window.

Viewing>

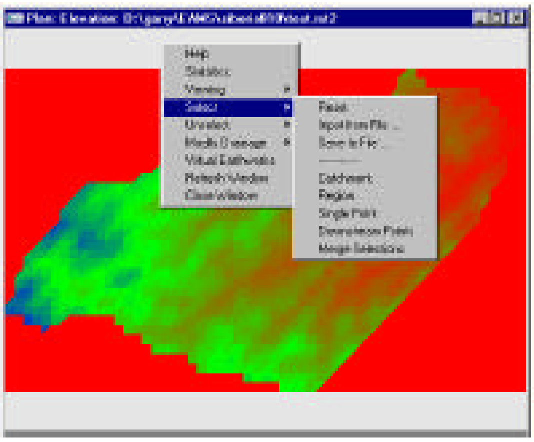


Content Region with Viewing popup menu visible

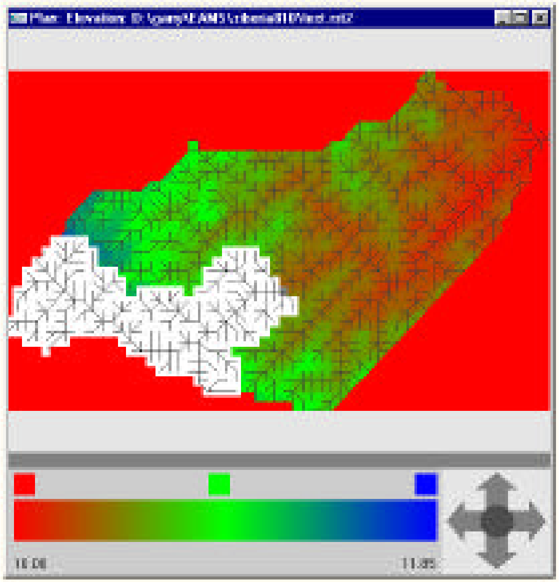
This menu item allows you input exact values for the position of the data within the window. This menu item performs the same role as the manipulator but more precisely.

For each of the menu items selected a dialog will brought up in which you input the required data. The default data in these dialogs is the current information for the region.

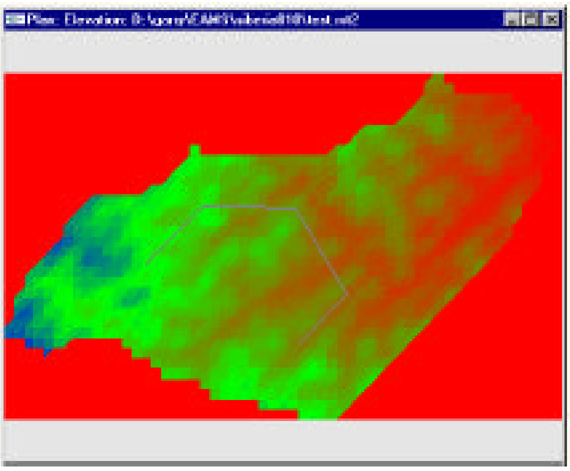
Select>



Content Region with Select popup menu visible



After Select>Catchment (note that the grey square has been identified as the catchment outlet because of the use of Modify Drainage)



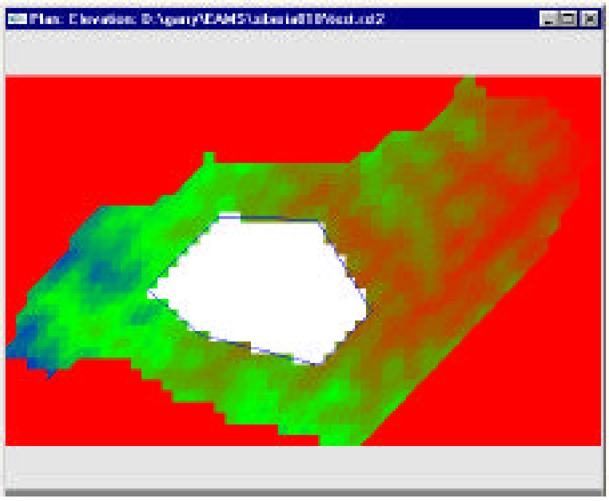
Rubber banding during Select>Region

This menu item allows you select a portion (called a ‘region’) of the domain. This region can then be output to a file so that it may be sued by SIBERIA to specify regions within the domain that have different runoff or erosion properties.

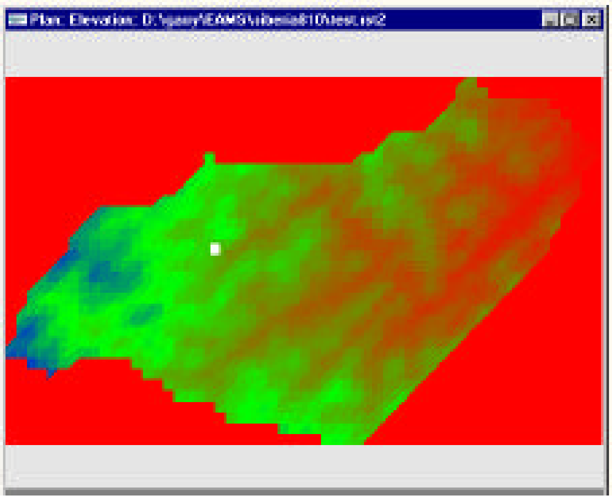
There are 5 options for selecting a region. They are

(1) *Catchment*: In this option you can select an entire catchment from catchment divide all the way down to the catchment outlet. Simply click somewhere inside the catchment you are interested in. It is generally useful to have Overlay>Network selected to make it easier to see which catchment you are selecting (see the pulldown menu options in the Viewer window). You can also select just part of catchment by using the Modify Drainage menu item (see below) to break the drainage paths at your desired point and then selecting the catchment either upstream or downstream of that point. After the operation the catchment area selected will be highlighted in white.

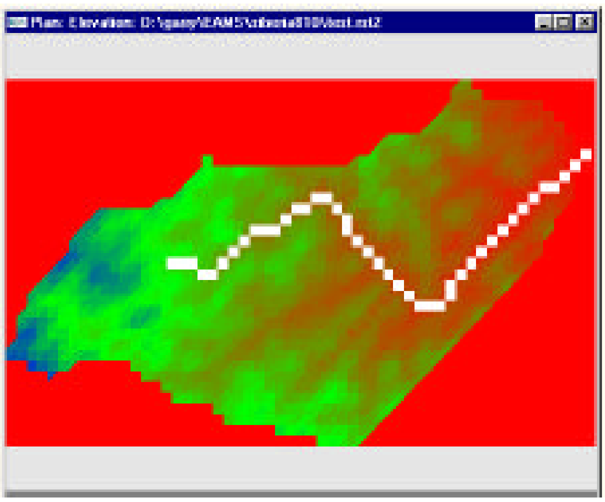
(2) *Region*: In this option you can select an irregular region by using the mouse. Click at the first point in the region and then click at each corner of the region (a grey line will be rubber banded while you are moving the mouse around). To finish the selection f region double click and the region will be closed with the last line being added to meet up with the



The final result from Select>Region



The result of the selection of by Select>Point



The result of a selection by Select>Downstream Points

first point. At this point a white irregular region will be overlayed showing those nodes that have been selected inside this region. The original region input will also be shown.

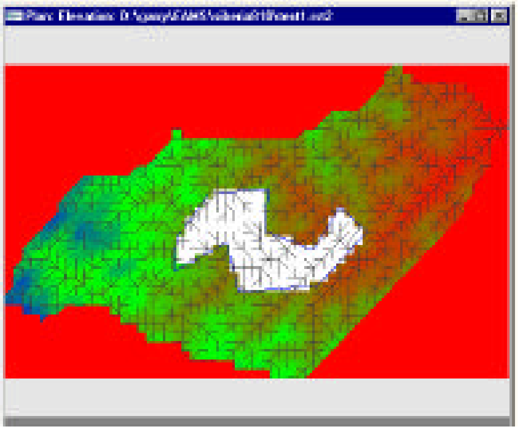
(3) *Single Point*: This option allows you select a single point. Simply click at or next to the point you wish to select. You might find it useful for this option to have Overlay>Grid or Overlay>Nodes selected when you do this to allow you to better identify the point you want to select.

(4) *Downstream Points*: This allows you to select a drainage path downstream of a selected point. In this case click on the node at the upstream end of the drainage path and all points downstream of this point will be selected. You might find it useful for this option to have Overlay>Network selected when you do this to allow you to better identify the point you want to select

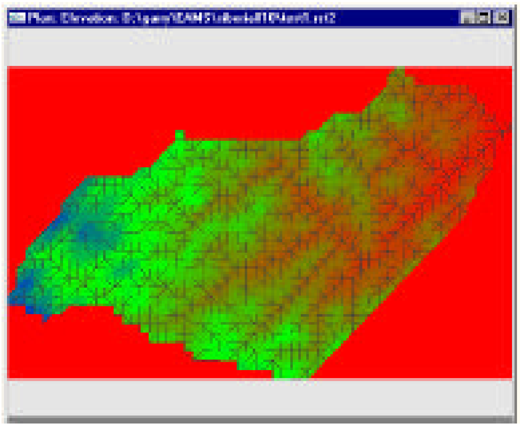
(5) *Merge Selections*: When you read in region file the selected region from the region file is kept separate from any region that you might already have selected (this allows us to undo anything that might already be done). This is why a file that is read in is shown as grey instead of white. There are times when you want to create a region that is merger of two separate region files or of a region file and a

selection on the screen. In this case if you merge selections then all the current selected areas are merged together so that when you output to a region file everything is output. Otherwise only those areas that are white are output. After Select>Merge Selections all selected area will be white.

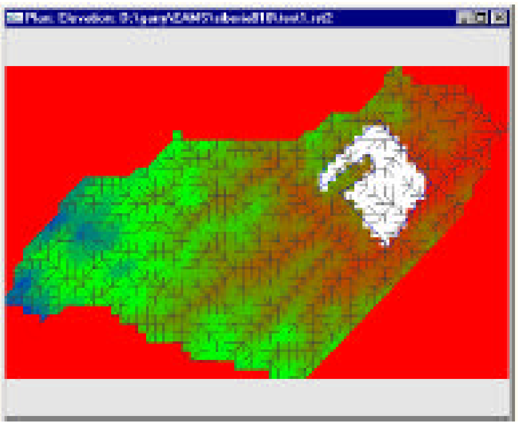
The figure below exemplifies the process. In step 1 a region has been selected with the mouse. In between Step1 and 2 this selection has output to a region file (called



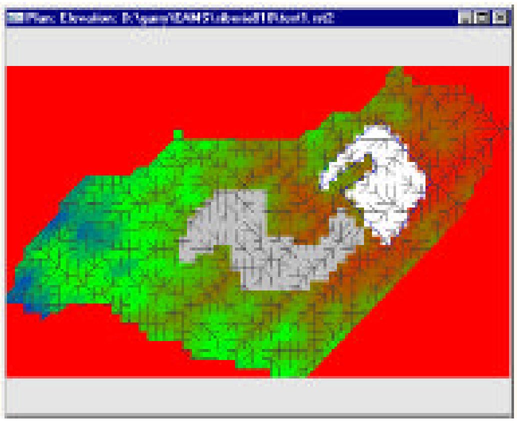
1. After making Selection 1



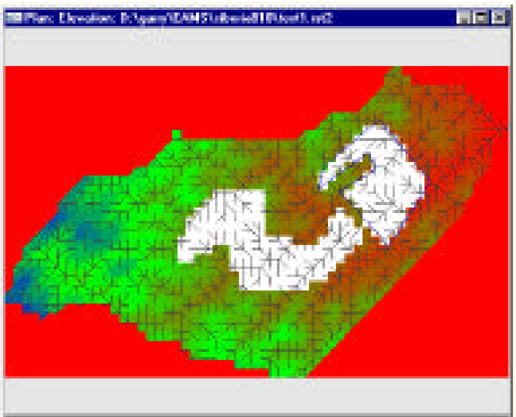
2. After saving Selection 1 and clearing the selection using Select>Reset



3. After making Selection 2



4. After reading in region file created at Step 2. Selection 2 is in white while Selection 1, read from the region file, is in grey



5. After merging selections.

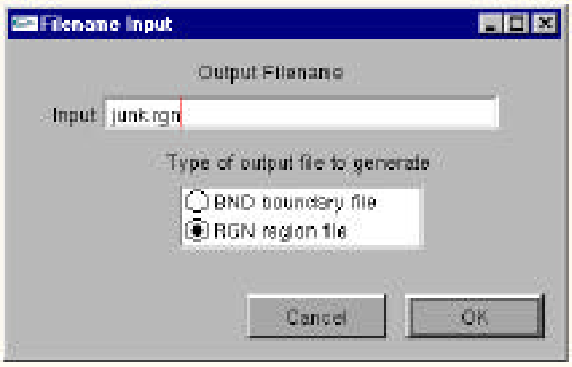
'selection1.rgn', region files are expected to have the extension '.rgn') and then the active selection has been cleared with Select>Reset. In Step 3 another selection is made, called Selection 2. In Step 4 Selection is read in from the region file created between steps 1 and 2. The active selection (i.e. the one that is used by Viewer if it does operations on selections) is in white, while the one read in from the file is grey. If read a second file in then it would also be grey, as well as the original file. In Step 5 these selection have been merged, indicated by the whole region being white. Now if Viewer does something based on the region it used the two white areas whereas in Step 4 it would only use the right-most white area and not use the grey area. For instance, if you output a region file after Step 5 it will use the two areas in white, whereas in Step it would only output the right-hand area.

In addition to creating selected areas there are a number of menu options in Select. They are

Reset: This erases any active selections

Input from File: This allows you to input a previously saved selection

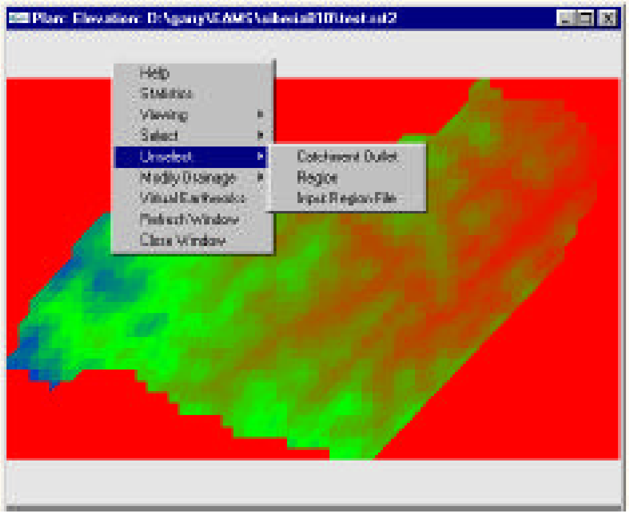
Save to File: This saves the current active selection, typically in a region file. A region file is almost identical to a SIBERIA boundary file with one exception. A SIBERIA boundary file must have a catchment outlet(s) selected whereas this is not necessary for a region file. When you ask to output a selection you Are presented with a dialog as to the left. You need to select what type of file to output (the name must end in .bnd for a SIBERIA boundary file and .rgn for a region file).



Save file dialog for a Selection

The default directory for the save is the Program Files directory on the system disk. If this is not where you want it then you will need to specify the full file path name.

UnSelect>



Content Region with Unselect Popup menu visible

This menu item is the exact reverse of the Select menu item. This allows you to unselect or exclude points from an already selected region. Note that you do not have to do the unselection from the same window that you did the original selection in. This allows then to select based on one or more characteristics (e.g. elevation) and then unselect parts of that already selected region on the basis of some other characteristics of the data (e.g. slope)

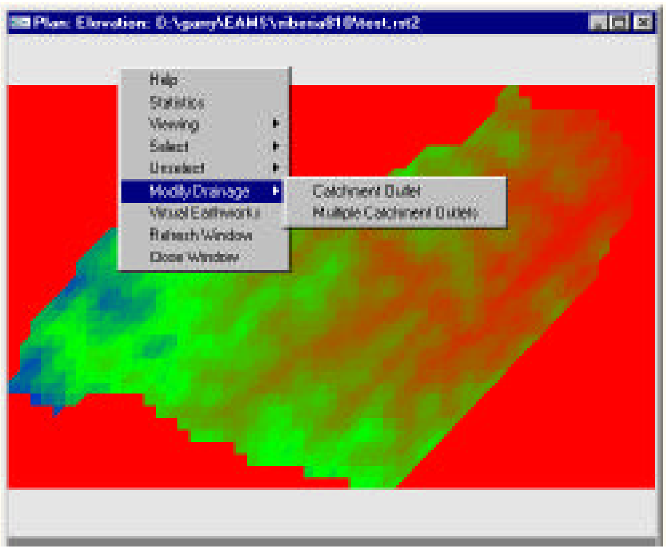
There are three options on this menu item.

(1) *Catchment Outlet*: This allows us to unselect an entire catchment by using the mouse to identify the outlet of the catchment you want to unselect. Simply click on or next to a node and the catchment upstream of that point will be unselected.

(2) *Region*: In the same way that you selected a region you mouse around a region and that region will be unselected.

(3) *Input Region File*: Read in an already output region file (presumably from a previous selection operation) and the region in the file will be unselected.

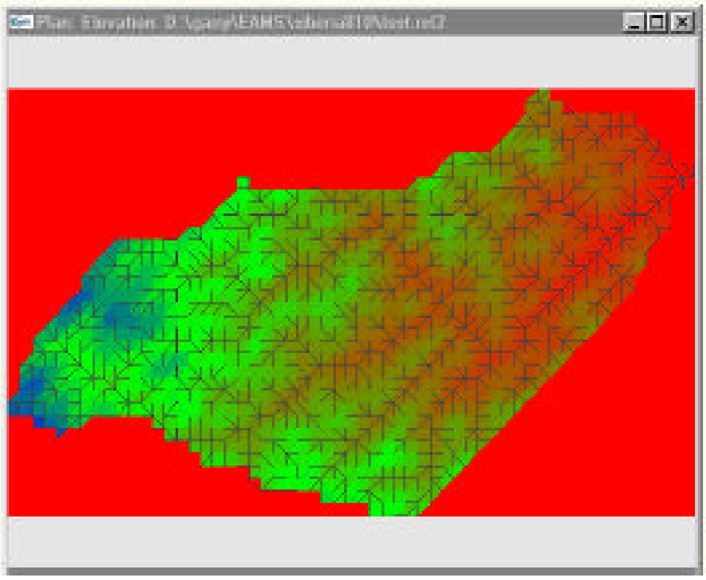
Modify Drainage>



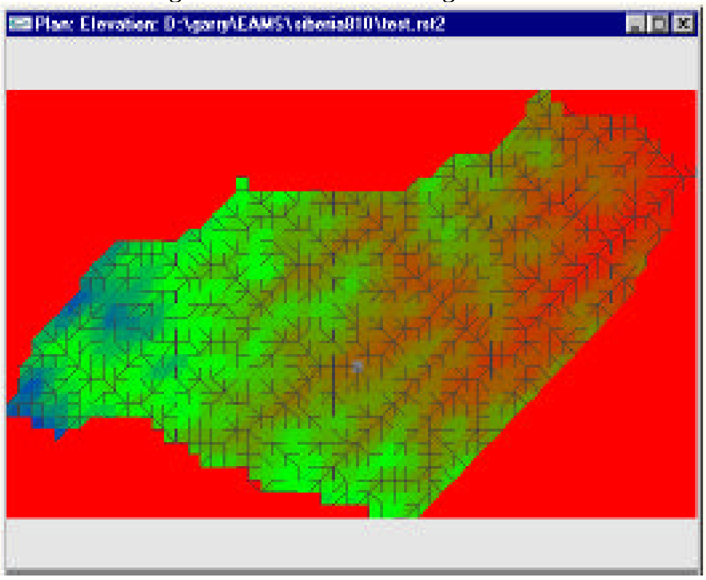
Plan Window and the Modify Drainage popup menu

This option allows you to modify the drainage pattern of the landform. Essentially you use the mouse to break the drainage paths so that one node no longer drains to another node. This is most useful when you are selecting a region using the catchment option. By breaking the drainage path you are then able to select a catchment (see the ‘Select>Catchment’ menu option) above a given point in the landform instead having to select the entire catchment all way down the edge of the boundary of the landform. Note that in both cases the drainage directions of the landform are permanently modified inside Viewer but the original file that was input is not affected (i.e. you can always read it into Viewer again) There are two options on this menu item.

(1) *Catchment Outlet*: This option allows you to break the drainage at a single node in the domain. Simply click at or near the node in question and the node will have its drainage path broken. Its useful when doing this to have Overlay>Network (from pull down menus in the Viewer Window) turned on as this will immediately confirm that you have selected the right point because the blue line linking that node with the one downstream will disappear. This indicated that the selected point no longer drains into the node that used to be downstream.

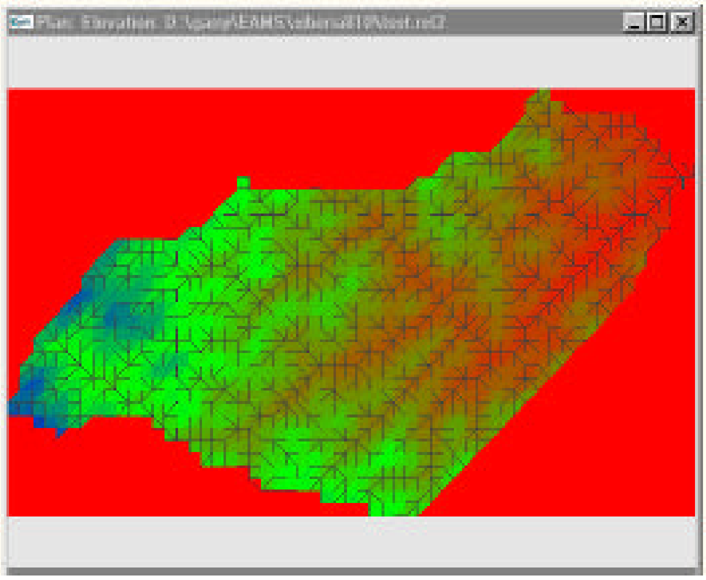


Drainage network before Drainage Modification

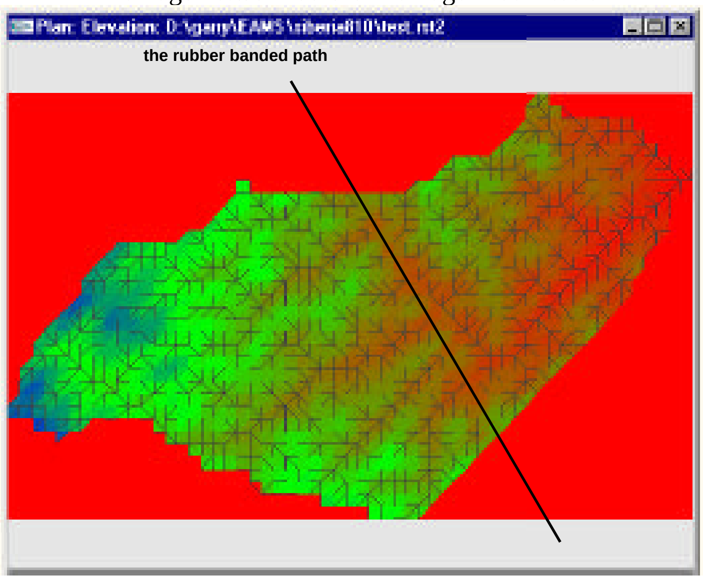


Drainage Network after Drainage Modification of a single point identified by the small grey square in the centre of picture. Note that the drainage line down stream of the grey square has dissappeared.

(2) *Multiple Catchment Outlet*: In this case you rubberband a line (i.e. press the mouse down at the start of the line and hold the mouse down while you mouse to where you want the end of the line to be, then release the mouse). All drainage paths that cross that line will be broken. As with the *Catchment Outlet* option above it is useful to have Overlay>Network turned on when doing this. This option is a useful way to break a large number of drainage paths simultaneously as, for instance, when you want to select all catchments above some feature in the landform. This is useful if you want to select all nodes upstream of some feature (e.g. the edge of the rehabilitated landform) but to one side of a drainage divide because all you then to do is then select each of the catchments upstream of the breaks. You could do this with Select>Region but it can often be difficult to follow the catchment divide accurately even with Overlay>Network turned on.



Drainage network before Drainage Modification



Drainage Network after Multiple Drainage Modification (the rubber banded line was from top to bottom in the middle of the picture). Note that the drainage lines in the centre of picture have dissappeared.

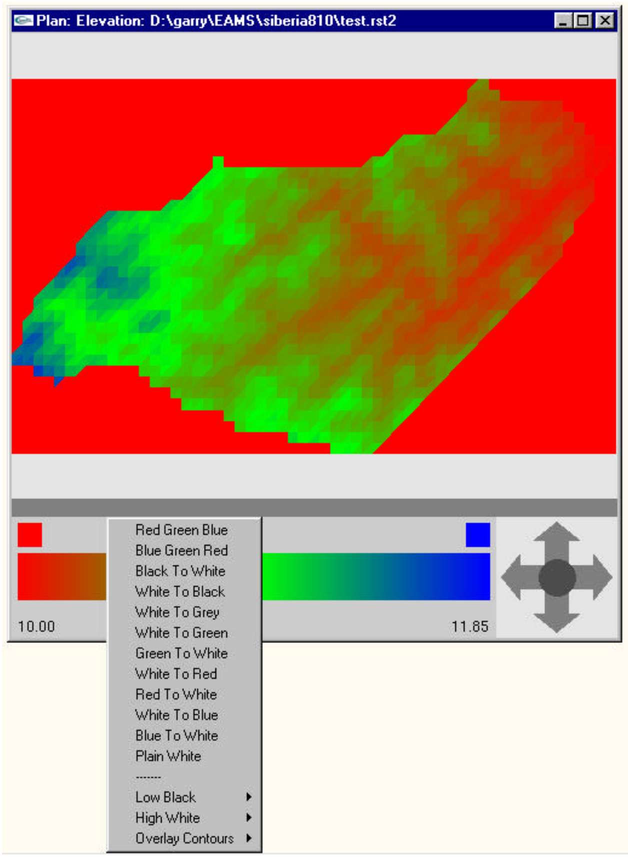
Virtual Earthworks>

Virtual Earthworks are a set of simple tools to manipulate the shape of the landform. It is most useful for rehabilitation design. This part of the Viewer package is document in Section 6. Once triggered it operates relatively independently of the Viewer software.

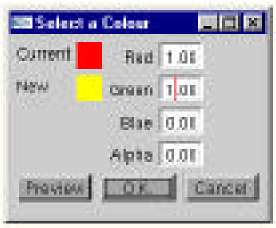
Refresh Window>

This menu item exists for when you set Manual Refresh in the Viewer Window pulldown menus. When you have lots of windows open in Viewer the automatic refreshing may become quite distracting and sometimes (depending on your graphics hardware) quite slow. In those it is suggested you turn off Automatic Refreshing and use the Refresh Window option.

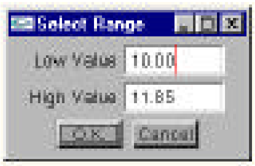
5.2.5 Plan Window Components: Colour Bar



Colour Bar Popup menu.



The Colour Selection Dialog.



The Range Selection Dialog.

The colour bar controls the colours used for visualisation, contouring and the range used to clip the display of the data (i.e. data outside the range is clipped to the display range). Like the other regions of the window the colour bar has a popup menu that is accessed by use of the right mouse button (see left). The colour bar is straightforward to use.

To change the range of the data to be displayed click on one of the two numbers values underneath the colour bar. You will prompted with a dialog to change one or other of these minimum and maximum value.

To change the colours for the top, mid point or bottom colour click in the appropriate box above the colour bar. Again you will prompted with a dialog to change those values. Alternatively a range of colour schemes are provided in the popup menu.

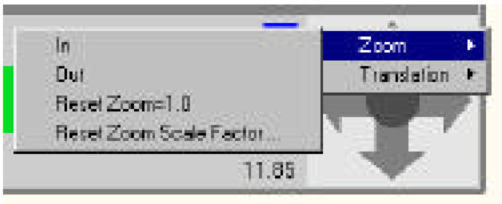
To display all values below the minimum value as black select the menu item Low Black>On. To display all values above the maximum value as white select the menu item High White>On.

Finally to draw pseudo-contours on top of whatever colour scheme you have adopted select one of the options under Overlay Contours. Sinusoidal provides a number of cycles of smooth lightening and darkening from the bottom of the data range to the top. Positive Ramp is a gradual decrease in lightness then a sudden increase followed by another gradual decrease, etc . Negative Ramp is the opposite of Positive Ramp.

5.2.6 Plan Window Components: Manipulator

The manipulator is the small set of arrows in the bottom right hand corner of the window. It provides control of the position of the drawing within the window.

The Translation popup menu



The Zoom popup menu

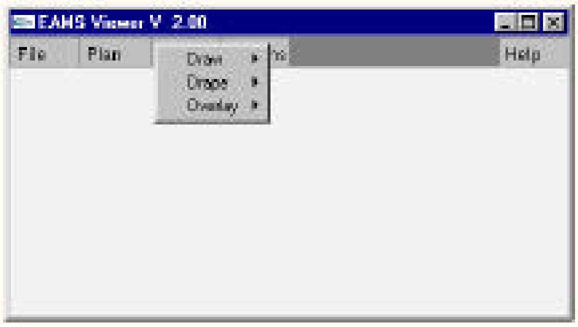
To move the picture click on the left arrow, right the right arrow, etc. The increment that is moved each time can be modified in the popup menu in the manipulator region (using the right mouse button) by setting a number for the Translation Scale factor

To zoom in click in the circle in the centre of the arrows. The zoom out hold down the control(ctrl) key while clicking in the circle. Again the increment that is zoomed each time you click can be controlled with the popup menu.

The popup menus also allows you to return to the original view before any manipulation.

Finally the same effect as clicking in the arrow keys can be obtained by placing the mouse in the content portion of the window and clicking on the left, right, up and down cursor buttons on the keyboard. The '+' key zooms in and '-' zooms out. Finally if you set the numeric keypad on the buttons '4', '8', '6', and '2' can be used to move the picture around (note the similarity to a cross of the arrangement of these four keys).

5.3 SURFACE Menu

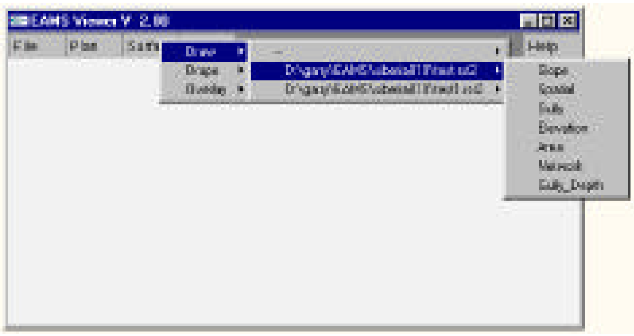


Viewer Window and the Surface pulldown menu

This menu controls the display of a plan of a data set a plan window. The plan window is the main window in which data analysis is done and is the work house of Viewer.

There are three menu items on this menu and they are discussed below.

5.3.1 Draw

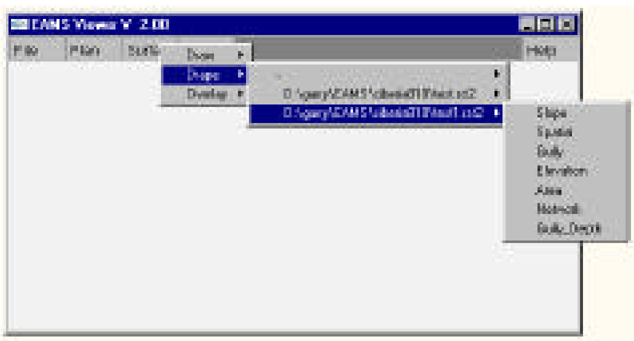


Viewer Window and the Surface>Draw menu.

This menu item allows you to select the data set that you wish to display on the screen. Initially when Viewer is started there are no submenus to this menu item but each time a file is opened its data is added to the menu item as shown at the left.

To select a dataset to be displayed simply select the data set in this drop down menu. After selection of this item the surface window will be drawn on the screen using defaults for the size of the window and the data range. The appearance of this window can then be manipulated in the surface window using the popup menu options(see below).

5.3.2 Drape

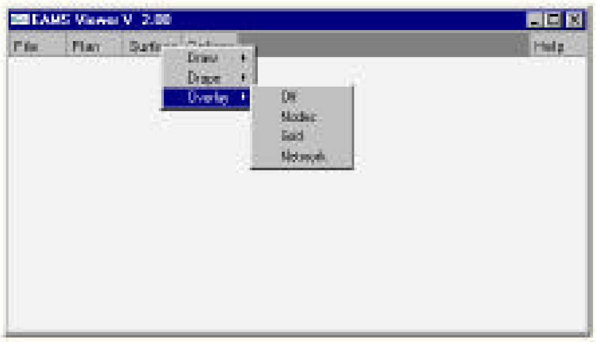


Viewer Window and the Surface>Drape menu.

This menu item allows you to select the data set that you wish to display on the screen. This date set is then draped over the elevations of the landform. Initially when Viewer is started there are no submenus to this menu item but each time a file is opened its data is added to the menu item as shown at the left.

To select a dataset to be displayed simply select the data set in this drop down menu. After selection of this item the surface window will be drawn on the screen using defaults for the size of the window and the data range. The appearance of this window can then be manipulated in the surface window using the popup menu options (see below).

5.3.3 Overlay



Viewer Window and the Surface>Overlay menu.

The overlay menu item allows you to overlay various pieces of information on top of all existing plan windows. This overlay is always positioned on top of whatever changes occur to the plan picture of the data. When the overlay option is selected it is applied to ALL plan windows.

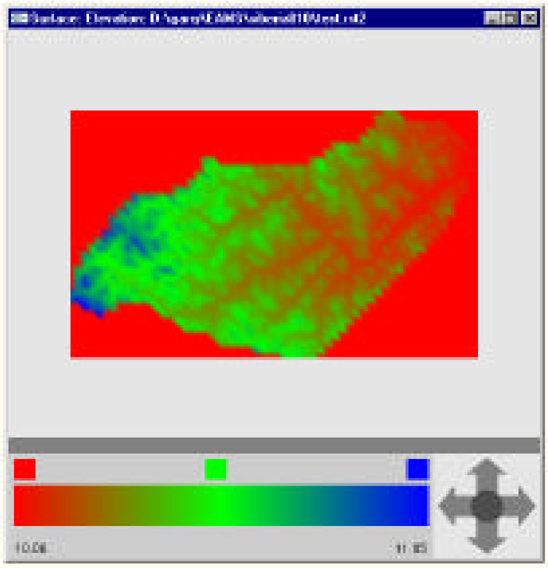
The data than can be overlayed are

(1) the position of each data point in the domain

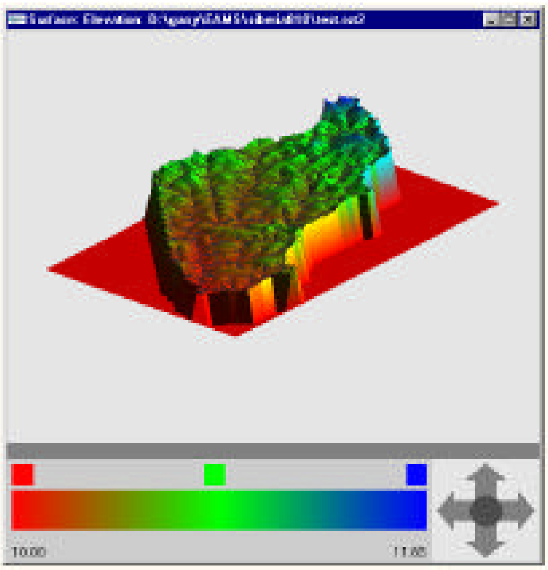
(2) the grid that the data is displayed on

(3) the drainage network for the domain.

5.3.4 Surface Window Components: Content Portion



The default view when the SURFACE window opens



The surface after it is rotated to provide a good viewing angle and lighting is turned on.

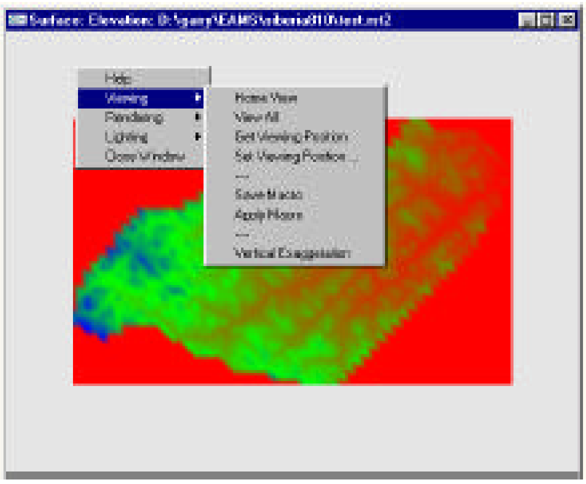
The content region of the window is that region in which the data is displayed. There is a pop-up menu in this region (accessed using the right mouse button) that provides tools for data manipulation. The main purpose of the Surface window is not data analysis but data visualisation. For data analysis you should use the PLAN window.

Initially by default the view of the landform is shown from above and looks very similar to the plan window. However, by rotating the landform (see the Manipulator section below) and turning on the lighting the three-dimensional viewing of this landform becomes clear (see left).

There are a number of popup menu options from the content part of the window and we explain them now.

To be able to set repeatable display characteristics from window to window there is an easy way to save the light position. See the discussion of the Viewing>Save Macro popup menu item for this capability.

Viewing>



The Viewing popup menu

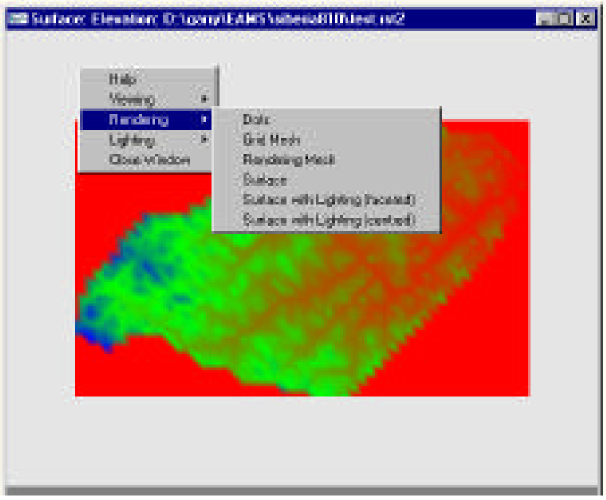
This menu controls how the data are displayed.

The first four options ('Home View', 'View All', Get Viewing Position', and 'Set Viewing Position'), control the orientation of the data. Less precise, but easier to use, orientation of the data is also available through the manipulator portion of the window (see below).

The macro options allow the saving of the current format of the picture so it may used again at a later data, or for another surface window. When 'Save Macro' is selected the current setting for the window are saved to a file. These include (1) colour scheme, (2) vertical exaggeration, (3) orientation of the data, (4) lighting and rendering options. When 'Apply macro' is selected the saved settings are applied to the current surface window. Only one set of settings may be saved at any one time.

The 'Vertical Exaggeration' setting allows you to control how vertically exaggerated the surface is. Typically if no exaggeration is used the surface will look flat because the vertical dimension is typically much less than the horizontal dimensions. Viewer initially selects a reasonable exaggeration based on the data ranges and use will need to change those with this menu item.

Rendering>



Rendering popup menu.

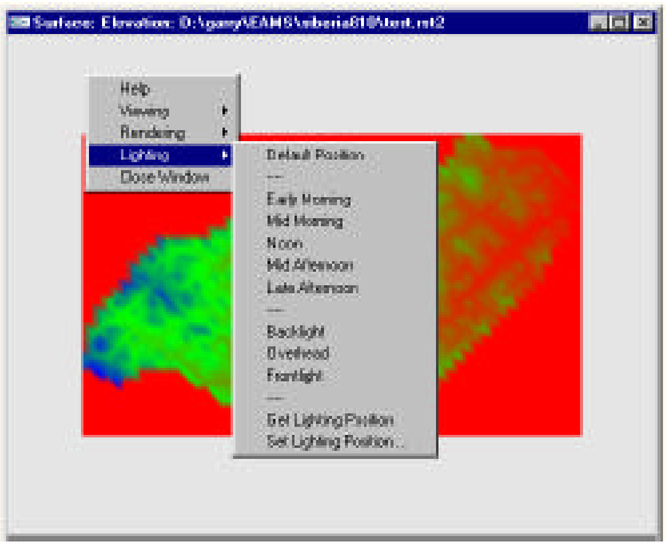
Rendering controls how the surface is displayed. There are a variety of options.

The two Rendering options require lighting. A default position is set by Viewer initially and the position of this light can then manipulated by the Lighting popup menu.

The other rendering options do not use lighting so the Lighting options have no effect on these.

To be able to set repeatable rendering from window to window there is an easy way to save the light position. See the discussion of the Viewing>Save Macro popup menu item for this capability.

Lighting>



The Surface lighting popup menu

The lighting options are not active until one of Rendering options that require lighting are selected. By default when the surface window is displayed lighting is not activated.

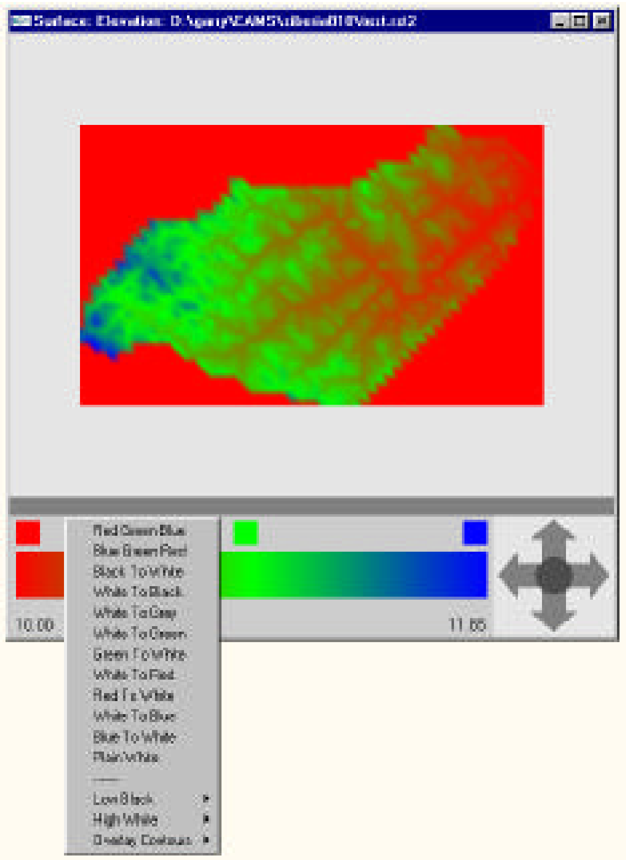
In the default position (see left) the left edge is the western edge, the right edge the eastern edge, etc. The lighting positions based on the time of day are consistent with this orientation.

The backlight/frontlight options correspond roughly to winter position of the sun for the Northern and Southern Hemispheres.

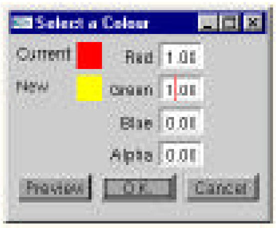
The ‘Get Lighting Position’ option puts up a dialog with the exact coordinates of the sun, while the ‘Set Lighting Position …” option allows the input of the exact lighting position.

To be able to set repeatable lighting from window to window, however, there is an easy way to save the light position. See the discussion of the Viewing>Save Macro popup menu item for this capability.

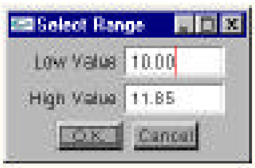
5.3.5 Surface Window Components: Colour Bar



Colour Bar Popup menu.



The Colour Selection Dialog.



The Range Selection Dialog.

The colour bar controls the colours used for visualisation, contouring and the range used to clip the display of the data (i.e. data outside the range is clipped to the display range). Like the other regions of the window the colour bar has a popup menu that is accessed by use of the right mouse button (see left). The colour bar is straightforward to use.

To change the range of the data to be displayed click on one of the two numbers values underneath the colour bar. You will prompted with a dialog to change one or other of these minimum and maximum value.

To change the colours for the top, mid point or bottom colour click in the appropriate box above the colour bar. Again you will prompted with a dialog to change those values. Alternatively a range of colour schemes are provided in the popup menu.

To display all values below the minimum value as black select the menu item Low Black>On. To display all values above the maximum value as white select the menu item High White>On.

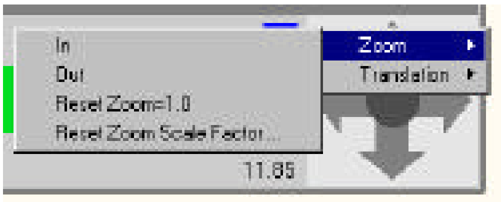
Finally to draw pseudo-contours on top of whatever colour scheme you have adopted select one of the options under Overlay Contours. Sinusoidal provides a number of cycles of smooth lightening and darkening from the bottom of the data range to the top. Positive Ramp is a gradual decrease in lightness then a sudden increase followed by another gradual decrease, etc . Negative Ramp is the opposite of Positive Ramp.

5.3.6 Surface Window Components: Manipulator

The manipulator is the small set of arrows in the bottom right hand corner of the window. It provides control of the position of the drawing within the window.

The Translation popup menu

To move the picture click on the left arrow, right the right arrow, etc. To rotate the picture hold the control key while you press on the arrows. The increment that is moved each time can be modified in the popup menu in the manipulator region (using the right mouse button) by setting a number for the Translation Scale factor



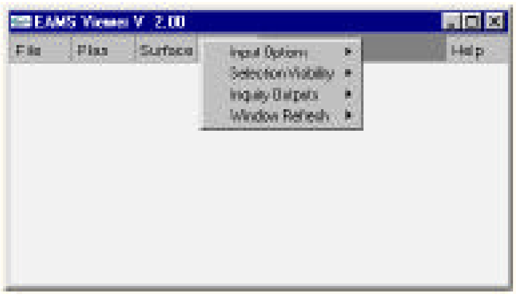
The Zoom popup menu

To zoom in click in the circle in the centre of the arrows. The zoom out hold down the control(ctrl) key while clicking in the circle. Again the increment that is zoomed each time you click can be controlled with the popup menu.

The popup menus also allows you to return to the original view before any manipulation.

Finally by placing the mouse in the content portion of the window and clicking on the left, right, up and down cursor buttons on the keyboard the surface can be rotated. The '+' key zooms in and '-' zooms out. Finally if you set the numeric keypad on the buttons '4', '8', '6', and '2' can be used to rotate the picture around (note the similarity to a cross of the arrangement of these four keys). In addition, the keys '7' and '9' can be used to tilt the picture.

5.4 OPTIONS Menu



The options menu allows control over various default ways that Viewer operates.

5.4.1 Input Options

This controls what happens when you input the data files. You may swap the data north-south, or east-west. This option is ONLY active when you input a file so if you input and then change this option all previously input files are unaffected but all subsequent file are.

5.4.2 Selection Visibility

When yopu make selection in a PLAN window this option determines what other windows this selection appears. By default only the window in which you do the selection displays the selection but can also (1) have it displayed in all other PLAN windows from the same file, and (2) have it displayed in any corresponding SURFACE windows as well.

5.4.3 Inquiry outputs

When you inquire a point this option determines whether you only get the data from that window in which you inquire or whether you get data for data sets from the corresponding file.

5.4.4 Window Refresh

Be default all windows refresh automatically when they need to be updated (when the data in them changes, when that window is brought to the front for some reason, etc). If you have a large number of windows open this behaviour may become quite tiresome. In this case you can set the windows to manual refresh and use the popup menu options in the PLAN and SURFACE display windows to update the windows manually when you need to.

6 Virtual Earthworks

7 References

Bell L C, Loch R J, Haneman D, and Willgoose G R. 1993. “A post-mining landform research program for open-cut mines”. *Australian Minerals Industry Council Environmental Workshop*. Melbourne, Australian Institute of Mining and Metallurgy.

Bras R L and Rodriguez-Iturbe I. 1985. *Random Functions and Hydrology*, Addison-Wesley, New York.

Davis W M. 1954. *Geographical Essays*, Dover, New York.

Dunne T. 1989. “Hydrology, mechanics and geomorphic implications of erosion by subsurface flow”, *Groundwater Geomorphology*, Higgins, (Ed) Geological Society of America,

Evans K G. 1998. *Runoff and erosion characteristics of a post-mining rehabilitated landform at Ranger Uranium Mine, Northern Territory, Australia and the implications for topographic evolution*. Ph.D. thesis, The University of Newcastle.

Gilbert G. 1909. “The convexity of hillslopes”, *Journal of Geology*, 17, pp. 344-350.

Gyasi-Agyei Y, and Willgoose G R. 1997. *Calibration of SIBERIA parameters for 15 mine sites using runoff and erosion data obtained from the QDPI rainfall simulator: Derivation of the Version 1 Database*, Research Report 146.05.1997. Department of Civil, Surveying and Environmental Engineering, The University of Newcastle, Callaghan, NSW.

Hancock G R. 1997. *Experimental testing of the SIBERIA landscape evolution model*. Ph.D. thesis, The University of Newcastle.

Horton R E. 1945. “Erosional development of steams and their drainage basins; hydrophysical approach to quantitative morphology”, *Bulletin of the Geological Society of America*, 56, pp. 275-370.

Huggett R J. 1988. “Dissipative systems: Implications for geomorphology”, *Earth Surface Processes and Landforms*, 13, pp. 45 - 49.

Kirkby M J. 1971. “Hillslope process-response models based on the continuity equation”, in *Slopes: form and process*, Institute of British Geographers Special Publications, 3, London, Institute of British Geographers, pp. 15 - 30.

Kirkby M J. 1987. “Modelling some influences of soil erosion, landslides and valley gradient on drainage density and hollow development”, Ahnert F, *Geomorphological Models: Theoretical and Empirical Aspects*, Aachen, Germany, 1986.

Knisel W. 1980. *CREAMS: A field-scale model for chemicals, runoff, and erosion from Agricultural Management Systems*, Conservation Research Report No. 26, US Department of Agriculture, Washington, USA

Loewenherz-Lawrence D S. Hydrodynamic description for advective sediment transport processes and rill initiation. *Water Resources Research*. 1994; 30(11), 3203-3212.

Meinhardt H A. 1982. *Models of biological pattern formation*, Academic Press, Berlin.

Moore I D, and Burch G J. 1986. Sediment transport capacity of sheet and rill flow : Application of unit stream power theory. *Water Resources Research*, 22(8), pp 1350-1360.

Shreve R L. 1966. “Statistical law of stream numbers”, *Journal of Geology*, 74, pp. 17-37.

Smith T R and Bretherton F P. 1972. “Stability and the conservation of mass in drainage basin evolution”, *Water Resources Research*, 8(6), pp. 1506 - 1529.

Strahler A N. 1964. “Quantitative geomorphology of drainage basins and channel networks”, *Handbook of applied hydrology*, Chow V T, (Ed.) New York, McGraw Hill, pp. 4-39 - 4-76.

Tarboton D G. 1997. “A new method for the determination of flow directions and upslope areas in grid digital elevation models”. *Water Resources Research*, 33(2), pp 309-19.

Tucker G and Bras R L. 1999.

Valdes J B, Fiallo Y and Rodriguez-Iturbe I. 1979. “A rainfall-runoff analysis of the geomorphologic IUH”, *Water Resources Research*, 15(6), pp. 1421-1434.

Willgoose G R. 1993. “Hydrology and erosion”, *Proceedings of the Symposium on the Management and Rehabilitation of Waste Rock Dumps*, Darwin, 7-8 October, 1993.

Willgoose G R. 1994a. “A physical explanation for an observed area-slope-elevation relationship for declining catchments”, *Water Resources Research*, 30(2), pp. 151-159.

Willgoose G R. 1994b. “A statistic for testing the elevation characteristics of landscape simulation models”, *Journal of Geophysical Research*, 99(B7), pp. 13987-13996.

Willgoose G R, and Kuczera G A. 1995. “Estimation of sub-grid scale kinematic wave parameters for hillslopes”. *Hydrological Processes*, 9(3/4), 469-82.

Willgoose G R and Riley S. J. 1993. “The assessment of the long-term erosional stability of engineered structures of a proposed mine rehabilitation”, Chowdhury R N and Sivakumar M (Ed), *Environmental Management: Geowater and Engineering Aspects*, Wollongong University, 8-11 February, 1993.

Willgoose G R and Riley S J. 1994. “Long-term erosional stability of mine spoils”, *Australian mining looks north - The challenges and choices*, Australian Institute of Mining and Metallurgy National Conference, Darwin, 5-9 August, 1994.

Willgoose G R, and Riley S J. 1998a. An assessment of the long-term erosional stability of a proposed mine rehabilitation. *Earth Surface Processes and Landforms*, 23, 237-59.

Willgoose G R, and Riley S J. 1998b. *Application of a catchment evolution model to the prediction of long term erosion on the spoil heap at Ranger Uranium Mines: Initial analysis*. 107p . Supervising Scientist Report 132. Canberra, Australian Government Publishing Service.

Willgoose G R, Bras R L and Rodriguez-Iturbe I. 1989. *A physically based channel network and catchment evolution model*, TR 322, Ralph M. Parsons Laboratory, Dept. of Civil Engineering, MIT, Boston, MA.

Willgoose G R, Bras R L and Rodriguez-Iturbe I. 1991a. "A physically based coupled network growth and hillslope evolution model: 1 Theory", *Water Resources Research*, 27(7), pp. 1671-1684.

Willgoose G R, Bras R L and Rodriguez-Iturbe I. 1991b. "A physically based coupled network growth and hillslope evolution model: 2 Applications", *Water Resources Research*, 27(7), pp. 1685-1696.

Willgoose G R, Bras R L and Rodriguez-Iturbe I. 1991c. "A physical explanation of an observed link area-slope relationship", *Water Resources Research*, 27(7), pp. 1697-1702.

Willgoose G R, Bras R L and Rodriguez-Iturbe I. 1991d. "Results from a new model of river basin evolution", *Earth Surface Processes and Landforms*, 16, pp. 237-254.

Willgoose G R, Bras R L and Rodriguez-Iturbe I. 1994. "Hydrogeomorphology modelling with a physically based river basin evolution model", *Process Models and Theoretical Geomorphology*, Kirkby M J (Ed), Chichester, Wiley, pp. 271-294.

APPENDICES

Appendix A: Revision History

Version 1 (1998-2002)

Version 1 was developed for mine site rehabilitation applications as part of the Queensland Coal Association "Post-Mining Landscapes" project. It involved the development of an interface to a series of pre-existing codes as well as the development of a database of parameters for erosion prediction based on field data collected in this project. This version of the software was not widely distributed beyond research colleagues. It was also restricted to running on Windows 95 and 98 because of limitations in the graphics software used to develop the GUI interface.

Version 2 (2002-date)

A major upgrade for EAMS. This involved rewriting the code to use a Fortran 90 interface to the OpenGL graphics and other support libraries. This allowed the support of Windows 2000, XP and ME.

- 2.00: Implementation of F90 interface and updating to Windows 2000/XP/ME.
- 2.01: Maintenance upgrade. Numerous small bug fixes in Moscow. Viewer unchanged.