ARCVIEW INTERFACE FOR SWAT2000

USER'S GUIDE

M. DI LUZIO, R. SRINIVASAN, J.G. ARNOLD, S.L. NEITSCH

2002

BLACKLAND RESEARCH & EXTENSION CENTER
TEXAS AGRICULTURAL EXPERIMENT STATION
720 EAST BLACKLAND ROAD ○ TEMPLE, TEXAS 76502

GRASSLAND, SOIL AND WATER RESEARCH LABORATORY
USDA AGRICULTURAL RESEARCH SERVICE
808 EAST BLACKLAND ROAD ○ TEMPLE, TEXAS 76502

Blackland Research and Extension Center, Temple, Texas
BRC Report 02-07

Grassland, Soil & Water Research Laboratory, Temple, Texas
GSWRL Report 02-03

Published 2002 by Texas Water Resources Institute, College Station, Texas
TWRI Report TR-193

CONTENTS

Section 1: Introduction 1

Section 2: Installing the ArcView Interface 6

Section 3: Preparing ArcView Input 10
- **3.1: Required ArcView Map Themes** 10
- **3.2: Optional ArcView Map Themes** 12
- **3.3: ArcView Tables and Text Files** 13

Section 4: Start ArcView Interface 36
- **4.1: Main Interface Commands** 38
  - 4.1.1: New Project 38
  - 4.1.2: Open Project 39
  - 4.1.3: Copy/Import Project 40
  - 4.1.4: Delete Project 41
  - 4.1.5: Exit ArcView 42
  - 4.1.6: Edit SWAT Databases 42
  - 4.1.7: Save Current Project 44
  - 4.1.8: Environmental Variables 44
  - 4.1.9: Return to Current Project 46
- **4.2: Project Views** 47
  - 4.2.1: The Avswat Menu 48
  - 4.2.2: The Input Menu 49
  - 4.2.3: The Edit Input Menu 51
  - 4.2.4: The Simulation Menu 52
  - 4.2.5: The Reports Menu 53
- **4.3: Pop-Up Menu** 55

Section 5: Watershed Delineation 61
- **5.1: Watershed Delineation Dialog Box** 62
- **5.2: DEM Setup** 63
- **5.3: Stream Definition** 78
- **5.4: Outlet and Inlet Definition** 79
- **5.5: Main Watershed Outlet Selection and Definition** 86
- **5.6: Reservoirs—Optional** 92

Section 6: Land Use/Soil Characterization 95
- **6.1: Land Use/Soil Definition and Overlay** 95
  - 6.1.1: Get Started 96
  - 6.1.2: Land Use Data Layer 97
  - 6.1.3: Soil Data Layer 108

6.1.4: Overlay of Land Use and Soil Maps 119
6.2: HRU Distribution 120

Section 7: Import Weather Data 128

Section 8: Creation of Input 135
8.1: Write All 136
8.2: Individual Write Commands 141

Section 9: Input Modification—Point Sources 146

Section 10: Input Modification—Inlet Discharges 154

Section 11: Input Modification—Reservoirs 162

Section 12: Input Modification—Subbasins 171
12.1: Edit Soil Physical Data (.sol) 173
12.2: Edit Weather Generator Input Data (.wgn) 179
12.3: Edit General Subbasin Input Data (.sub) 183
12.4: Edit General HRU Input Data (.hru) 186
12.5: Edit Main Channel Input Data (.rte) 190
12.6: Edit Groundwater Input Data (.gw) 193
12.7: Edit Consumptive Water Use Input Data (.wus) 197
12.8: Edit Management Input Data (.mgt) 200
12.9: Edit Soil Chemical Input Data (.chm) 217
12.10: Edit Pond/Wetland Input Data (.pnd) 223
12.11: Edit Stream Water Quality Input Data (.swq) 227

Section 13: SWAT Output 231
13.1: Run SWAT 231
13.2: Read Results 246
13.3: Calibration Tool 248
13.4: Create Delivered Load Table 254
13.5: Reports: Map-Chart 257

Section 14: SWAT Databases 261
14.1: User Soils Database 263
14.2: User Weather Stations Database 270
14.3: Land Cover/Plant Growth Database 278
14.4: Fertilizer Database 287
14.5: Pesticide Database 293
14.6: Tillage Database 299
14.7: Urban Database 304

Section 15: The Example Data Set 313
15.1: Create SWAT Run with Example Dataset 314
15.1.1: Processing the Elevation Map Grid 319
15.1.2: Processing the Land Use/Soil Map Grids 329
15.1.3: Land Use/Soil Distribution 335
15.1.4: Weather Stations 336
15.1.5: Create ArcView Databases & SWAT Input Files 338
15.1.6: Run SWAT 339
15.1.7: View Results 340

Appendix 1: Watershed Delineation Output Data 343

Appendix 2: State FIPS Codes 345

SECTION 1: INTRODUCTION

Purpose

AVSWAT-2000 (version 1.0) (Di Luzio *et al.*, 2002) is an ArcView extension and a graphical user interface for the **SWAT (Soil and Water Assessment Tool)** model (Arnold *et al.*, 1998). **SWAT** is a river basin, or watershed, scale model developed to predict the impact of land management practices on water, sediment, and agricultural chemical yields in large, complex watersheds with varying soils, land use, and management conditions over long periods of time. The model is physically based and computationally efficient, uses readily available inputs, and enables users to study long-term impacts. For a detailed description of **SWAT**, see *Soil and Water Assessment Tool Theoretical Documentation* and *User's Manual, Version 2000* (Neitsch *et al.*, 2001a; 2001b), published by the Agricultural Research Service and the Texas Agricultural Experiment Station, in Temple, Texas.

The SWAT model can be applied to support various watershed and water quality modeling studies. Examples of such studies include the following:

- National and regional scale water resource assessment considering both current and projected management conditions.
- Bosque River TMDL in Erath County, Texas. The project determined sediment, nitrogen, and phosphorus loadings to Lake Waco from various sources including dairy waste application areas, waste treatment plants, urban areas, conventional row crops, and rangeland. Numerous land management practices were simulated and analyzed (Saleh *et al.*, 2000).
- Poteau River TMDL in Oklahoma/Arkansas. This project determined sediment, nitrogen, and phosphorus loadings in Wister Lake and dissolved oxygen, temperature, algae, and CBOD in the river. Management scenarios regarding poultry waste were analyzed (Srinivasan *et al.*, 2000).
- DDT in the Yakima River basin, Washington. SWAT was used to simulate past and future sediment contamination by DDT in the Yakima River basin.
- The EPA office of pesticide registration is evaluating SWAT for use in landscape/watershed scale evaluation for pesticide registration.
- SWAT is being used extensively in the U.S. and Europe to assess the impact of global climate on water supply and quality (Rosenberg *et al*, 1999).

The AVSWAT-2000 ArcView extension evolved from AVSWAT, an ArcView extension developed for an earlier version of SWAT (Di Luzio *et al.*, 1998). Important functional components and the analytical capability of ArcView GIS are implemented in several sets of customized and user friendly tools designed to: (1) generate specific parameters from user-specified GIS coverages; (2) create SWAT input data files; (3) establish agricultural management scenarios; (4) control and calibrate SWAT simulations; (5) extract and organize SWAT model output data for charting and display. The most relevant components of the system are: (1) a complete and advanced watershed delineator, (2) a tool for the definition of the Hydrologic Response Units, and (3) the latest version of the SWAT model with a relative interface. AVSWAT software is developed as an extension of ArcView GIS for the Personal Computer (PC) environment.

Within this system (Figure 1.1) ArcView provides both the GIS computation engine and a common Windows-based user interface. AVSWAT is organized in a sequence of several linked tools grouped in the following eight modules: (1) Watershed Delineation; (2) HRU Definition; (3) Definition of the Weather Stations; (4) AVSWAT Databases; (5) Input Parameterization, Editing and Scenario Management; (6) Model Execution; (7) Read and Map-Chart Results; (8) Calibration tool. Once AVSWAT is loaded, the modules get embedded into ArcView, and the tools are accessed through pull-down menus and other controls, which are introduced in various ArcView graphical user interfaces (or GUIs) and custom dialogs. The basic map inputs required for the AVSWAT include digital elevation, soil maps, land use/cover, hydrography (stream lines), and climate. In addition, the interface requires the designation of land use, soil, weather, groundwater, water use, management, soil chemistry, pond, and stream water quality data, as well as the simulation period, to ensure a successful simulation.

This tool is being applied worldwide and can support water quality analysis at the watershed level as well as at single stream segments, scales required for the support of most Total Maximum Daily Load (TMDL) programs.

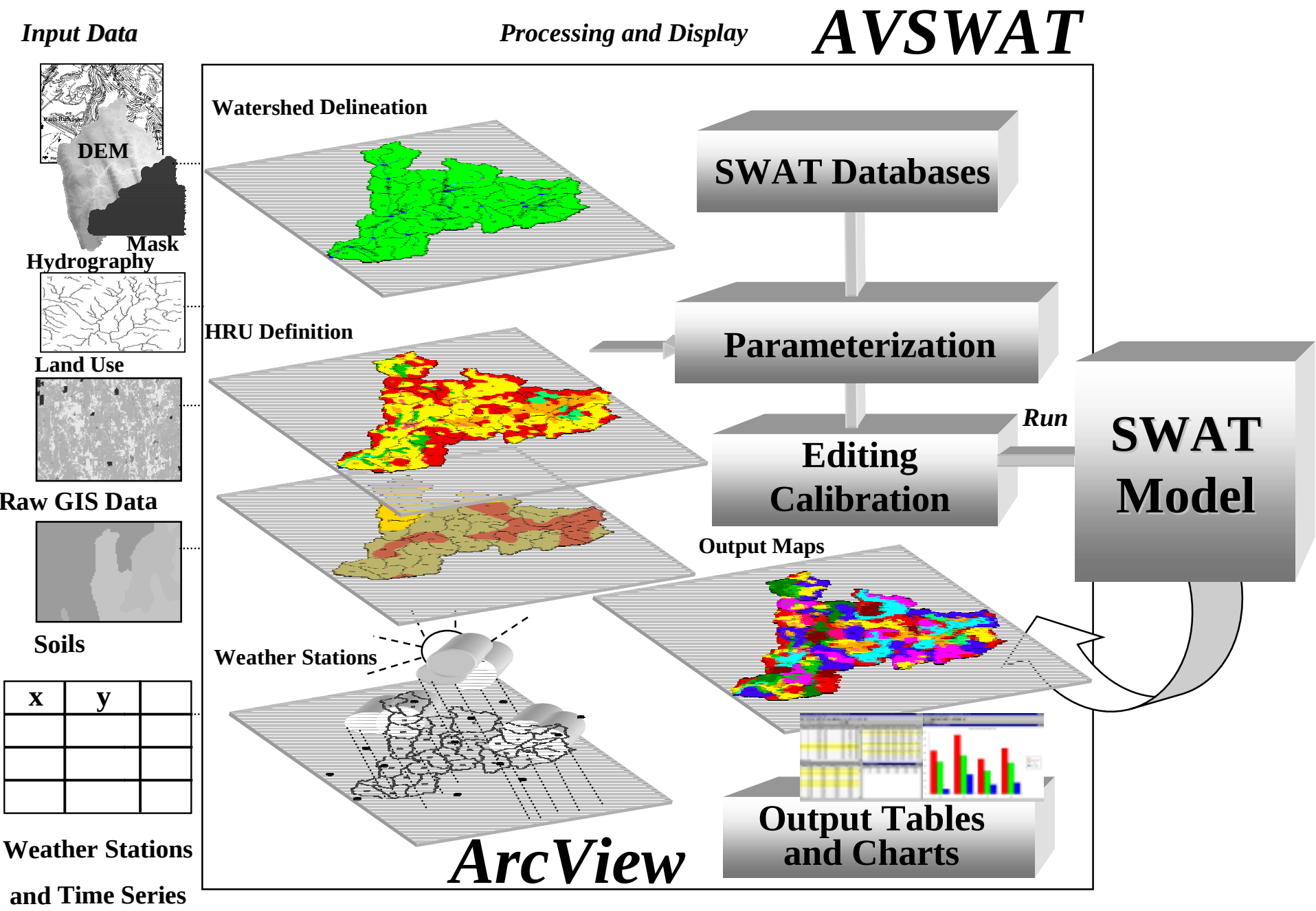

Input Data
Processing and Display
AVSWAT
DEM
Mask
Hydrography
Land Use
Raw GIS Data
Soils
x
y
Weather Stations
and Time Series
Watershed Delineation
HRU Definition
Weather Stations
ArcView
SWAT Databases
Parameterization
Editing
Calibration
Run
SWAT
Model
Output Maps
Output Tables
and Charts

Figure 1.1

Application

SWAT can be used to simulate a single watershed or a system of multiple hydrologically connected watersheds. Each watershed is first divided into subbasins and then into hydrologic response units (HRUs) based on the land use and soil distributions.

Procedures

Key Procedures

- Load or select the AVSWAT2000 extension
- Delineate the watershed and define the HRUs
- (Optional) Edit SWAT databases
- Define the weather data
- Apply the default input files writer
- (Optional) Edit the default input files
- Set up (requires specification of simulation period, PET calculation method, etc.) and run SWAT
- (Optional) Apply a calibration tool
- (Optional) Analyze, plot and graph SWAT output

References

Arnold, J.G., R. Srinivasan, R.S. Muttiah, and J.R. Williams. 1998. Large area hydrologic modeling and assessment part I: model development. J. American Water Resources Association 34(1):73-89.

Di Luzio, M., R. Srinivasan, and J.G. Arnold. 2002. A GIS-Hydrological Model System for the Watershed Control of Agricultural Nonpoint and Point Sources of Pollution. Submitted to Transaction of the ASAE.

Di Luzio, M., R. Srinivasan, and J.G. Arnold. 1998. Watershed oriented non-point pollution assessment tool. In: Computers in Agriculture - 7th international conference. October 26-30th 1998, Orlando, FL. ASAE. 233-241.

Neitsch, S.L., J.G. Arnold, J.R. Kiniry and J.R. Williams. 2001a. Soil and Water Assessment Tool Theoretical Documentation, Version 2000.

Neitsch, S.L., J.G. Arnold, J.R. Kiniry and J.R. Williams. 2001a. Soil and Water Assessment Tool User's Manual, Version 2000.

Rosenberg, N.J., D.L. Epstein, D. Wang, L. Vail, R. Srinivasan, and J.G. Arnold. 1999. Possible impacts of global warming on the hydrology of the Ogallala aquifer region. J. of Climate 42:677-692.

Saleh, A., J.G. Arnold, P.W. Gassman, L.W. Hauck, W.D. Rosenthal, J.R. Williams, and A.M.S. McFarland. 2000. Application of SWAT for the upper north Bosque watershed. Transactions of the ASAE 43(5):1077-1087.

Srinivasan, R., J.G. Arnold, T.S. Ramanarayanan, and S.T. Bednarz. 2000. Modeling Wister lake watershed with the soil and water assessment tool (SWAT) J. American Water Resources Association (in review)

SECTION 2: INSTALLING THE ARCVIEW INTERFACE

System Requirements

The SWAT2000/Arcview Interface requires:

- Personal computer using a Pentium I processor or higher, which runs at 166 megahertz or faster
- 64 megabytes RAM minimum
- Microsoft Windows 95, 98, NT 4.0 or Win2000 operating system with most recent kernel patch*
- VGA graphics adapter and monitor. *The interface works best when the resolution is set to 800 x 600 or 1024 x 768 pixels, the color palette is set to 8-bit (256 colors) or 16-bit (32768 colors) and the display font size is set to* ***small fonts****.*
- 50 megabytes free memory on the hard drive for minimal installation and up to 300 megabytes for a full installation
- ArcView 3.1 or 3.2 (software)
- Spatial Analyst 1.1 or later (software)
- Dialog Developer 3.1 or later (software)

While 50 MB is adequate memory for installing the interface, you may need considerably more memory to store the tables generated when the interface processes the map layers.† We have found that a 2 gigabyte hard drive works very well for storing ArcView, the SWAT/ArcView interface, and project maps and tables.

* Microsoft constantly updates the different versions of windows. This interface was developed with the latest version of Windows and may not run with earlier versions. Patches are available from Microsoft.

† The space required to create a SWAT project with the SWAT/ArcView interface depends on the resolution of the maps used. While testing the interface, a 10-meter resolution DEM map layer taking up only 6 MB of space was processed. At one point in the analysis of the map, the interface had filled 350 MB of storage with data.

To install the interface

The ArcView interface for SWAT2000 has been formatted to create a separate directory structure from that used by the ArcView interface for SWAT99.2. This allows users to keep both versions installed on their machine.

1. If you have not done so, turn on your computer. Download the archive file and extract ArcView interface installation files.
2. Start ArcView.
3. On the **File** menu, click **Open Project**.
4. Move to the directory containing the installation files.
5. Choose **setup.apr** and click **OK**.
6. The installation program for the SWAT2000 ArcView interface will be activated.
7. Soils data for Texas, Pennsylvania, and Wisconsin is automatically installed with the interface. If you want to use the soils data for another state, you will need to install the additional data separately.

   To install the data:

   a. In the directory drive:/*Installation dir*/AvSwatDB/AllUs/statsgo, create a directory for each state of interest using the 2-letter alpha code for the state (e.g. TX for Texas, MS for Mississippi, etc.). Appendix 2 lists numeric and alpha codes for all states.

   b. From the CD or the web site, copy the zip file containing the soils data for the state to the directory created in step a.

   c. Unzip the soils data using WinZip or pkunzip.

If you have installed the SWAT2000/ArcView interface on your machine and wish to upgrade the interface to a new version, please perform the following steps prior to installing the new version: 1) Uninstall the old version of the interface. 2) Make sure the *Installation dir* directory and all subdirectories are removed from your machine. 3) Delete :/ESRI/Av_gis30/ArcView/Ext32/avswat.avx.

The directory structure created by the interface is displayed in Figure 2.1.

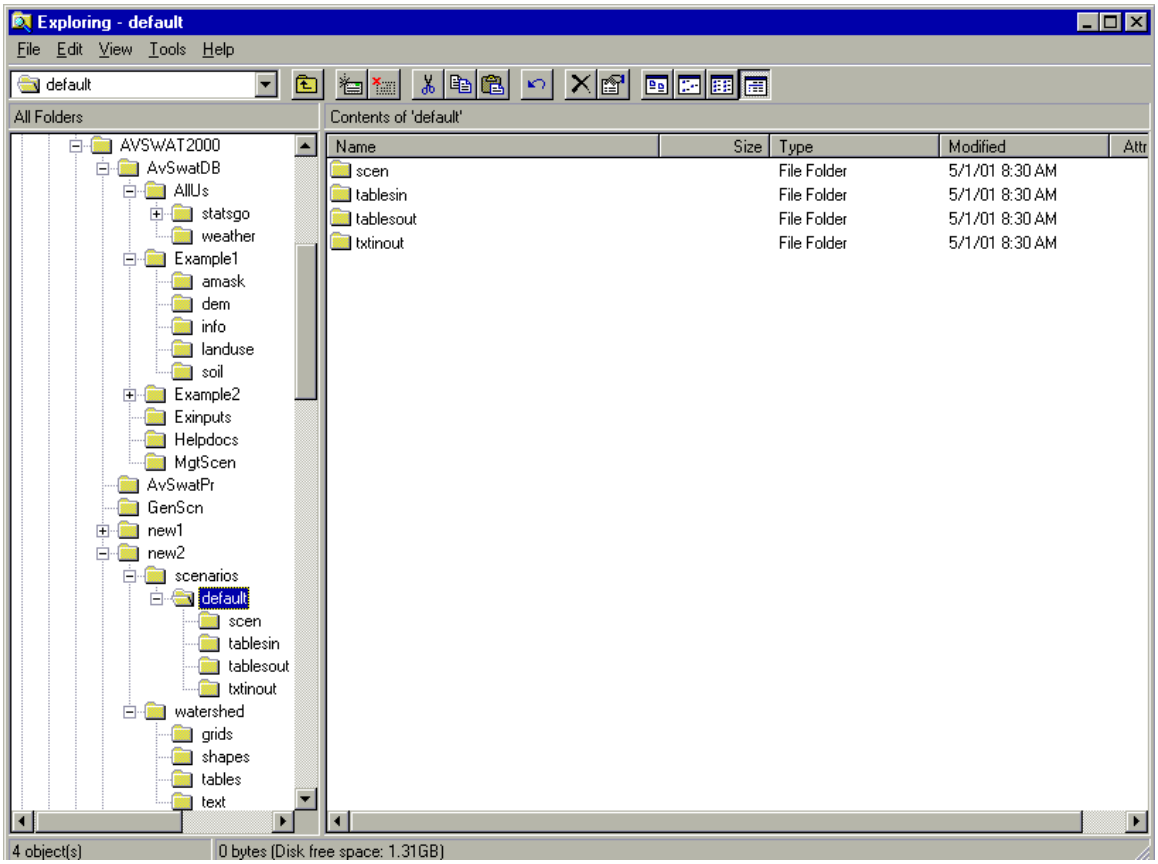


Figure 2.1

The *Installation dir* directory is created during installation and initially contains 3 subdirectories: AvSwatDB, AvSwatPr, and GenScn. When projects are created by the user, the interface will by default make a directory for each project in the *Installation dir* directory (new1 and new2 are project directories in Figure 2.1). However, the user can also specify a different location for the project directory if desired.

The AvSwatDB directory contains all the databases. The default U.S. soils data is stored in AvSwatDB/AllUS/statsgo while the default U.S. weather generator database is stored in AvSwatDB/AllUS/weather. The AvSwatDB directory also contains the example data sets, help documents, and management scenarios created for use in more than one project.

The AvSwatPr directory contains the swat2000 executable and the dll used to calculate the number of total potential heat units for a plant in an HRU.

The GenScn directory contains documentation and installation files for GenScn, an output analyzer. This output processor currently doesn't work properly.

The project directory contains all files generated during the course of creating and running the project. Two subdirectories are created within the project directory: scenarios and watershed. The watershed directory contains all ArcView maps, tables, and reports generated during the project. The scenarios directory contains the default subdirectory. Within the default directory, four subdirectories are created: scen (stores scenario runs created with the Calibration tool in the interface); tablesin (stores all .dbf tables that contain SWAT input variables); tablesout (stores .dbf tables created when SWAT output is loaded in the interface); txtinout (contains ASCII files read and written by the SWAT2000 executable).

SECTION 3: PREPARING ARCVIEW INPUT

To create a SWAT dataset, the interface will need to access ArcView map themes and database files, which provide certain types of information about the watershed. The necessary maps and database files need to be prepared prior to running the interface. Examples of each of the different types of maps and tables can be viewed in the demonstration dataset.

SECTION 3.1: REQUIRED ARCVIEW MAP THEMES

- ArcInfo-ArcView GRID—Digital Elevation Model (DEM)

  The interface allows the DEM to use integer or real numbers for elevation values. The units used to define the map resolution and the elevation are not required to be identical. For example, the map resolution may be in meters while the elevation may be in feet.

  The map resolution must be defined in one of the following units: meters, kilometers, feet, yards, miles, decimal degrees.

  The elevation must be defined in one of the following units: meters, centimeters, yards, feet, inches.

- ArcInfo-ArcView GRID or Shape—Land Cover/Land Use

  The categories specified in the land cover/land use map will need to be reclassified into SWAT land cover/plant types. The user has three options for reclassifying the categories.

  The first option is to use USGS category codes when creating the map (or use a USGS land use/land cover map). The interface contains an ArcView table that identifies the different SWAT land cover/plant types used to model the various USGS land uses.

  The second option is to select the SWAT land cover/plant type or urban code for each category when the land cover/land use map theme is loaded in the interface.

  The third option is to create a look up table that identifies the 4-letter SWAT code for the different categories of land cover/land use on the map. The format of the look up table is described in Section 3.3.

- ArcInfo-ArcView GRID or Shape—Soil

  The categories specified in the soil map will need to be linked to the soil database (U.S. soils data only) included with the interface or to the User Soils database, a custom soil database designed to hold data for soils not included with the U.S. soil database. The user has four options for linking

the map to the U.S. soil database.

One method is to use the STATSGO polygon (MUID) number. Because the soils database contains information for the entire U.S., the 3-digit state STATSGO number must be prefixed with the 2-digit numeric code for the state. (The 2-digit numeric codes are listed in Section 8.) For every polygon, the soil database contains data for all soil phases found within the polygon.

When the "Stmuid" option is chosen, data for the dominant soil phase in the polygon is used for the map category. The "Stmuid + Seqn" option allows the user to specify the polygon number and the soil sequence number. This allows the user to choose a soil other than the dominant within the polygon. For example, if Seqn is set to 3, data for the third most common soil phase will be used to represent the map category. The "Name + Stmuid" option allows the user to specify a soil series within the STATSGO polygon by name. The interface will use data for the dominant phase of the soil series to represent the map category.

The user may also link the soils map to the database via Soils5ID number. When the "S5id" option is chosen, data for the specified soil series is used to represent the map category. In order to use the "S5id" option, the soil database for the entire US must be installed.

The final option, "Name", is chosen when soils data from the User Soils database are to be utilized. The user will import SWAT soil files (.sol) or type the soil data into the User Soils database for each of the map categories prior to creating the project. The "Name" specified for each of the map categories is the name of the soil in the User Soils database.

To reclassify the map categories, the information may be manually entered within the interface. Alternatively, a look up table may be loaded which has this information listed. Section 3.3 summarizes the format of the look up table used to specify the soils information.

The map themes may be created in any projection (the same projection must be used for all maps). The user will identify the type of projection and the projection settings within the interface when the working DEM is loaded.

Note: All of the input maps listed above and below must be located in the computer disk. None should be located in the local network.

SECTION 3.2: OPTIONAL ARCVIEW MAP THEMES

- ArcInfo-ArcView GRID or Shape or Draw Manually—DEM Mask

  The interface allows a mask to be superimposed on the DEM. The interface differentiates the mask grid into areas classified as category 0 (no data) and areas classified as any category > 0. Areas of the DEM grid for which the Mask grid has a value of 0 will not be processed for stream delineation.

- ArcInfo-ArcView Shape—Stream Delineation

  The interface allows a polyline shape file with the stream delineation to be superimposed on the DEM. The stream delineation shape file is needed for areas where the relief is so low the DEM map grid is unable to accurately predict the location of the streams.

SECTION 3.3: ARCVIEW TABLES AND TEXT FILES

❖ **Subbasin Outlet Location Table** (dBase Table)

The subbasin outlet location table is used to specify the location of: additional subbasin outlet locations (for example, stream gaging locations).

The use of a location table to import locations for subbasin outlets is recommended when the user plans to compare observed or measured data with SWAT results.

Table Format: Preferred (5 fields)

Field name	Field format	Definition
XPR	floating point	X coordinate in the defined projection
YPR	floating point	Y coordinate in the defined projection
LAT	floating point	Latitude in decimal degrees
LONG	floating point	Longitude in decimal degrees
TYPE	string 1 char	"O" should always be entered in this field

Only the subbasin outlets (Type "O") are allowed.

❖ **Watershed Inlet Location Table** (dBase Table)

The watershed inlet location table is used to specify the location of: point sources and drainage watershed inlets.

Table Format: Preferred (5 fields)

Field name	Field format	Definition
XPR	floating point	X coordinate in the defined projection
YPR	floating point	Y coordinate in the defined projection
LAT	floating point	Latitude in decimal degrees
LONG	floating point	Longitude in decimal degrees
TYPE	string 1 char	"D" for Point Source "I" Draining Watershed Inlet

Only the point source (Type "D") or draining watershed inlet (Type "I") are allowed.

❖ **Land Use Look Up Table** (dBase or ASCII)

The land use look up table is used to specify the SWAT land cover/plant code or SWAT urban land type code to be modeled for each category in the land use map grid. Because this information can be entered manually, this table is not required to run the interface.

This table may be formatted as a dBase table or as a comma delimited text table.

The first row of the land use look up table must contain the field names. The remaining rows will hold the required data. An example land use look up table can be found in the included dataset.

dBase Table Format (2 fields)

Field name[†]	**Field format**	**Definition**
VALUE	String	Number of map category
LANDUSE	string 4 chars	Corresponding SWAT land use or urban code

Note: Land use codes starting with a numerical character are not allowed.

ASCII (.txt) Table Format

An example land use look up file is:

```
"Value","Landuse"
1,RNGE
2,PAST
3,FRSD
4,WATR
5,AGRL
6,URBN
```

[†] Specific field names must be used in all tables for the interface to properly access the information.

❖ **Soil Look Up Table** (dBase or ASCII)

The soil look up table is used to specify the type of soil to be modeled for each category in the soil map grid. The format of the table will vary depending on the option chosen to link the soil data to the soil map. Because this information can be entered manually, this table is not required to run the interface.

The first row of the soil look up table must contain the field names. The remaining rows will hold the required data. An example soil look up table can be found in the included dataset.

dBase Table Format: **Stmuid** option (2 fields)

Field name	Field format	Definition
VALUE	string	Number of map category
STMUID	string 5 chars	5-digit number: digits 1-2: numeric code for state; digits 3-5:STATSGO polygon number

dBase Table Format: **S5id** option (2 fields)

Field name	Field format	Definition
VALUE	string	Number of map category
S5ID	string 6 chars	6-character alpha-numeric code for SOILS-5 data for the soil series

dBase Table Format: **Name** option (2 fields)

Field name	Field format	Definition
VALUE	string	Number of map category
NAME	string (30 chars max)	Name of the soil. The name entered into this field must correspond with the name of a soil in the User Soils database.

Note: Soil names starting with a numerical character are not allowed.

dBase Table Format: **Stmuid + Seqn** option (3 fields)

Field name	Field format	Definition
VALUE	string	Number of map category
STMUID	string 5 chars	5-digit number: digits 1-2: numeric code for state; digits 3-5:STATSGO polygon number
SEQN	string	Sequence number of soil within the STATSGO polygon. (2^{nd} most dominant soil, SEQN=2; 3^{rd} most dominant soil, SEQN=3, etc.)

dBase Table Format: **Stmuid + Name** option (3 fields)

Field name	Field format	Definition
VALUE	string	Number of map category
STMUID	string 5 chars	5-digit number: digits 1-2: numeric code for state; digits 3-5:STATSGO polygon number
NAME	string (30 chars max)	Name of soil within the STATSGO polygon

ASCII (.txt) Table Format

An example soil look up file for the Stmuid option is:

```
"Value","Stmuid"
1,48047
2,48236
3,48357
4,48619
5,48620
6,48633
```

ASCII look-up tables for other options will contain data for the joining attributes listed in the dBase format summaries for the different linkage options.

❖ Weather Generator Gage Location Table (dBase)

A database included with the interface has weather generator data summarized for 1,041 weather stations across the United States. If this database will not be used in the simulation, a location table is required to provide the location of custom weather generator stations. Custom weather generator data should be entered into the User Weather Stations database prior to creating the project (see Section 14).

dBase Table Format: Preferred (4 fields)

Field name	Field format	Definition
ID	integer	Gage identification number
NAME	string max 8 chars	Corresponding gage[‡] name string
XPR	floating point	X coordinate in the defined projection
YPR	floating point	Y coordinate in the defined projection

dBase Table Format: Alternative[*] (4 fields)

This format can be used only if the projection is defined in the project.

Field name	Field format	Definition
ID	integer	Gage identification number
NAME	string max 8 chars	Corresponding gage name string
LAT	floating point	Latitude in decimal degrees
LONG	floating point	Longitude in decimal degrees

dBase Table Format: Alternative[*] (6 fields)

This format must be used if the projection is ***not*** defined in the project.

Field name	Field format	Definition
ID	integer	Gage identification number
NAME	string max 8 chars	Corresponding gage name string
XPR	floating point	X coordinate in the defined projection
YPR	floating point	Y coordinate in the defined projection
LAT	floating point	Latitude in decimal degrees
LONG	floating point	Longitude in decimal degrees

The user will provide a record for each station to be used: the "Name" field will contain the string used as the station name in the User Weather Stations database.

[‡] The gage name must be the same as the gage name listed for the weather generator data in the User Weather Stations database.

[*] When using the alternative table format, the projection of the maps used in the interface must be defined in order for the interface to convert the latitude and longitude to the proper coordinate values.

[*] When using the alternative table format, the projection of the maps used in the interface must be defined in order for the interface to convert the latitude and longitude to the proper coordinate values.

Note: Station names starting with a numerical character are not allowed.

❖ Precipitation Gage Location Table (dBase)

When measured precipitation data are to be used, a table is required to provide the locations of the rain gages. The precipitation gage location table is used to specify the location of rain gages.

dBase Table Format: Preferred (5 fields)

Field name	Field format	Definition
ID	integer	Gage identification number (not used by interface)
NAME	string max 8 chars	Corresponding table[‡] name string
XPR	floating point	X coordinate in the defined projection
YPR	floating point	Y coordinate in the defined projection
ELEVATION	integer	Elevation of rain gage (m)

dBase Table Format: Alternative[*] (5 fields)

This format can be used only if the projection is defined in the project.

Field name	Field format	Definition
ID	integer	Gage identification number (not used by interface)
NAME	string max 8 chars	Corresponding table name string
LAT	floating point	Latitude in decimal degrees
LONG	floating point	Longitude in decimal degrees
ELEVATION	integer	Elevation of rain gage (m)

[‡] The name of the table which holds the precipitation data is "NAME".dbf or "NAME".txt.

[*] When using the alternative table format, the projection of the maps used in the interface must be defined in order for the interface to convert the latitude and longitude to the proper coordinate values.

dBase Table Format: Alternative* (7 fields)

This format must be used if the projection is ***not*** defined in the project.

Field name	Field format	Definition
ID	integer	Gage identification number (not used by interface)
NAME	string max 8 chars	Corresponding table name string
XPR	floating point	X coordinate in the defined projection
YPR	floating point	Y coordinate in the defined projection
LAT	floating point	Latitude in decimal degrees
LONG	floating point	Longitude in decimal degrees
ELEVATION	integer	Elevation of rain gage (m)

The user will provide a record for each station to be used: the "Name" field will contain the string used to name the linked precipitation data table.

Note: Station names starting with a numerical character are not allowed.

* When using the alternative table format, the projection of the maps used in the interface must be defined in order for the interface to convert the latitude and longitude to the proper coordinate values.

❖ **Precipitation Data Table** (dBase or ASCII)

The precipitation data table is used to store the daily precipitation for an individual rain gage. This table is required if the raingage option is chosen for rainfall in the weather data dialog box. There will be one precipitation data table for every location listed in the rain gage location table.

The name of the precipitation data table is "*name*.dbf" or "*name*.txt" where *name* is the character string entered for NAME in the rain gage location table.

This table may be formatted as a dBase table or as a comma delimited text table.

dBase (.dbf) Table Format: (2 fields)

Field name	**Field format**	**Definition**
DATE	date (yyyymmdd)	Day of precipitation
PCP	floating point (f5.1)	Amount of precipitation (mm)

Note: An example precipitation table is in *\Installation dir\avswatdb\exinputs\precip.dbf*

ASCII (.txt) Table Format:

Line	**Field format**	**Definition**
First	Yyyymmdd string	Starting day of precipitation
All other lines	Floating point (f5.1) string number	Amount of precipitation (mm)

The daily records must be listed in sequential order.

Note: An example precipitation table is in *\Installation dir\avswatdb\exinputs\precip.txt*

The precipitation data tables must be placed in the User Data directory (see Section 4.1.8).

❖ Temperature Gage Location Table (dBase)

When measured temperature data are to be used, a table is required to provide the locations of the temperature gages. The temperature gage location table is used to specify the location of temperature gages.

dBase Table Format: Preferred (5 fields)

Field name	Field format	Definition
ID	Integer	Gage identification number (not used by interface)
NAME	string max 8 chars	Corresponding table[‡] name string
XPR	floating point	X coordinate in the defined projection
YPR	floating point	Y coordinate in the defined projection
ELEVATION	Integer	Elevation of temperature gage (m)

dBase Table Format: Alternative[*] (5 fields)

This format can be used only if the projection is defined in the project.

Field name	Field format	Definition
ID	Integer	Gage identification number (not used by interface)
NAME	string max 8 chars	Corresponding table name string
LAT	floating point	Latitude in decimal degrees
LONG	floating point	Longitude in decimal degrees
ELEVATION	Integer	Elevation of temperature gage (m)

dBase Table Format: Alternative[*] (7 fields)

[‡] The name of the table which holds the temperature data is "NAME".dbf or "NAME".txt.

[*] When using the alternative table format, the projection of the maps used in the interface must be defined in order for the interface to convert the latitude and longitude to the proper coordinate values.

[*] When using the alternative table format, the projection of the maps used in the interface must be defined in order for the interface to convert the latitude and longitude to the proper coordinate values.

This format can be used only if the projection is defined in the project.

Field name	Field format	Definition
ID	Integer	Gage identification number (not used by interface)
NAME	string max 8 chars	Corresponding table name string
XPR	floating point	X coordinate in the defined projection
YPR	floating point	Y coordinate in the defined projection
LAT	floating point	Latitude in decimal degrees
LONG	floating point	Longitude in decimal degrees
ELEVATION	Integer	Elevation of temperature gage (m)

The user will provide a record for each station to be used: the "Name" field will contain the string used to name the linked temperature data table.

Note: Station names starting with a numerical character are not allowed.

❖ **Temperature Data Table** (dBase or ASCII)

The temperature data table is used to store the daily maximum and minimum temperatures for a weather station. This table is required if the climate station option is chosen for temperature in the weather data dialog box. There will be one temperature data table for every location listed in the climate station location table.

The name of the temperature data table is "*name*.dbf" or "*name*.txt" where *name* is the character string entered for NAME in the temperature gage location table.

This table may be formatted as a dBase table or as a comma delimited text table.

dBase Table Format: (3 fields)

Field name	**Field format**	**Definition**
DATE	date (yyyymmdd)	Day of measure
MAX	floating point (f5.1)	Daily maximum temperature (°C)
MIN	floating point (f5.1)	Daily minimum temperature (°C)

Note: An example temperature data table is in *\Installation dir\avswatdb\exinputs\temper.dbf*

ASCII (.txt) Table Format:

Line	**Field format**	**Definition**
First	yyyymmdd string	Starting day of data
All other lines	floating point (f5.1), floating point (f5.1) string numbers use comma to separate values	Daily maximum and minimum temperature (°C)

The daily records must be listed in sequential order.

Note: An example temperature data table is in *\Installation dir\avswatdb\exinputs\temper.txt*

The temperature data tables must be placed in the User Data directory (see Section 4.1.8).

❖ Solar Radiation, Wind Speed, or Relative Humidity Gage Location Table (dBase)

When measured solar radiation, wind speed, or relative humidity data are to be used, a table is required to provide the locations of the gages. The location table format described below may be used for any of these three types of records.

Remember, a separate location table is used for each type of weather data.

dBase Table Format: Preferred (4 fields)

Field name	Field format	Definition
ID	Integer	Gage identification number (not used by interface)
NAME	string max 8 chars	Corresponding table[‡] name string
XPR	floating point	X coordinate in the defined projection
YPR	floating point	Y coordinate in the defined projection

dBase Table Format: Alternative[*] (4 fields)

This format can be used only if the projection is defined in the project.

Field name	Field format	Definition
ID	integer	Gage identification number (not used by interface)
NAME	string max 8 chars	Corresponding table name string
LAT	floating point	Latitude in decimal degrees
LONG	floating point	Longitude in decimal degrees

dBase Table Format: Alternative[*] (6 fields)

This format must be used if the projection is ***not*** defined in the project.

Field name	Field format	Definition
ID	integer	Gage identification number (not used by interface)
NAME	string max 8 chars	Corresponding table name string
XPR	floating point	X coordinate in the defined projection
YPR	floating point	Y coordinate in the defined projection
LAT	floating point	Latitude in decimal degrees
LONG	floating point	Longitude in decimal degrees

The user will provide a record for each station to be used: the "Name" field will contain the string used to name the linked solar radiation, wind speed, or relative humidity data table.

[‡] The name of the table which holds the solar radiation, wind speed or relative humidity data is “NAME” .dbf or “NAME”.txt.

[*] When using the alternative table format, the projection of the maps used in the interface must be defined in order for the interface to convert the latitude and longitude to the proper coordinate values.

[*] When using the alternative table format, the projection of the maps used in the interface must be defined in order for the interface to convert the latitude and longitude to the proper coordinate values.

Note: Station names starting with a numerical character are not allowed.

❖ **Solar Radiation Data Table** (dBase or ASCII)

The solar radiation data table is used to store the total daily amounts of solar radiation reaching the ground that are recorded at a specific weather station. This table is required if the Solargages option is chosen for solar radiation in the weather data dialog box. There will be one solar radiation data table for every location listed in the solar radiation location table.

The name of the solar radiation data table is "*name*.dbf" or "*name*.txt" where *name* is the character string entered for NAME in the solar radiation gage location table.

This table may be formatted as a dBase table or as a comma delimited text table.

dBase Table Format: (2 fields)

Field name	**Field format**	**Definition**
DATE	date (yyyymmdd)	Day of measure
SLR	floating point (f8.3)	Daily solar radiation (MJ/m^2/day)

ASCII (.txt) Table Format:

Line	**Field format**	**Definition**
First	yyyymmdd string	Starting day of data
All other lines	floating point (f8.3) string number	Daily solar radiation (MJ/m^2/day)

The daily records must be listed in sequential order.

The solar radiation data tables must be placed in the User Data directory (see Section 4.1.8).

❖ **Wind Speed Data Table** (dBase or ASCII)

The wind speed data table is used to store the average daily wind speeds recorded at a specific weather station. This table is required if the Windgages option is chosen for wind speed data in the weather data dialog box. There will be one wind speed data table for every location listed in the wind speed location table.

The name of the wind speed data table is "*name*.dbf" or "*name*.txt", where *name* is the character string entered for NAME in the wind speed gage location table.

This table may be formatted as a dBase table or as a comma delimited text table.

dBase Table Format: (2 fields)

Field name	**Field format**	**Definition**
DATE	date (yyyymmdd)	Day of measure
WND	floating point (f8.3)	Daily average wind speed (m/s)

ASCII (.txt) Table Format:

Line	**Field format**	**Definition**
First	yyyymmdd string	Starting day of data
All other lines	floating point (f8.3) string number	Daily average wind speed (m/s)

The daily records must be listed in sequential order.

The wind speed data tables must be placed in the User Data directory (see Section 4.1.8).

❖ **Relative Humidity Data Table** (dBase or ASCII)

The relative humidity data table is used to store the fraction relative humidity recorded at a specific weather station. This table is required if the Rel. Humidity gages option is chosen for relative humidity data in the weather data dialog box. There will be one relative humidity data table for every location listed in the relative humidity location table.

The name of the relative humidity data table is "*name*.dbf" or "*name*.txt", where *name* is the character string entered for NAME in the relative humidity gage location table.

This table may be formatted as a dBase table or as a comma delimited text table.

dBase Table Format: (2 fields)

Field name	**Field format**	**Definition**
DATE	date (yyyymmdd)	Day of measure
HMD	floating point (f8.3)	Daily relative humidity (fraction)

ASCII (.txt) Table Format:

Line	**Field format**	**Definition**
First	yyyymmdd string	Starting day of data
All other lines	floating point (f8.3) string number	Daily relative humidity (fraction)

The daily records must be listed in sequential order.

The relative humidity data tables must be placed in the User Data directory (see Section 4.1.8).

❖ **Point Discharge Data Table—Annual Loadings** (dBase or ASCII)

Point source or inlet discharge data may be summarized in one of four methods: constant daily loadings, average annual loadings, average monthly loadings, or daily loadings. If the discharge data is summarized as constant daily loadings, the data will be entered in the Point Discharges Data dialog box. For the other three methods, the interface requires a file to be previously created that contains the point discharge data.

This section describes the format of the point discharge data table for annual loadings. The table may be formatted as a dBase table or as a comma delimited text table.

dBase (.dbf) Table Format: (14 fields)

Field name	Field format	Definition
YEAR	integer i4	Year of measured data
FLOYR	floating point (f12.3)	Average daily water loading for year (m3/day)
SEDYR	floating point (f12.3)	Average daily sediment loading for year (kg/day)
ORGNYR	floating point (f12.3)	Average daily organic N loading for year (kg/day)
ORGPYR	floating point (f12.3)	Average daily organic P loading for year (kg/day)
NO3YR	floating point (f12.3)	Average daily nitrate loading for year (kg/day)
MINPYR	floating point (f12.3)	Average daily soluble P loading for year (kg/day)
NH3YR	floating point (f12.3)	Average daily ammonia loading for year (kg/day)
NO2YR	floating point (f12.3)	Average daily nitrite loading for year(kg/day)
CMTL1YR	floating point (f12.3)	Average daily loading of conservative metal # 1 for year (kg/day)
CMTL2YR	floating point (f12.3)	Average daily loading of conservative metal # 2 for year (kg/day)
CMTL3YR	floating point (f12.3)	Average daily loading of conservative metal # 3 for year (kg/day)
BACTPYR	floating point (f12.3)	Average daily loading of persistent bacteria for year (# bact/100ml)
BACTLPYR	floating point (f12.3)	Average daily loading of less persistent bacteria for year (# bact/100ml)

Note: An example annual point discharge table is in *\Installation dir\avswatdb\exinputs\pointsyearly.dbf*

ASCII (.txt) Table Format

The ASCII table format for yearly records will be a comma delimited text file with the same data reported above for the dBASE format. The first line of the file will contain the field names while the remaining lines will contain the loadings summarized on an annual basis.

Note: An example annual point discharge table is in *\Installation dir\avswatdb\exinputs\ pointsyearly.txt*

❖ **Point Discharge Data Table—Monthly Loadings** (dBase or ASCII)

Point source or inlet discharge data may be summarized in one of four methods: constant daily loadings, average annual loadings, average monthly loadings, or daily loadings. If the discharge data is summarized as constant daily loadings, the data will be entered in the Point Discharges Data dialog box. For the other three methods, the interface requires a file to have been previously created that contains the point discharge data.

This section describes the format of the point discharge data table for monthly loadings. The table may be formatted as a dBase table or as a comma delimited text table.

dBase (.dbf) Table Format: (15 fields)

Field name	Field format	Definition
MONTH	integer i2	Month of measured data
YEAR	integer i4	Year of measured data
FLOMON	floating point (f12.3)	Average daily water loading for month (m3/day)
SEDMON	floating point (f12.3)	Average daily sediment loading for month (kg/day)
ORGNMON	floating point (f12.3)	Average daily organic N loading for month (kg/day)
ORGPMON	floating point (f12.3)	Average daily organic P loading for month (kg/day)
NO3MON	floating point (f12.3)	Average daily nitrate loading for month (kg/day)
MINPMON	floating point (f12.3)	Average daily soluble P loading for month (kg/day)
NH3MON	floating point (f12.3)	Average daily ammonia loading for month (kg/day)
NO2MON	floating point (f12.3)	Average daily nitrite loading for month (kg/day)
CMTL1MON	floating point (f12.3)	Average daily loading of conservative metal # 1 for month (kg/day)
CMTL2MON	floating point (f12.3)	Average daily loading of conservative metal # 2 for month (kg/day)
CMTL3MON	floating point (f12.3)	Average daily loading of conservative metal # 3 for month (kg/day)
BACTPMON	floating point (f12.3)	Average daily loading of persistent bacteria for month (# bact/100ml)
BACTLPMON	floating point (f12.3)	Average daily loading of less persistent bacteria for month (# bact/100ml)

Note: An example monthly point discharge table is in *\Installation dir\avswatdb\exinputs\pointsmonthly.dbf*

ASCII (.txt) Table Format

The ASCII table format for monthly records will be a comma delimited text file with the same data reported above for the dBASE format. The first line of the file will contain the field names while the remaining lines will contain the loadings summarized on a monthly basis.

Note: An example monthly point discharge table is in *\Installation dir\avswatdb\exinputs\pointsmonthly.txt*

❖ **Point Discharge Data Table—Daily Loadings** (dBase or ASCII)

Point source or inlet discharge data may be summarized in one of four methods: constant daily loadings, average annual loadings, average monthly loadings, or daily loadings. If the discharge data is summarized as constant daily loadings, the data will be entered in the Point Discharges Data dialog box. For the other three methods, the interface requires a file to have been previously created that contains the point discharge data.

This section describes the format of the point discharge data table for daily loadings. The table may be formatted as a dBase table or as a comma delimited text table.

dBase (.dbf) Table Format: (14 fields)

Field name	Field format	Definition
DATE	date (yyyyMMdd)	Day of measured data
FLODAY	floating point (f12.3)	Average daily water loading for the day (m3/day)
SEDDAY	floating point (f12.3)	Average daily sediment loading for the day (kg/day)
ORGNDAY	floating point (f12.3)	Average daily organic N loading for the day (kg/day)
ORGPDAY	floating point (f12.3)	Average daily organic P loading for the day (kg/day)
NO3DAY	floating point (f12.3)	Average daily nitrate loading for the day (kg/day)
MINPDAY	floating point (f12.3)	Average daily soluble P loading for the day (kg/day)
NH3DAY	floating point (f12.3)	Average daily ammonia loading for the day (kg/day)
NO2DAY	floating point (f12.3)	Average daily nitrite loading for the day (kg/day)
CMTL1DAY	floating point (f12.3)	Average daily loading of conservative metal # 1 for the day (kg/day)
CMTL2DAY	floating point (f12.3)	Average daily loading of conservative metal # 2 for the day (kg/day)
CMTL3DAY	floating point (f12.3)	Average daily loading of conservative metal # 3 for the day (kg/day)
BACTPDAY	floating point (f12.3)	Average daily loading of persistent bacteria for year (# bact/100ml)
BACTLPDAY	floating point (f12.3)	Average daily loading of less persistent bacteria for the day (# bact/100ml)

Note: An example daily point discharge table is in *\Installation dir\avswatdb\exinputs\ pointsdaily.dbf*

ASCII (.txt) Table Format

Line	Field format	Definition
First	yyyymmdd string	Starting day of data
All other lines	13 floating point (f12.3) string numbers, comma delimited	Set of daily discharge data. From left to right: FLODAY, SEDDAY, ORGNDAY, ORGPDAY, NO3DAY, MINPDAY, NH3DAY, NO2DAY,CMTL1DAY, CTML2DAY, CMTL3DAY, BACTPDAY, BACTLPDAY

The daily records must be listed in sequential order.

Note: An example daily point discharge table is in *\Installation dir\avswatdb\exinputs\pointsdaily.txt*

❖ **Reservoir Monthly Outflow Data Table** (dBase or ASCII)

One option allowed to define reservoir outflow is to provide average daily outflow values for every month of simulation.

This section describes the format of the reservoir monthly outflow data table. The table may be formatted as a dBase table or as a comma delimited text table.

dBase (.dbf) Table Format: (13 fields)

Field name	**Field format**	**Definition**
YEAR	integer i4	Year of measured data
RESOUT1	floating point (f10.1)	Measured average daily outflow for January (m^3/s)
RESOUT2	floating point (f10.1)	Measured average daily outflow for February (m^3/s)
RESOUT3	floating point (f10.1)	Measured average daily outflow for March (m^3/s)
RESOUT4	floating point (f10.1)	Measured average daily outflow for April (m^3/s)
RESOUT5	floating point (f10.1)	Measured average daily outflow for May (m^3/s)
RESOUT6	floating point (f10.1)	Measured average daily outflow for June (m^3/s)
RESOUT7	floating point (f10.1)	Measured average daily outflow for July (m^3/s)
RESOUT8	floating point (f10.1)	Measured average daily outflow for August (m^3/s)
RESOUT9	floating point (f10.1)	Measured average daily outflow for September (m^3/s)
RESOUT10	floating point (f10.1)	Measured average daily outflow for October (m^3/s)
RESOUT11	floating point (f10.1)	Measured average daily outflow for November (m^3/s)
RESOUT12	floating point (f10.1)	Measured average daily outflow for December (m^3/s)

Note: An example reservoir monthly outflow table is in *\Installation dir\avswatdb\exinputs\resmonthly.dbf*

ASCII (.txt) Table Format

The ASCII table format for monthly records will be a comma delimited text file with the same data reported above for the dBASE format. The first line of the file will contain the field names while the remaining lines will contain the monthly reservoir outflow.

Note: An example reservoir monthly outflow table is in *\Installation dir\avswatdb\exinputs\ resmonthly.txt*

❖ **Reservoir Daily Outflow Data Table** (dBase or ASCII)

One option allowed to define reservoir outflow is to provide outflow values for every day of simulation.

This section describes the format of the reservoir daily outflow data table. The table may be formatted as a dBase table or as a comma delimited text table.

dBase (.dbf) Table Format: (2 fields)

Field name	**Field format**	**Definition**
DATE	date (yyyyMMdd)	Day of measure
RESOUTFLOW	floating point (f8.2)	Water release rate for the day (m^3/s)

Note: An example reservoir daily outflow table is in *\Installation dir\avswatdb\exinputs\resdaily.dbf*

ASCII (.txt) Table Format

Line	**Field format**	**Definition**
First	Yyyymmdd string	Starting day of data
All other lines	Floating point (f8.2) string number	Water release rate for the day (m^3/s)

The daily records must be listed in sequential order.

Note: An example daily point discharge table is in *\Installation dir\avswatdb\exinputs\resdaily.txt*

❖ **Potential ET Data Table** (dBase or ASCII)

One option allowed for defining potential evapotranspiration is to provide values for every day of simulation.

This section describes the format of the potential ET daily data table. The table may be formatted as a dBase table or as a comma delimited text table.

dBase (.dbf) Table Format: (2 fields)

Field name	**Field format**	**Definition**
DATE	date (yyyyMMdd)	Day of measure
RESOUTFLOW	floating point (f5.1)	Potential evapotranspiration (mm)

ASCII (.txt) Table Format

Line	**Field format**	**Definition**
First	yyyymmdd string	Starting day of data
All other lines	floating point (f5.1) string number	Potential evapotranspiration (mm)

The daily records must be listed in sequential order.

SECTION 4: START ARCVIEW INTERFACE

To start the ArcView Interface from Windows

1. Start ArcView.
2. On the **File** menu, click **Extensions.**
3. Scroll the list of available extensions until you locate **AVSWAT2000**. Click the box beside **AVSWAT2000**—a check mark will appear in the box.
4. Click **OK**.

Note:	***Do not*** check the **Make Default** option on the ArcView Extension dialog box. This option has a bug that prevents ArcView from starting once you have exited the program after turning on this option.

Once the extension is loaded, the SWAT ArcView main interface screen will be displayed (Figure 4.1).

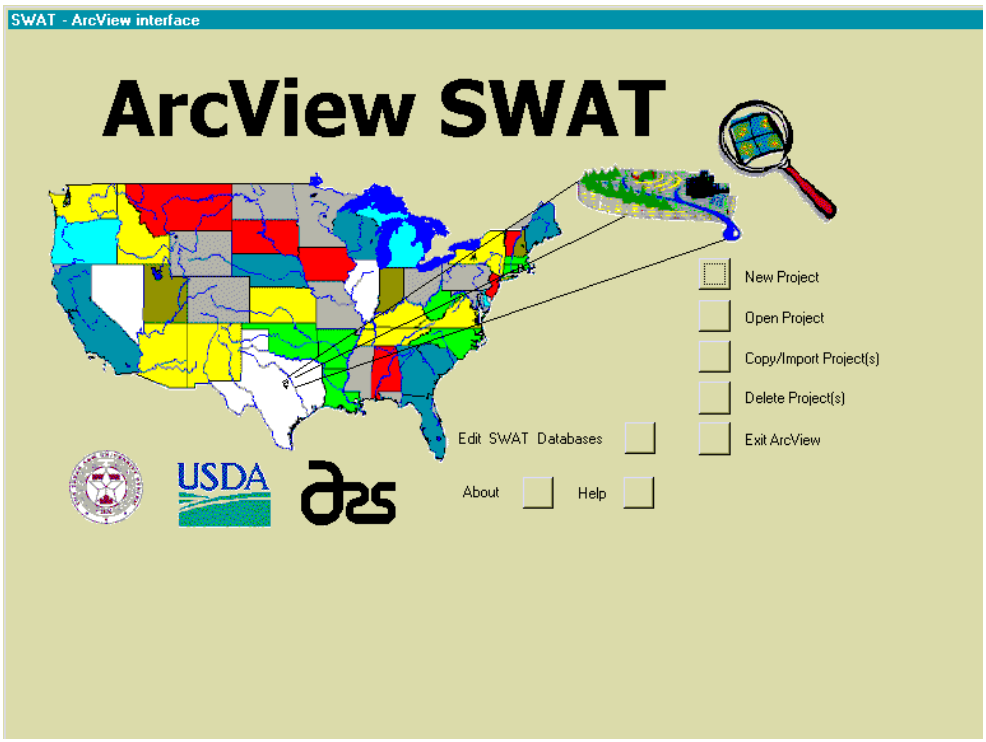


Figure 4.1

The main interface offers the user 8 options:

- Create a new SWAT project
- Open an existing SWAT project
- Copy/Import an existing SWAT project
- Delete an existing SWAT project
- Exit ArcView
- Edit SWAT Databases
- About
- Help

Three additional options are displayed on the main interface once a project has been opened or created (Figure 4.2).

- Save the active project
- Set environmental variables
- Return to current project

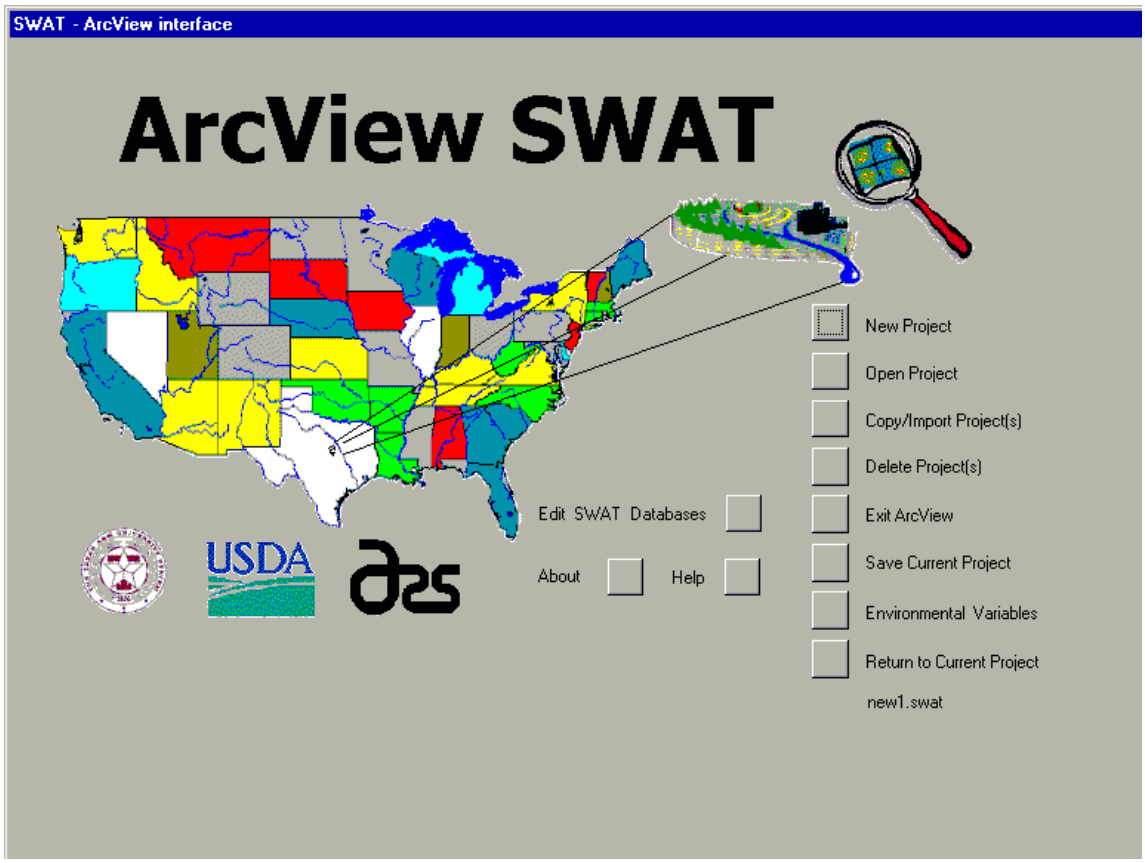


Figure 4.2

SECTION 4.1: MAIN INTERFACE COMMANDS

The following sections describe the use of the command buttons listed on the main interface screen.

SECTION 4.1.1: NEW PROJECT

To create a new SWAT Project

1. On the main interface, click the button beside **New Project**.
2. A browser will be displayed requesting a name for the new project (Figure 4.3). The browser shows the directory tree where the data associated with the new project will be stored. The interface will default the project directory location to the *Installation dir* directory. The default project name and directory location may be used or a new name/directory entered.

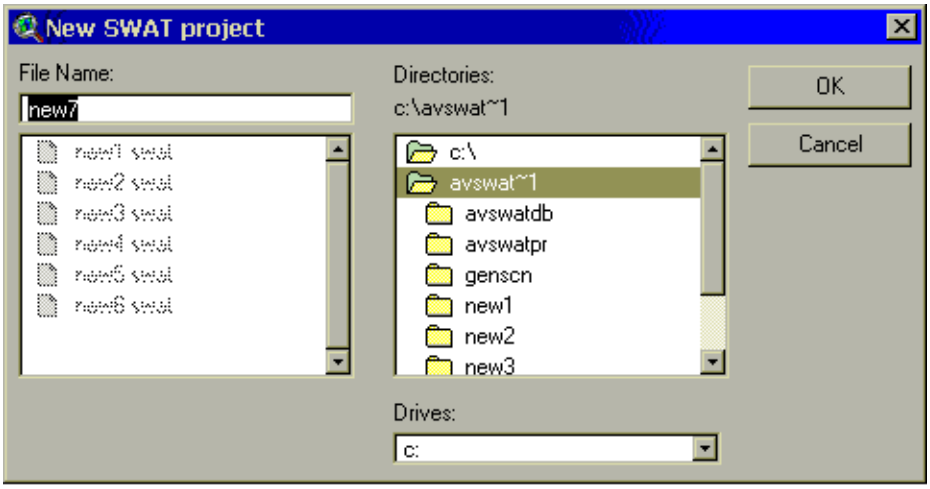


Figure 4.3

3. (Optional) Move to the target directory in the disk.
4. Type a project name in the text box called File Name (a default name for the project is provided).
5. Change the project directory location if desired.
6. Once the desired project name is displayed in the browser, click **OK**.

> Within the directory specified in the browser, the SWAT ArcView interface creates a subdirectory to store the project files generated for SWAT. The subdirectory has the same name as the project. A project management file or log file is also created for the project and is stored in the main directory. The log file is of the file type **SWAT project**. The log file name is the project name with the extension **.swat**.

5. A project window for the new SWAT will be displayed.

SECTION 4.1.2: OPEN PROJECT

To open an existing SWAT Project

1. On the main interface, click the button beside **Open Project**.
2. A browser will be displayed (Figure 4.4). If necessary, move to the proper directory. Click on the name of the SWAT project log file (**.swat**) belonging to the project you wish to open.

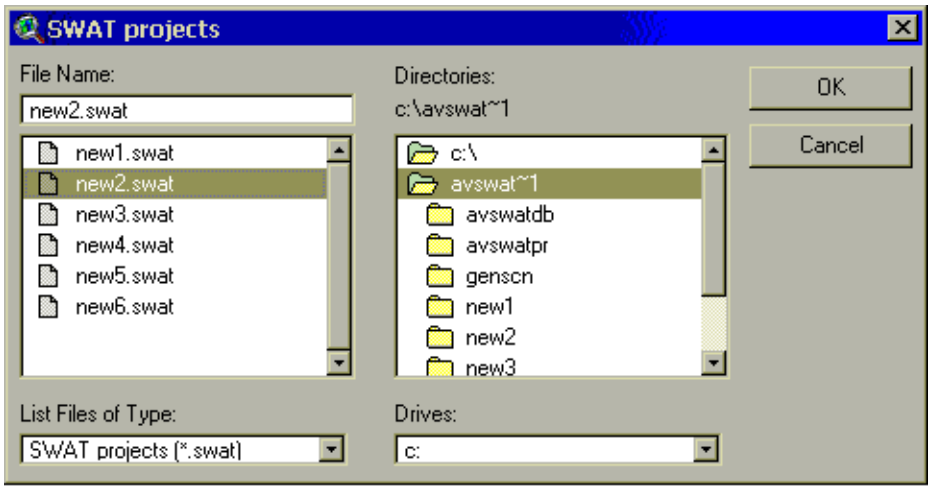


Figure 4.4

3. Once the desired project name is displayed in the text box called File Name, click **OK**.
4. A prompt box will be displayed which offers the option of making a backup copy of the project. If you wish to make a backup copy of the project, click **Yes**. If not, click **No**.
5. The project window for the SWAT project will be displayed. The layout of the window is recreated from the last time the project was saved.

> The user may also open a project without loading the AVSWAT2000 extension. To open a project directly from ArcView, select **Open Project** on the ArcView **File** menu. A browser will be displayed. Use the directory tree to open the project directory. Within the project directory, an ArcView project (.apr) file is stored. Select this file and click **OK**. ArcView will load the SWAT interface and open the project in one step.

SECTION 4.1.3: COPY/IMPORT PROJECT

To copy an existing SWAT Project

1. On the main interface, click the button beside **Copy/Import Project**.
2. A browser will be displayed (Figure 4.5). If necessary, move to the proper directory. Type or click the name of the SWAT project log file (**.swat**) belonging to the project you wish to copy.

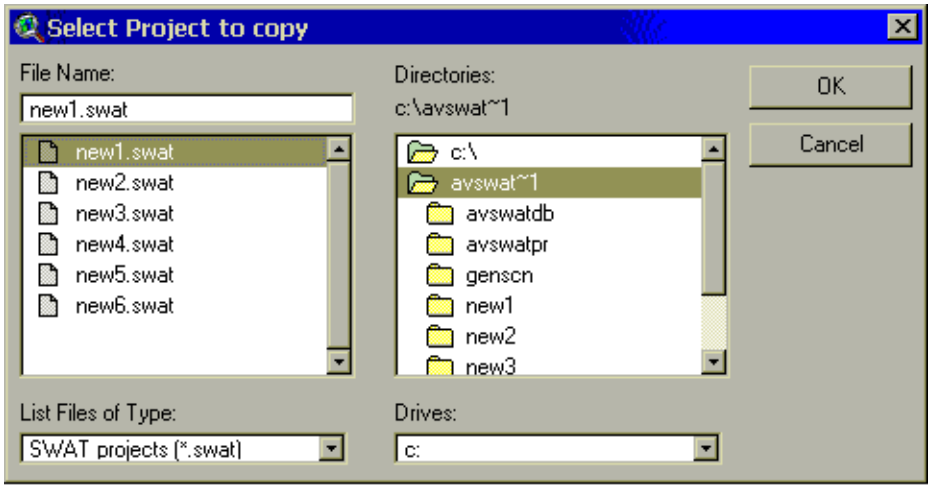


Figure 4.5

3. Once the desired project name is displayed in the File Name text box, click **OK**.
4. A new browser will be displayed (Figure 4.6). If you wish to copy the project to a different directory, move to the desired directory. Type the new project name in the text box and click **OK**.

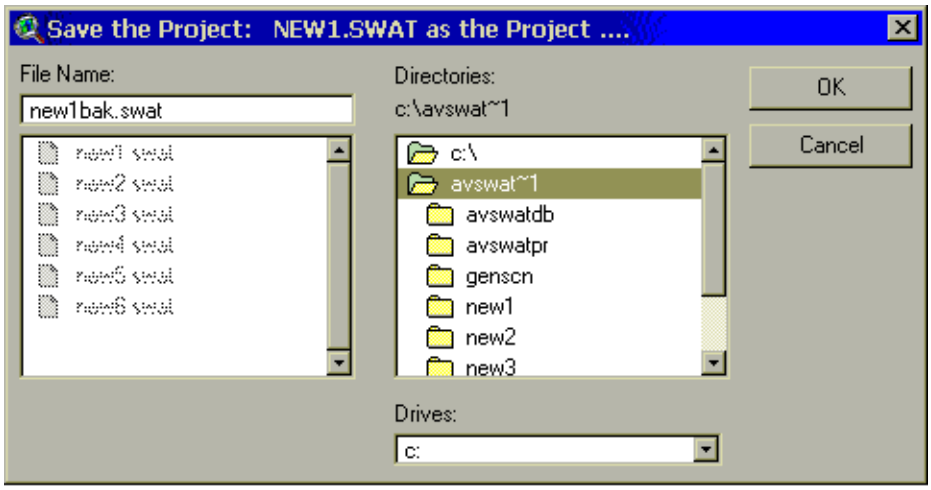


Figure 4.6

5. As the interface copies the project, links to any maps or data stored in a directory other than the *Installation dir* directory must be verified. This feature

allows path information to be altered when a project has been moved from one computer to another. Browsers will be displayed for every unique file path the interface encounters as it copies a project. Set the path to the proper directory and click **OK**.

6. A prompt box will be displayed once the copy process is complete. Click **OK**.

SECTION 4.1.4: DELETE PROJECT

To delete an existing SWAT project

1. On the main interface, click the button beside **Delete Project**.
2. A browser will be displayed (Figure 4.7). If necessary, move to the proper directory. Type or click the name of the SWAT project log file (**.swat**) belonging to the project you wish to delete.

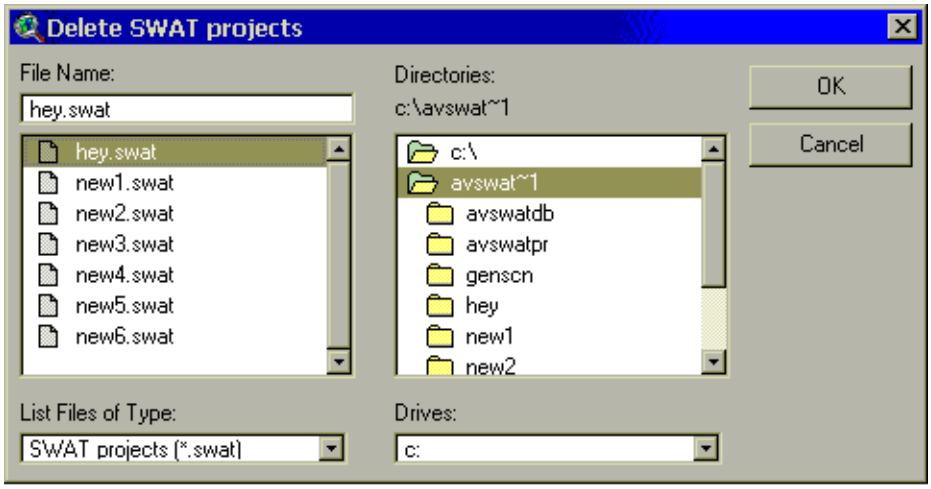


Figure 4.7

3. Once the desired project name is displayed in the File Name text box, click **OK**.
4. A prompt box will appear to verify the delete command. To continue with the delete process, click **Yes**. To abandon the delete process, click **No.**
5. The selection mouse pointer will be replaced with an hourglass symbol during the delete process. Once the delete process is complete, a prompt box will be displayed (Figure 4.8). Click **OK**.

Figure 4.8

SECTION 4.1.5: EXIT ARCVIEW

To close the SWAT ArcView interface and ArcView

1. On the main interface, click the button beside **Exit ArcView**.
2. A prompt box will be displayed requesting the user to verify the command (Figure 4.9)

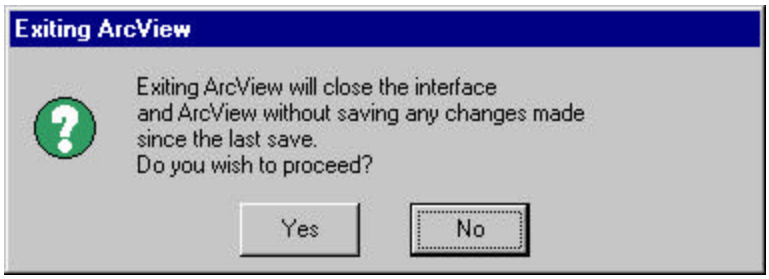


Figure 4.9

3. To close the SWAT ArcView interface and ArcView program, click **Yes**. To abandon the exit procedure, and return to the main interface screen, click **No**.

SECTION 4.1.6: EDIT SWAT DATABASES

To edit the SWAT Databases

1. On the main interface, click the button beside **Edit SWAT Databases**.
2. A dialog box will appear with a list of the databases (Figure 4.10).

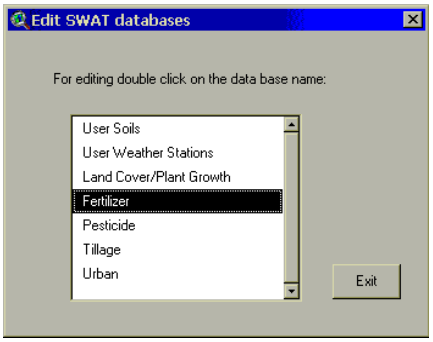


Figure 4.10

Seven databases are available for editing:

User Soils

This database is used to store custom soil data. Data is entered into this database for soil maps that do not use the soil database included with the interface.

User Weather Stations

This database is used to store custom weather generator station data. Statistical weather data is entered into this database for weather simulation stations not included in the provided database.

Land Cover/Plant Growth

This database contains SWAT plant growth parameters. While users are given the option of modifying existing land cover/plant parameters or adding additional plant species to the database, we would like to emphasize that changes to the plant database should be based on experimental data. The typical user will not need to make changes to this database.

Fertilizer

This database contains SWAT fertilizer parameters. Both inorganic and organic (manure) fertilizer data is stored in this database.

Pesticide

This database contains SWAT pesticide parameters.

Tillage

This database contains SWAT tillage parameters.

Urban

This database contains SWAT urban land type parameters.

Section 14 describes the procedures used to modify the databases.

SECTION 4.1.7: SAVE CURRENT PROJECT

To save the active SWAT project

1. On the main interface, click the button beside **Save Project**.
2. A prompt box will appear once the project has been saved. Click **OK** to return to the main interface.

> The project ***must*** be saved via the **Save Project** option on the main interface menu or by the **Save Project** button (diskette symbol or **File** menu item) for the user to be able to fully restore the project in the interface at a future time.

SECTION 4.1.8: ENVIRONMENTAL VARIABLES

To set the environmental variables for a project

While the user specifies the directory where *output* from the interface is stored with the **New Project** command, the **Environmental Variables** command is used to identify the directory where data accessed by the interface is stored, i.e. interface *input*. The types of data and map themes required to run the interface are reviewed in Section 3.

1. On the main interface, click the button beside **Environmental Variables**.
2. A prompt box will appear which shows the locations of data and programs used by the interface (Figure 4.11). While the SWAT ArcView interface will automatically set the locations of **SWAT Programs** and **SWAT Databases** upon installation, the **SWAT User Data** directory needs to be defined for every new project.

> Whenever a browser is activated, it will automatically open the SWAT Users Data directory. While the maps for the interface can be located any place the user wishes, the precipitation, temperature, solar radiation, relative humidity, wind speed, and weather generator data files must be stored in this directory.

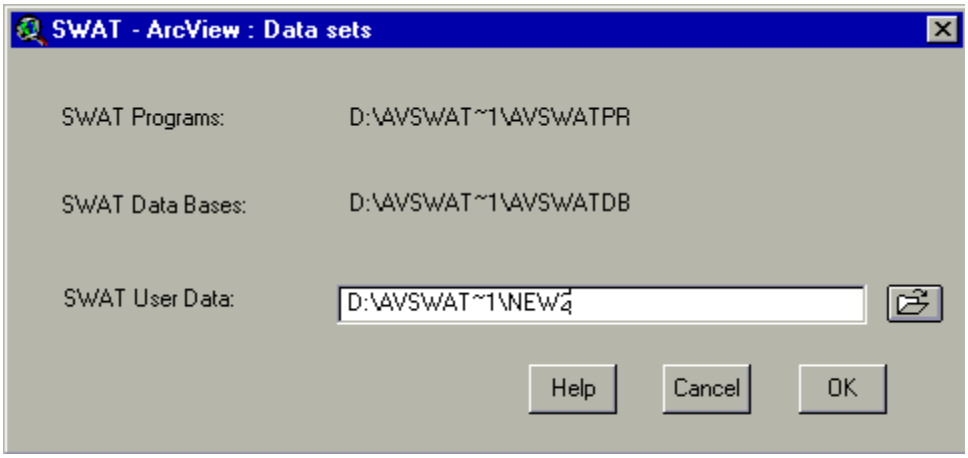


Figure 4.11

The directory initially specified in the text field is the directory where interface output for the project is stored--the project directory. This needs to be replaced with the location of the interface data directory.

3. Type in the directory path or press the button to activate a browser (Figure 4.12).

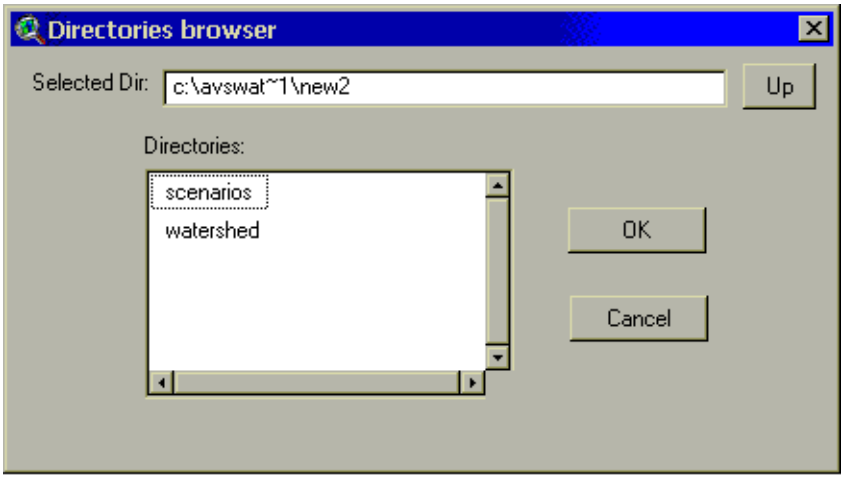


Figure 4.12

4. When searching for the directory, click **Up** to close the current directory. To open or choose a subdirectory, find the name of the directory in the listing and click on it.
5. In order to change drive, select the entire path and type the drive letter.
6. Once the directory path listed in the text bar is correct (Figure 4.13), click **OK**.

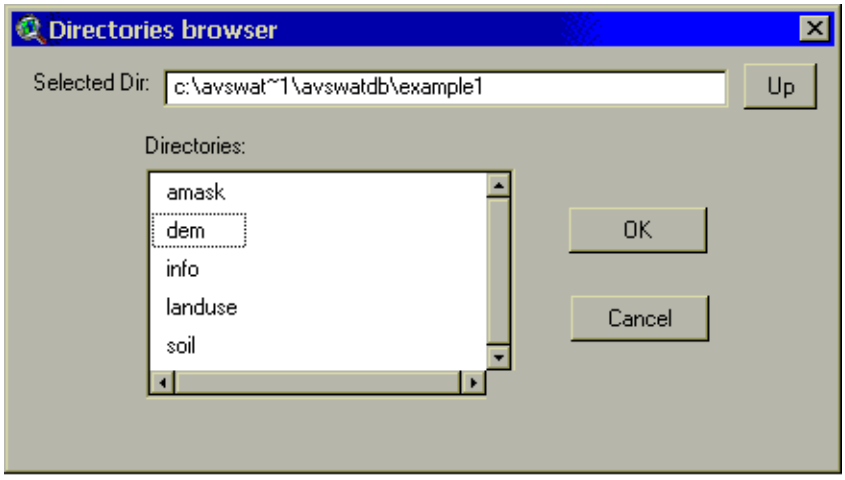


Figure 4.13

7. The prompt window will be shown with the selected directory (Figure 4.14). If the listed directory is correct, click **OK**.

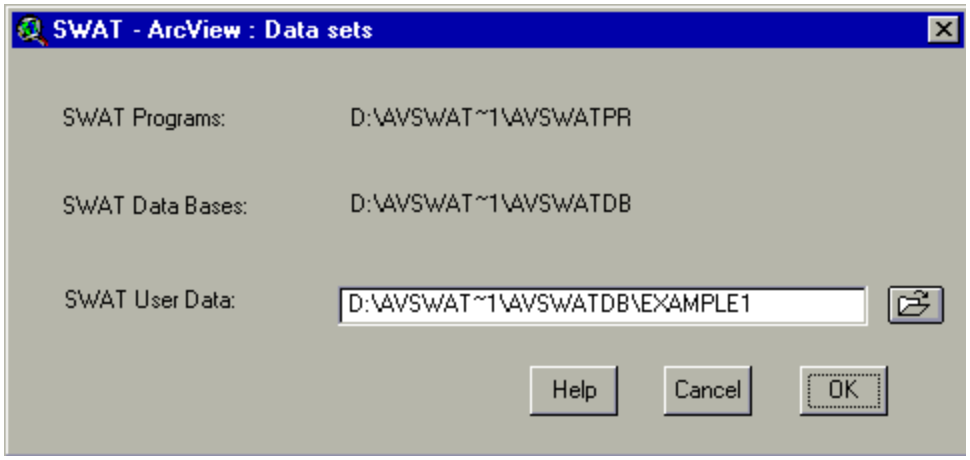


Figure 4.14

8. The project window will be restored to the screen.

SECTION 4.1.9: RETURN TO CURRENT PROJECT

To return to the project window of an open project

1. On the main interface, click the button beside **Return to current project**.

SECTION 4.2: PROJECT VIEWS

The SWAT ArcView interface creates two views: the Watershed View and the SWAT View. The Watershed View is used to process all maps while the SWAT View is used to edit input data, run the SWAT model, and analyze output.

The Watershed View menu bar contains the same pull-down menus as the basic ArcView window, 2 Spatial Analyst pull-down menus (**Analysis** and **Surface**) plus one menu unique to the SWAT ArcView Watershed View—**Avswat**.

The SWAT View menu bar contains the same pull-down menus as the basic ArcView window plus 4 menus unique to the SWAT View. These are **Input**, **Edit Input**, **Simulation**, and **Reports**. The SWAT View is created once all map processing has been completed in the Watershed View. To shift between the Watershed and SWAT Views, position the mouse cursor in an open area in the map display, click the right mouse button, and select **SWAT View** from the pop-up menu (see Section 4.3 for more information about the pop-up menu).

Figure 4.15 shows the ArcView View menu and tool bars, Figure 4.16 shows the Watershed View menu and tool bars while Figure 4.17 shows the SWAT View menu and tool bars.

Figure 4.15

Figure 4.16

Figure 4.17

SECTION 4.2.1: THE AVSWAT MENU

The Avswat menu contains all the commands required to import and process the ArcView themes and tables used for delineation of subbasins and HRUs within the watershed. Figure 4.18 displays the Avswat menu.

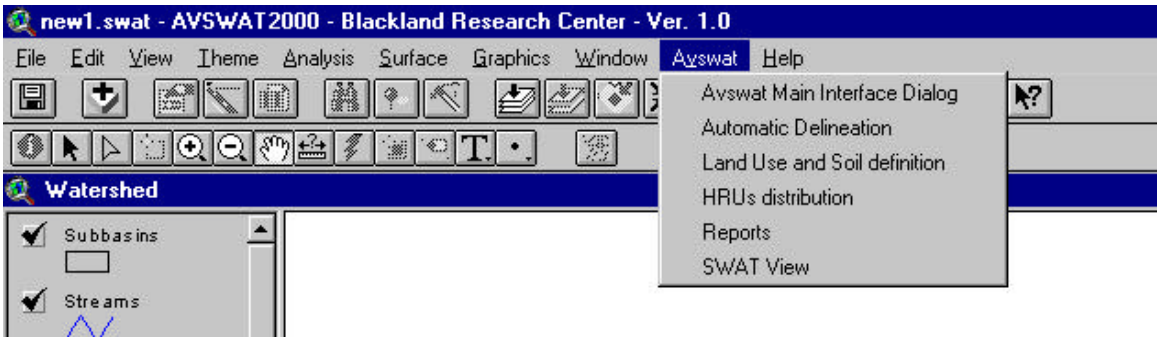


Figure 4.18

The Avswat Menu: Avswat main interface dialog

The **Avswat main interface dialog** command returns the user to the main interface screen (Figure 4.2).

The Avswat Menu: Automatic Delineation

The **Automatic Delineation** command accesses the dialog box used to import topographic maps and delineate the watershed. This procedure is reviewed in Section 5.

The Avswat Menu: Land Use and Soil definition

The **Land Use and Soil definition** command accesses the dialog box used to import land use and soil maps, link the maps to SWAT databases, and perform an overlay. This procedure is reviewed in Section 6.

The Avswat Menu: HRUs distribution

The **HRUs distribution** command accesses the dialog box used to define the number of HRUs created within each subbasin in the watershed. This procedure is reviewed in Section 6.

The Avswat Menu: Reports

The **Reports** command lists various reports generated by the interface. To access a particular report, highlight the name of the report and click the left mouse button. The report of interest will be displayed in a text editor.

The Avswat Menu: SWAT View

The **SWAT View** command activates the SWAT View.

SECTION 4.2.2: THE INPUT MENU

The Input menu contains the commands which generate the ArcView database files used by the interface to store input values for the SWAT model. Figure 4.19 displays the input menu.

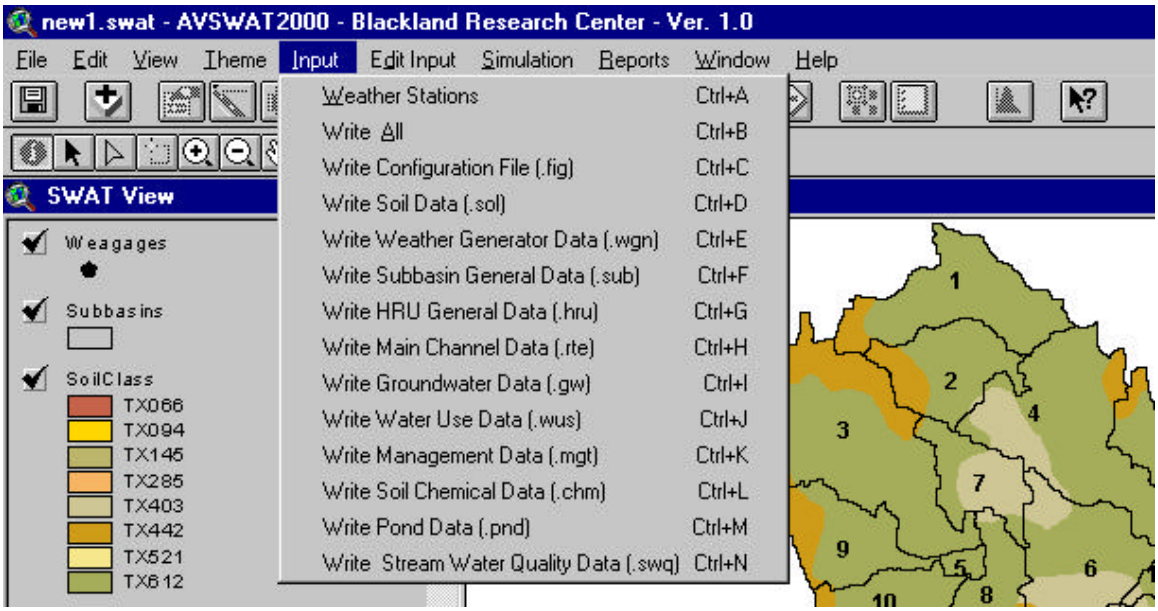


Figure 4.19

The Input Menu: Weather Stations

The **Weather Stations** command loads weather station locations and data for use.

The Input Menu: Write All

The **Write All** command creates ArcView tables (.dbf) that store values for SWAT input parameters. Initial SWAT ASCII input files are also generated.

The Input Menu: Write Configuration File (.fig)

The **Write Configuration File (.fig)** command creates an ArcView table (fig.dbf) that stores values for the watershed configuration input parameters. The SWAT ASCII watershed configuration input file is also generated.

The Input Menu: Write Soil Data (.sol)

The **Write Soil Data (.sol)** command creates an ArcView table (sol.dbf) that stores values for SWAT soil input parameters. Initial SWAT ASCII .sol input files are also generated.

The Input Menu: Write Weather Generator Data (.wgn)

The **Write Weather Generator Data (.wgn)** command creates an ArcView table (wgn.dbf) that stores values for SWAT weather generator input parameters. Initial SWAT ASCII .wgn input files are also generated.

The Input Menu: Write Subbasin General Data (.sub)

The **Write Subbasin General Data (.sub)** command creates an ArcView table (sub.dbf) that stores values for SWAT subbasin input parameters. Initial SWAT ASCII .sub input files are also generated.

The Input Menu: Write HRU General Data (.hru)

The **Write HRU General Data (.hru)** command creates an ArcView table (hru.dbf) that stores values for SWAT HRU input parameters. Initial SWAT ASCII .hru input files are also generated.

The Input Menu: Write Main Channel Data (.rte)

The **Write Main Channel Data (.rte)** command creates an ArcView table (rte.dbf) that stores values for SWAT main channel input parameters. Initial SWAT ASCII .rte input files are also generated.

The Input Menu: Write Groundwater Data (.gw)

The **Write Groundwater Data (.gw)** command creates an ArcView table (gw.dbf) that stores values for SWAT groundwater input parameters. Initial SWAT ASCII .gw input files are also generated.

The Input Menu: Write Water Use Data (.wus)

The **Write Water Use Data (.wus)** command creates an ArcView table (wus.dbf) that stores values for SWAT water use input parameters. Initial SWAT ASCII .wus input files are also generated.

The Input Menu: Write Management Data (.mgt)

The **Write Management Data (.mgt)** command creates an ArcView table (mgt.dbf) that stores values for SWAT management input parameters. Initial SWAT ASCII .mgt input files are also generated.

The Input Menu: Write Soil Chemical Data (.chm)

The **Write Soil Chemical Data (.chm)** command creates an ArcView table (chm.dbf) that stores values for SWAT soil chemical input parameters. Initial SWAT ASCII .chm input files are also generated.

The Input Menu: Write Pond Data (.pnd)

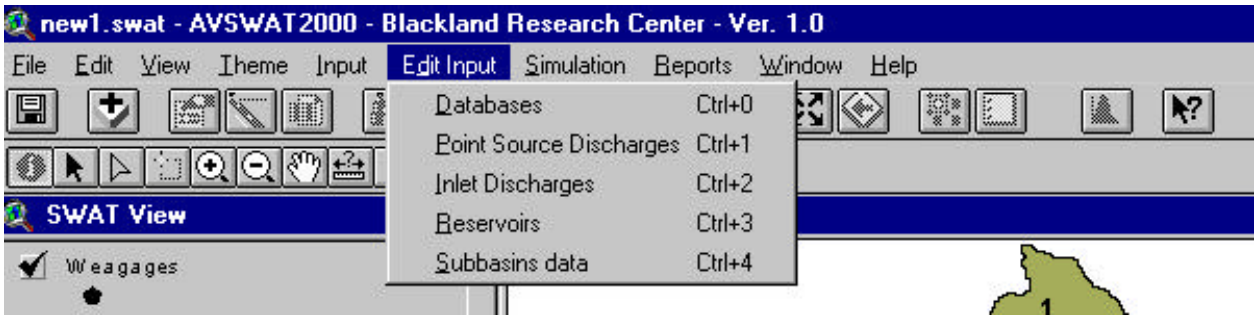
The **Write Pond Data (.pnd)** command creates an ArcView table (pnd.dbf) that stores values for SWAT pond/wetland input parameters. Initial SWAT ASCII .pnd input files are also generated.

The Input Menu: Write Stream Water Quality Data (.swq)

The **Write Stream Water Quality Data (.swq)** command creates an ArcView table (swq.dbf) that stores values for SWAT stream water quality input parameters. Initial SWAT ASCII .swq input files are also generated.

SECTION 4.2.3: THE EDIT INPUT MENU

The **Edit Input** menu is one of four new menus introduced in the SWAT View. The **Edit Input** menu allows you to edit the SWAT model databases and the



watershed database files containing the current inputs for the SWAT model. Five items are listed on the **Edit Input** menu (Figure 4.20).

Figure 4.20

The Edit Input Menu: Databases

The **Databases** command allows the user to access the SWAT model databases from within a project.

The Edit Input Menu: Point Source Discharges

The **Point Source Discharges** command allows the user to access/define the point source loadings for all subbasins with point source discharges.

The Edit Input Menu: Inlet Discharges

The **Inlet Discharges** command allows the user to access/define loadings for upstream sections of the watershed not directly modeled in the current project.

The Edit Input Menu: Reservoirs

The **Reservoirs** command allows the user to access/edit input parameters for any reservoirs located within the watershed.

The Edit Input Menu: Subbasins Data

The **Subbasins Data** command allows the user to access/edit input parameters for land areas, channels, ponds/wetlands, and groundwater systems within the watershed.

SECTION 4.2.4: THE SIMULATION MENU

The **Simulation** menu is one of four new menus introduced in the SWAT View. The **Simulation** menu allows you to run the SWAT model and import results to the interface. The **Simulation** menu also contains tools that allow the user to rapidly calibrate the model simulation and determine delivery loads from the different areas of the watershed. Four items are listed on the **Simulation** menu (Figure 4.21).

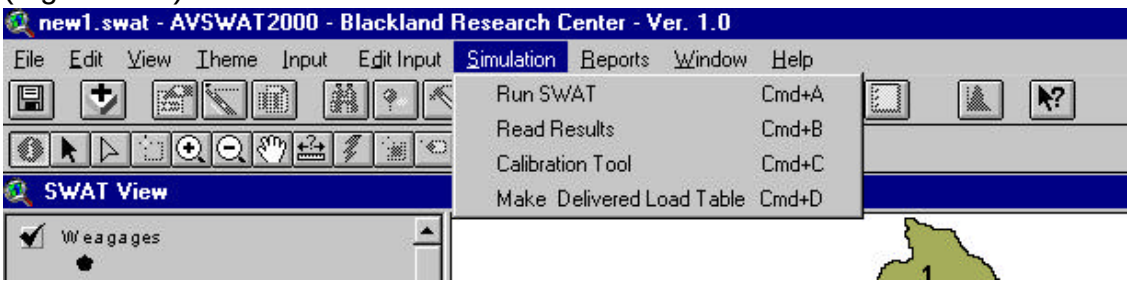


Figure 4.21

The Simulation Menu: Run SWAT

The **Run SWAT** command allows the user to modify parameters in three SWAT input files, the input control code file (.cod), the basin input file (.bsn), and the watershed water quality input file (.wwq), as well as set up and run the SWAT model.

The Simulation Menu: Read Results

The **Read Results** command imports output from the SWAT run into ArcView tables (.dbf) so that the simulation results may be displayed within the interface.

The Simulation Menu: Calibration Tool

The **Calibration Tool** command allows the user to perform global edits on SWAT input variables commonly modified during calibration. The interface saves these modified SWAT runs in separate directories so that the user can compare results from several different calibration runs.

The Simulation Menu: Make Delivered Load Table

The **Make Delivered Load Table** command analyzes SWAT output to calculate the source of loadings at any point along the channel network.

SECTION 4.2.5: THE REPORTS MENU

The **Reports** menu is one of four new menus introduced in the SWAT View. The **Reports** menu provides the user access to ArcView and ASCII tables generated during the project. Five items are listed on the **Reports** menu (Figure 4.22).

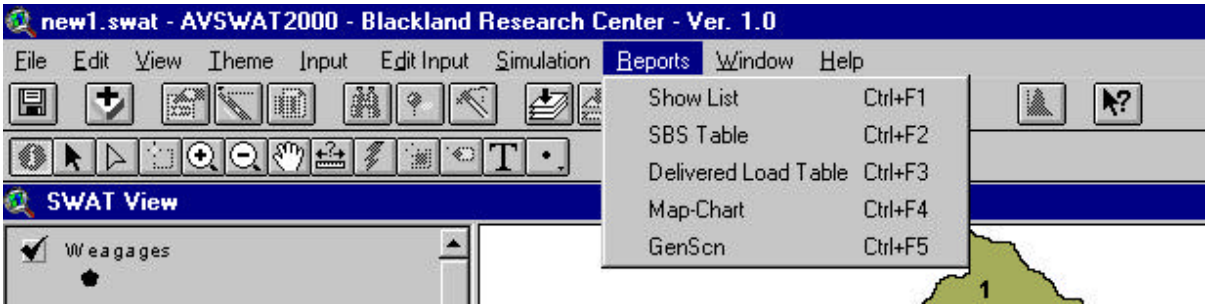


Figure 4.22

The Reports Menu: Show List

The **Show List** command displays a list of all ASCII reports generated in the project as well as several of the SWAT input files that users might wish to review.

The Reports Menu: SBS Table

The **SBS Table** command displays the ArcView table containing information loaded from the SWAT sbs output file (HRU output) (this command is enabled only when the simulation printout frequency is yearly).

The Reports Menu: Delivered Load Table

The **Delivered Load Table** command displays the ArcView table containing the loadings from the various subbasins for the point along the stream network defined when the **Make Delivered Load Table** command under the **Simulation** menu was performed.

The Reports Menu: Map-Chart

The **Map-Chart** command provides a dialog box that allows the user to map and chart information from the SWAT .bsb and .rch output files and the Delivered Load Table. See Section 13.6 for a full description of this tool.

The Reports Menu: GenScn

The **GenScn** command allows the user to analyze the outputs using GENeration and analysis of model simulation SceNarios (GenScn) tool (U.S.G.S., 1998). The installation package is located in:

\Installation dir\GenScn

U.S. Geological Survey, Department of the Interior. 1998. A Tool for the Generation and Analysis of Model Simulation Scenarios for Watersheds (GenScn). Water-Resources Investigations Report 98-4134. Decatur, GA, Reston, VA, 152 p.

SECTION 4.3: POP UP MENU

A shortcut menu to several tools is provided once the user has completed all procedures in the Watershed View and activated the SWAT View. This shortcut allows you to switch the current working View, edit the Point Source of Discharge, Inlet Discharge, Reservoir, and all the Subbasin inputs.

1. Right click the mouse within the map display area. A menu will appear (Figure 4.23)

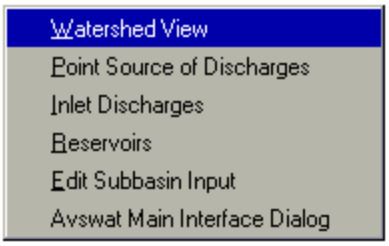


Figure 4.23

2. The pop-up menu contains 6 items:
   a. **Watershed View**. Selecting this option will switch the screen to the Watershed View. You can return to the SWAT View using the analog pop up menu or by selecting **SWAT View** under the **AvSwat** menu contained in Watershed View.
   b. **Point Source of Discharges**.
      i. If the mouse cursor was located outside the watershed when right clicked, a message box will be displayed (Figure 4.24).

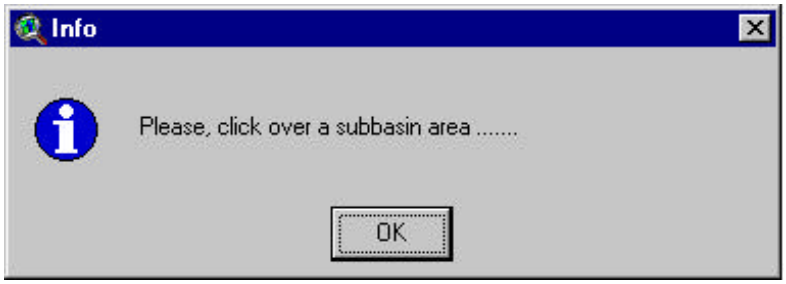


Figure 4.24

Click **OK** to proceed.

      ii. If the mouse cursor was located over a subbasin with no point discharges when right clicked, a message box will notify the user (Figure 4.25).

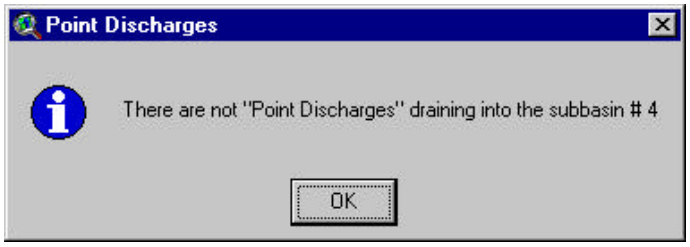


Figure 4.25

Click **OK** to proceed.

iii. If the mouse cursor was located over a subbasin with point discharges when right clicked, a dialog box will display the subbasin point source input values (Figure 4.26).

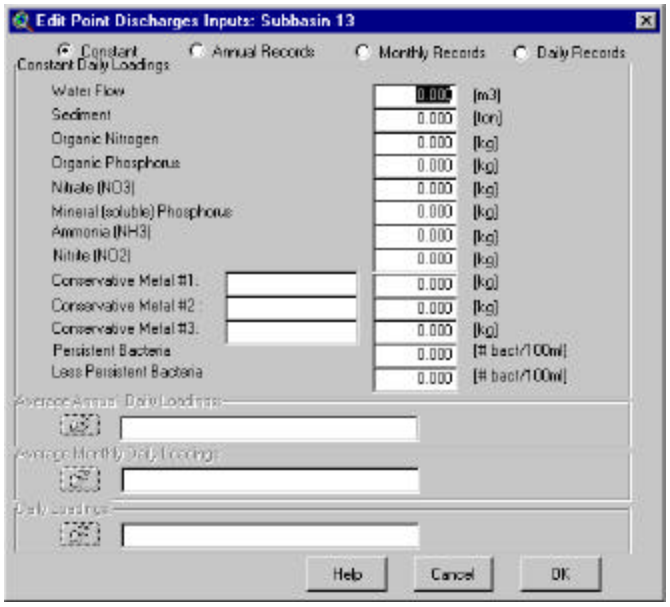


Figure 4.26

This is the same dialog box that is accessed by clicking **Point Source Discharges** under the **Edit Input** menu. See Section 9 for more information about the dialog box.

c. **Inlet Discharges**:

i. If the mouse cursor was located outside the watershed when right clicked, a message box will notify the user (Figure 4.27).

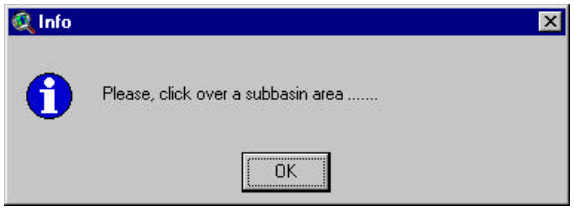


Figure 4.27

Click **OK** to proceed.

ii. If the mouse cursor was located over a subbasin with no inlet discharges when right clicked, a message box will notify the user (Figure 4.28).

Figure 4.28

Click **OK** to proceed.

iii. If the mouse cursor was located over a subbasin with inlet discharges when right clicked, a dialog box will display the subbasin inlet discharge data (Figure 4.29).

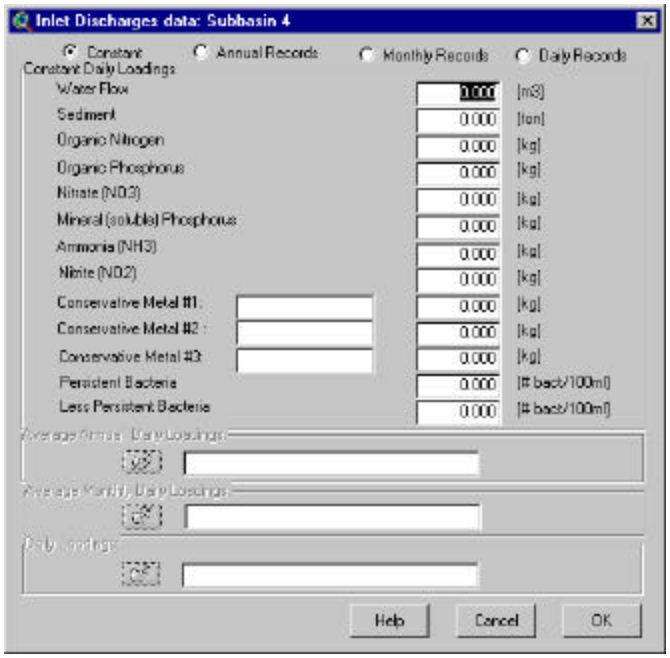


Figure 4.29

This is the same dialog box that is accessed by clicking **Inlet Discharges** under the **Edit Input** menu. See Section 10 for more information about the dialog box.

d. **Reservoirs**:

i. If the mouse cursor was located outside the watershed when right clicked, a message box will notify the user (Figure 4.30).

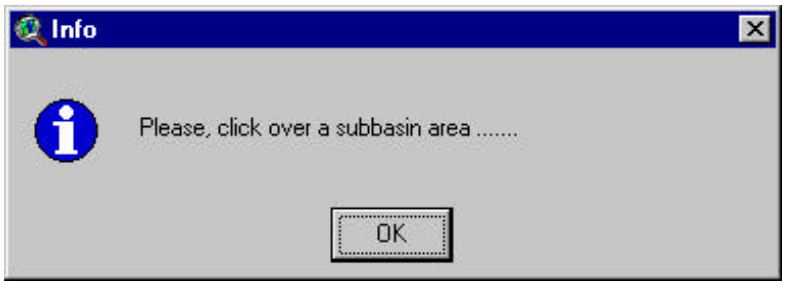


Figure 4.30

Click **OK** to proceed.

ii. If the mouse cursor was located over a subbasin with no reservoir when right clicked, a message box will notify the user (Figure 4.31).

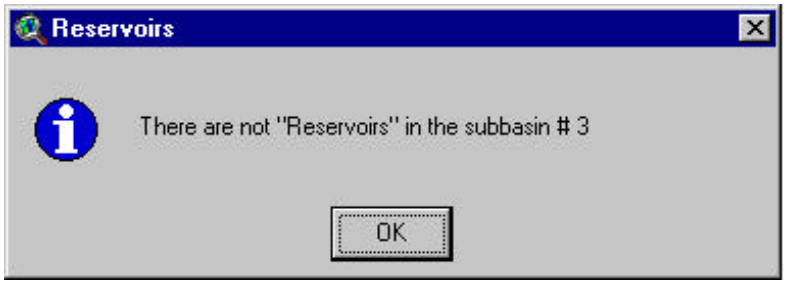


Figure 4.31

iii. If the mouse cursor was located over a subbasin with a reservoir when right clicked, a dialog box will display the reservoir input data (Figure 4.32).

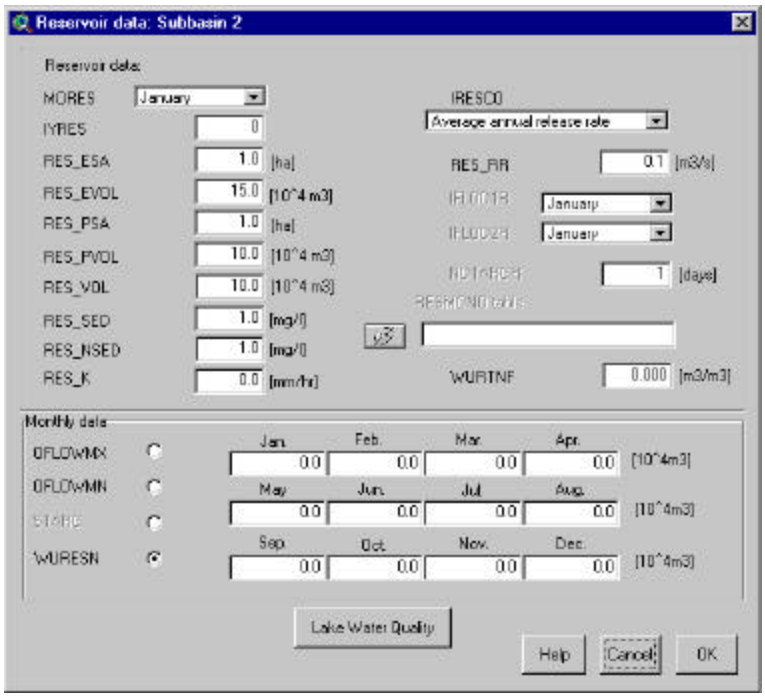


Figure 4.32

This is the same dialog box that is accessed by clicking **Reservoirs** under the **Edit Input** menu. See Section 11 for more information about the dialog box.

e. **Edit Subbasin Input**:

i. If the mouse cursor was located outside the watershed when right clicked, a message box will notify the user (Figure 4.33).

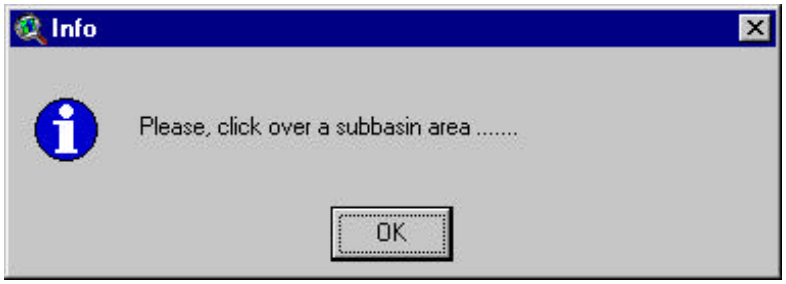


Figure 4.33

Click **OK** to proceed.

ii. If the mouse cursor was located over any subbasin area when right clicked, the Edit Subbasin Inputs dialog pops up with the subbasin number selected (Figure 4.34).

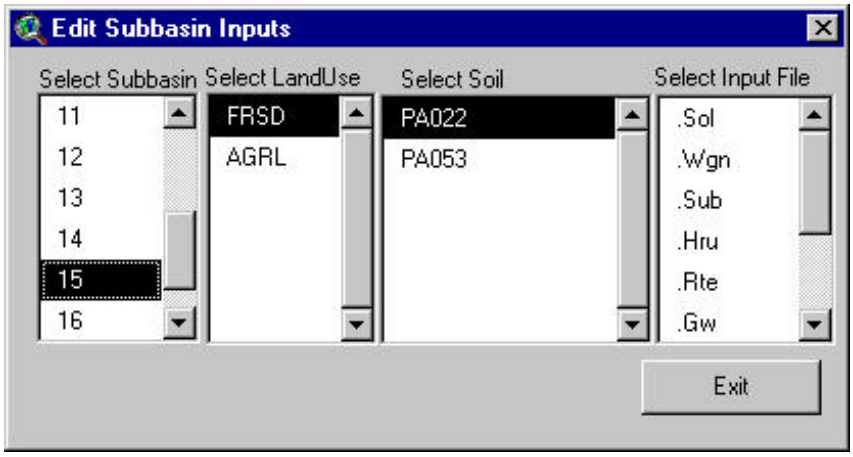


Figure 4.34

This is the same dialog box that is accessed by clicking **Subbasins Data** under the **Edit Input** menu. See Section 12 for more information about the dialog box.

Tip:	Before clicking the mouse, move the mouse cursor on top of the subbasin area you want to analyze. This is convenient when using all the pop-up menu items except the first one.

f. **Avswat Main Interface Dialog**. Selecting this option will switch the screen to the Main Interface screen (Figure 4.35). You can return to the project by clicking the button labeled **Return to Current Project**.

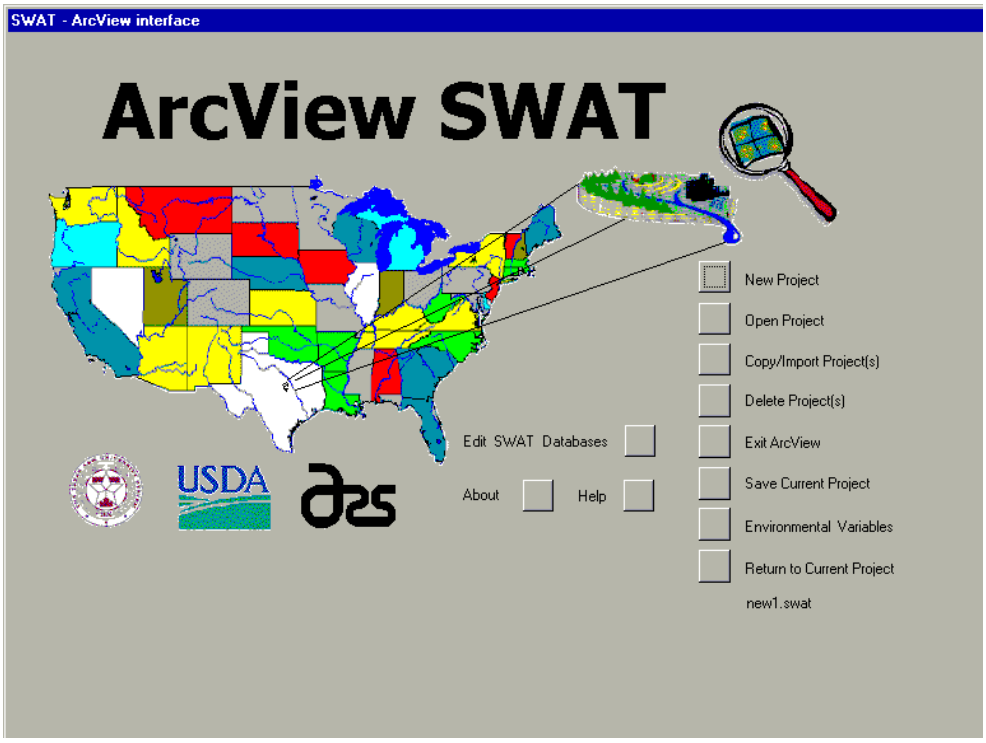


Figure 4.35

SECTION 5: WATERSHED DELINEATION

This tool allows the user to delineate subwatersheds based on an automatic procedure using Digital Elevation Model (DEM) data. User specified parameters provide limits that influence the size and number of subwatersheds created.

Purpose

The *Watershed Delineation* carries out advanced GIS functions to aid the user in segmenting watersheds into several "hydrologically" connected sub-watersheds for use in watershed modeling with SWAT.

Application

The *Watershed Delineation* tool uses and expands ArcView and Spatial Analyst extension functions to operate watershed delineations. The delineation process requires a Digital Elevation Model (DEM) in ArcInfo grid format. The user also has the option of importing and using a pre-digitized stream network in ArcView shape (PolyLine) format.

Once the delineation is finished, a detailed report (*Topographic Report*) is added to the current project, and the following resulting themes will be added to the *Watershed View*: *Subbasins*, *Streams*,*Outlets* and (optional) *Reservoirs*. See Watershed Delineation Output Data for the content of the respective table of attributes. The topographic report describes the elevation distribution within the watershed (or "hydrologically" not connected watersheds) and within each sub-watershed unit (subbasin). The themes carry the parameters of the watershed(s) characterization.

Key Procedures

- Load the DEM
- (Optional) Define the working area (Focused Area)
- (Optional) Load the stream network to be used for the delineation
- Preprocess the DEM
- Specify the minimum sub-watershed area (critical source area)
- Review and edit the stream network points
- Run the calculation of the subbasin parameters

- (Optional) Locate the Reservoirs

> **Note:** Only the DEM grid file is required for the delineation process.

SECTION 5.1: WATERSHED DELINEATION DIALOG BOX

When a new project is created, the Watershed Delineation dialog box will automatically be displayed. If the dialog box is not displayed, on the **Avswat** menu, click **Automatic Delineation**. (The **Avswat** menu is present only in the Watershed View.) The Watershed Delineation dialog will open (Figure 5-1).

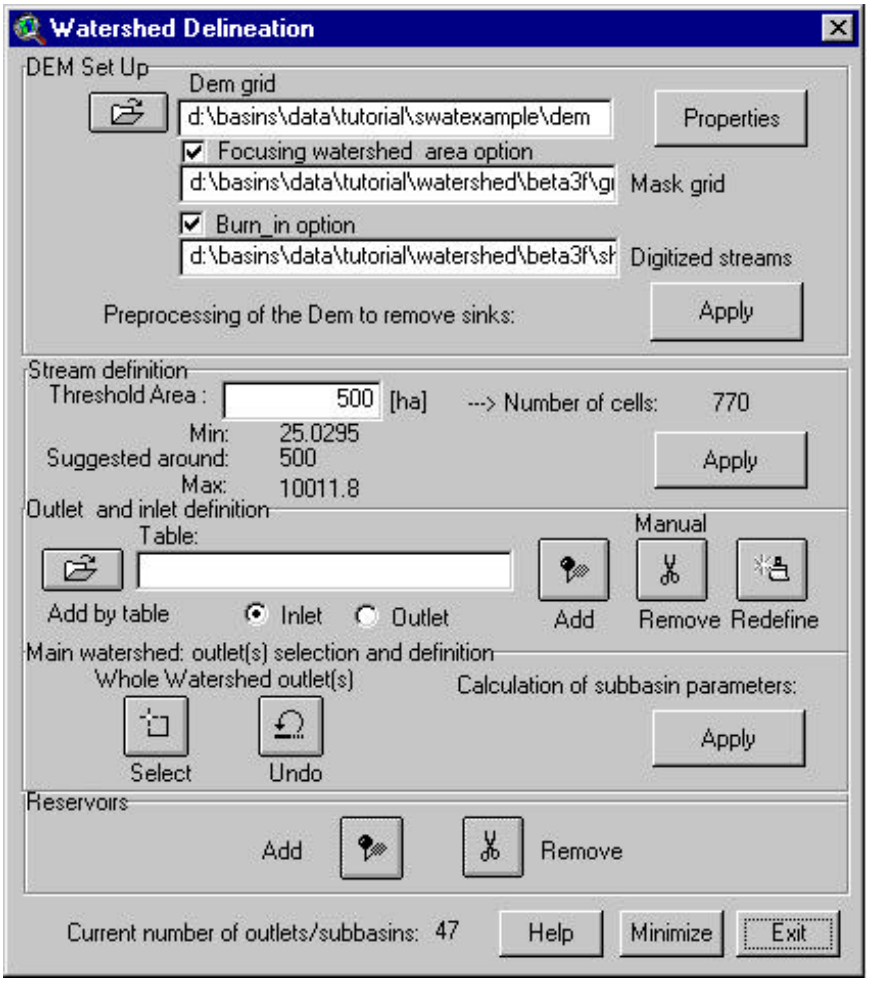


Figure 5.1

The dialog is divided into five sections: DEM Setup, Stream Definition, Outlet and Inlet Definition, Main Watershed Outlet(s) Selection and Definition, and Reservoirs.

SECTION 5.2: DEM SETUP

1. The DEM Setup section is shown in Figure 5.2.

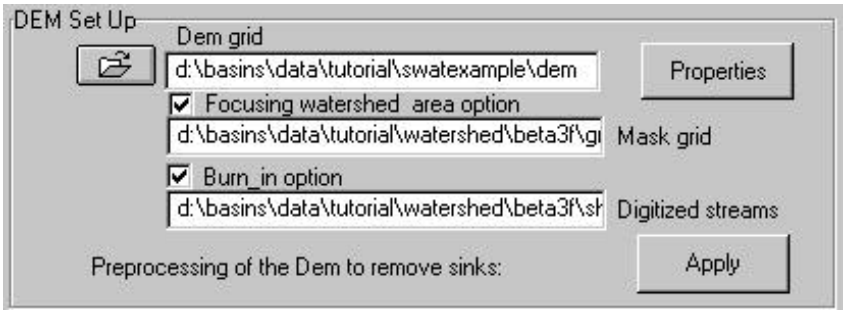


Figure 5.2

One button loads the DEM grid map used to calculate all subbasin/reach topographic parameters. Two check boxes (options) load or create a mask grid and/or load a stream shape file.

2. To load or select the DEM grid, click the button beside the text box labeled "DEM grid"

3. A dialog box is opened to specify which DEM map grid to use (Figure 5.3).

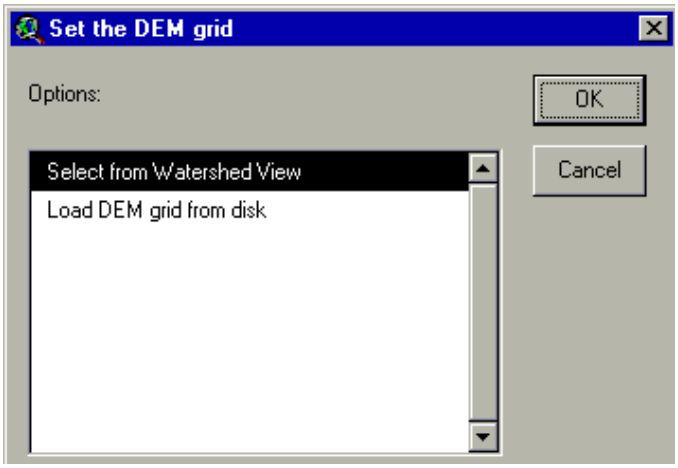


Figure 5.3

You may select a DEM grid that has previously been added to the *Watershed View* or load a DEM.

Click **OK** after the selection. If the first option was selected, the list of the grid themes in the *Watershed View* is shown (Figure 5.4); otherwise a grid dataset file browser (Figure 5.5) will appear to allow you to specify which DEM will be used.

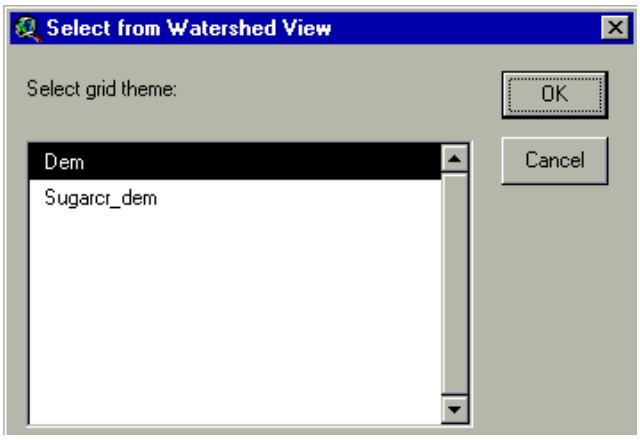


Figure 5.4

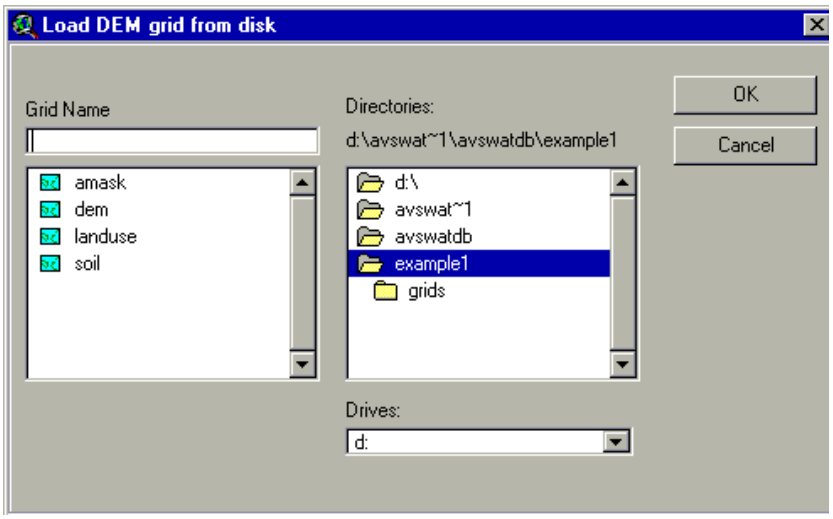


Figure 5.5

Select the name of the DEM map grid and Click **OK** (or double click the selection.)

4. Once the DEM is loaded, the grid data set path will be shown in the text box labeled “DEM Grid”. A prompt box will be displayed reminding the user to verify DEM properties (Figure 5.6)

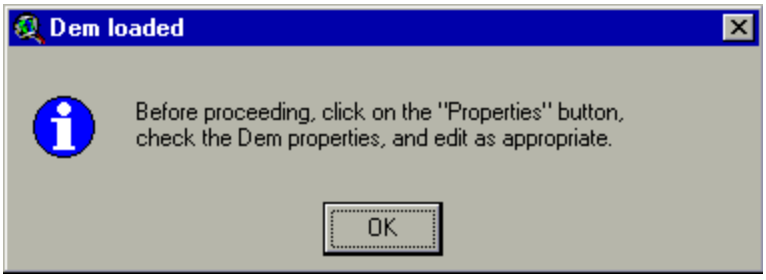


Figure 5.6

5. Click the properties button

next to the DEM grid text box.

   The DEM properties dialog box allows DEM vertical and horizontal units of measure and the projection to be verified or edited (Figure 5.7).

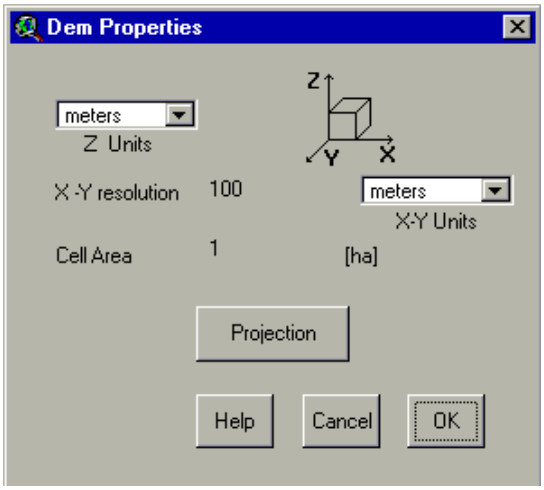
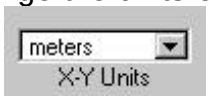
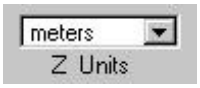


Figure 5.7

   To change the units of measure, click on the arrows to display drop-down boxes(

or 

) and select one of the listed units.

Note:	Careful!! The DEM properties dialog should correctly report the horizontal and vertical units. Incorrect settings will affect the results of the watershed geomorphic parameterization.

6. To verify or edit the projection, click the projection button

.

Note:	The projection does not have to be defined in order to use the interface. If the projection is not defined: • All location tables must use XPR and YPR coordinates to define the location of the objects of interest (Latitude and Longitude cannot be used to denote the locations) • The US weather generator database cannot be used • The PHU values cannot be calculated for each individual plant in the watershed by the interface.

   A prompt box will appear listing projection details for the DEM map. When a new DEM map is loaded, the projection will be undefined (Figure 5.8).

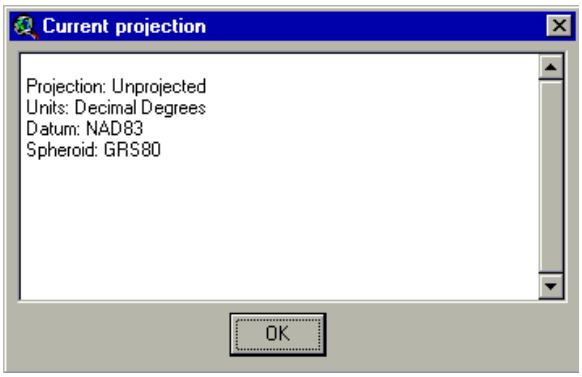


Figure 5.8

Click **OK**. A prompt box will appear (Figure 5.9)

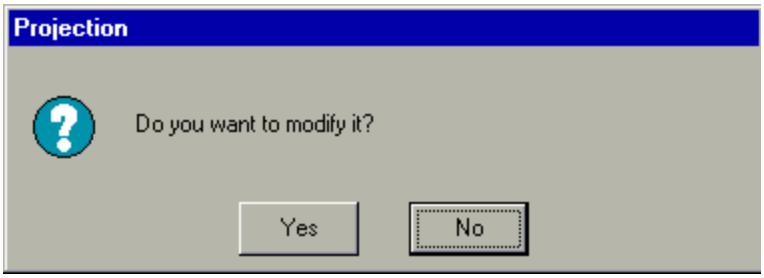


Figure 5.9

If the projection listed was correct, click **No**. To change the projection click **Yes**.

7. If **Yes** was clicked, a prompt box is displayed offering two options: **predefined projection** and **custom projection** (Figure 5.10). Click either **predefined projection** or **custom projection**.

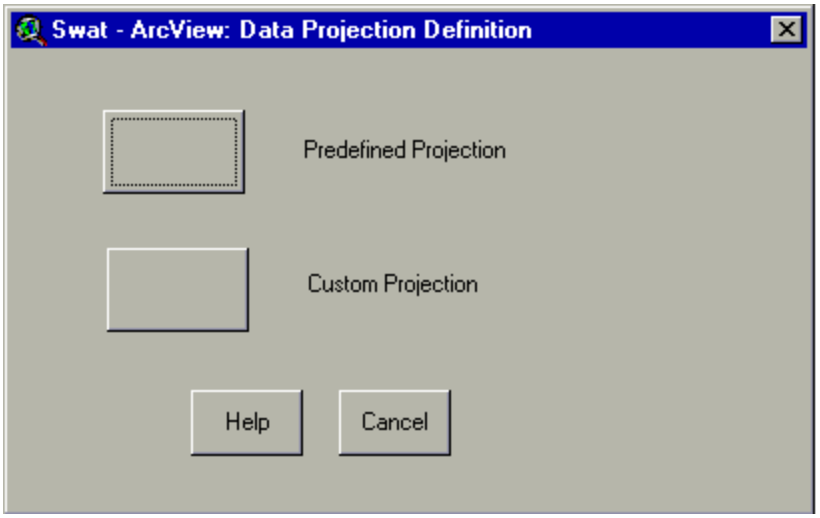


Figure 5.10

8. If **Predefined projection** was chosen, the dialog box in Figure 5.11 will be displayed. Only three projections are predefined in the interface: Geographic (decimal degrees), Albers Equal Area (conterminous US), and Wisconsin TM. Click the project map projection. Once the proper projection is highlighted, click **OK**.

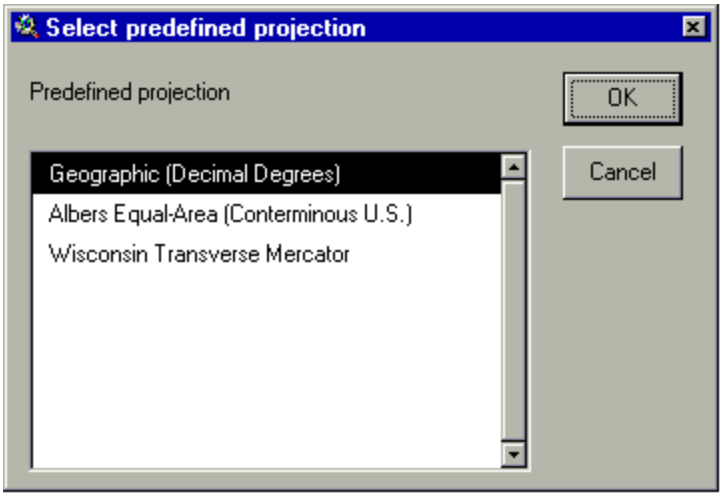


Figure 5.11

9. If **Custom projection** was chosen, the dialog box in Figure 5.12 will be displayed. Scroll the drop-down menus for Category and Type to find the project map projection and click the appropriate choices. Once the projection has been selected, standard settings for the projection will be displayed in the lower portion of the dialog box. If the projection settings need to be altered, click the circle beside **custom** and type the correct values for each of the projection properties (Figure 5.13).

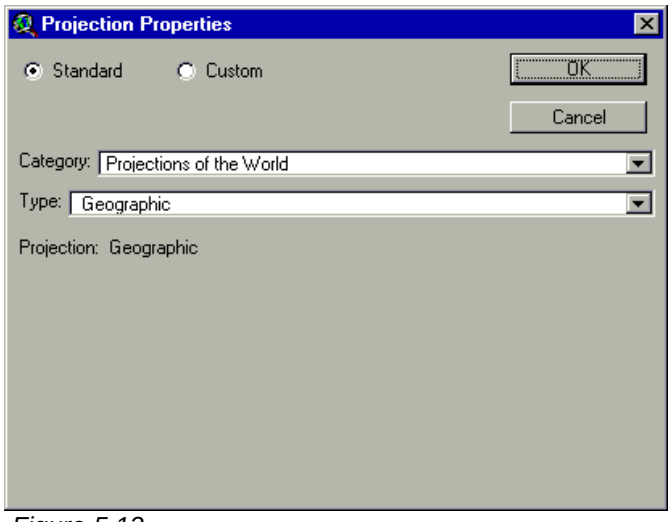


Figure 5.12

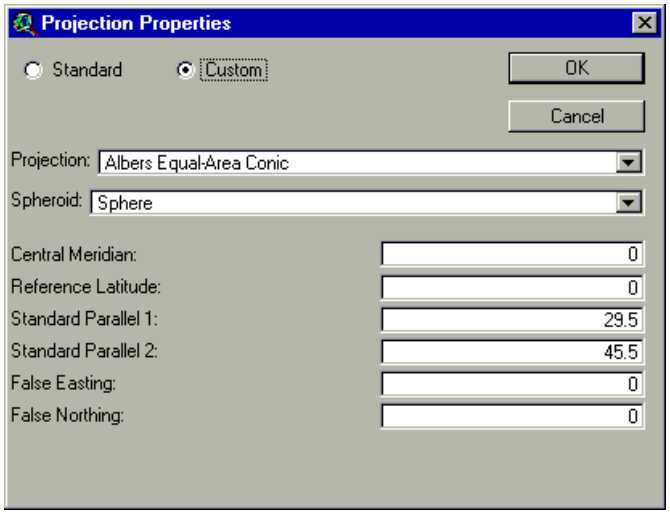


Figure 5.13

10. Once all projection properties have been set, click **OK**. This will close the projection dialog box.

11. Once the units of measure and projection are properly set, click **OK** on the properties dialog box.

12. Once the DEM properties are set, the **Apply** button (labeled "Preprocessing of the DEM to remove sinks" will be enabled. The following two options need to be set before proceeding with the preprocessing of the DEM.

13. **Focus watershed area** ***optional***

    The first option in the DEM setup section allows you to import or create a grid map that masks out a part of the DEM grid. Only the portion of the DEM covered by the mask will be processed by the interface. This map is not required but will reduce the processing time of the GIS functions.

    Click the box beside **Focusing watershed area option**. A check will appear in the box ☑ Focusing watershed area option and a prompt dialog will open (Figure 5.14)

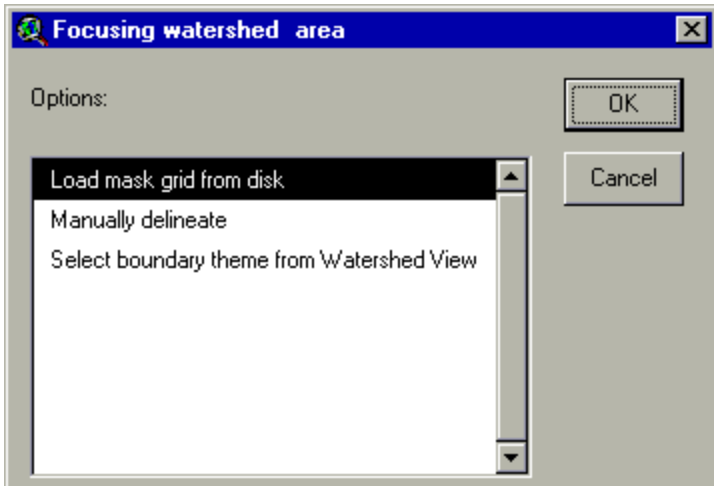


Figure 5.14

The user has three options for masking an area of the DEM map. To activate one option, highlight the option and click **OK** or double-click the selection.

a The first option, **Load mask grid from disk**, allows the user to import a grid map from a disk drive. If this option is selected, a grid data set browser is opened (Figure 5.15).

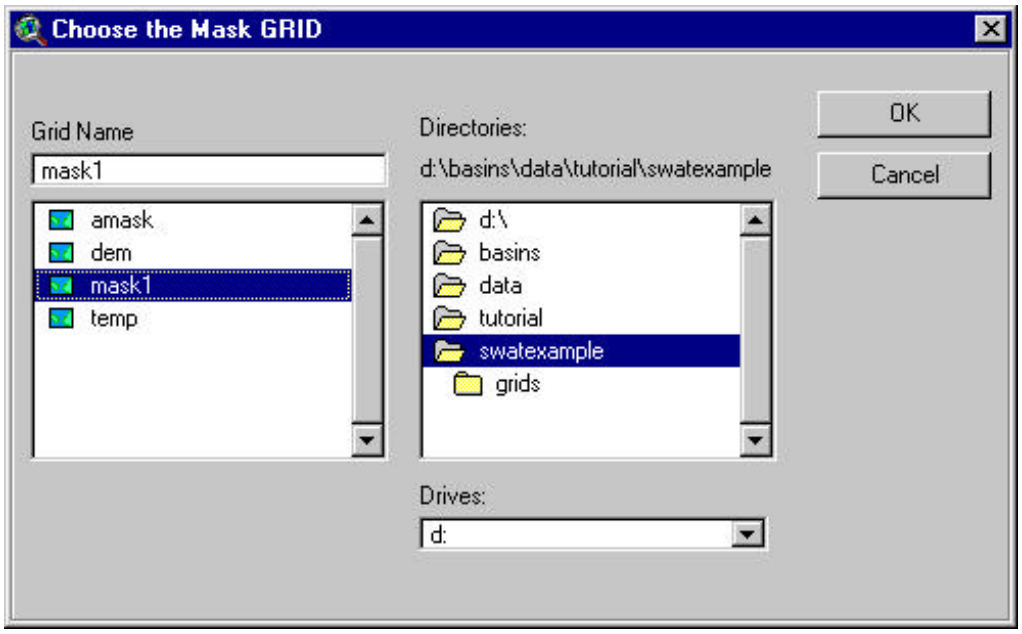


Figure 5.15

Select the name of the mask grid and click **OK** (or double click the selection). If a mask was set previously, a prompt box will appear asking if the new mask should be merged with the existing one (Figure 5.16).

Figure 5.16

Click **Yes** to merge the new and previous mask grids.

Click **No** to skip merging.

b The second option, **Manually delineate**, allows the user to draw and edit a polygon mask using the manual delineation tool. (Figure 5.17)

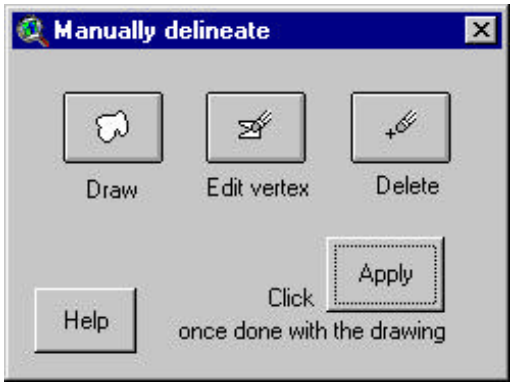


Figure 5.17

If a mask grid was previously defined, a prompt box will appear asking the user if the previous mask will be edited (Figure 5.18).

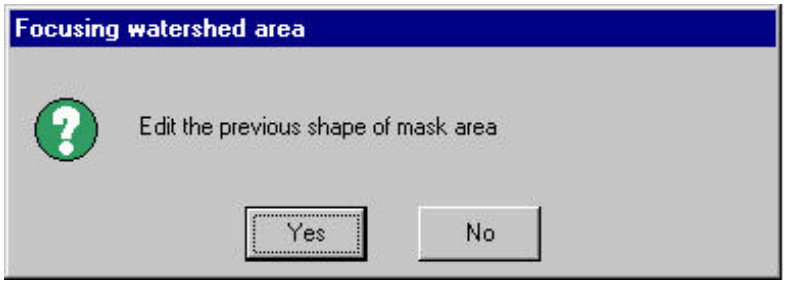
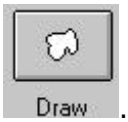


Figure 5.18

Click **Yes** to edit the previous mask. Otherwise click **No**.

While delineating the mask, the standard ArcView zoom-in and zoom-out tools can be used without closing the dialog.

Click the draw button 

.

The cursor appears as a loop. Click on the map to begin defining the polygon boundary. Click at each corner boundary, or vertex, of the polygon. Double-click the final vertex. The outline of the polygon will be displayed (Figure 5.19)

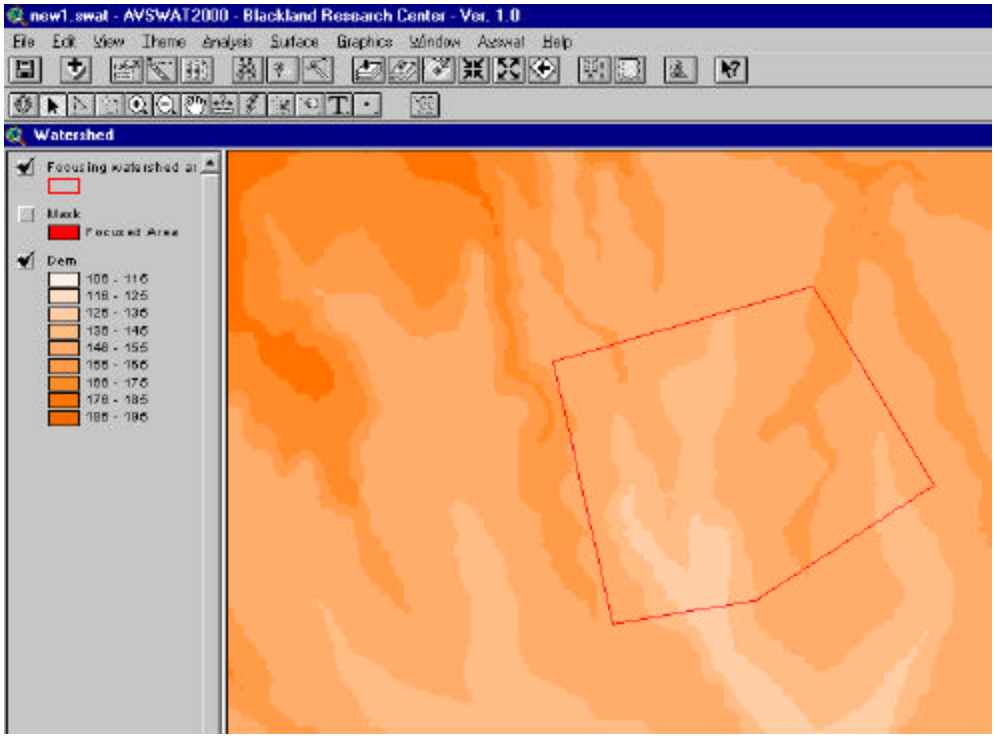
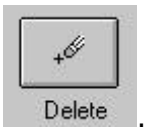


Figure 5.19

Repeat the same procedure for as many polygons as are needed. The polygons are displayed as the Focusing Watershed Area theme.

To add a new vertex to a polygon or move a vertex, click the **Edit Vertex** button

. Move the cursor to the position on the line where the new vertex will be located. When the cursor symbol changes to a target, click. To move a vertex, place the cursor on the vertex to be moved. When the cursor symbol changes to a crosshair, hold down the left mouse button and drag the vertex to the new position.

To delete a polygon, click the **Delete** button 

. Click on the polygon to be deleted. When the polygon is selected (Figure 5-20), press the `Delete` key on the keyboard and the polygon will be removed.

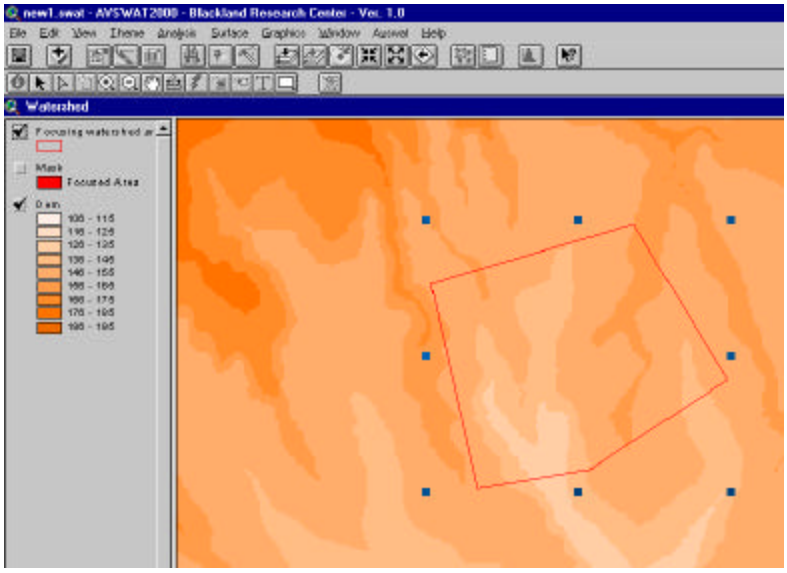


Figure 5.20

Once all drawing and editing on the grid mask is completed, click the **Apply** button.

c The third option, **Select boundary theme from Watershed View**, allows a polygon shape theme already loaded in the Watershed View to be selected as the mask.

A prompt box appears with a list of all polygon shape themes loaded in the Watershed View (Figure 5-21).

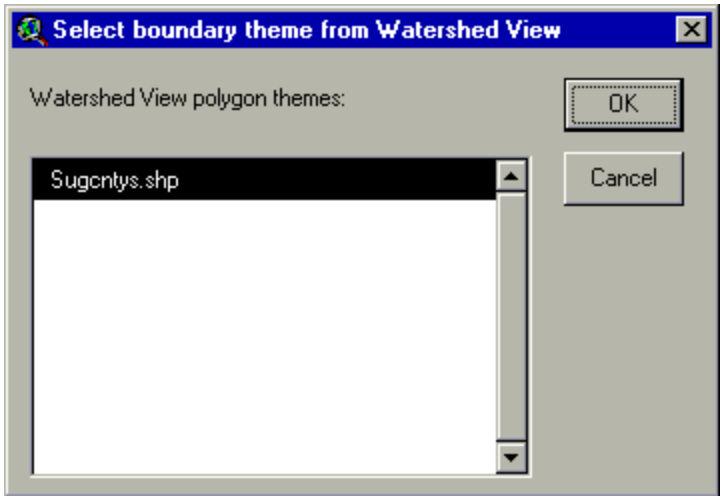


Figure 5.21

Select the name of the mask theme and click **OK**. If some features are selected on the theme, only these features will be converted: on the prompt dialog click **Yes** to continue just with the selected features or click **No** to abort. If a mask was set previously, a prompt box will appear asking if the new mask should be merged with the existing one.

d Once the mask grid is loaded, the grid data set path will be shown in the text box labeled “Mask Grid” in the Watershed Delineation dialog box, and a theme labeled “Mask-Focused Area” will be added to the Watershed View.

Note:	The Analysis Mask of Spatial Analyst Properties is now set. Applications of Spatial Analyst commands will be limited to the mask zone.

14. **Burn in a stream network** ***optional***

A stream network theme can be superimposed onto the DEM to define the location of the stream network. This feature is most useful in situations where the DEM does not provide enough detail to allow the interface to accurately predict the location of the stream network. Burning in a stream network theme improves hydrographic segmentation and sub-watershed boundary delineation. The theme must be a polyline shape file.

Click the box beside **Burn_in option**. A check will appear in the box ☑ Burn_in option and a prompt dialog will open (Figure 5.22)

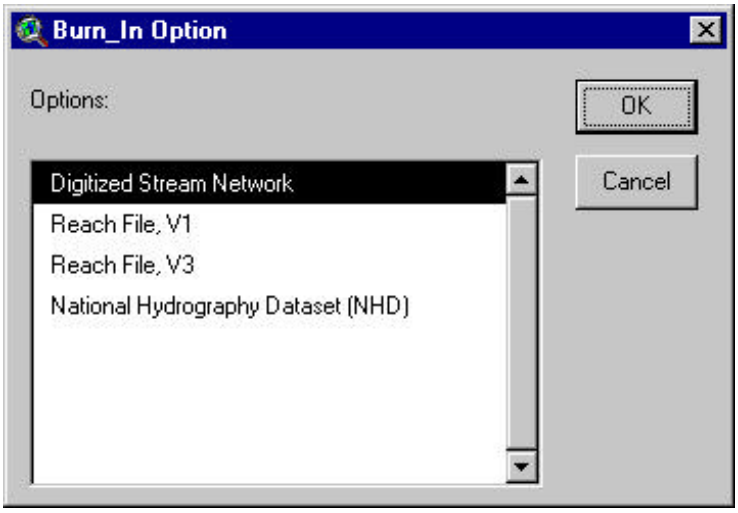


Figure 5.22

Four options are available to specify the stream network: Digitized Stream Network, Reach File V1, Reach File V3, and National Hydrography Dataset (NHD).

Digitized Stream Network: This option is selected when the user is supplying the stream network. The interface assumes the stream network delineated in the shape file contains no errors.

Reach File V1 is the oldest set of U.S. stream network delineations, Reach File V3 is the second generation set of U.S. stream network delineations, and NHD is the latest set of U.S. stream network delineations. The Reach File V3 streams data contains many spurious features (e.g. wide rivers, lakes, etc.). When stream delineations are imported from these datasets, the user should define the source. The interface will “clean” the stream delineation. NHD stream delineations are available for most of the U.S. from U.S. Geological Survey. This dataset is of much higher quality than V1 and V3, but it still needs to be filtered to retain only the lines of flowing bodies of water. When the NHD option is selected, the user needs to obtain the flow direction table (in dBase format). The name of this table must be the same as the NHD shape postfixed by “rflow”. Example NHD files may be viewed in the *installation dir*/avswatdb/example2/nhd directory.

Click **OK** after the selection has been made (or double click the selection).

A prompt box will open, and the user will select one of two options to load the stream delineation (Figure 5.23)

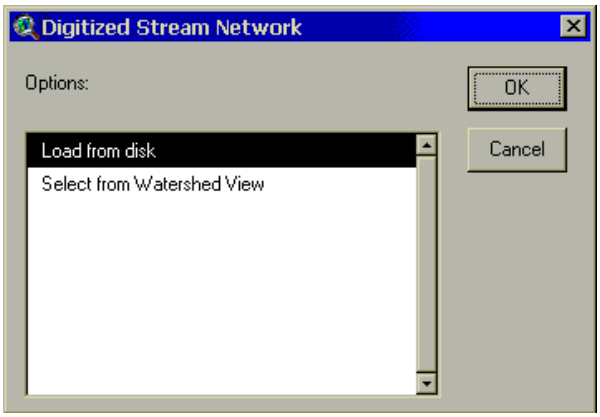


Figure 5.23

You may select a polyline theme that has previously been added to the *Watershed View* or load a polyline theme.

Click **OK** after the selection. If the first option was selected, the list of the polyline themes in the *Watershed View* is shown (Figure 5.24); otherwise a polyline dataset file browser (Figure 5.25) will appear to allow you to specify which shape file will be used.

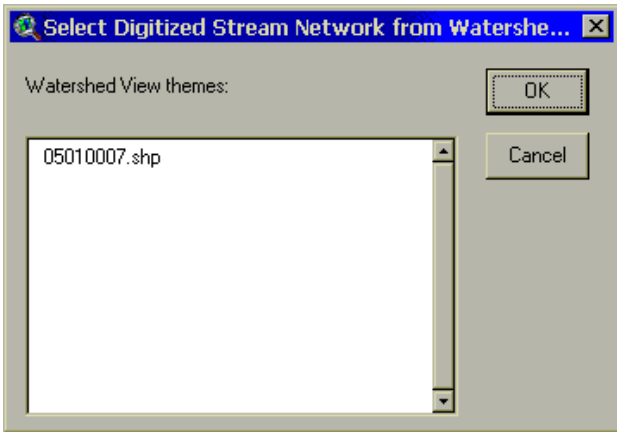


Figure 5.24

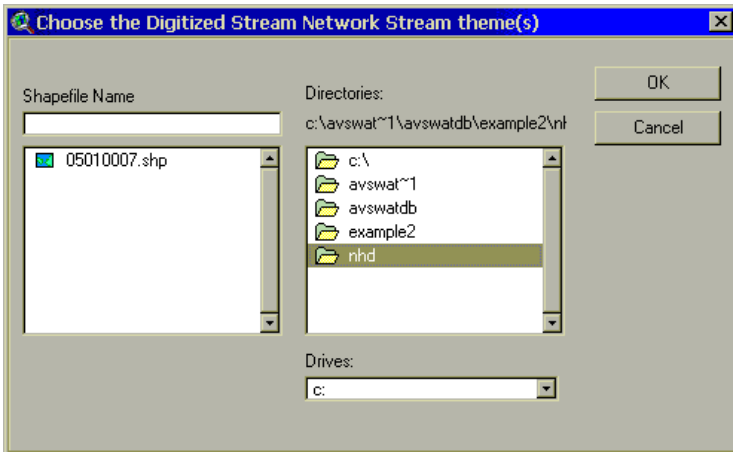


Figure 5.25

Select the name of the stream network theme (hold the `Shift` key for multiple selection) and click **OK**.

If any of the selected shape files are not polylines, a dialog box will report the error (Figure 5.26) and the process will be stopped.

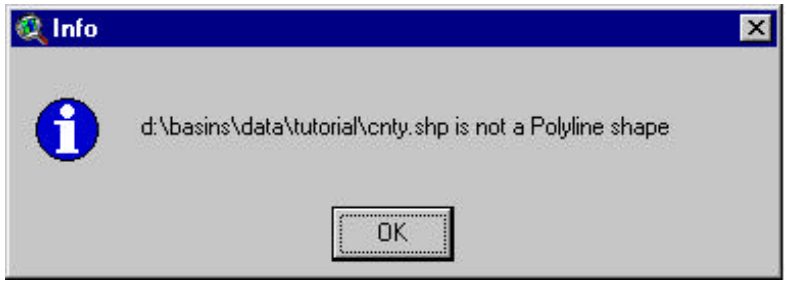


Figure 5.26

For Reach File V3 or NHD options, if the wrong data set is selected, a dialog box will display an error message noting that the fields for the shape file are not correct.

For the NHD option, if the NHD flow direction table is not located in the same directory as the shape file, a dialog box will display an error message.

Once the stream network is loaded, the shape file path will be shown in the text box labeled **Digitized streams**. The selected theme(s) will be displayed in the Watershed View as a single theme named “Digitized streams” (multiple selected data sets will be merged).

The filtered features (if any) from the selected themes are merged into a theme named “Removed streams” that is added to the Watershed View. A dialog box is displayed to indicate this to the user (Figure 5.27)

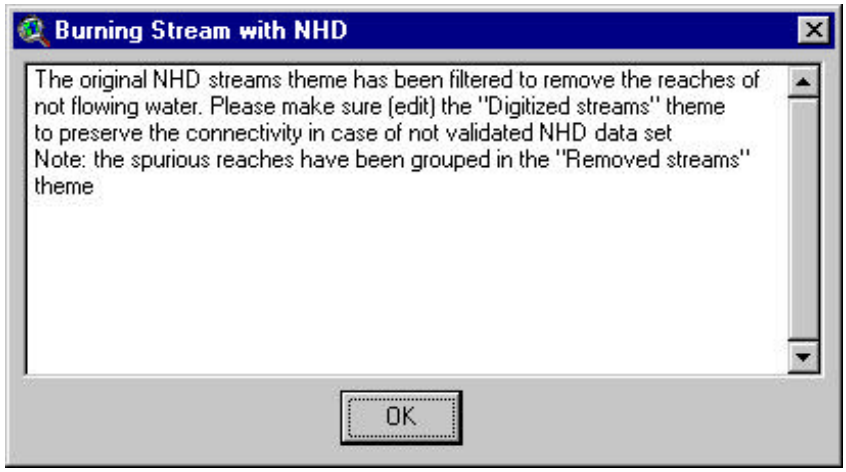


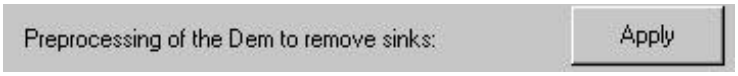
Figure 5.27

Tip: After filtering and before proceeding to the next preprocessing step, you should edit the “Digitized stream” theme to provide a continuous set of stream lines (e.g. draw lines through lakes and ponds, remove isolated reaches). For this task, you can start an editing section of the “Digitized stream” theme and use the ArcView default tools in the Watershed View .

Note: Other than outlet lines, the stream lines should not cross the edge of the DEM (or the Focused Area if a mask was set). Lines crossing the edge can affect the resulting flow direction.

15. **DEM Preprocessing**

Once the DEM grid is loaded, the properties have been reviewed, and optional themes are set, click the **Apply** button next to the label “Preprocessing of the DEM to remove sinks”.



The interface will process the DEM map grid to remove all of the non-draining zones (sinks).

While processing the DEM, a prompt box will appear (Figure 5.28).

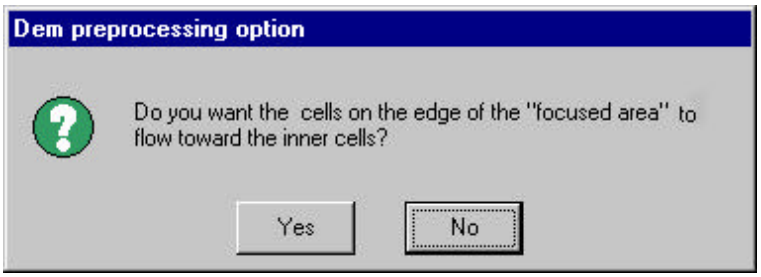


Figure 5.28

Click **No** if all cells on the edge of the working zone need to flow away from the zone. This is the recommended choice.

Click **Yes** if the flow is calculated normally for edge cells, with the edge being slightly lower than the cell.

A prompt box is displayed when map preprocessing is complete (Figure 5.29). Click **OK**.

Figure 5.29

SECTION 5.3: STREAM DEFINITION

In this section of the Watershed Delineation dialog box, the initial stream network and subbasin outlets are defined. The interface lists a minimum, maximum, and suggested sub-watershed area (in hectares), which are shown in the stream definition section (Figure 5.30)

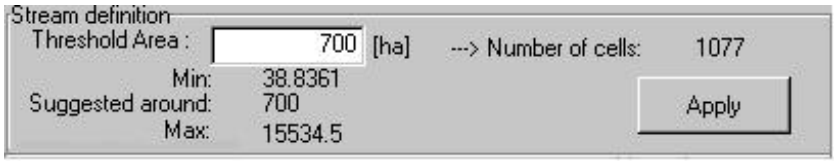


Figure 5.30

The user has the ability to set the minimum size of the subbasins. This function plays an important role in determining the detail of the stream network and the size and number of sub-watersheds. The threshold area, or critical source area, defines the minimum drainage area required to form the origin of a stream.

1. In the text box labeled “Threshold area”, type the upstream drainage area (in hectares) required to define the beginning of a stream. The smaller the specified number of hectares, the more detailed the drainage network delineated by the interface.
2. Click the **Apply** button.
3. Two themes are now added to the Watershed View and displayed over the DEM map grid: Streams (the current synthetic drainage network) and Outlets (the respective stream junction points) (Figure 5.31).

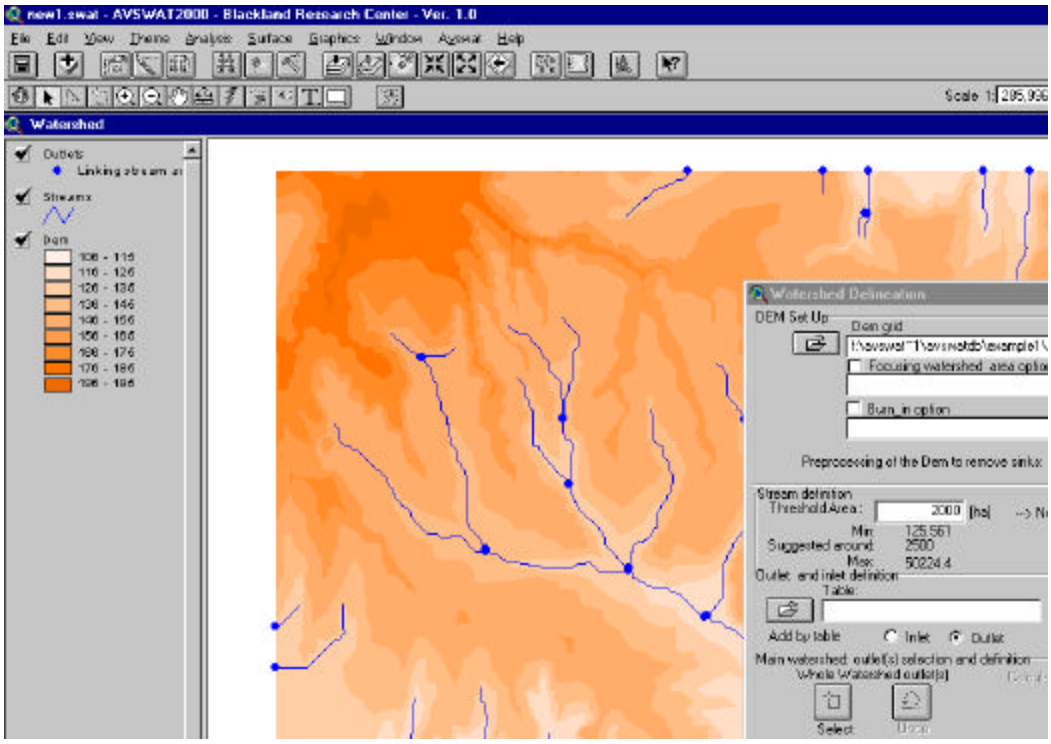


Figure 5.31

4. The user can change the threshold value and re-run the stream and outlet definition routine or proceed with the next section.

SECTION 5.4: OUTLET AND INLET DEFINITION

In this section of the Watershed Delineation dialog box, the user may refine the stream network and outlet configuration. Drainage inlets and sub-watershed outlets may be added, deleted, or redefined (Figure 5.32)

Figure 5.32

Sub-watershed outlets are the points in the drainage network of a sub-watershed where streamflow exits the sub-watershed area. Adding outlets at the location of monitoring stations is useful for comparison of measured and predicted flows and concentrations.

There are two types of drainage inlets: a point-source discharge or the outlet of a draining watershed. The second type of inlet is used when a portion of the watershed area is not directly modeled with SWAT. For both types of

inlets, the user provides discharge data records. The inlet discharge is routed through the stream network.

Inlets and outlets may be added to the stream network by importing a predefined table or manually clicking the mouse over the map on the screen. Two radio buttons allow you to switch the current definition between inlets and outlets

.

The legend for the Outlets theme in the Watershed View (Figure 5.33) distinguishes the types of inlets/outlets added to the view.

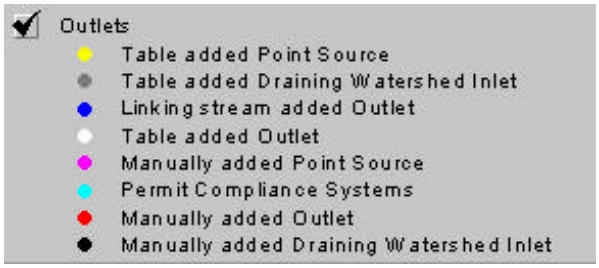


Figure 5.33

The following sections described the different methods used to add inlets and outlets.

Adding Outlets by Table

Outlet point locations (subbasin outlets) can be imported in the project using a dBASE table and the following steps:

1. Make sure the radio button labeled "Outlet" is selected

.
2. Click on the

button next to the "Table" text box. A file browser will appear (Figure 5.34) allowing you to select a dBASE table. Select the file name and click **OK** (or double click the selection).

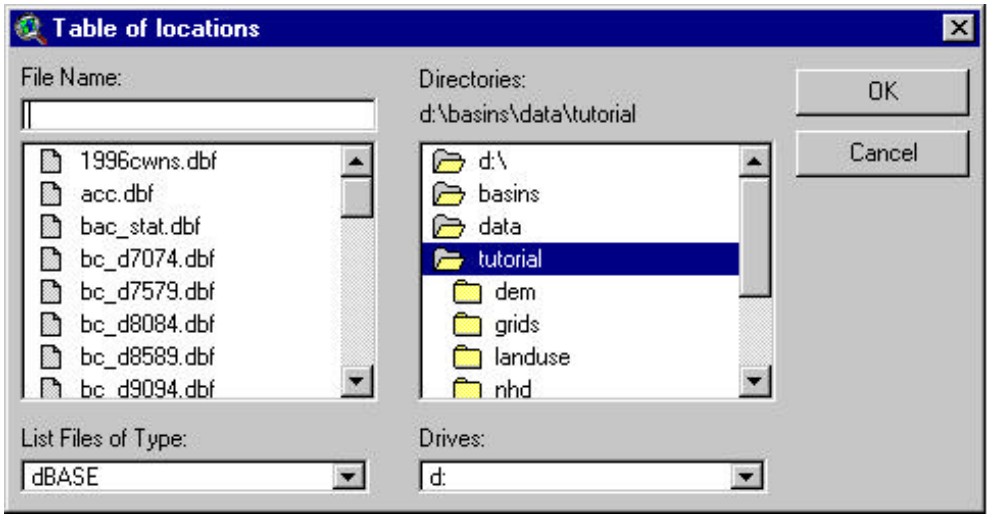


Figure 5.34

3. This table must have the same fields specified in Section 3.3 for Subbasin Outlet Location Table. All locations listed in the table must be outlet Type “O”. If a different "Type" value is specified, a dialog box will report an error like the one reported in Figure 5.35 and the loading process will stop.

Figure 5.35

4. Once geocoded, the outlet locations will snap automatically to the closest reach of the Streams theme.

Note:	Xpr and Ypr field data values have priority over the Lat and Long field data value for the definition of the point location on the map.

5. The DEM dialog box will automatically minimize, and the Watershed View will be redrawn showing the added points.

.

Adding Point Sources or Inlets of Draining Watersheds from Tables

Inlet point locations can be imported in the project using a dBASE table and the following steps:

1. Make sure the "Inlet" radio button is selected

.
2. In the Outlet Definition section of the DEM dialog box, click on the

button next to the "Table" text box. A dialog box will appear (Figure 5.36)

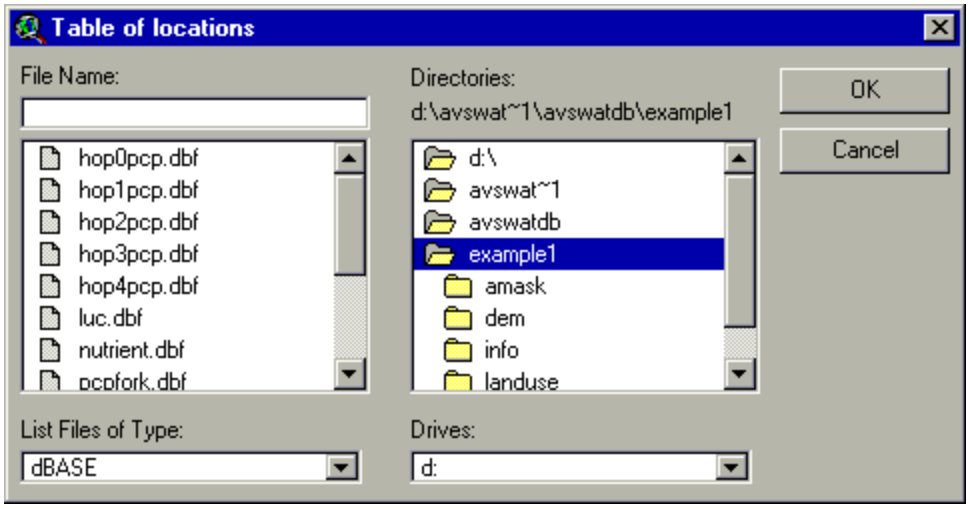


Figure 5.36

3. This table must have the same fields specified in Section 3.3 for Watershed Inlet Location Table. All locations listed in the table must be Type “D” (Point Sources) or Type “I” (Draining Watershed Inlets). If a different "Type" value is specified, a dialog box will report an error like the one reported in Figure 5.35 and the loading process will stop.
4. The DEM dialog box will automatically minimize, and the Watershed View will be redrawn showing the added points.

Manually Editing Outlets and Inlets

Outlets and inlets may be manually edited using the following steps.

Adding Outlets

1. In the Outlet Definition section of the DEM dialog box, make sure the "Outlet" radio button is selected

.

2. Click the

button.
3. The Watershed Delineation dialog will be minimized. Move the cursor to the desired location(s) and click the left mouse button. An outlet point will automatically snap to the closest stream line, and the feature will be added to the "Outlets" theme.
4. When all desired outlets have been added, maximize the dialog box.

Adding Inlets

1. In the Outlet Definition section of the DEM dialog box, make sure the Inlet radio button is selected

.
2. Click the

button.
3. The Watershed Delineation dialog will be minimized. Move the cursor to the desired location(s) and click the left mouse button. A dialog box will open (Figure 5.37) providing two options: *Point discharge* and *Inlet of Draining Watershed*.

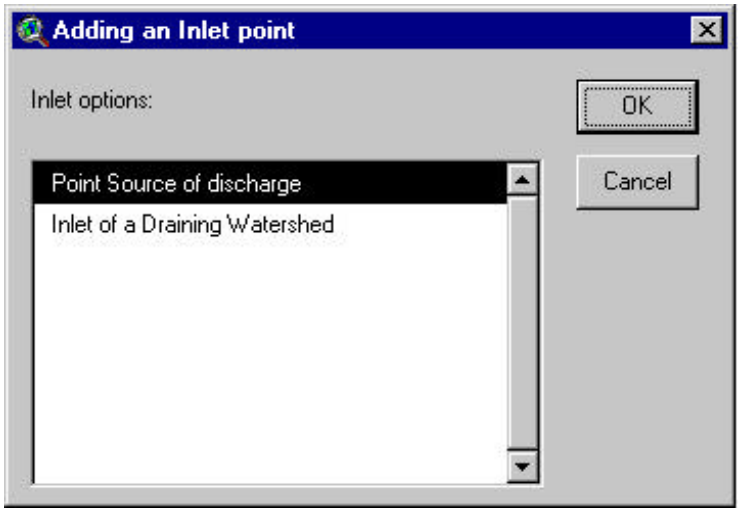


Figure 5.37

4. Select the inlet type. The inlet point will automatically snap to the closest stream line, and the feature will be added to the "Outlets" theme.
5. When all desired inlets have been added, maximize the dialog box.

When adding and deleting points:

Do not insert an inlet or outlet point in a junction cell. A close-up view of stream juncture points created by the interface is shown on the DEM map grid in Figure 5.38. The points are placed in the first cell of each branch of the stream. If these points are removed and replaced with one point in the junction cell (Figure 5.39), the interface will not be able to understand which branch of the stream is the correct stream line and will be unable to delineate the subbasins properly for the two stream branches.

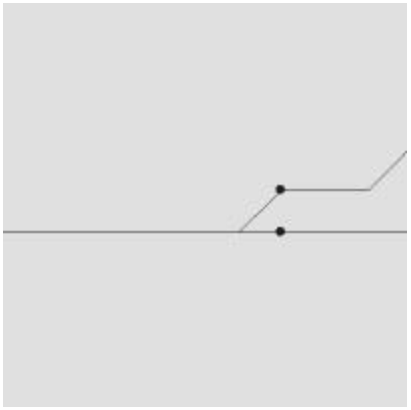


Figure 5.38

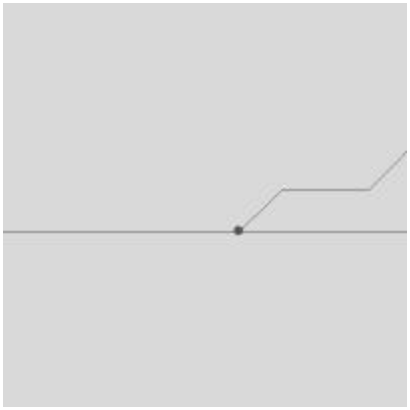


Figure 5.39

Deleting Outlet and/or Inlet Points

1. If necessary, zoom in on the inlet or outlet points to be deleted.
2. Click the

button.
3. The Watershed Delineation dialog will be minimized. Move the cursor to the location(s) you wish to remove.
4. Hold down the left mouse button, and move the mouse to draw a box around the point(s) you wish to remove. Release the left mouse button.
5. A prompt box will appear, asking you to verify removal of the selected point(s).(Figure 5.40)

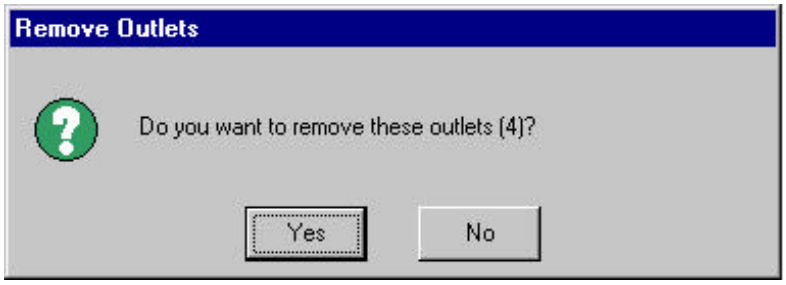


Figure 5.40

Redefining Outlets/Inlets

1. Click the

button.
2. The *Watershed Delineation* dialog will be minimized. Move the cursor to the desired location(s) and left click. Hold down the left mouse button, and draw a box around the point(s) you wish to redefine. Release the left mouse button. A prompt box will open (Figure 5.41). Click the **Cancel** button to exit.

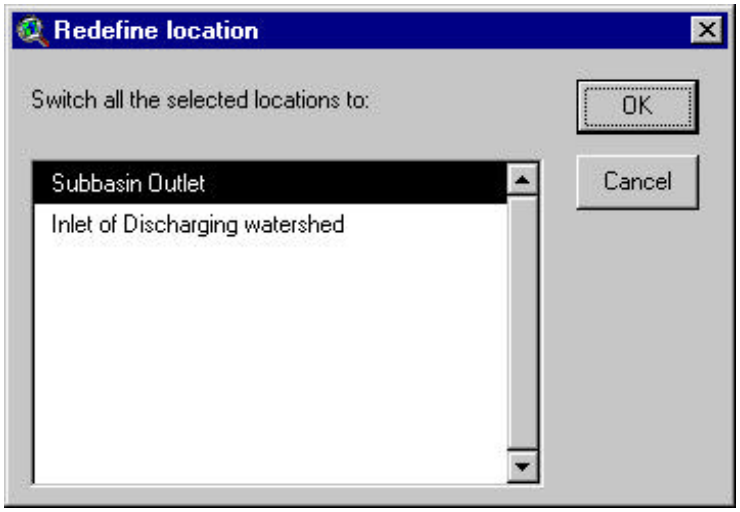


Figure 5.41

You may redefine one or more outlets to drainage watershed inlets and vice versa. Point Source inlets cannot be redefined. If one of these points is selected, a dialog box will report an error (Figure 5.42), and the process will stop.

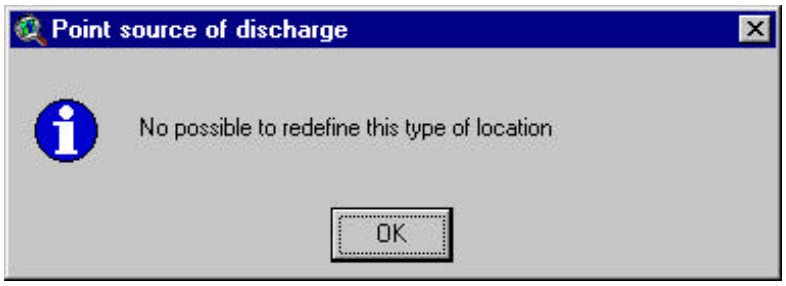


Figure 5.42

SECTION 5.5: MAIN WATERSHED OUTLET SELECTION AND DEFINITION

Sub-watershed delineation is completed in this section (Figure 5.43) of the Watershed Delineation dialog box.

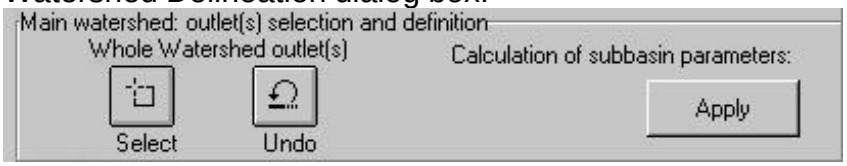


Figure 5.43

The interface will allow more than one watershed to be delineated at the same time. It is convenient to select just the most downstream outlet(s) of each target watershed(s).

Click the

button.

1. The *DEM dialog* will minimize and another dialog will appear (Figure 5.44).

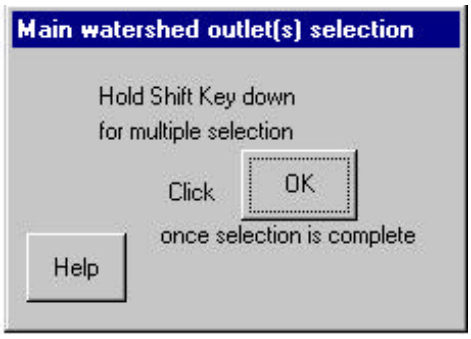


Figure 5.44

2. Press the ESC key to exit the selection.
3. To select one watershed outlet, position the cursor close to the point chosen to be the watershed outlet. Hold down the left mouse button and

move the mouse to form a box on the screen around the selected outlet. Release the left mouse button.

4. To select multiple watershed outlets that are not adjacent to one another, hold down the `Shift` key continuously while performing the actions described in Step 4 as many times as necessary to select all the watershed outlets of interest.

> **Tip:** The tool allows multiple main watershed outlets to be selected. This feature allows non-hydrologically connected watersheds to be simulated at the same time.

5. Once the outlet points are selected, click **OK** on the outlet selection prompt box.

   If a Point Discharge or Inlet of Draining Watershed is specified, a dialog box will report an error and processing will stop (Figure 5.45).

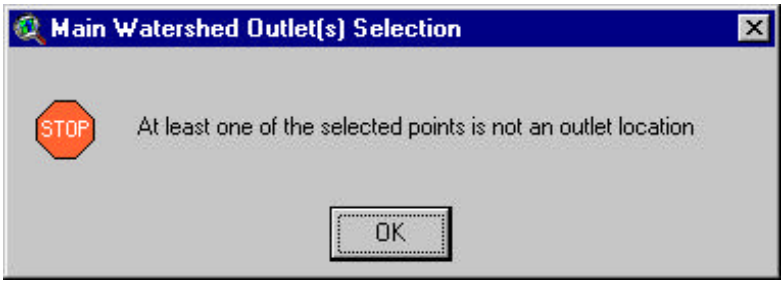


Figure 5.45

   Otherwise a confirmation prompt pops up (Figure 5.46).

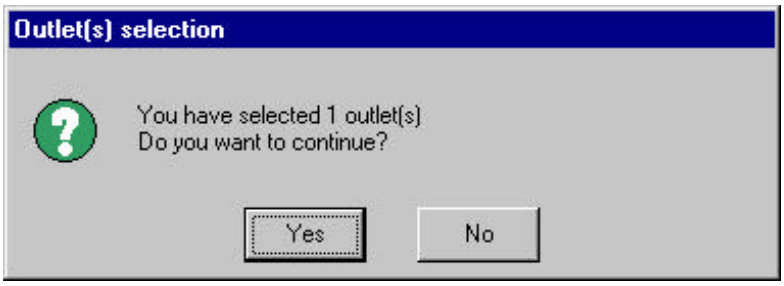


Figure 5.46

6. Click **Yes** to proceed.

Note: Repeated outlets (located in the same location) will be removed. In this case, a message dialog will pop up (Figure5.47).

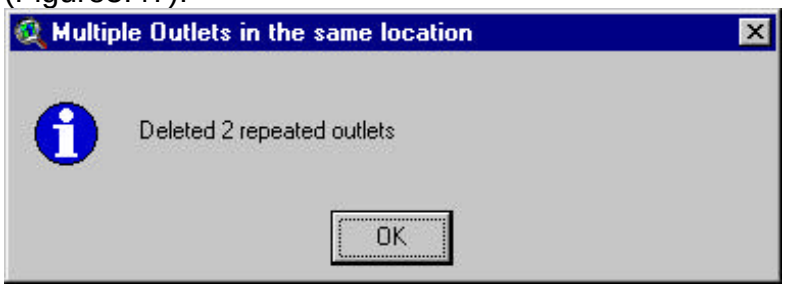


Figure 5.47

7. A map of the watershed (*Watershed* theme), sub-watersheds (*Subbasins* theme), and stream network restricted to the watershed (*Streams* theme) will appear on the screen when the interface has completed the watershed delineation (Figure 5.48).

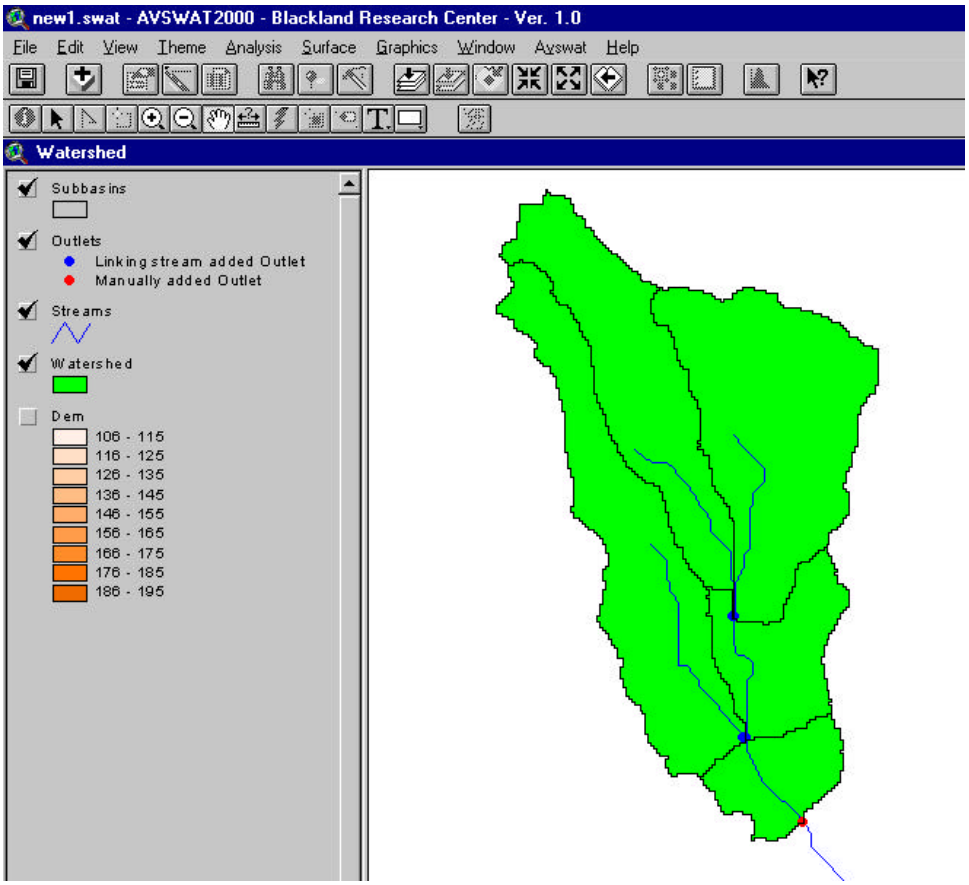


Figure 5.48

If one or more "Inlet of watershed" points are set on the stream network, the *Inlet Draining Watershed* theme is also added to the Watershed View (Figure 5.49).

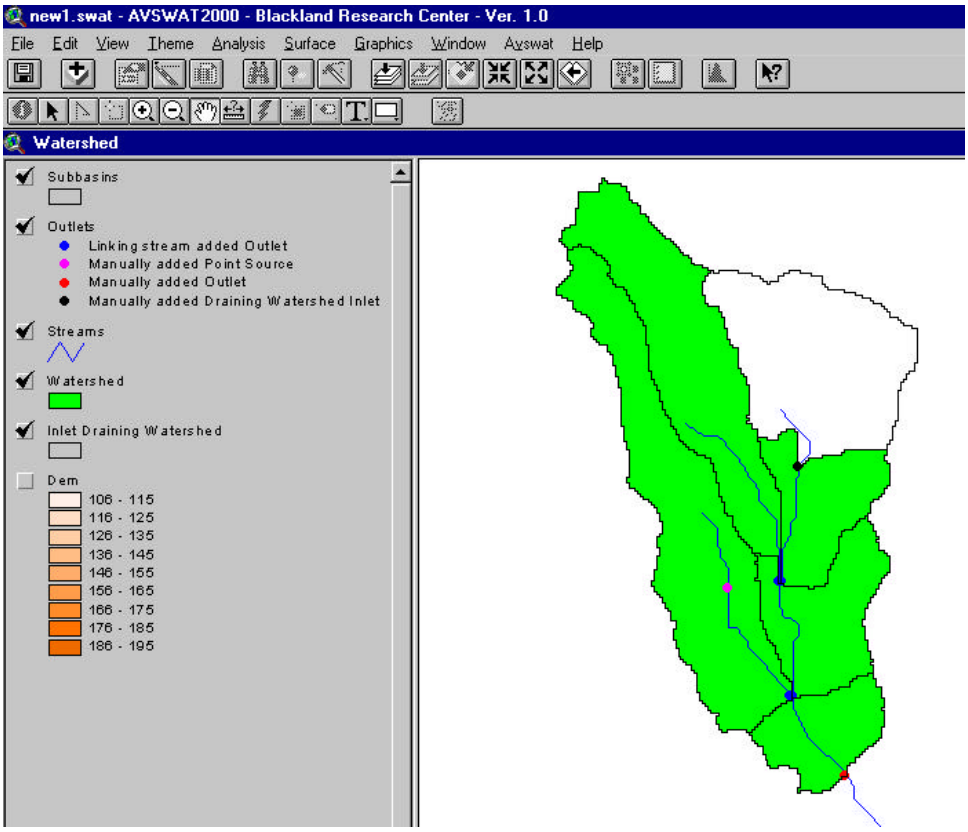


Figure 5.49

8. By clicking the

button, the user can step back and repeat steps 1-7. This action removes the watershed delineation and allows outlets and inlets to be edited.

   When the **Undo** button is clicked, a prompt box will appear asking the user to verify the undo operation (Figure 5.50)

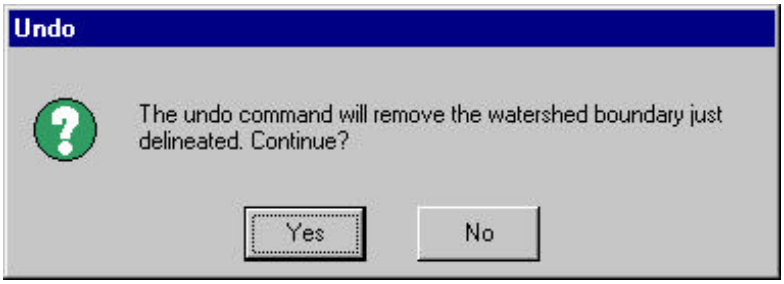


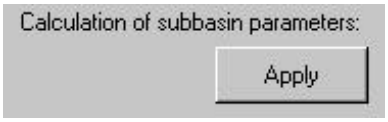
Figure 5.50

Click **Yes** to remove the watershed delineation.

Click **No** to continue with the current settings.

Calculation of Subbasin Parameters

This function calculates geomorphic parameters for each subbasin and the relative stream reach. The results of the calculations are stored in the table of attributes of the updated *Subbasin* and *Streams* themes. (See Appendix-Watershed Delineation Output Data for a description of the stored data.)

1. Click the 

button.
2. If the Projection for the watershed was not defined when the DEM was loaded, a message box will appear (Figure 5.51).

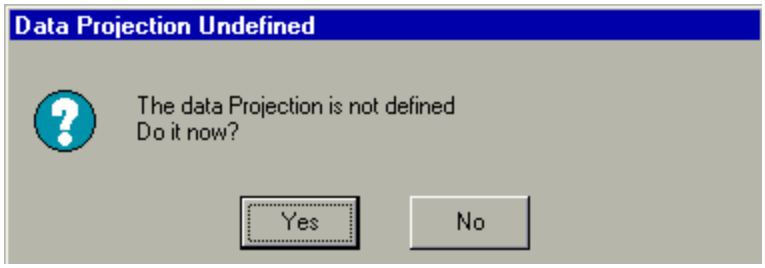


Figure 5.51

Click **Yes** to define the Projection or click **No** to leave the projection undefined.

a. If the projection is left undefined, another message box will appear (Figure 5.52).

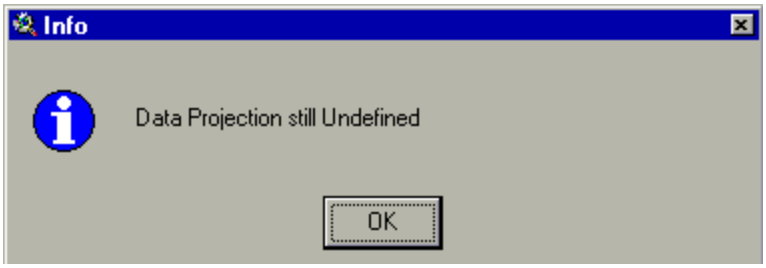


Figure 5.52

Click **OK**.

b. A prompt box is displayed (Figure 5.53).

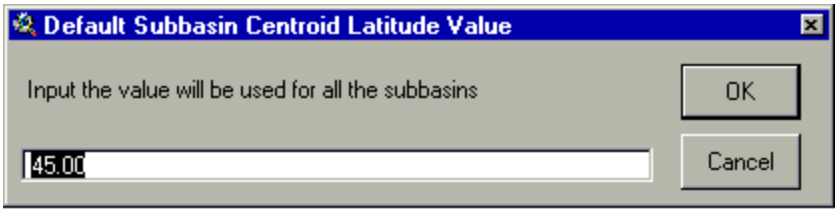


Figure 5.53

c. Type the latitude to be assigned to the subbasins and click **OK**.

3. When all parameters are calculated, a dialog box appears (Figure 5.54).

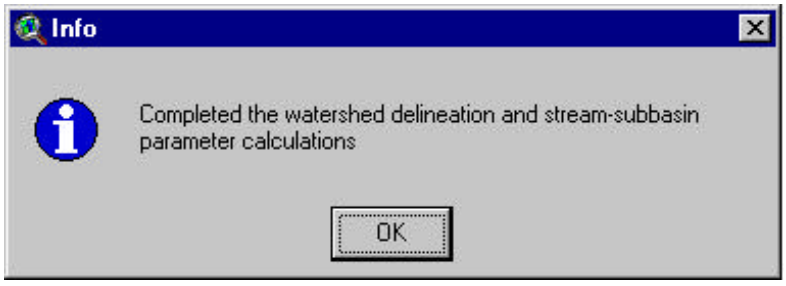


Figure 5.54

> **Note:** Each subbasin is coupled to a single stream reach. If the user removed any of the outlets defined by the interface during the initial analysis of the DEM, the main stem within the subbasin area is assumed to represent the single stream reach associated with the subbasin.

Each subbasin in the *Subbasins* theme is numbered, and the label is visible in the Watershed View.

The *Streams* theme is labeled as well if the burning option *Reach V3* or *NHD* has been used.

A new report, named *Topographic Report,* is now available (Figure 5.55). This report provides a statistical summary and distribution of discrete land surface elevations in the watershed and all the sub-watersheds.

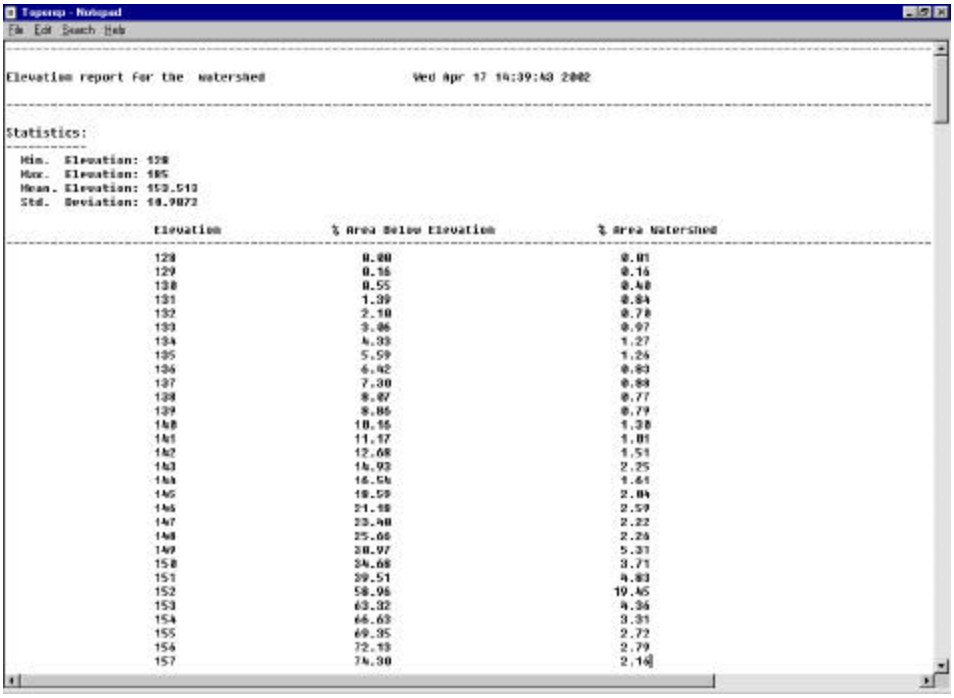


Figure 5.55

SECTION 5.6: RESERVOIRS—OPTIONAL

Once the delineation is complete, the user has the option of inserting/removing reservoir locations along the main channel network (Figure 5.56).

Figure 5.56

Adding a reservoir.

1. Click the button.

2. The dialog box will be minimized, and the cursor will become a crosshair. Click over the target subbasin area to add a reservoir.
3. A prompt box will be displayed asking for verification of the reservoir placement (Figure 5.57).

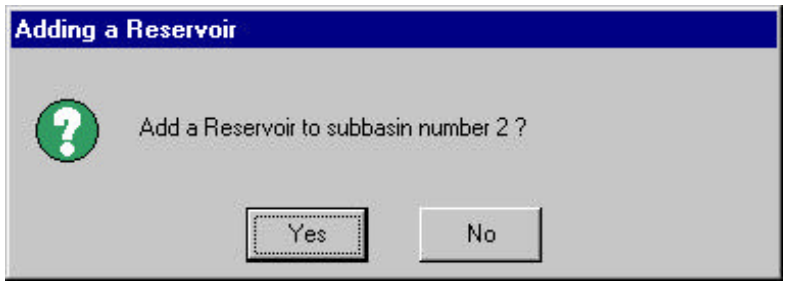


Figure 5.57

If the wrong subbasin is listed, click **No** and repeat step 2. If the correct subbasin is listed, click **Yes**. The new reservoir location will be placed at the outlet of the respective subbasin.

4. Once the first reservoir location is added, the new *Reservoirs* theme will be added to the Watershed View (Figure 5.58).

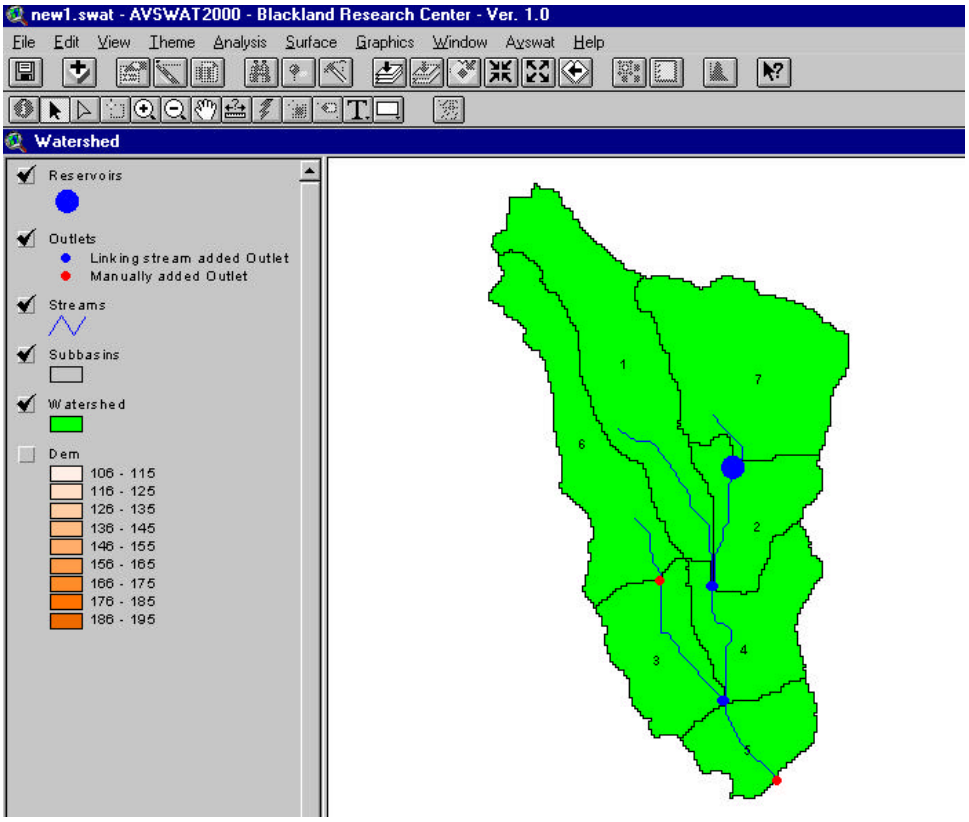


Figure 5.58

Tip: The user is allowed to add a single reservoir location for each subbasin. Refine the outlet set if more than one reservoir location needs to be set in the current subbasin area.

Removing a reservoir(s).

1. Click the

button.
2. The *Watershed Delineation* dialog box will be minimized and the mouse cursor will become a square. Draw a square around the reservoir(s) you wish to remove by holding down the left mouse button.
3. A prompt box will be displayed asking for verification of the reservoir(s) removal (Figure 5.59).

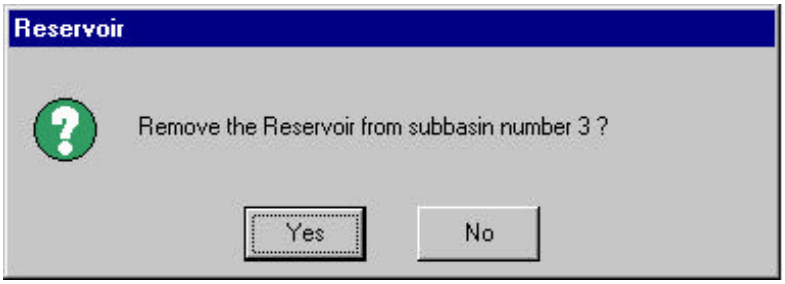


Figure 5.59

4. If the wrong subbasin is listed, click **No** and repeat step 2. If the correct subbasin is listed, click **Yes**.
5. If all the reservoir locations are removed, the *Reservoirs* theme will be removed from the Watershed View.

SECTION 6: LAND USE/SOIL CHARACTERIZATION

Land Use and Soil Characterization for a watershed is performed using two commands in the Avswat menu of the Watershed View. This tool allows users to load land use and soil themes into the current project and determine the land use/soil class combinations and distributions for the delineated watershed(s) and each respective sub-watershed. The themes can be either grid or shape format.

Once the land use and soil themes have been imported and linked to the SWAT databases, the user specifies criteria used in determining the HRU distribution. One or more unique land use/soil combinations (hydrologic response units or HRUs) can be created for each subbasin.

SECTION 6.1: LAND USE/SOIL DEFINITION AND OVERLAY

Purpose

The *Land Use and Soils Classification and Overlay* tool allows the user to load the land use and soil themes and determine land use/soil class combinations and distributions for the delineated watershed(s) and each respective sub-watershed. The themes can be either grid or shape theme(s). The land use and soil themes must be in the same projection as the DEM used in the watershed delineation.

Application

Hydrologic models like **SWAT** require land use and soil data to determine the area and the hydrologic parameters of each land-soil category simulated within each sub-watershed. The *Land Use and Soil Classification* tool guides the user through the process of specifying the data to be used.

Either shape or grid themes may be used. Shape files are automatically converted to grid, the format required by Spatial Analyst to compute cross-tabulated areas between land use and soil data sets.

Once the application is finished a detailed report is added to the current project. This report describes the land use and soil class distribution within the watershed and within each sub-watershed unit (subbasin).

Key Procedures

- Define the land use theme

- Reclassify the land use theme
- Define the soil theme
- Reclassify the soil theme
- Overlay land use and soil themes
- Click the Exit button

SECTION 6.1.1: GET STARTED

Initiate the *Landuse and Soil Classification and Overlay* tool by selecting **Land Use and Soil Definition** in the **Avswat** menu (Figure 6.1). (The **Avswat** menu is present only in the Watershed View.)

Figure 6.1

The *Definition of Landuse and Soil themes* dialog will open (Figure 6.2).

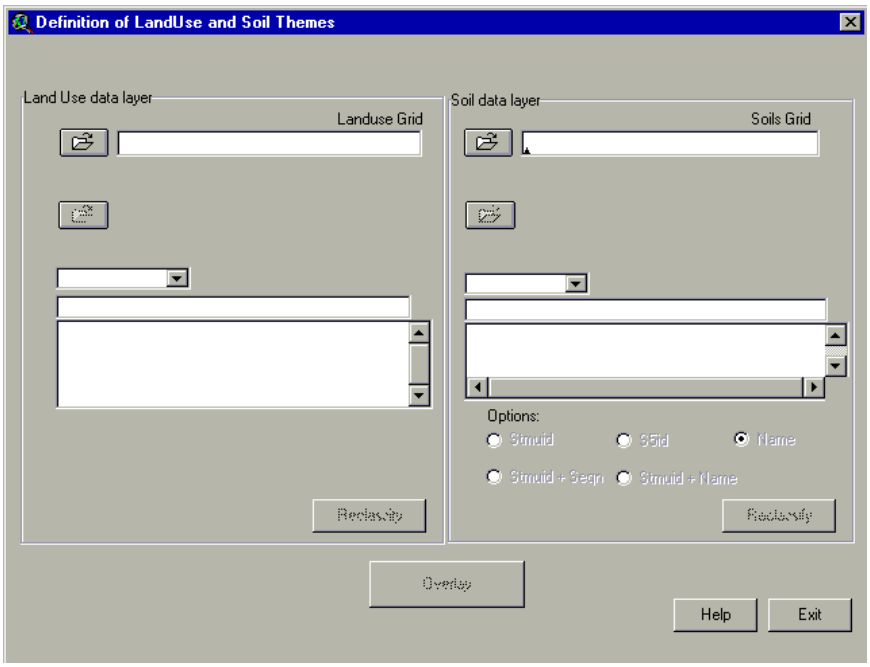


Figure 6.2

The dialog is divided into two main sections: *Land Use data layer* and *Soil data layer*.

SECTION 6.1.2: LAND USE DATA LAYER

Define Land Use/Land Cover theme:

1. Select the land use data layer by clicking the button next to the text box labeled **Land Use Grid**. A dialog box labeled *Set the LandUse Grid* will appear (Figure 6.3).

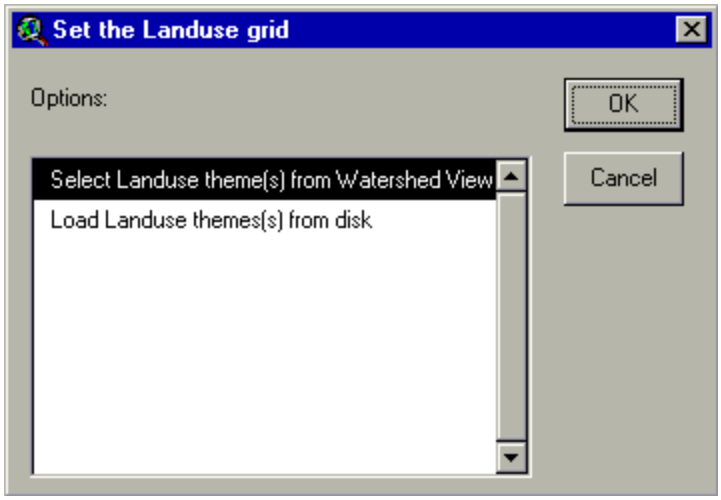


Figure 6.3

2. If the land use theme is already displayed in the *Watershed View*, highlight **Select Landuse theme(s) from Watershed View**. If the land use theme is not displayed, highlight **Load Landuse theme(s) from disk**. Click **OK**.

3. A new dialog is displayed for the user to define the type of theme being used (Figure 6.4). Select either **Shape** or **Grid** from the drop-down menu and click **OK**.

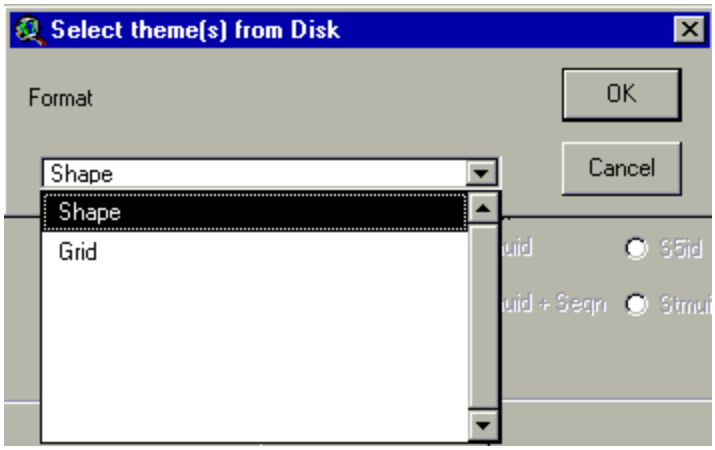


Figure 6.4

a. If you selected the **Shape** option:

   i. If loading from the *Watershed View*, a dialog is displayed (Figure 6.5) showing the list of shape themes.

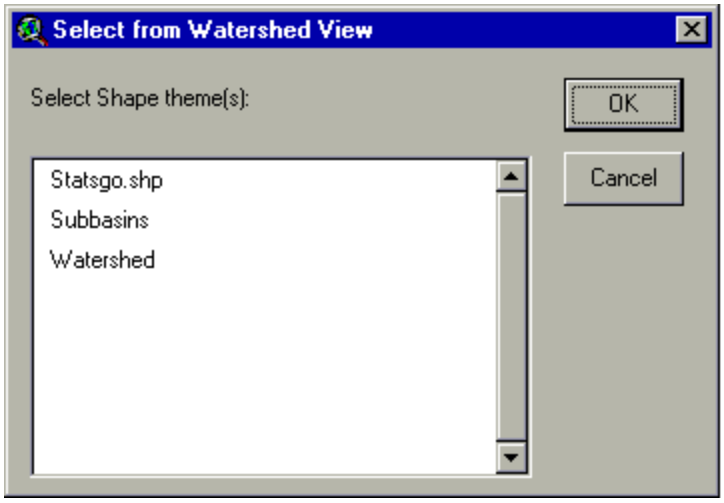


Figure 6.5

Select one or more themes (hold down the Shift key for a multiple selection) and click **OK**. The shape file(s) will be automatically converted to grid (and merged for a multiple selection).

   ii. If loading from disk, a file browser pops up (Figure 6.6).

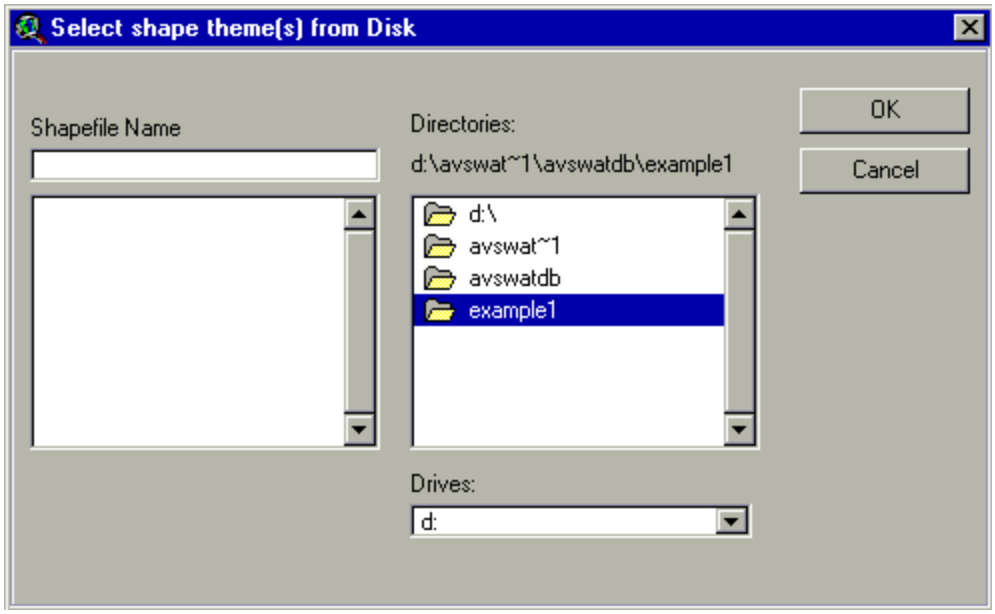


Figure 6.6

Select one or more themes (hold the Shift key for a multiple selection) and click **OK**.

The selected themes are converted into a grid theme with the cell size set to the base cell size. The base cell size is the cell size of the DEM theme.

By default, the script will look for the *Lucode* field in the table of attributes of the shape themes and use these values for the conversion to grid. If this field is not included, the script will look for the field types integer and string. If none of these fields are found, a dialog box will report the lack of usable fields (Figure 6.7) and stop the process.

Figure 6.7

If more than one field is usable, a dialog box will list them (Figure 6.8), and you can select one and click **OK**.

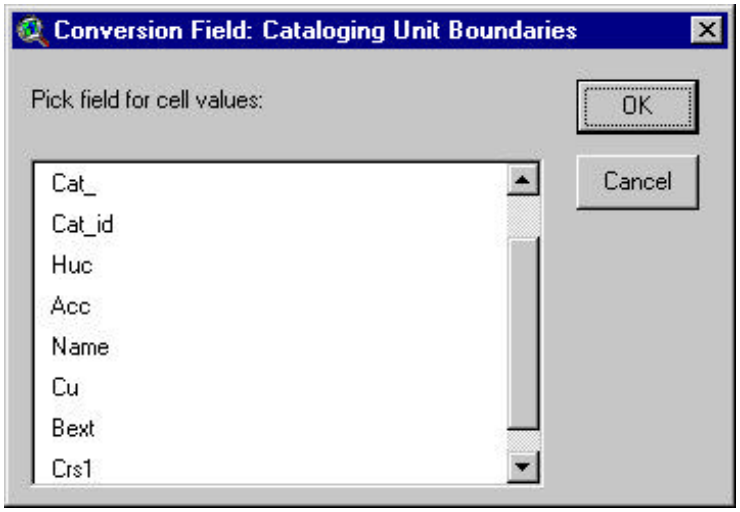


Figure 6.8

If more than one theme was selected, the interface will check all themes for the selected field. If one or more themes do not contain the field, the interface will stop the loading process.

b. If you selected the **Grid** option:

   i. If loading from the *Watershed View,* a dialog box is displayed (Figure 6.9) showing the list of grid themes.

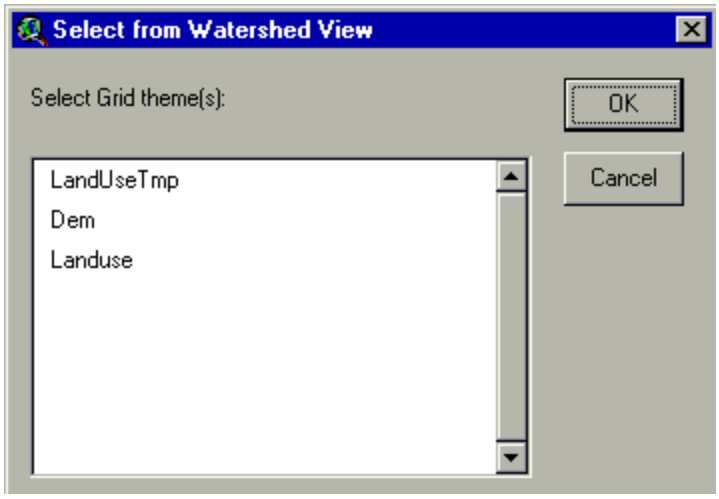


Figure 6.9

ii. If loading from disk, a prompt dialog pops up (Figure 6.10).

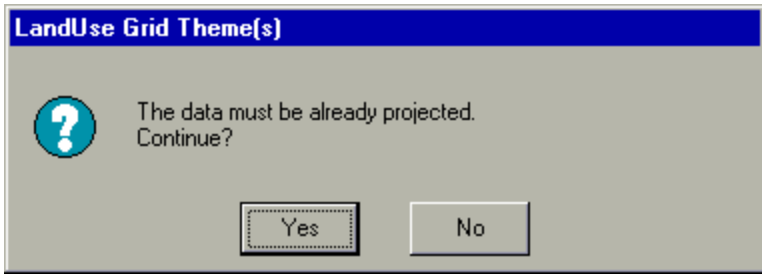


Figure 6.10

If the land use grid is not projected, click **No** to exit. This will stop map processing.

If the land use grid is defined in the same projection as the DEM, click **Yes** to proceed. A browser will be displayed allowing the user to select the land use grid (Figure 6.11).

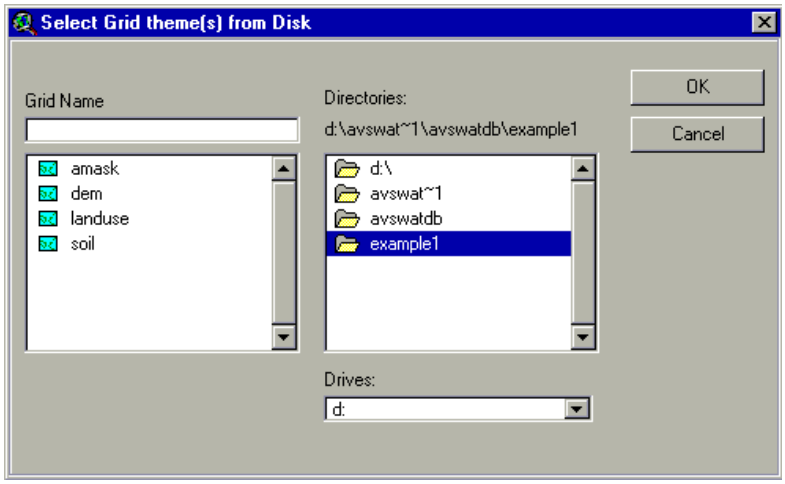


Figure 6.11

Note: The base cell size of the land use grid is automatically set to the same size as the DEM grid cell. This is required to properly overlay the different maps for comparison.

4. The *Load and Clip Land Use* process might return an unsuccessful report (Figure 6.12) if none of the selected theme(s), either grids or shape, overlay part of the watershed. Click **OK** and the process will be stopped.

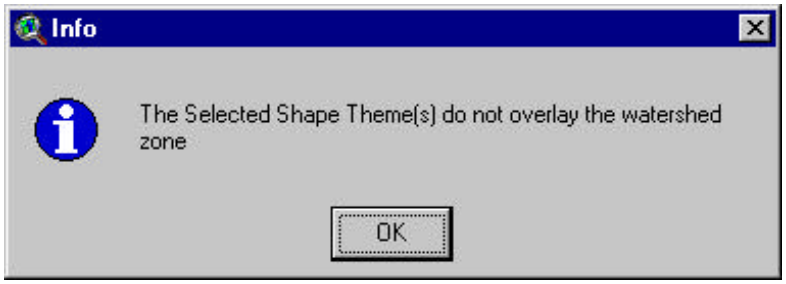


Figure 6.12

5. When this step is complete, a *Load and Clip Land Use* message box will appear (Figure 6.13) stating that the land use theme has been processed and clipped to the watershed boundaries. The message box also reminds the user to load a look-up table or manually define the land use classes for the theme.

Figure 6.13

6. Click **OK** to proceed. This will return you to the *Definition of LandUse and Soil Themes* dialog box.

7. A new theme, *LanduseTmp*, is added to the *Watershed View,* where the user may review the results of the map processing. The path of the resulting grid is shown in the text box labeled **Landuse Grid**. The list box displays the landuse grid values with percent watershed area. The *LanduseSwat* attribute that defines the land cover/plant from the SWAT databases is blank and must be defined (Figure 6.14).

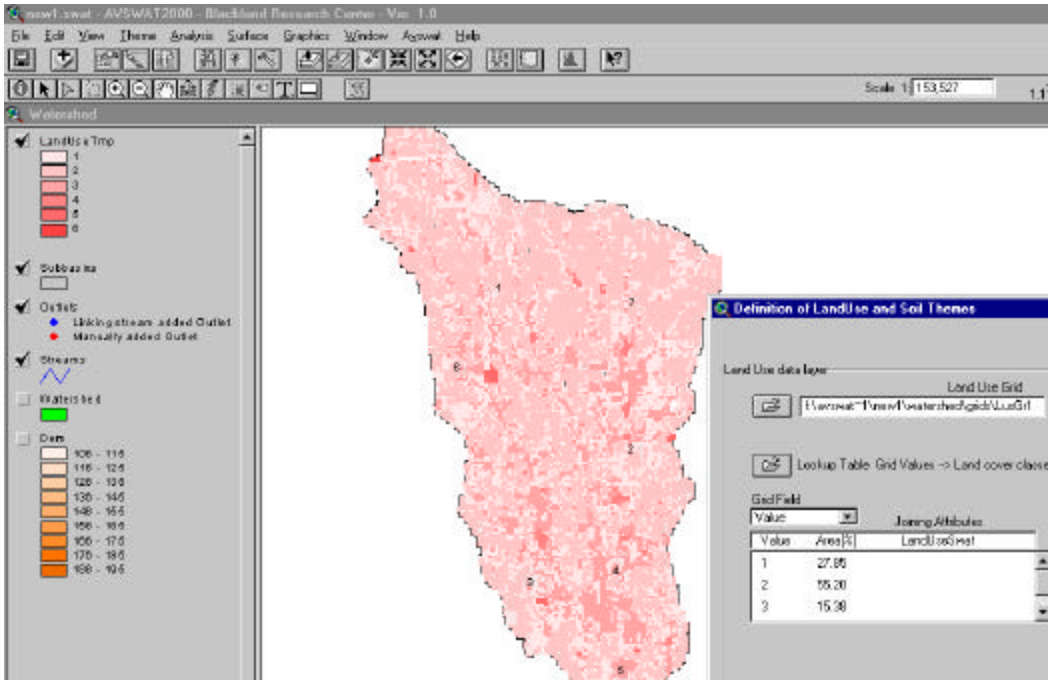
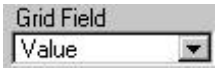


Figure 6.14

Define SWAT Land Cover associated with land use theme categories:

1. Select the Grid attribute field 

containing the codes/category values to be reclassified.
2. SWAT land cover/plant descriptions are assigned to land use theme categories using the *Landuse Reclass Section* of the dialog box (Figure 6.15).

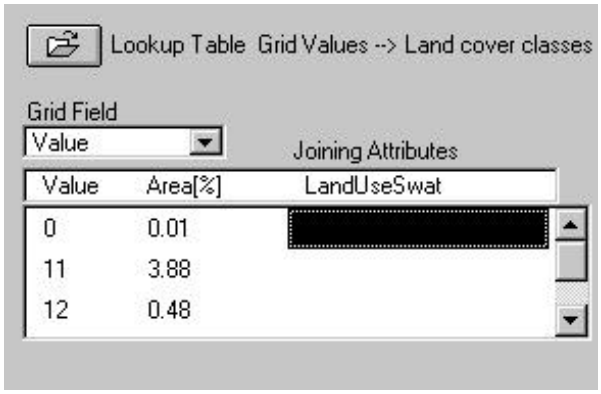


Figure 6.15

3. You can manually assign a land cover/plant code or use a look-up table.

   Manually assign land cover/plant codes:

a. To manually assign land cover/plant codes, double-click in the *LandUseSwat* spot in the *Joining Attributes* box (Figure 6.16).

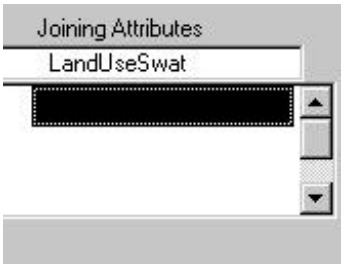


Figure 6.16

b. A dialog box will pop up listing two database files from which a SWAT land type can be selected: Land Cover/Plant or Urban (Figure 6.17).

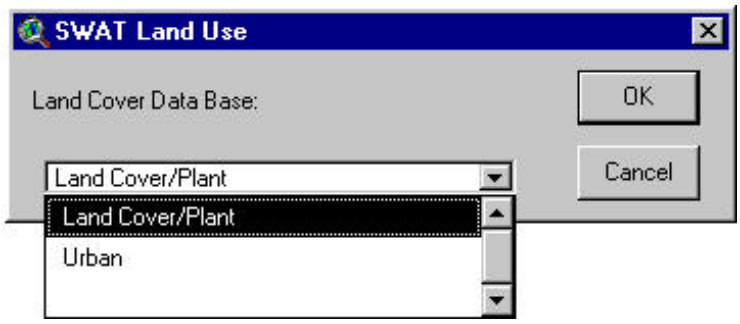


Figure 6.17

c. Highlight the desired database file by clicking on it. Click **OK** (or **Cancel** to exit). A dialog box will pop up listing the available SWAT land cover codes (Figure 6.18) or the available SWAT urban land type codes (Figure 6.19).

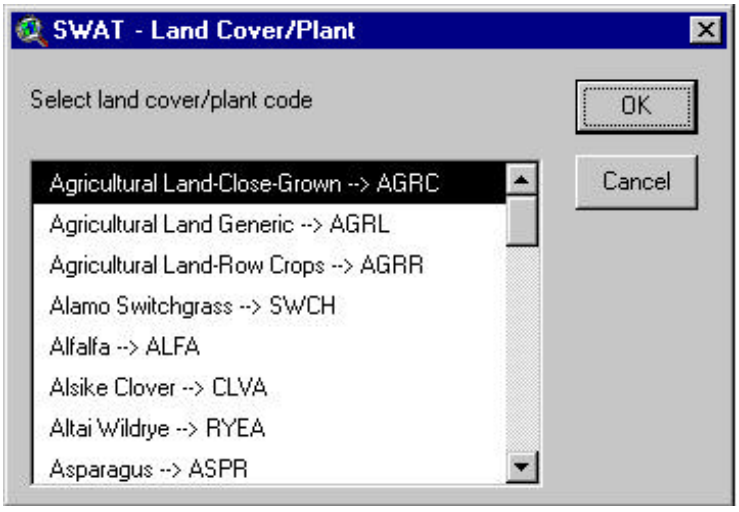


Figure 6.18

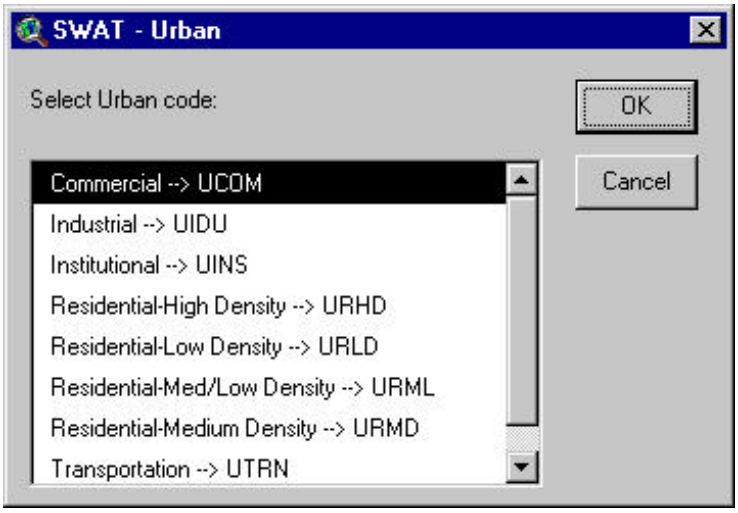


Figure 6.19

> **Tip:** Prior to loading the land use map in a project, edit the SWAT Land use/Plant Growth or Urban data base (see Section 14) to add any new types of land cover required for the land use map reclassification.

d. Scroll down the list and highlight the land cover code for the map category by clicking on it. Click **OK** (or double click the selection). The SWAT land cover or urban code will be displayed next to the corresponding land use map category in the Definition of *Land Use and Soil Themes* dialog box (Figure 6.20).

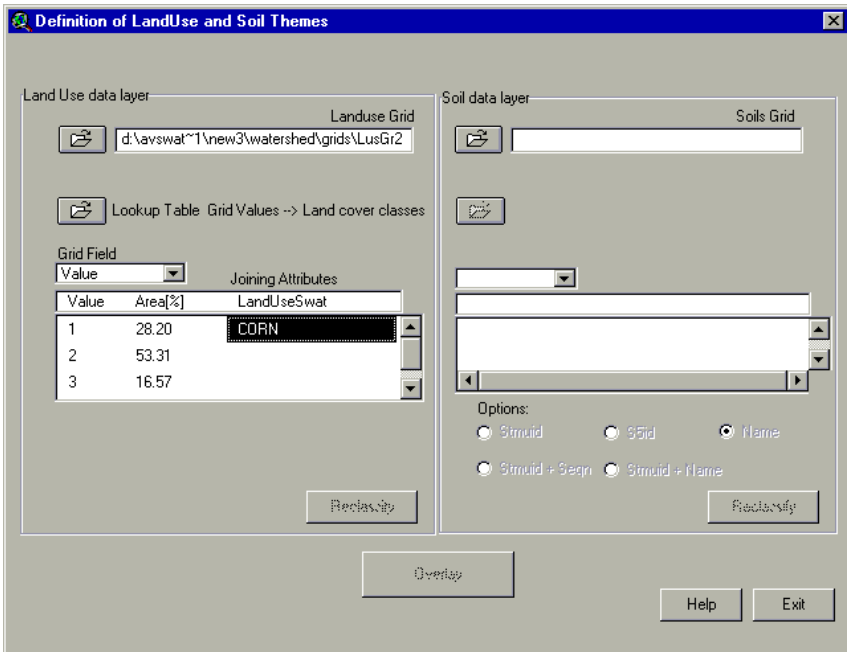


Figure 6.20

e. Repeat these steps until all LanduseSWAT codes are defined.

Assign land cover/plant codes using a look-up table:

a. To load a look-up table select the land use look-up table by clicking on the button labeled **Look-up table Grid Values→Land cover classes**

.

b. A dialog box will be displayed (Figure 6.21).

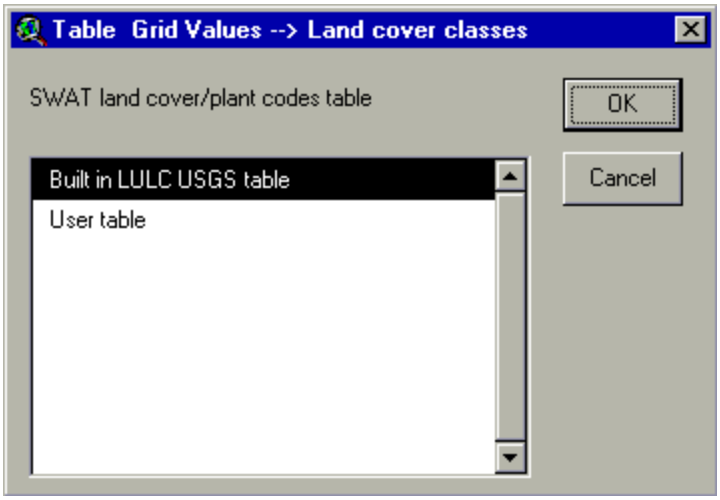


Figure 6.21

Two options are listed. If the land use map theme uses USGS land use/land cover classification codes, the built-in LULC USGS table may be selected. Otherwise the user will supply a table to be imported by the interface.

Use built-in USGS land use classification table.

i. Highlight **Built in LULC USGS table**. Click **OK**.

Use custom classification table.

i. Highlight **User table**. Click **OK**.

ii. A prompt box will appear that allows the user to specify the format of the land use classification table (Figure 6.22). The land use classification table may be formatted in dBASE (.dbf) or ASCII (.txt).

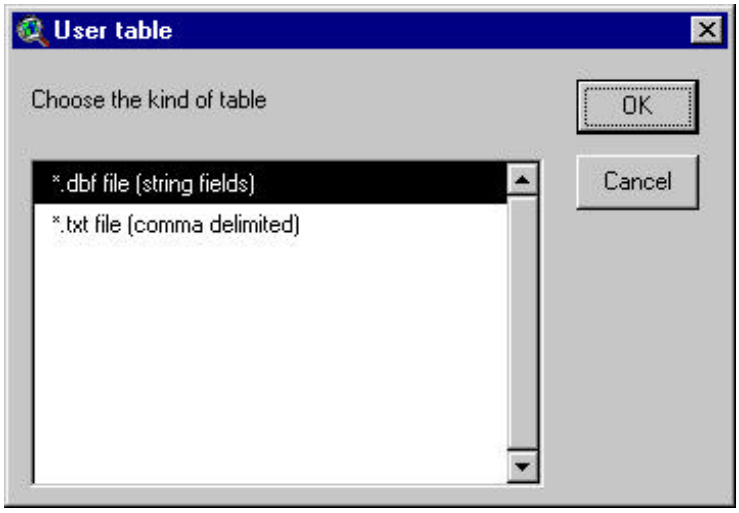


Figure 6.22

iii. Select the proper format and click **OK**.

iv. A browser will appear, allowing you to select and load the look-up table from disk (Figure 6.23).

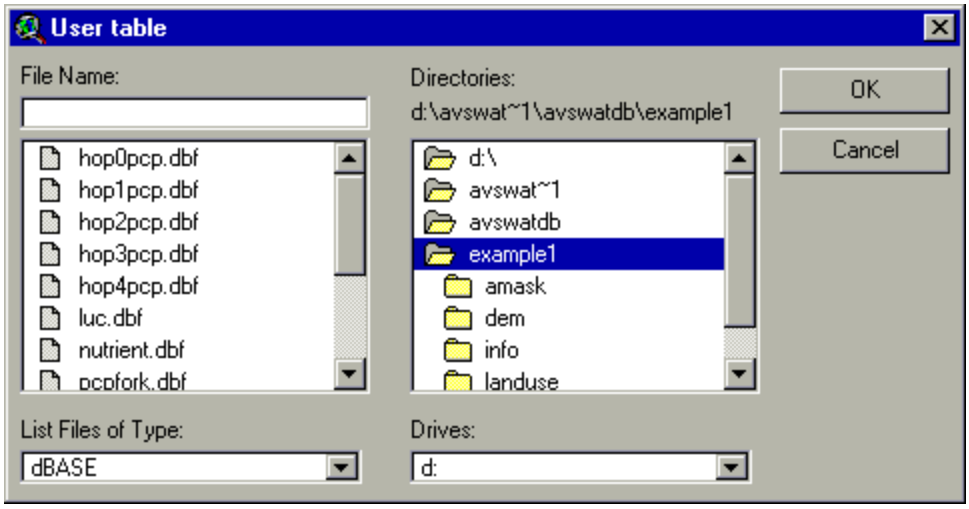


Figure 6.23

Note:	Information on the land use classification or look-up table format (dBASE and ASCII) is provided in Section 3.

v. Highlight the name of the look-up table and click **OK**.

vi. The interface will insert LanduseSWAT codes from the table into the list box (Figure 6.24).

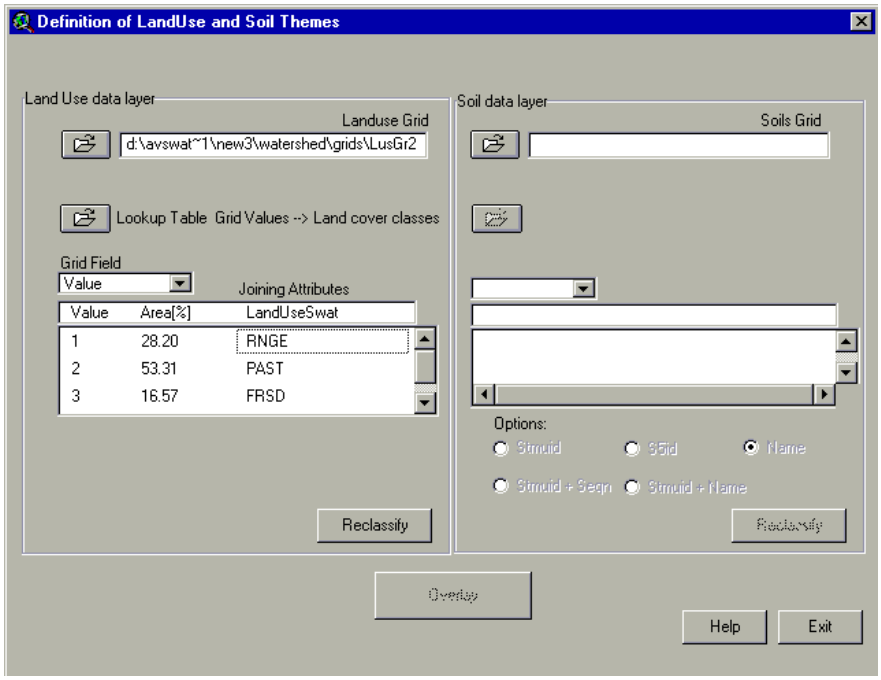
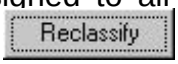


Figure 6.24

4. Once a *LandUseSwat* code has been assigned to all map categories, the **Reclassify** button will be enabled. Click the  button.

5. A new theme named "SwatLanduseClass" will be displayed in the *Watershed View* (Figure 6.25).

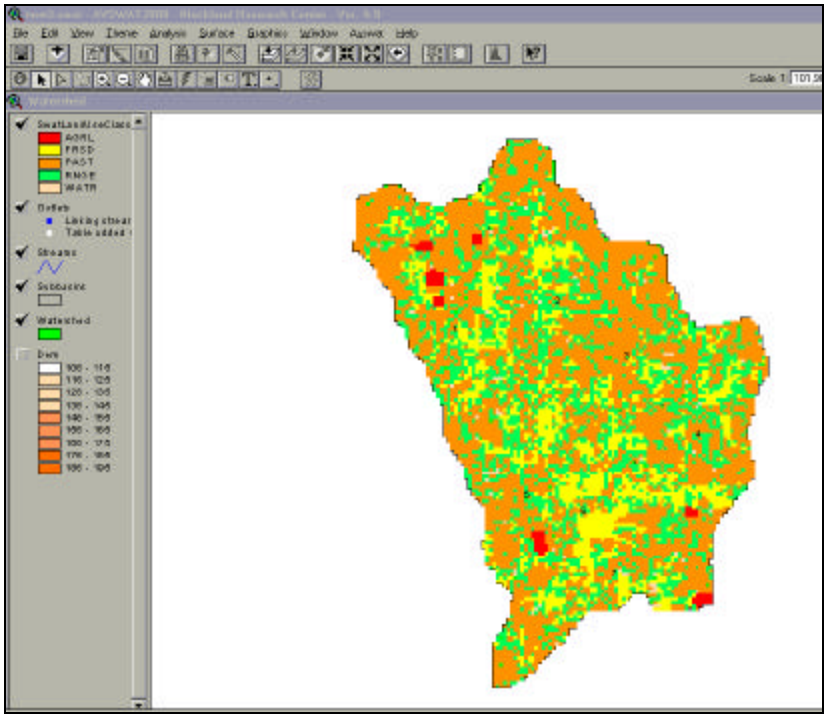


Figure 6.25

6. The land use data layer is now loaded.

SECTION 6.1.3: SOIL DATA LAYER

Define Soil theme:

1. Select the soil data layer by clicking the button next to the text box labeled **Soil Grid**. A dialog box labeled *Set the Soil Grid* will appear (Figure 6.26).

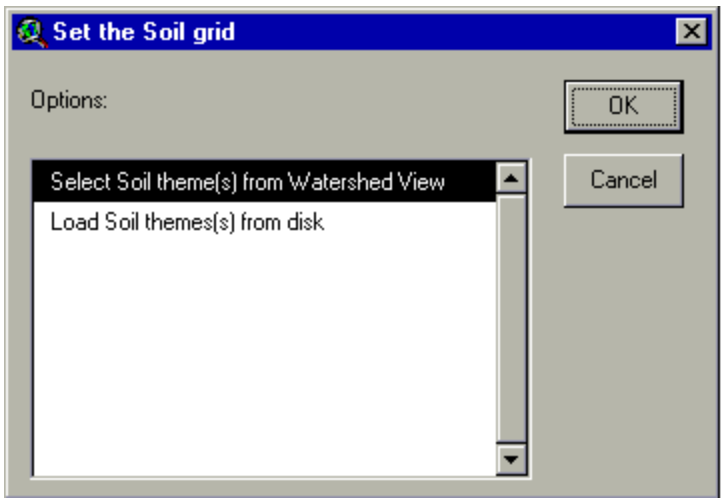


Figure 6.26

2. If the soil theme is already displayed in the *Watershed View*, highlight **Select Soil theme(s) from Watershed View**. If the soil theme is not displayed, highlight **Load Soil theme(s) from disk**. Click **OK**.

3. A new dialog is displayed for the user to define the type of theme being used (Figure 6.27). Select either **Shape** or **Grid** from the drop-down menu and click **OK**.

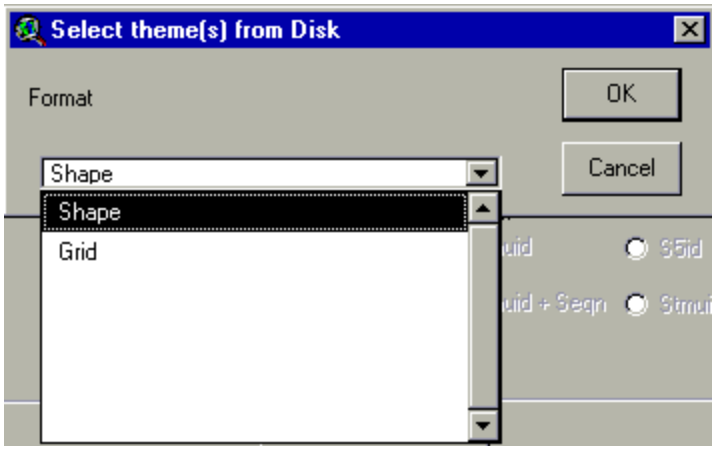


Figure 6.27

a. If you selected the **Shape** option:

i. If loading from the *Watershed View*, a dialog is displayed (Figure 6.28) showing the list of shape themes.

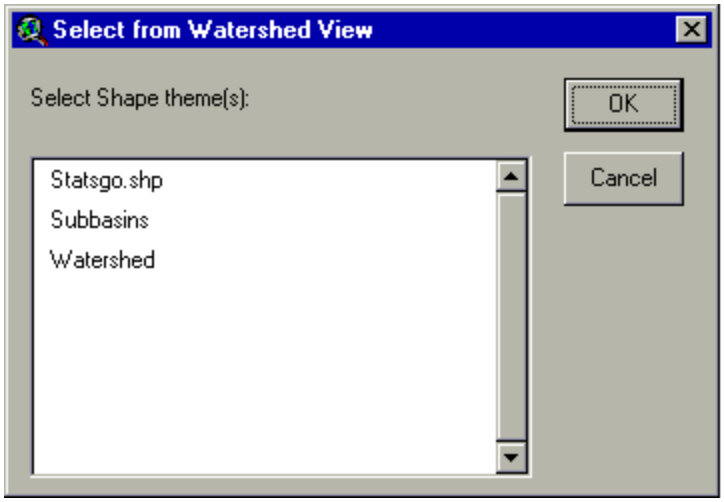


Figure 6.28

Select one or more themes (hold down the Shift key for a multiple selection) and click **OK**. The shape file(s) will be automatically converted to grid (and merged for a multiple selection).

ii. If loading from disk, a file browser pops up (Figure 6.29).

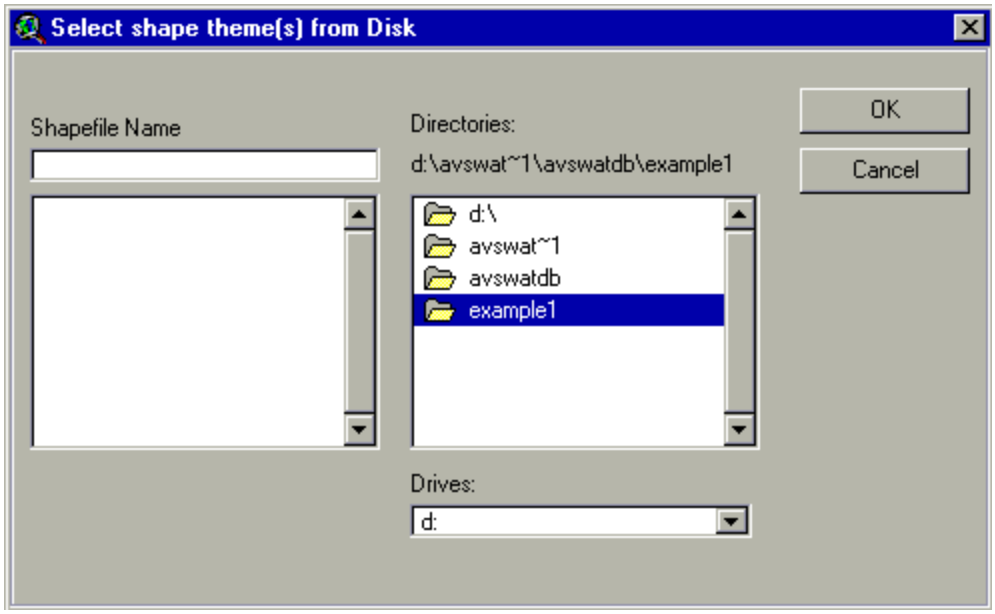


Figure 6.29

Select one or more themes (hold the Shift key for a multiple selection) and click **OK**.

The selected themes are converted in a grid theme with the cell size set to the base cell size. The base cell size is the cell size of the DEM theme.

By default, the interface will look for one of the follow fields (listed in order of priority) *Muid, Stmuid, Name, S5id,* and *Seqn* in the table of attributes of the shape themes and use these values for the conversion to grid. If one of these fields is not included, the script will look for the field types integer and string. If none of these fields are found, a dialog box will report the lack of usable fields (Figure 6.30) and stop the process.

Figure 6.30

If more than one field is usable, a dialog box will list them (Figure 6.31), and you can select one and click **OK**.

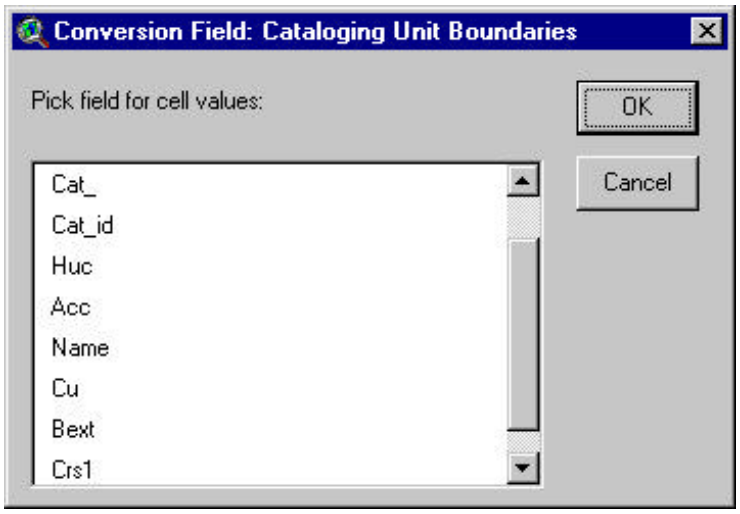


Figure 6.31

If more than one theme was selected, the interface will check all themes for the selected field. If one or more themes do not contain the field, the interface will stop the loading process.

b. If you selected the **Grid** option:

i. If loading from the *Watershed View,* a dialog box is displayed (Figure 6.32) showing the list of grid themes.

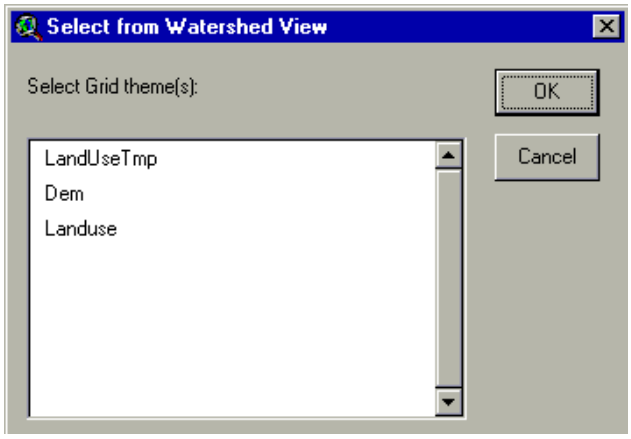


Figure 6.32

ii. If loading from disk, a prompt dialog pops up (Figure 6.33).

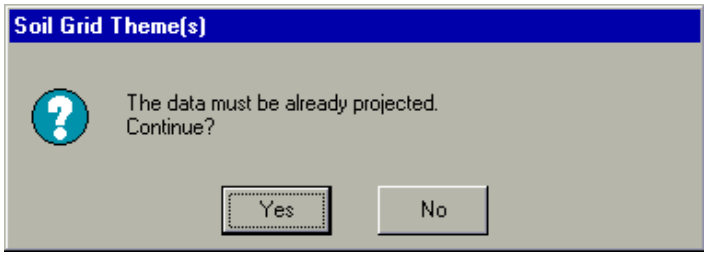


Figure 6.33

If the soil grid is not projected, click **No** to exit. This will stop map processing.

If the soil grid is defined in the same projection as the DEM, click **Yes** to proceed. A browser will be displayed allowing the user to select the soil grid (Figure 6.34).

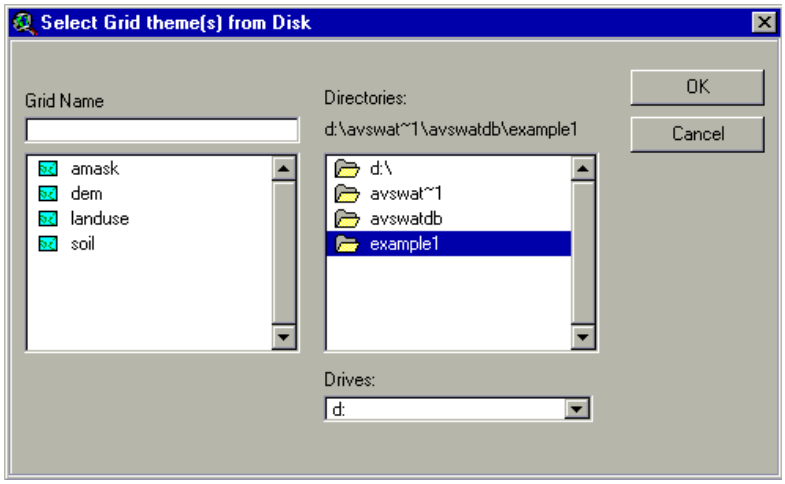


Figure 6.34

Note: The base cell size of the soil grid is automatically set to the same size as the DEM grid cell. This is required to properly overlay the different maps for comparison.

4. The *Load and Clip Soil* process might return an unsuccessful report (Figure 6.35) if none of the selected theme(s), either grids or shape, overlay part of the watershed. Click **OK** and the process will be stopped.

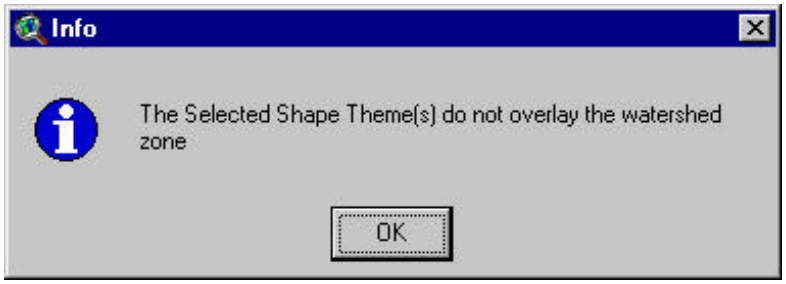


Figure 6.35

5. When this step is complete, a *Load and Clip Soil* message box will appear (Figure 6.36) stating that the soil theme has been processed and clipped to the watershed boundaries. The message box also reminds the user to load a look-up table or manually define the soil classes for the theme.

Figure 6.36

6. Click **OK** to proceed. This will return you to the *Definition of LandUse and Soil Themes* dialog box.

7. A new theme, *SoilTmp*, is added to the *Watershed View* where the user may review the results of the map processing. The path of the resulting grid is shown in the text box labeled **Soils Grid**. The list box displays the soil grid values with percent watershed area as well as attributes that will store (some could be already populated) the soil description information (Figure 6.37).

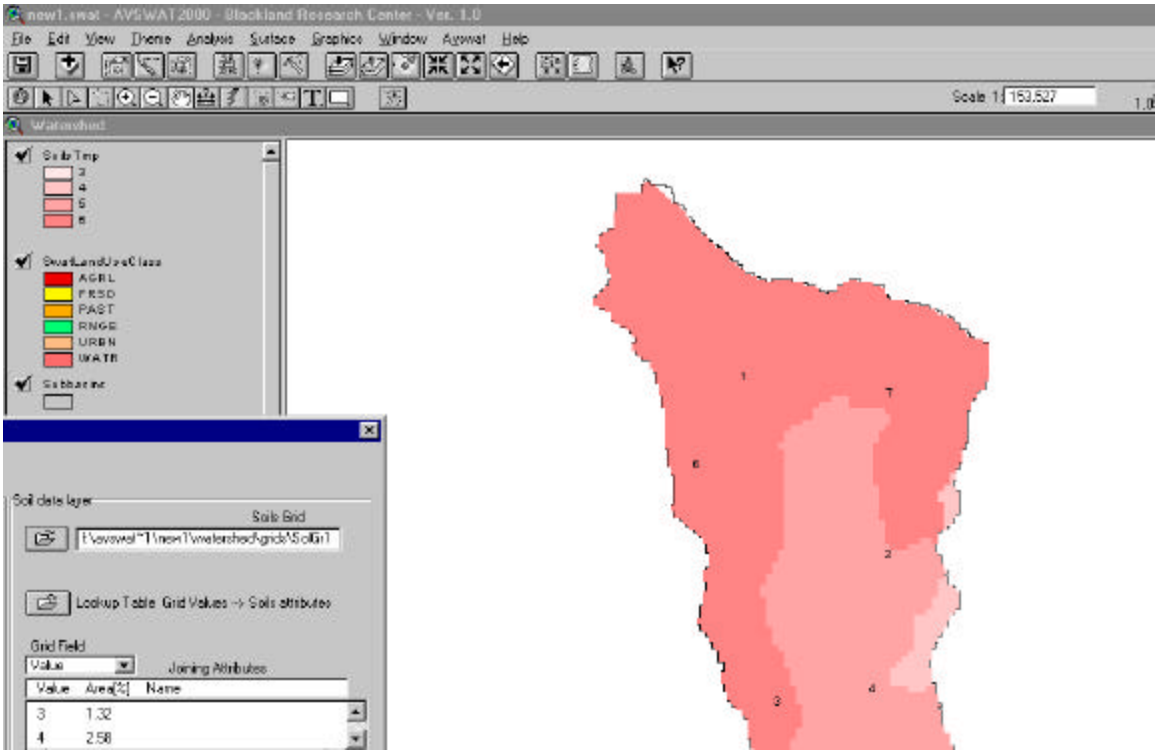
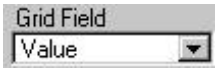


Figure 6.37

Define SWAT Soil associated with soil theme categories:

1. Select the Grid attribute field 

containing the codes/category values to be reclassified.

2. Soil map categories must be linked to one of two databases: the U.S. STATSGO database or the User Soils database. STATSGO data for three states, Texas, Pennsylvania, and Wisconsin, are loaded when the interface is installed. Data for other states can be downloaded from the SWAT website and loaded following the directions provided in Section 2. Information on the User Soils database is provided in Section 14.

   The interface allows map categories to be linked to the soil databases in one of five ways. The database and linkage options are defined by the soil option radio buttons (Figure 6.38).

Figure 6.38

To link to the User Soils database, the **Name** radio button is selected. To link a STATSGO map to the STATSGO soil database, the **Stmuid**, **Stmuid + Seqn**, or **Stmuid + Name** radio buttons are used. STATSGO maps are general soil distribution maps. STATSGO map associations or polygons are comprised of many soil series. The **Stmuid** button will assign data for the dominant soil series in the STATSGO polygon to HRUs. The **Stmuid + Seqn** or **Stmuid + Name** radio buttons provide users with methods to assign data from a soil series other than the dominant to HRUS. To link a soil series or Soils5 map to the STATSGO database, the **S5id** radio button is selected.

3. As for landuse, the attribute data required to link the map categories to soil information in one of the two databases may be entered manually or loaded from a look-up table.

   Manually assign soil attribute data:

   To manually assign soil attribute data, double-click in the empty records below the *Joining Attributes* label (Figure 6.39).

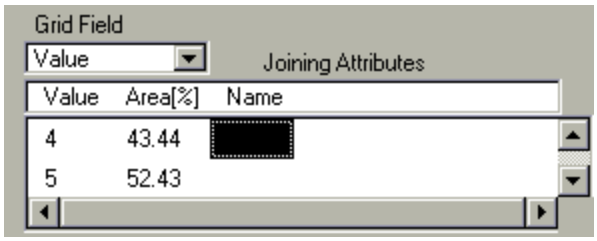


Figure 6.39

   For user provided soil data click the radio button labeled **Name**

.

   a. Only the joining attribute records labeled *Name* are available. Double click a record.

   b. A dialog box is displayed listing all soils in the User Soil database (Figure 6.40).

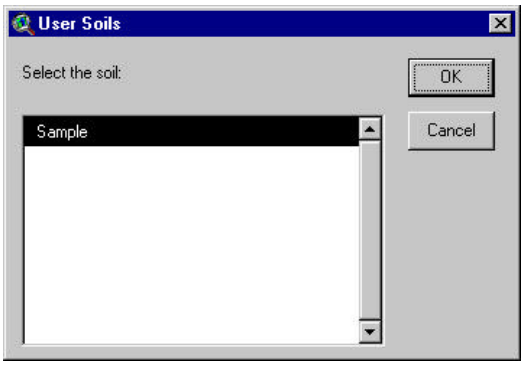


Figure 6.40

   c. Select the soil and click **OK** (or double click the selection).

> **Tip:** Input your soils entry and data sets in the *User Soils* data base (See Section 14) before you reclassify the Soil grid.

For the use of the STATSGO database, the user has four options:

a. **Stmuid**. The State STATSGO polygon number is specified by the user, and the interface selects the dominant soil phase in the STATSGO polygon to represent the soil attributed in the area mapped in the polygon.

   i. Click the **Stmuid** radio button

.

   ii. Double click the respective record. A dialog box will be displayed that allows the user to enter the State STATSGO polygon number (Figure 6.41).

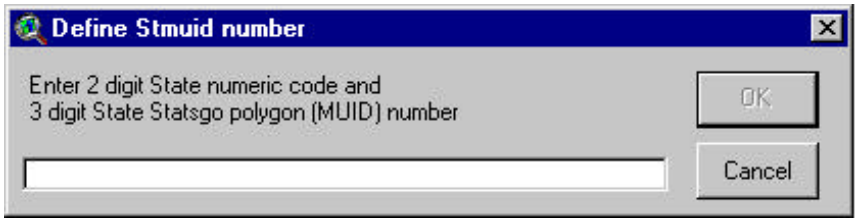


Figure 6.41

   iii. Type the Stmuid number in the text box. Click **OK**. The entry is now set in the selected record.

b. **S5id**. The Soils5 ID number for USDA soil series data is specified by the user.

   i. Click the **S5id** radio button

.

   ii. Double click the respective record. A dialog box will be displayed that allows the user to enter the Soils5 ID number (Figure 6.42).

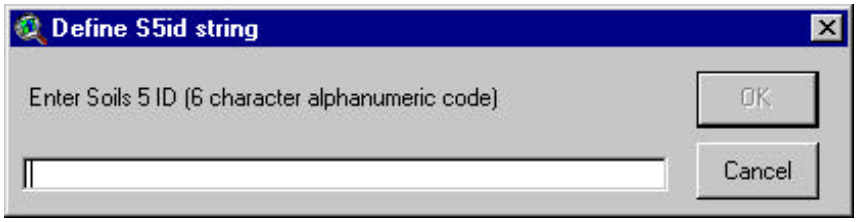


Figure 6.42

   iii. Type the Soils5 ID number in the text box. Click **OK**. The entry is now set in the selected record.

c. **Stmuid+Seqn**. The State STATSGO polygon number and sequence number of soil phase is specified by the user.

i. Click the **Stmuid + Seqn** radio button

.

ii. Two joining attribute records must be defined for each map category.

iii. When the **Stmuid** record is double-clicked, a dialog box (Figure 6.41) is displayed. Enter the State STATSGO polygon number and click **OK**. The Stmuid number is set in the selected record.

iv. Double click the **Seqn** record. A dialog box will be displayed that allows the user to enter a sequence number. This number is the ranking in dominance (1=dominant, 2=second most dominant, etc.) used to select the soil series data assigned to HRUs containing the STATSGO polygon. (Figure 6.43)

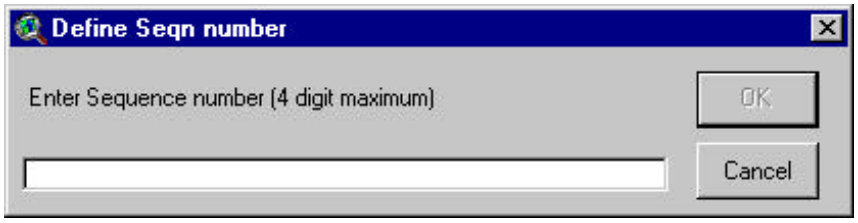
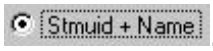


Figure 6.43

v. Type the sequence number in the text box. Click **OK**. The entry is now set in the selected record.

d. **Stmuid + Name**. The State STATSGO polygon number and soil series name is specified by the user.

i. Click the **Stmuid + Name** radio button 

.

ii. Two joining attribute records must be defined for each map category.

iii. When the **Stmuid** record is double-clicked, a dialog box (Figure 6.41) is displayed. Enter the State STATSGO polygon number and click **OK**. The Stmuid number is set in the selected record.

iv. Double click the **Name** record. A dialog box will be displayed that allows the user to enter a soil series name (Figure 6.44).

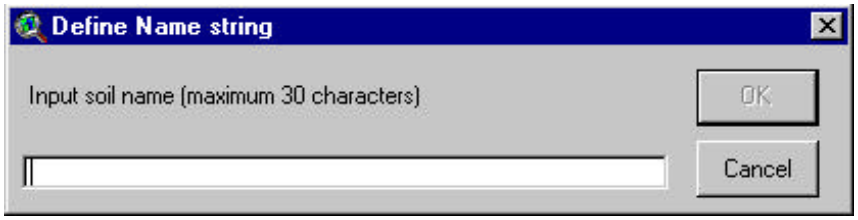


Figure 6.44

v. Type the soil series name in the text box. Click **OK**. The entry is now set in the selected record.

vi. Repeat these steps until all soil joining attribute codes are defined.

Assign soil attribute information using a look-up table:

a. To load a look-up table, select the soil look-up table by clicking on the button labeled **Look-up table Grid Values→Soil Attributes**

.

b. A prompt box will appear that allows the user to specify the format of the land use classification table (Figure 6.45). The land used classification table may be formatted in dBASE (.dbf) or ASCII (.txt).

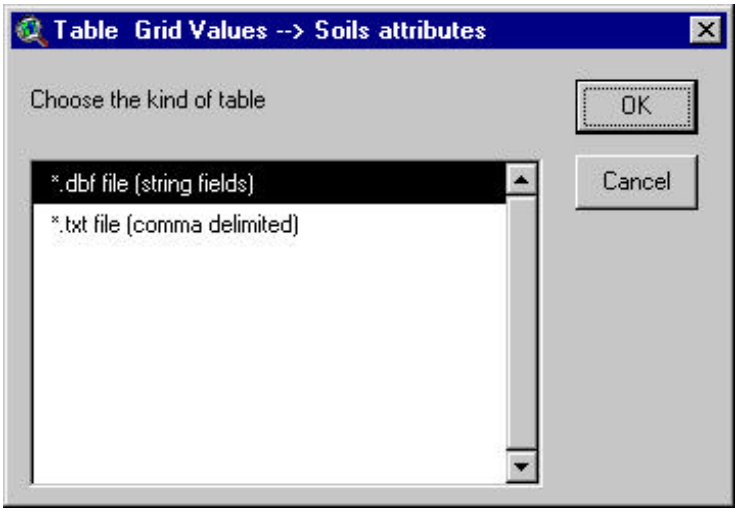


Figure 6.45

c. Select the proper format and click **OK**.

d. A browser will appear, allowing you to select and load the look-up table from disk (Figure 6.46).

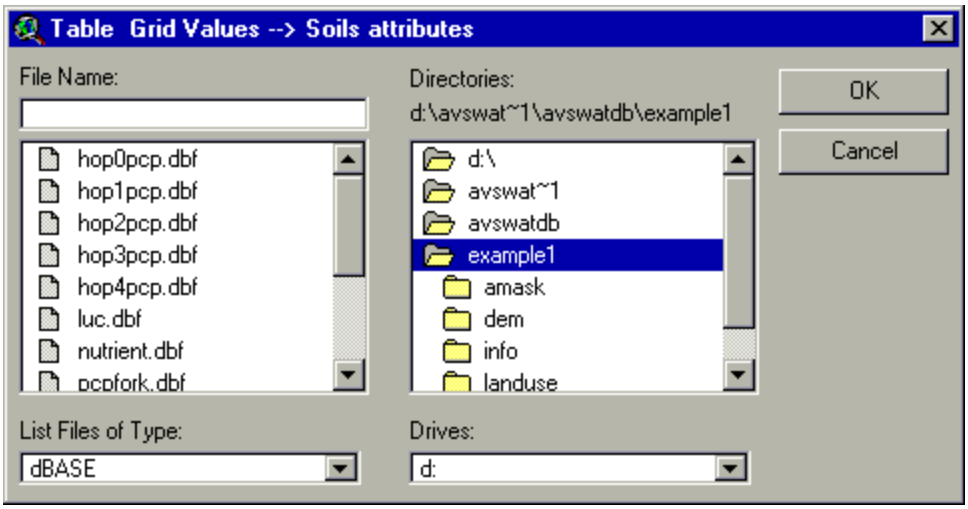


Figure 6.46

> **Note:** Information on the soil classification or look-up table format (dBASE and ASCII) is provided in Section 3.

e. Highlight the name of the look-up table and click **OK**.

f. The interface will insert soil joining attribute codes from the table into the list box (Figure 6.47).

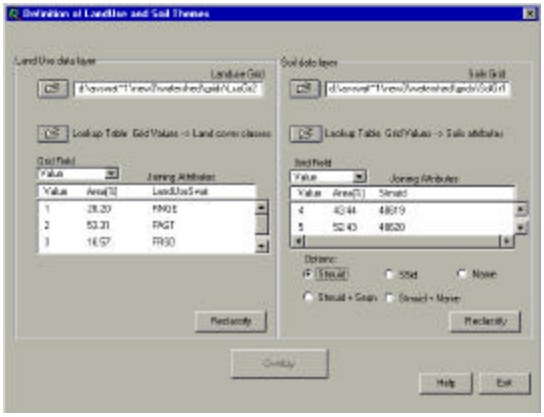
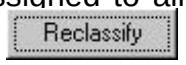


Figure 6.47

7. Once the joining attribute codes have been assigned to all map categories, the **Reclassify** button will be enabled. Click the 

button.

8. A new theme named "SoilClass" will be displayed in the *Watershed View* (Figure 6.48).

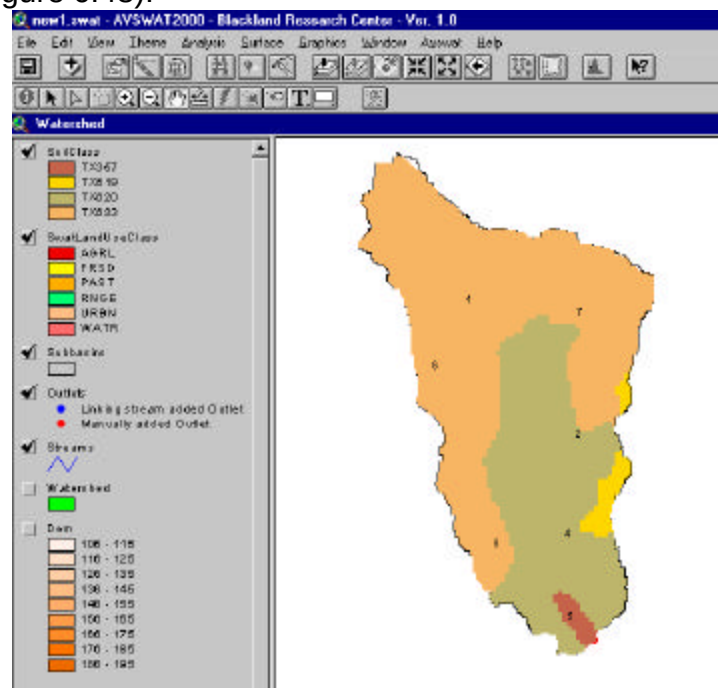


Figure 6.48

9. The soil data layer is now loaded.

SECTION 6.1.4: OVERLAY OF LANDUSE AND SOIL MAPS

1. When both the landuse and soil grids are reclassified, the

button is enabled. Click the **Overlay** button.

> **Note:** If *LandUseSwat* is set to *NOCL* for any landuse map category, a message box will be displayed requesting the user to redefine *LandUseSwat* (Figure 6.49).
>
>
>
>
> *Figure 6.49*
>
> If this message box appears, click **OK**, replace the *NOCL* codes with SWAT land use codes, and reclassify.

2. A message box signals the end of the overlay process (Figure 6.50)

Figure 6.50

3. A report named *SWAT model: LandUse and Soil Distribution* is generated during the overlay process (Figure 6.51). This report provides a detailed description of the distribution of the landuse and soil classes in the watershed and all the sub-watersheds. To access this report, click on **Reports** under the **Avswat** menu. Select **SWAT model: LandUse and Soil Distribution** and click **OK**.

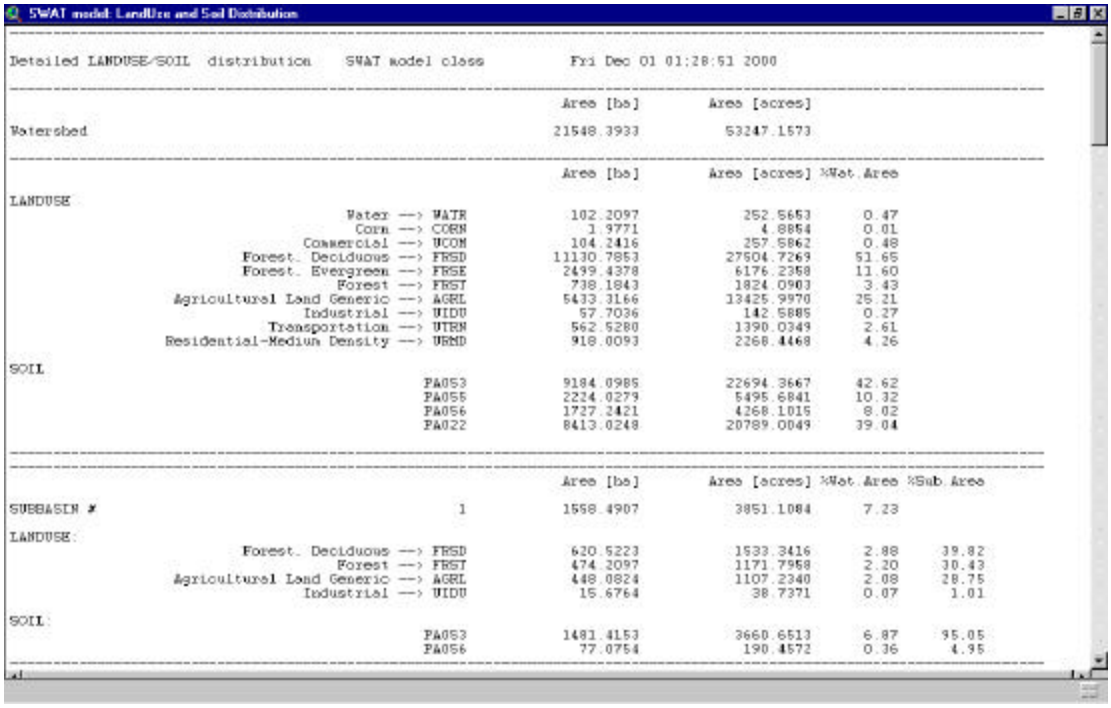


Figure 6.51

4. Once the overlay process is complete, the user may proceed with determination of hydrologic response units (HRUs).

SECTION 6.2: HRU DISTRIBUTION

Purpose

Once the land use and soil data layers have been imported, the distribution of hydrologic response units (HRUs) within the watershed must be determined. The **HRUs Distribution** command in the **Avswat** menu allows the user to specify criteria used in determining the HRU distribution. One or more unique land use/soil combinations (hydrologic response units or HRUs) can be created for each subbasin.

Application

Subdividing the watershed into areas having unique land use and soil combinations enables the model to reflect differences in evapotranspiration and other hydrologic conditions for different land covers/crops and soils. Runoff is predicted separately for each HRU and routed to obtain the total runoff for the watershed. This increases the accuracy of load predictions and provides a much better physical description of the water balance.

The user has two options in determining the HRU distribution: assign a single HRU to each subwatershed or assign multiple HRUs to each subwatershed. If a single HRU per subbasin is selected, the HRU is determined by the dominant land use category and soil type within each watershed. If multiple HRUs are selected, the user may specify sensitivities for the land use and soil data that will be used to determine the number and kind of HRUs in each watershed.

Key Procedures

- Select single or multiple HRUs per subwatershed.
- For multiple HRUs, define land use and soil threshold levels.
- Click the OK button to determine the HRU distribution.

Detailed Operations

1. Select **HRU distribution** from the *Watershed View* menu **Avswat** (Figure 6.52).

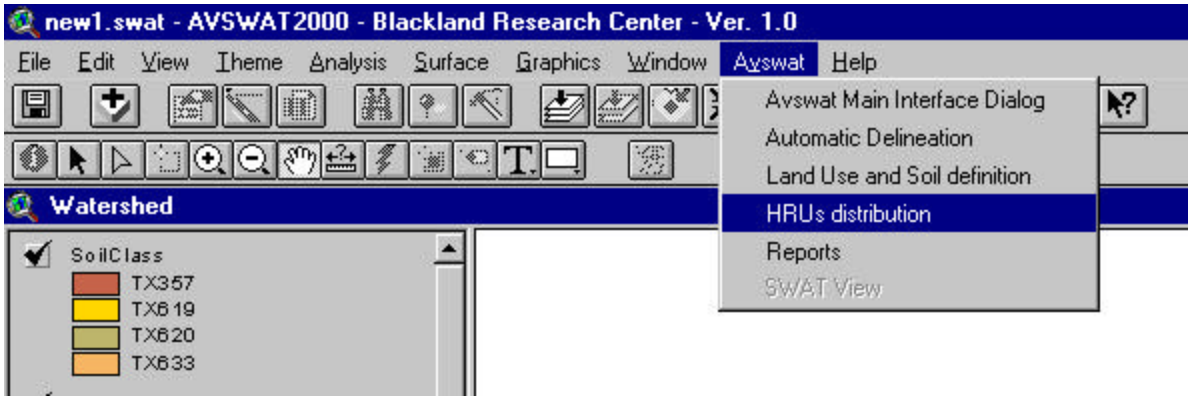


Figure 6.52

2. The *Swat Model: definition of the land use / soil distribution* dialog box will be displayed (Figure 6.53).

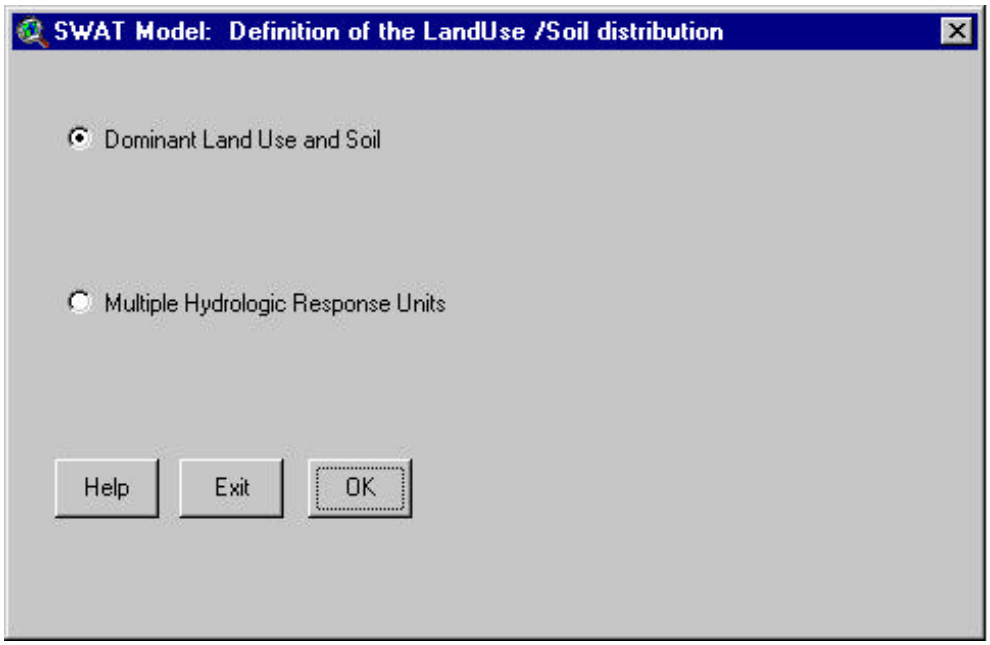
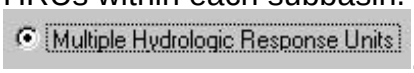


Figure 6.53

3. The dialog box has two radio buttons: **Dominant Land Use and Soil** and **Multiple Hydrologic Response Units**. The user must select the button for the method used to create HRUs.

   a. The **Dominant Land Use and Soil** option will create one HRU for each subbasin. The dominant land use and soil class in the subbasin are simulated in the HRU. To activate this option, select the radio button

.

   b. The **Multiple Hydrologic Response Units** option will create multiple HRUs within each subbasin. To activate this option, select the radio button 

.

   Two slide bars are now visible (Figure 6.54).

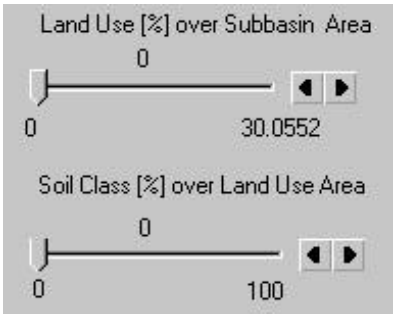


Figure 6.54

> **Note:** The creation of multiple HRUs is a two-step process. First, land uses are chosen. Once the land uses to be modeled are determined, the different soils for each land use are chosen. One HRU is created for each unique land use/soil combination.

The first scale controls the threshold level used to eliminate minor land uses in each subbasin. Land uses that cover a percentage of the subbasin area less than the threshold level are eliminated. After the elimination process, the area of the remaining land uses is reapportioned so that 100% of the land area in the subbasin is modeled.

For example, assume there is a subbasin that contains

- 35% agricultural land in corn
- 30% pasture
- 21% forest
- 10% agricultural land in orchard
- 4% urban

If the threshold level for land use is set to 20%, HRUs would be created for pasture, forest, and corn. The areas of modeled land uses would be modified as follows:

- corn: (35% ÷ 86%) x 100% = 41%
- pasture: (30% ÷ 86%) x 100% = 35%
- forest: (21% ÷ 86%) x 100% = 24%

where 86% was the percentage of the subbasin originally covered by pasture, forest, and corn.

The second scale controls the creation of additional HRUs based on the distribution of the selected land uses over different soil types. This scale is used to eliminate minor soils within a land use area. As with the land use areas, once minor soil types are eliminated, the area of remaining soils is reapportioned so that 100% of the land use area is modeled.

For example, assume that the overlay performed by the interface during the processing of the land use and soil maps identified the following soil distribution for pastureland in the subbasin:

- 20% Houston Black
- 25% Branyon
- 15% Heiden
- 10% Austin

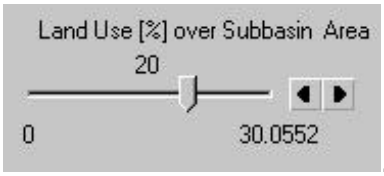
- 7% Stephen
- 6% Denton
- 5% Frio
- 4% Purves
- 3% Bastrop
- 2% Altoga
- 1% Eddy
- 1% San Saba
- 1% Ferris

If the threshold level for soils within a land use area is set to 10%, the following HRUs will be created for this example:

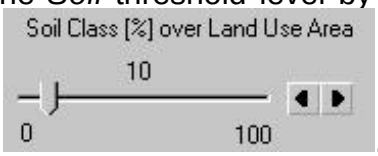
- pasture/Houston Black
- pasture/Branyon
- pasture/Heiden
- pasture/Austin

This process is performed for every land use modeled in the subbasin.

The threshold levels set for multiple HRUs is a function of the project goal and the amount of detail desired by the modeler. For most applications, the default settings for land use threshold (20%) and soil threshold (10%) are adequate.

i. Specify the *Landuse* threshold level by moving the pointer on the first slide bar 

.

ii. Specify the *Soil* threshold level by moving the pointer on the second slide bar 

.

4. Click **OK**.
5. If the **Multiple Hydrologic Response Units** option was selected and the soil class threshold was set too high, a message dialog is displayed (Figure 6.55).

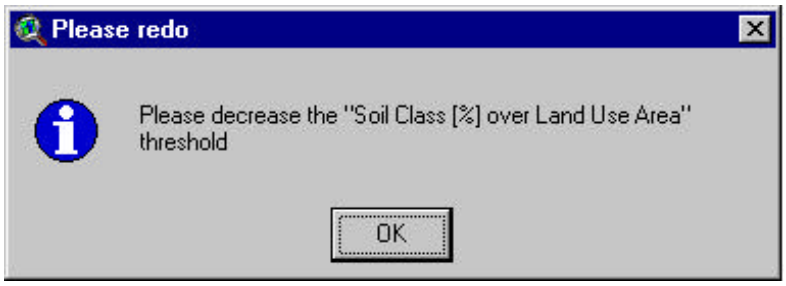


Figure 6.55

If this message is displayed, close the message box, decrease the soil threshold value, and click **OK** on the *Swat Model: definition of the land use / soil distribution* dialog box to repeat the HRU definition process.

6. Once the HRUs are created a message dialog pops up (Figure 6.56).

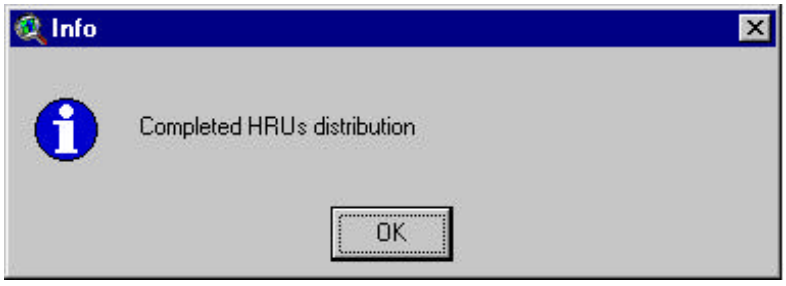


Figure 6.56

7. Click **OK**.

8. A report named *SWAT model: LandUse and Soil Distribution (after threshold application)* is generated during the HRU definition process (Figure 6.57). This report provides a detailed description of the distribution of the landuse and soil classes after application of thresholds for the watershed and all the sub-watersheds. The number of HRUs with the land use/soil classes and areal extent are listed for each subbasin. To access this report, click on **Reports** under the **Avswat** menu in the *Watershed View*. Select **SWAT model: LandUse and Soil Distribution (after threshold application)** and click **OK**.

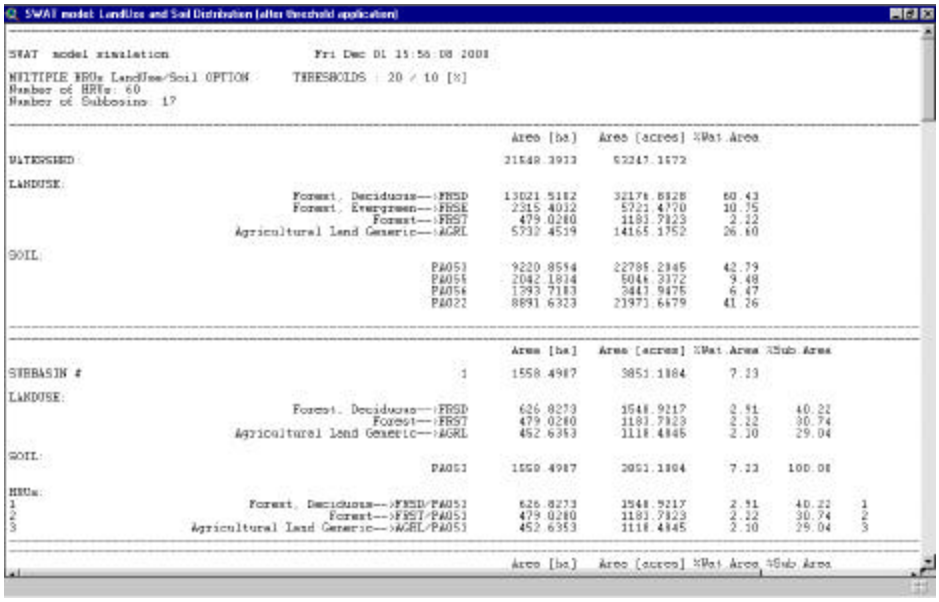


Figure 6.57

9. An ArcView table, *Distrswat,* is also created that provides a detailed distribution of the HRUs, landuse and soil classes in the watershed and all subwatersheds (Figure 6.58).

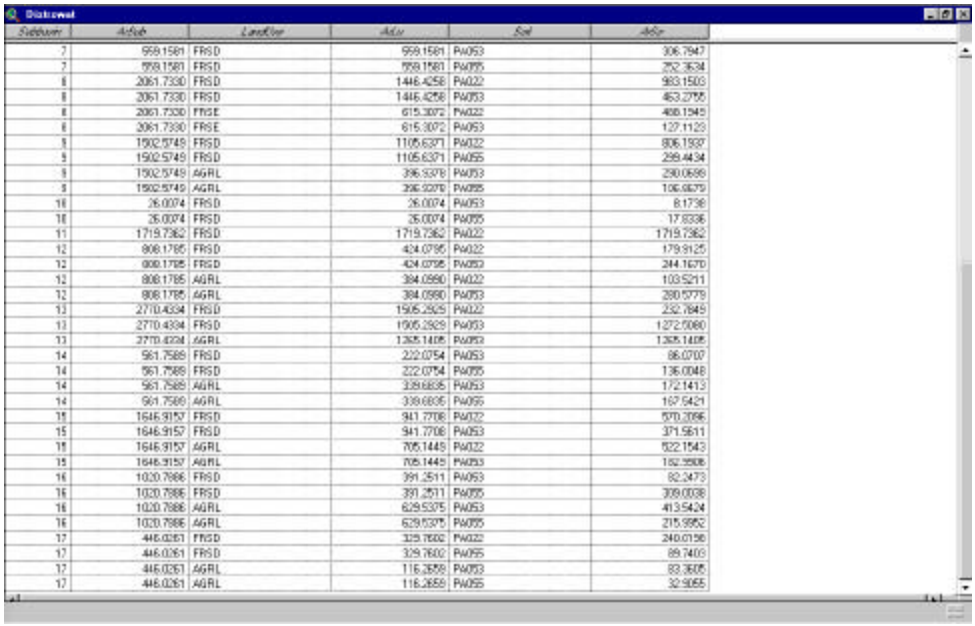


Figure 6.58

10. When the HRU distribution has been defined, the *SWAT View* is automatically created. A message box will appear notifying the user of this fact (Figure 6.59).

Figure 6.59

Click **OK**.

11. The interface will activate the *SWAT View* (Figure 6.60).

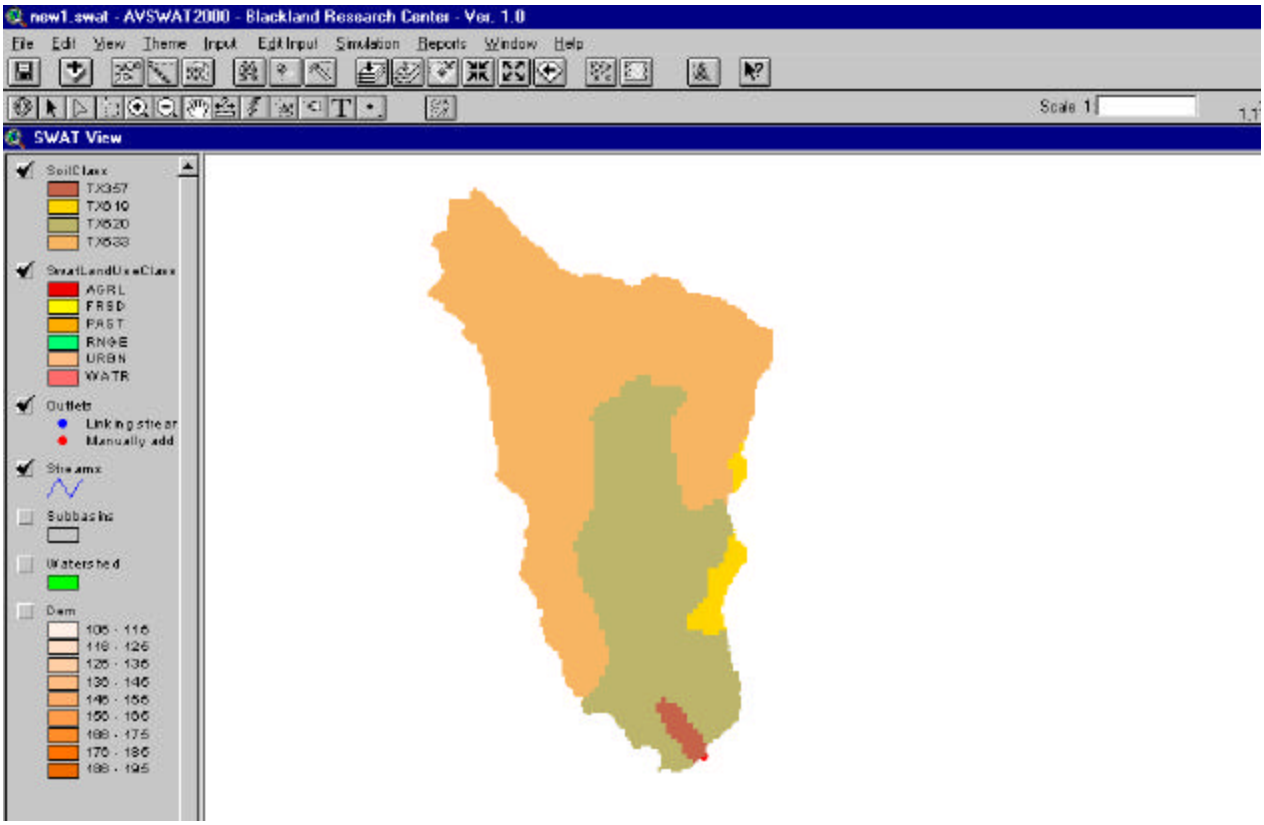


Figure 6.60

SECTION 7: IMPORT WEATHER DATA

Weather data to be used in a watershed simulation is imported once the HRU distribution has been defined. Weather data is loaded using the first command in the Input menu of the *SWAT View*. This tool allows users to load weather station locations into the current project and assign weather data to the sub-watersheds. For each type of weather data loaded, each sub-watershed is linked to one gage.

1. Select **Weather Stations** from the **Input** menu on the *SWAT View*. The *Weather Data Definition* dialog is displayed (Figure 7.1).

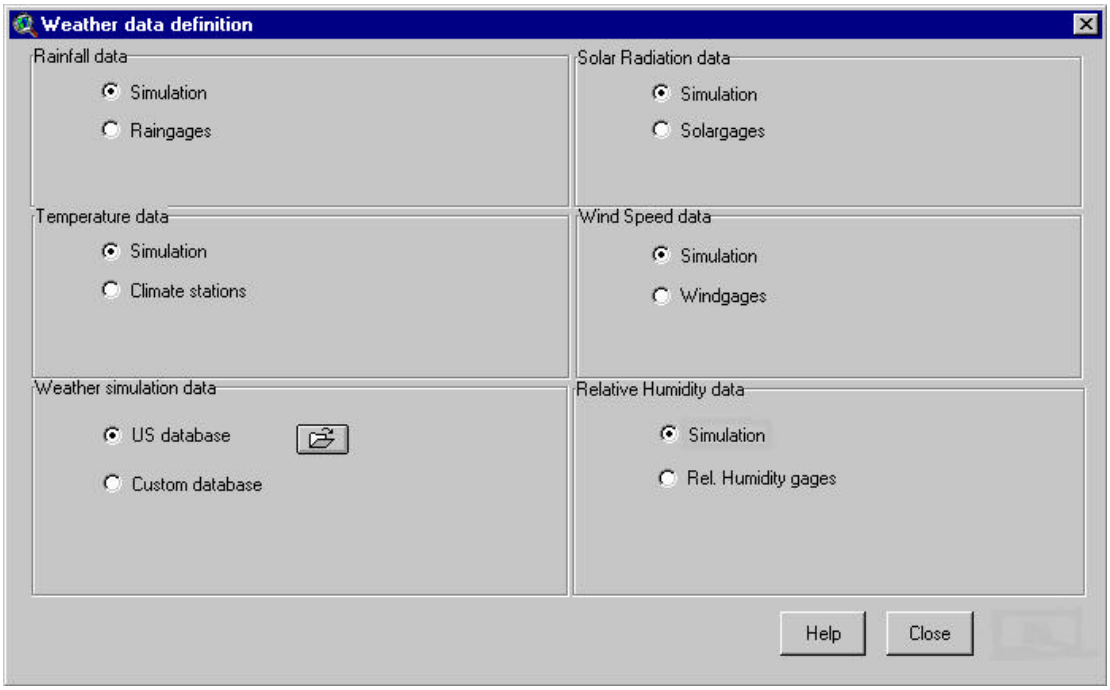


Figure 7.1

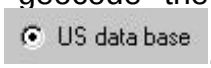
2. The *Weather Data Definition* dialog is divided into six sections: *Weather simulation data*, *Rainfall data*, *Temperature data*, *Solar Radiation data*, *Wind Speed data,* and *Relative Humidity data.* The first section listed, *Weather simulation data,* **must** be set. The interface will not allow the user to perform other input data processing until the *Weather simulation data* is defined. The other five sections allow the user to choose between simulated or measured climate data for specific types of data.

   a. **Weather simulation data***:* In this section, the user must define the data used to generate various weather parameters. Data loaded in this section is used to build .wgn files for the dataset. For more information on the type

of data used to generate weather data, please see the SWAT2000 Theoretical Documentation and the SWAT2000 User's Manual.

Weather station locations and weather generator data are obtained from one of two sources: the built-in *US database* or the *User Weather Stations* database.

i. **US database**: The US database contains weather information for 1,041 stations around the United States and is provided with the interface.

   In order to load and geocode the US database, select the **US Database** radio button 

.

   Click the button.

ii. **Custom database**: This option is used to load custom weather generator data stored in the *User Weather Stations* database.

   Select the **Custom database** radio button

.

   Click the open file folder button next to the text box that appeared with the radio button that was selected

.

   A file browser (Figure 7.2) allows you to select the dBASE location table for the weather generator stations. The location table is prepared by the user following the format described in Section 3.

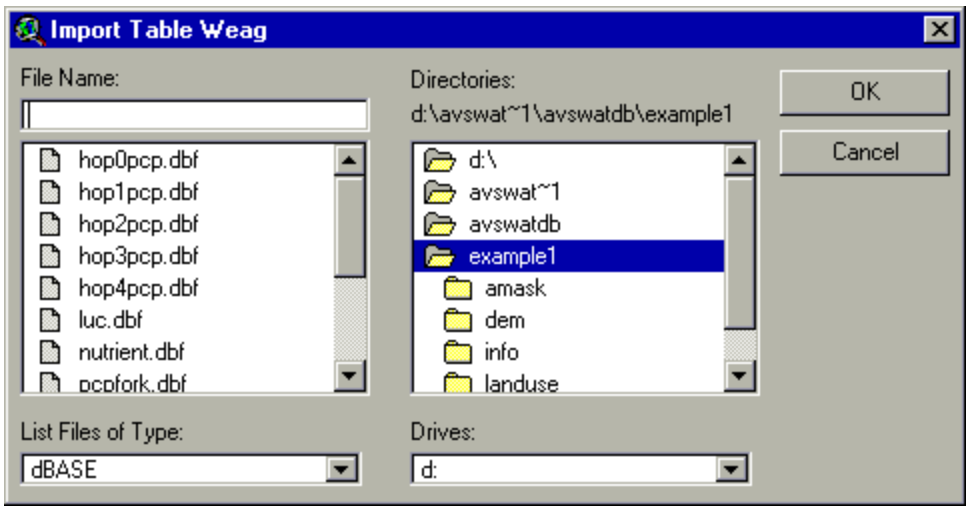


Figure 7.2

   Highlight the name of the weather generator location table and click **OK** or double click the selection.

iii. The weather locations will be loaded in the *SWAT View* as the *Weagages* point theme (with the *custom database* option, the data path will be displayed in the text box beside the button).

b. **Rainfall data** (optional). In this section, the user can import measured precipitation data for use in the project.

To use measured precipitation data, select the **Raingages** radio button

. Click the open file folder button next to the text box that appeared with the radio button that was selected

.

A file browser (Figure 7.3) allows you to select the dBASE Precipitation gage location table. The rain gage location table must be prepared by the user following the format described in Section 3.

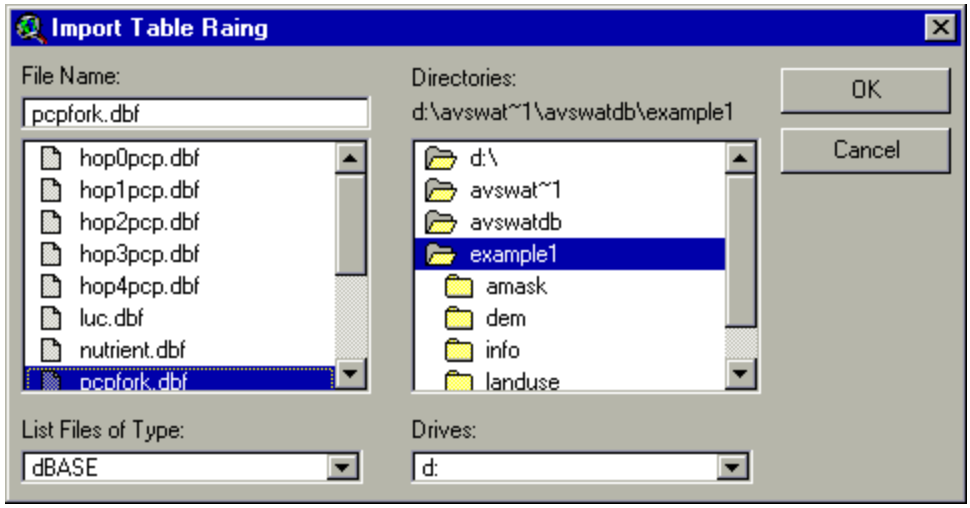
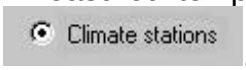


Figure 7.3

Highlight the name of the precipitation gage location table and click **OK** or double click the selection.

The precipitation gage locations will be loaded in the *SWAT View* as the *Raingages* point theme and the data path will be displayed in the text box.

c. **Temperature data** (optional). In this section, the user can import measured temperature data for use in the project.

To use measured temperature data, select the **Climate Stations** radio button 

. Click the open file folder button next to the text

box that appeared with the radio button that was selected

.

A file browser (Figure 7.4) allows you to select the dBASE Temperature gage location table. The temperature gage location table is prepared by the user following the format described in Section 3.

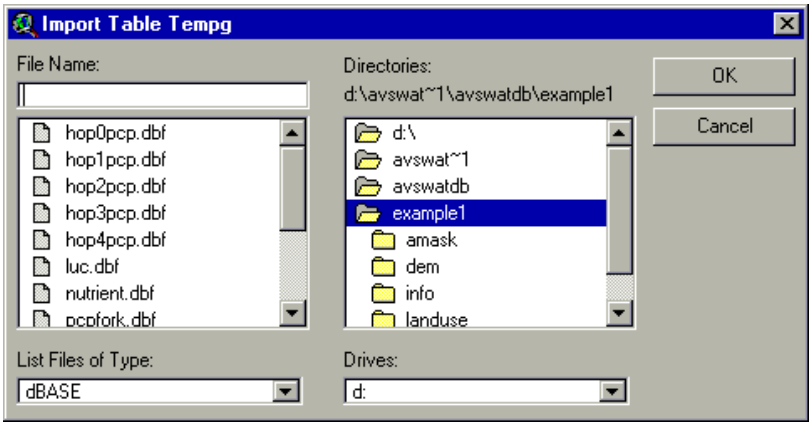


Figure 7.4

Highlight the name of the temperature gage location table and click **OK** or double click the selection.

The temperature gage locations will be loaded in the *SWAT View* as the *Tempgages* point theme, and the data path will be displayed in the text box.

d. **Solar Radiation data** (optional). In this section, the user can import measured solar radiation data for use in the project.

To use measured solar radiation data, select the **Solargages** radio button

. Click the open file folder button next to the text box that appeared with the radio button that was selected

.

A file browser (Figure 7.5) allows you to select the dBASE Solar radiation gage location table. The solar radiation gage location table must be prepared by the user following the format described in Section 3.

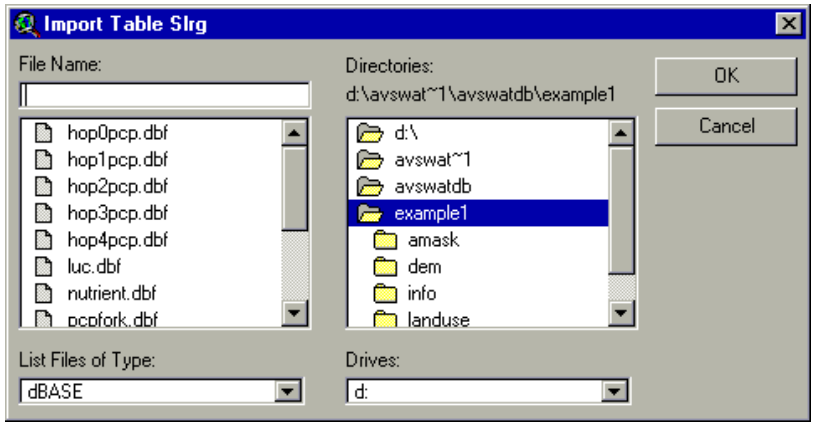


Figure 7.5

Highlight the name of the solar radiation gage location table and click **OK** or double click the selection.

The solar radiation gage locations will be loaded in the *SWAT View* as the *Solargages* point theme and the data path will be displayed in the text box.

e. **Wind Speed data** (optional). In this section, the user can import measured wind speed data for use in the project

To use measured wind speed data, select the **Windgages** radio button

. Click the open file folder button next to the text box that appeared with the radio button that was selected

.

A file browser (Figure 7.6) allows you to select the dBASE Wind speed gage location table. The wind speed gage location table must be prepared by the user following the format described in Section 3.

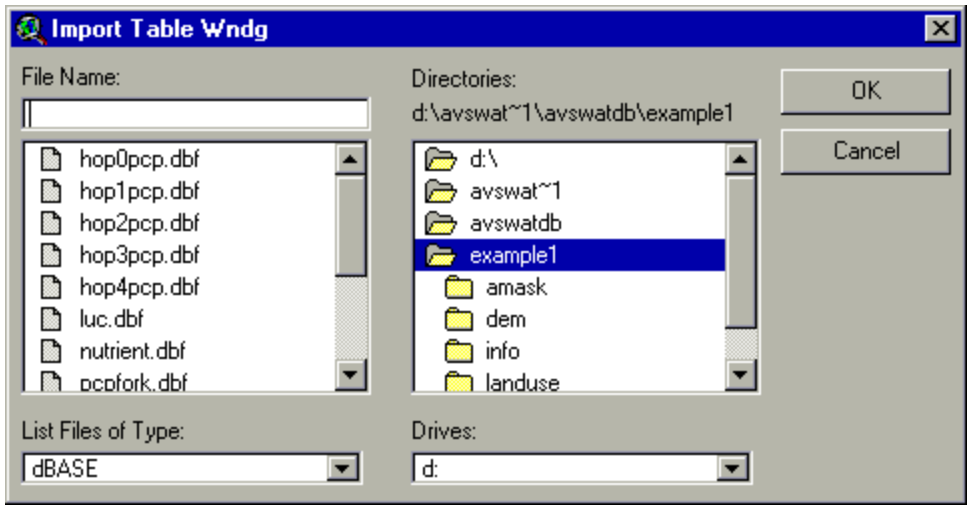


Figure 7.6

Highlight the name of the wind speed gage location table and click **OK** or double click the selection.

The wind speed gage locations will be loaded in the *SWAT View* as the *Windgages* point theme, and the data path will be displayed in the text box.

f. **Relative humidity data** (optional). In this section, the user can import measured relative humidity data for use in the project

To use measured relative humidity data, select the **Rel. Humidity gages** radio button

. Click the open file folder button next to the text box that appeared with the radio button that was selected

.

A file browser (Figure 7.7) allows you to select the dBASE Relative humidity gage location table. The relative humidity gage location table must be prepared by the user following the format described in Section 3.

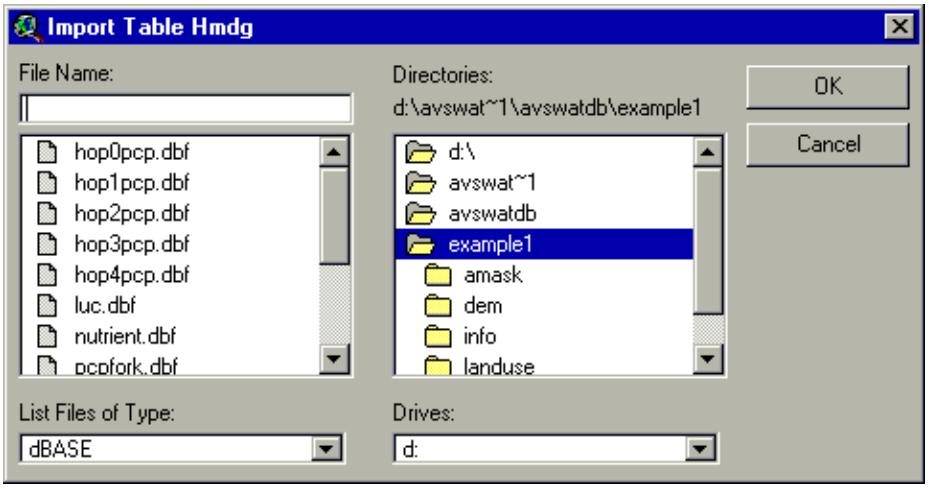


Figure 7.7

Highlight the name of the relative humidity gage location table and click **OK** or double click the selection.

The relative humidity gage locations will be loaded in the *SWAT View* as the *Humgages* point theme, and the data path will be displayed in the text box.

3. Once all weather data is specified and station locations are loaded, an **OK** button will appear in the lower right corner of the *Weather Data Definition* dialog box (Figure 7.8).

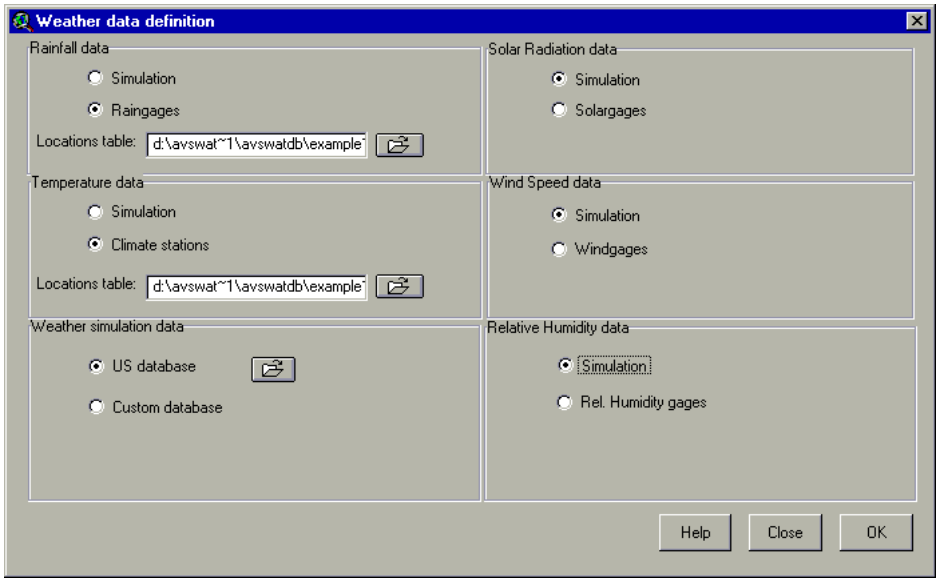


Figure 7.8

Click the **OK** button. This starts the set up of the weather database:

a. Data assigned to a subbasin is obtained from the closest station.

b. A -99.0 value is used to fill in skipped daily data and to fill in measured climate records so that all records have the same starting and ending date. The starting date use for measured climate data is the earliest starting date listed in any record while the ending date is the latest ending date listed in any record. The –99.0 value is used to call the weather generator to generate a value to replace the missing data during run time.

4. When setup of the weather database is complete, a message dialog is displayed (Figure 7.9).

Figure 7.9

Click **OK**.

SECTION 8: CREATION OF INPUT

The **Input** menu is one of four new menus introduced in the SWAT View. The items contained in the Input menu allow the user to build database files containing the information needed to generate default input for SWAT. Select the **Input** menu using the mouse or pressing the ALT+I keys. Several commands are listed on the **Input** menu (Figure 8.1). These commands are enabled in sequence (the next command is enabled only after the steps associated with the previous command are completed) and need to be processed only once for a project. However, if the user modifies the HRU distribution (see Section 6.1 and 6.2) after building the input database files, the **Input** menu commands will need to be processed again.

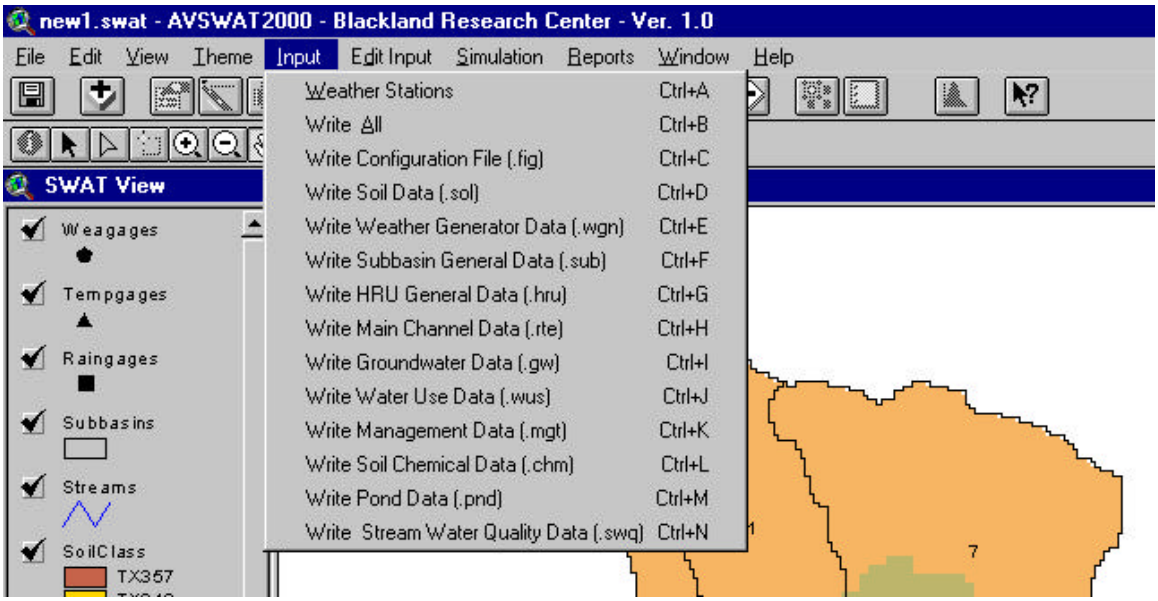


Figure 8.1

Before SWAT can be activated, the initial watershed input values must be defined. These values are set automatically based on the watershed delineation and land use-soil characterization (see Section 5 and Section 6) or from defaults.

There are two ways to build the initial values: activate the **Write All** command or the individual **Write** commands on the **Input** menu. The individual Write commands have been left on the menu for debugging purposes. The majority of users will perform the first option.

SECTION 8.1: WRITE ALL

1. Select the **Write All** item from the **Input** menu on the *SWAT View* (or press `Ctrl+B`) (Figure 8.2).

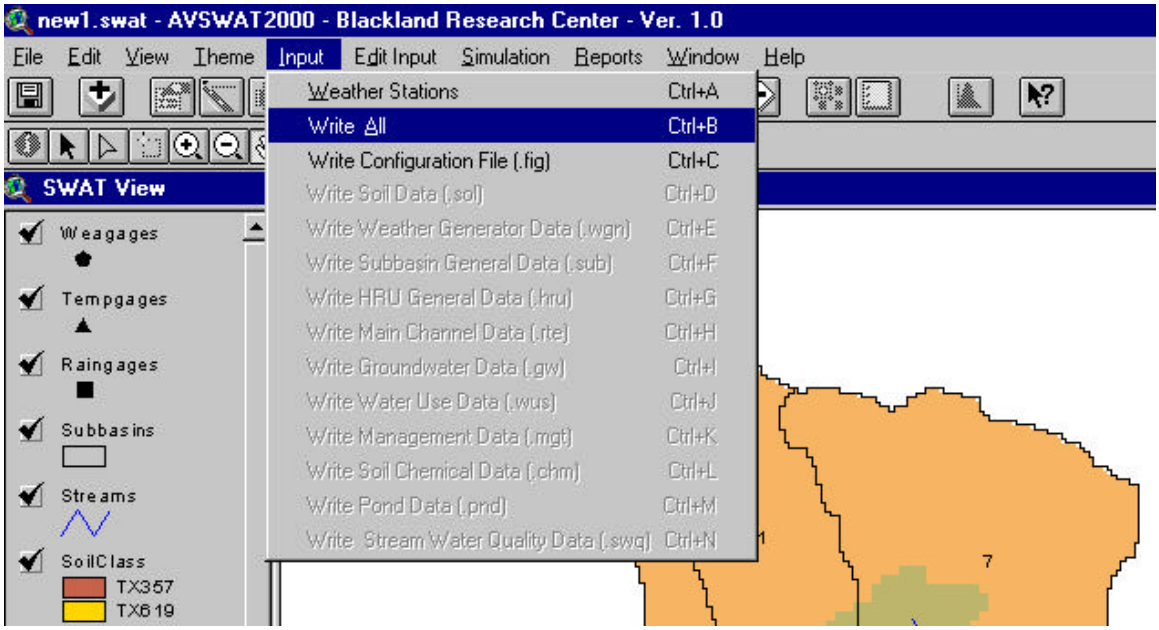


Figure 8.2

a. The Write All Inputs dialog box will appear to verify that all input is to be generated at once (Screen 8.3). Click **No** to abort. Click **Yes** to proceed.

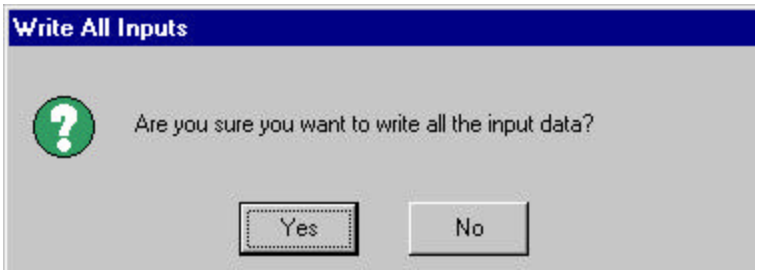


Figure 8.3

b. The Current Status of Input Data message box that lists the different databases being generated appears (Figure 8.4). As each database is completed, a check mark appears next to the database name and a beep sounds.

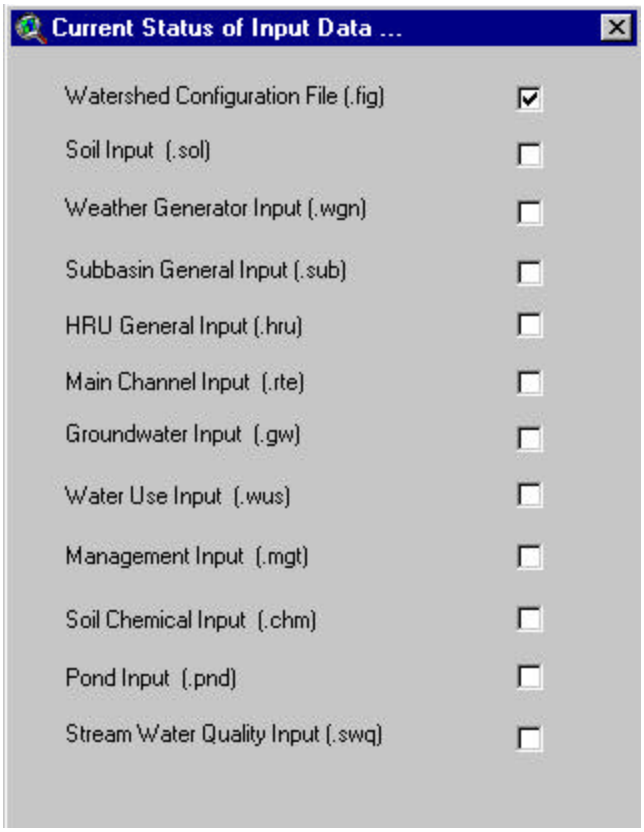


Figure 8.4

c. When writing of the *.sub* (subbasin general input) database begins, a prompt box appears (Figure 8.5) offering two options for defining Manning's roughness factor for the tributary channels in all subbasins.

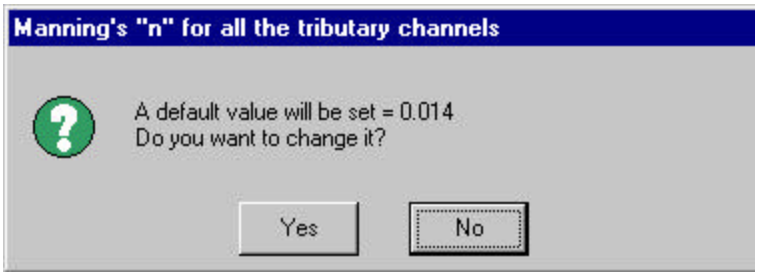


Figure 8.5

Click **No** to accept the default value (0.014). Click **Yes** to open the Manning's Roughness Factor "n" dialog (Figure 8.6).

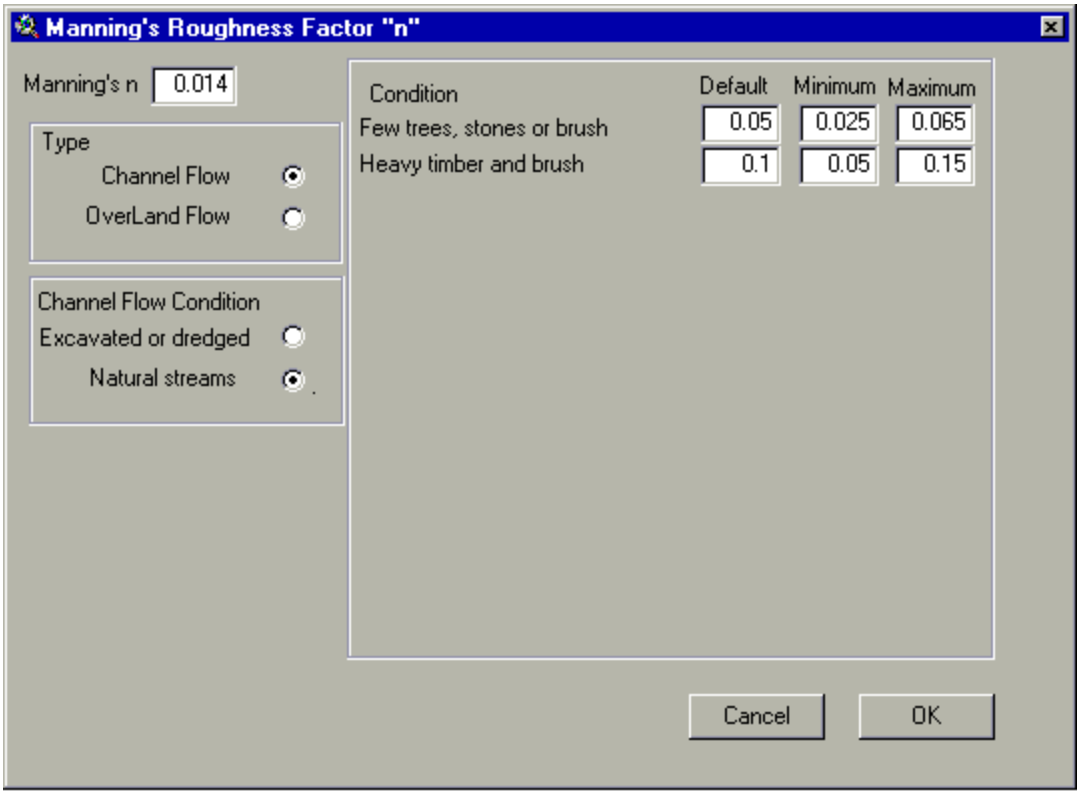


Figure 8.6

Enter the input value in the box labeled "Manning's n" or set its value by browsing the tabulated values within the dialog. Click **Cancel** to use the default value (0.014). Click **OK** to use the current value in the input box.

d. When writing of the *.rte* (main channel input) database begins, a prompt box appears (Figure 8.7) offering two options for defining Manning's roughness factor for the main channels in all subbasins.

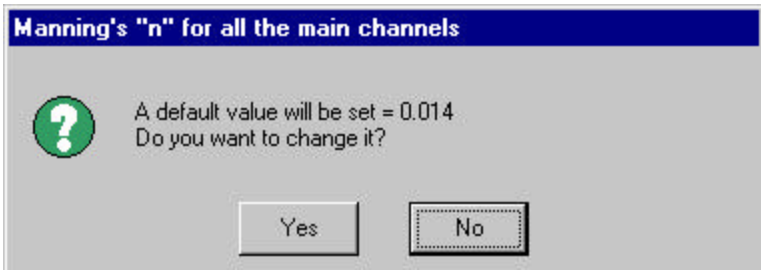


Figure 8.7

Click **No** to accept the default value (0.014). Click **Yes** to open the Manning's Roughness Factor "n" dialog (Figure 8.6).

Enter the input value in the box labeled "Manning's n" or set its value by browsing the tabulated values within the dialog. Click **Cancel** to use the default value (0.014). Click **OK** to use the current value in the input box.

e. When writing of the *.mgt* (management input) database begins, a prompt box appears with two options for defining plant growth heat units (Figure 8.8):

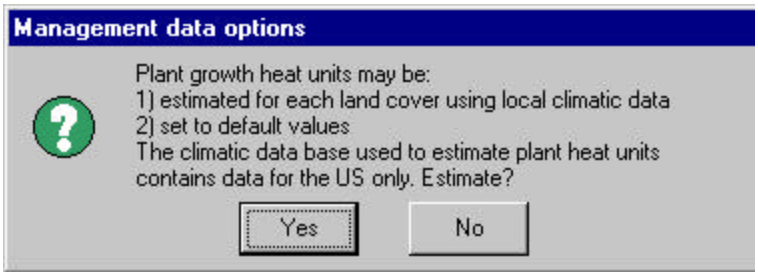


Figure 8.8

i. **Yes** may be chosen only for watersheds in the United States. If you click **Yes**, the plant heat units will be calculated from local climatic parameters stored in an internal weather generator database.

ii. If you click **No**, a dialog box will appear asking for a default heat unit value (Figure 8.9). The default heat unit value will be used for all land cover/plants within the watershed. Click **Cancel** to abort. Click **OK** to use the current value in the input box.

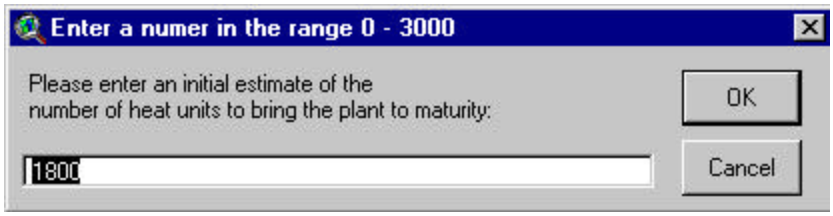


Figure 8.9

f. When all databases have been built, a message box will be displayed (Figure 8.10).

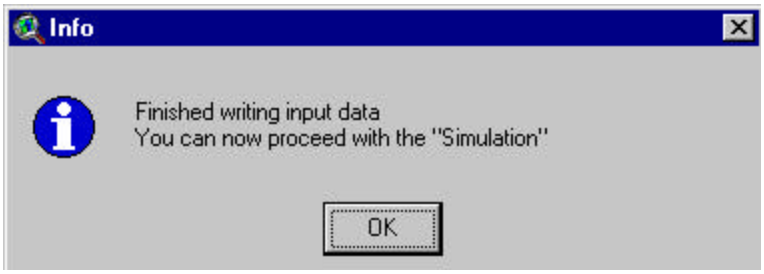


Figure 8.10

Click **OK** to proceed.

You can activate the **Write All** command in the same way at any later time. In this case, a confirmation dialog pops up (Figure 8.11).

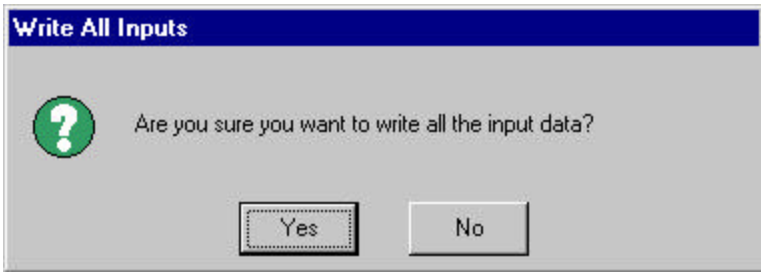


Figure 8.11

Click **No** to abort the process. Click **Yes** to proceed.

> Note: If you make changes to inputs using the commands under the **Edit Inputs** menu and then reactivate the **Write All** command under the **Input** menu, all your editing changes will be overwritten by the default values.

You can abort any current writing process by clicking the stop button located in the lower right corner of the ArcView screen near the status bar. You can activate the **Write All** command at any later time. In this case a different confirmation dialog pops up (Figure 8.12).

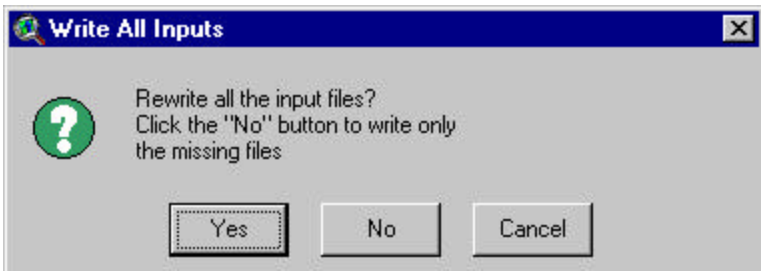


Figure 8.12

Click **No** to write only the missing initial inputs. Click **Yes** to proceed to rewrite all the initial inputs. Click **Cancel** to abort any writing action.

g. When all of the default inputs have been generated, you can move to the **Simulation** menu and make a SWAT run (See Section 10) or edit the default inputs using the editors activated under the **Edit Inputs** menu (See Section 9).

SECTION 8.2: INDIVIDUAL WRITE COMMANDS

1. As an alternative to writing all the input files at once, they can be written individually. If this method is used to write the input database files, the database files must be written (selected) in the sequence in which they are presented in the *Input* menu: watershed configuration file (Figure 8.13), soil data, weather generator data, general subbasin data, hru general data, main channel data, groundwater data, water use data, management data, soil chemical data, pond data, and stream water quality data.

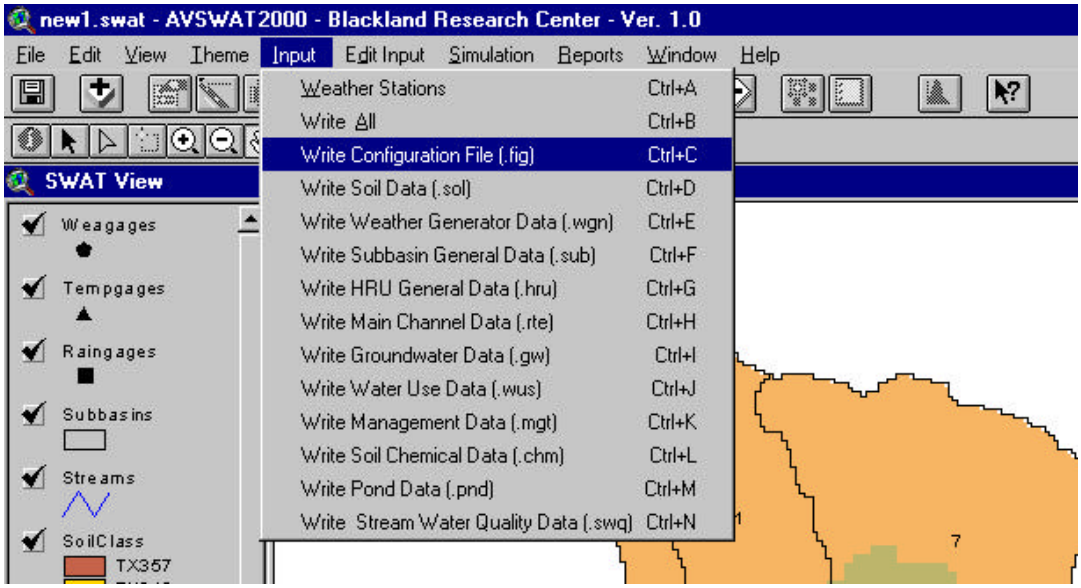


Figure 8.13

a. To write SWAT input databases individually, begin by selecting the **Write Configuration File (.fig)** command from the **Input** menu (or press `Ctrl+C`). This selection generates the watershed configuration file (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 2 and Appendix B). When the file has been written, a message box will pop up (Figure 8.14).

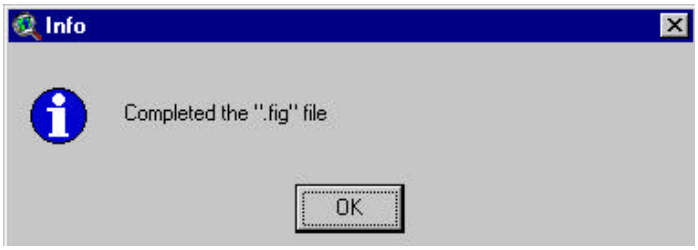


Figure 8.14

Click **OK** to proceed.

b. From the **Input** menu, select **Write Soil Data (.sol)** (or press `Ctrl+D`). This selection generates the HRU soil data (see the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 22). When the data files have been written, a message box will pop up (Figure 8.15).

Figure 8.15

Click **OK** to proceed.

c. From the **Input** menu, select **Write Weather Generator Data (.wgn)** (or press `Ctrl+E`). This selection generates the subbasin weather generator data (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 13). When the data files have been written, a message box will pop up (Figure 8.16).

Figure 8.16

Click **OK** to proceed.

d. From the **Input** menu, select **Write General Subbasin Data (.sub)** (or press `Ctrl+F`). This selection generates the general subbasin data (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 6). A prompt box pops up (as described in 1.c under Section 8.1). When the data files have been written, a message box will pop up (Figure 8.17).

Figure 8.17

Click **OK** to proceed.

e. From the **Input** menu, select **Write General HRU Data (.hru)** (or press `Ctrl+G`). This selection generates the general HRU data (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 19). When the data files have been written, a message box will pop up (Figure 8.18).

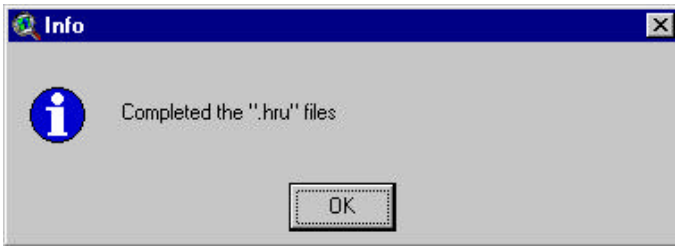


Figure 8.18

Click **OK** to proceed.

f. From the **Input** menu, select **Write Main Channel Data (.rte)** (or press `Ctrl+H`). This selection generates the subbasin main channel data (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 25). A prompt box pops up (as described in 1.d under Section 8.1). When the data files have been written, a message box will pop up (Figure 8.19).

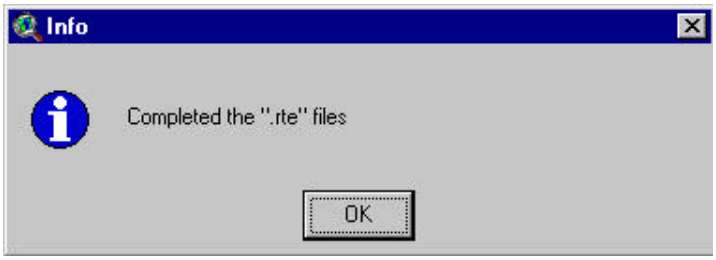


Figure 8.19

Click **OK** to proceed.

g. From the **Input** menu, select **Write Groundwater Data (.gw)** (or press `Ctrl+I`). This selection generates the HRU groundwater data (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 24). When the data files have been written, a message box will pop up (Figure 8.20).

Figure 8.20

Click **OK** to proceed.

h. From the **Input** menu, select **Write Water Use Data (.wus)** (or press `Ctrl+J`). This selection generates the subbasin water use data (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 21). When the data files has been written, a message box will pop up (Figure 8.21).

Figure 8.21

Click **OK** to proceed.

i. From the **Input** menu, select **Write Management Data (.mgt)** (or press `Ctrl+K`). This selection generates the HRU management data (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 20). A prompt box pops up (see description under 1.e under Section 8.1). When the data files have been written, a message box will pop up (Figure 8.22).

Figure 8.22

Click **OK** to proceed.

j. From the **Input** menu, select **Write Soil Chemical Data (.chm)** (or press `Ctrl+L`). This selection generates the HRU soil chemical data (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 23). When the data files have been written, a message box will pop up (Figure 8.23).

Figure 8.23

Click **OK** to proceed.

k. From the **Input** menu, select **Write Pond Data (.pnd)** (or press `Ctrl+M`). This selection generates the subbasin pond data (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 28). When the data files have been written, a message box will pop up (Figure 8.24).

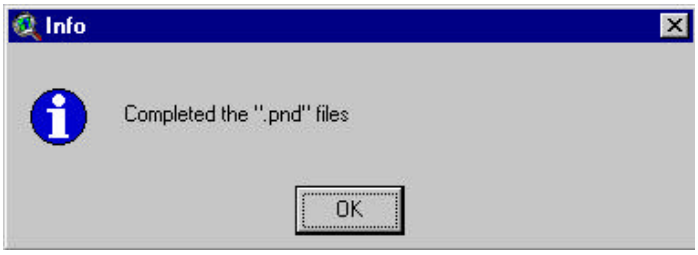


Figure 8.24

Click **OK** to proceed.

l. From the **Input** menu, select **Write Stream Water Quality Data (.swq)** (or press `Ctrl+N`). This selection generates the subbasin stream water quality data (see *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 27). When the data files have been written, a message box will pop up (Figure 8.25).

Figure 8.25

Click **OK** to proceed.

m. When all of the default inputs have been generated, you can move to the **Simulation** menu and make a SWAT run (See Section 10) or edit the default inputs using the editors activated under the **Edit Inputs** menu (See Section 9).

SECTION 9: INPUT MODIFICATION—POINT SOURCES

The **Edit Input** menu is one of four new menus introduced in the SWAT View. The **Edit Input** menu allows you to edit the SWAT model databases and the watershed database files containing the current inputs for the SWAT model. Select the **Edit Input** menu using the mouse or by pressing the ALT+D keys. Five items are listed on the **Edit Input** menu (Figure 9.1).

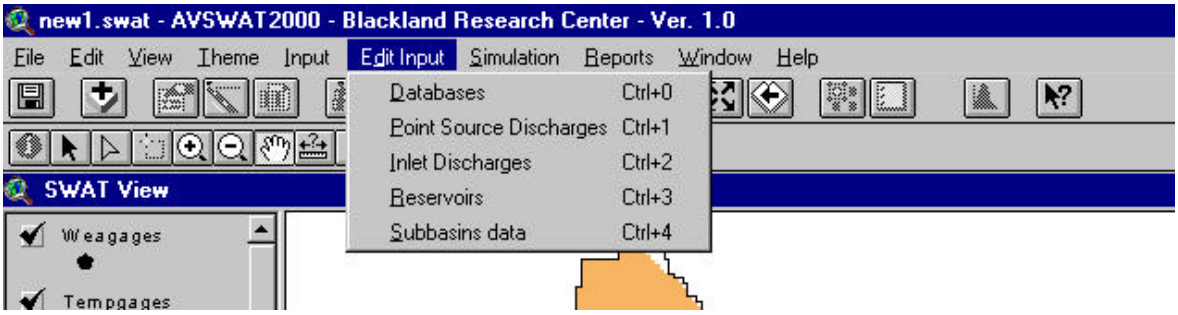


Figure 9.1

The second item of the **Edit Input** menu allows the user to edit point source discharge loadings. Point source discharges are added to the watershed configuration during the watershed discretization (see Section 5).

1. Select **Point Source Discharges** from the **Edit Input** menu (or press Ctrl+1) (Figure 9.2).

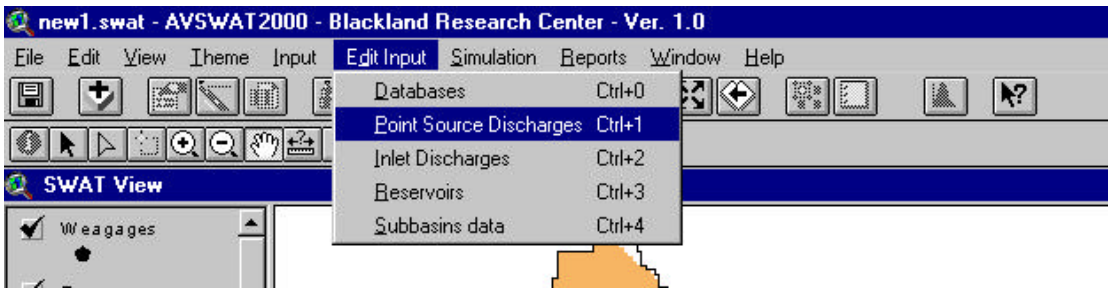


Figure 9.2

If there are no point sources in the watershed (See Section 5), a dialog box warns the user (Figure 9.3).

Figure 9.3

The Edit Point Discharges Inputs dialog pops up (Figure 9.4) if at least one point source of discharge was set in the watershed (see Section 5).

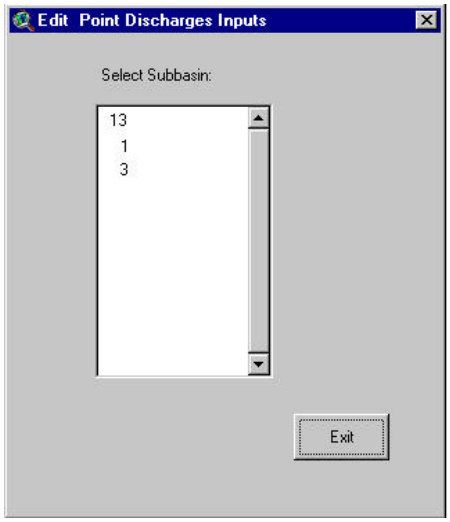


Figure 9.4

All subbasins containing point sources are listed.

2. To edit point source data for a subbasin, click on the number of the subbasin on the list. The Point Discharges Data dialog box will open, displaying the data for the point source in the selected subbasin (Figure 9.5).

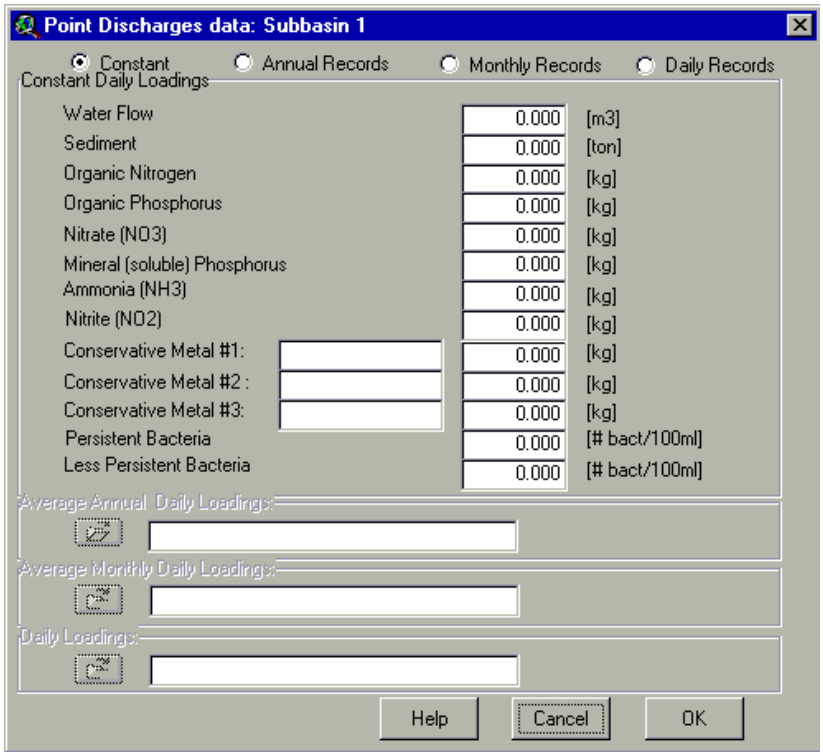


Figure 9.5

Point source data may be summarized in one of four formats: constant daily loadings, average annual loadings, average monthly loadings, and daily loadings. Four radio buttons are located at the top of the dialog box for the user to select the data format they prefer.

a. **Constant Daily Loadings.** By default, this radio button will be enabled. If the point source loadings are to be input as constant daily loadings, verify that the radio button labeled "Constant" is selected.

This option allows the user to enter the following data: average daily water loading [m^3], sediment loading [tons], organic Nitrogen [kg], organic Phosphorus [kg], Nitrate (NO_3) [kg], mineral (soluble) Phosphorus loading [kg], Ammonia (NH_3) [kg], Nitrite (NO_2) [kg], Conservative Metal #1 [kg], Conservative Metal #2, Conservative Metal #3 [kg], Persistent Bacteria and Less Persistent Bacteria (both in [# bacteria/100 ml]) as required by the SWAT model (see the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 31). This data represents average daily loadings input to the stream network every day of the simulation.

To define the constant daily loadings, enter the data in the section of the dialog box labeled Constant Daily Loadings (Figure 9.6). If data for any conservative metal is input, the type of metal should be specified.

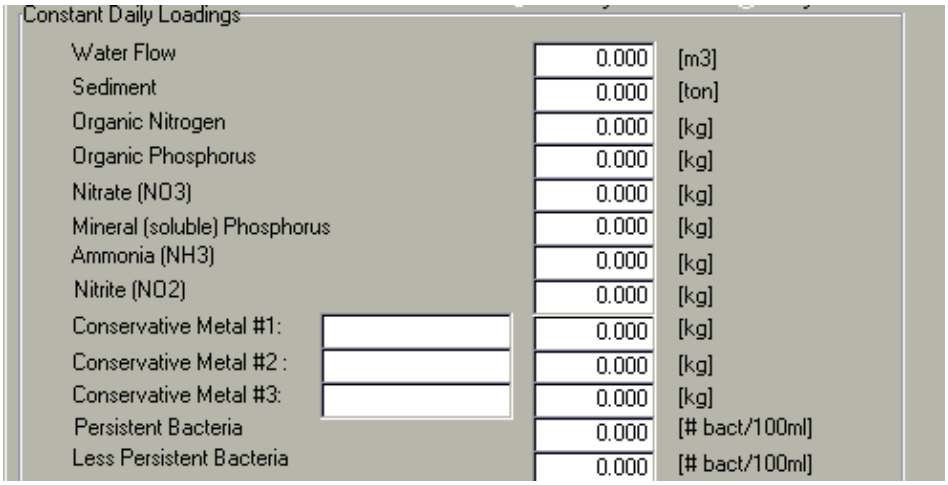
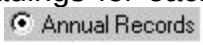


Figure 9.6

b. **Average annual daily loadings.** If the point source loadings are to be input as average daily loadings for each year, verify that the radio button labeled "Annual Records" 

is selected.

The Average Annual Daily Loadings section in the dialog box is now enabled (Figure 9.7).

Figure 9.7

This option requires the user to summarize daily loadings by year (see the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 31). Prior to creating the project, a data table should be set up following the format specified for Annual Loadings in Section 3.3.

i. Click the open file folder button to the left of the text box.

ii. A prompt box will appear so that the user can specify the format of the file containing the average annual point source loadings (Figure 9.8). Select dBASE (.dbf) or text file (.txt) and click **OK**.

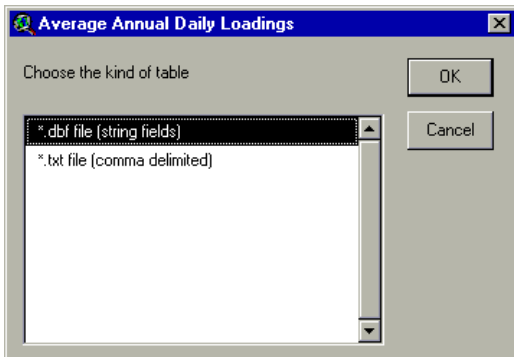


Figure 9.8

iii. A browser will be displayed, allowing the user to select the dBASE file (.dbf) or text file (.txt) file containing the point source data from disk (Figure 9.9).

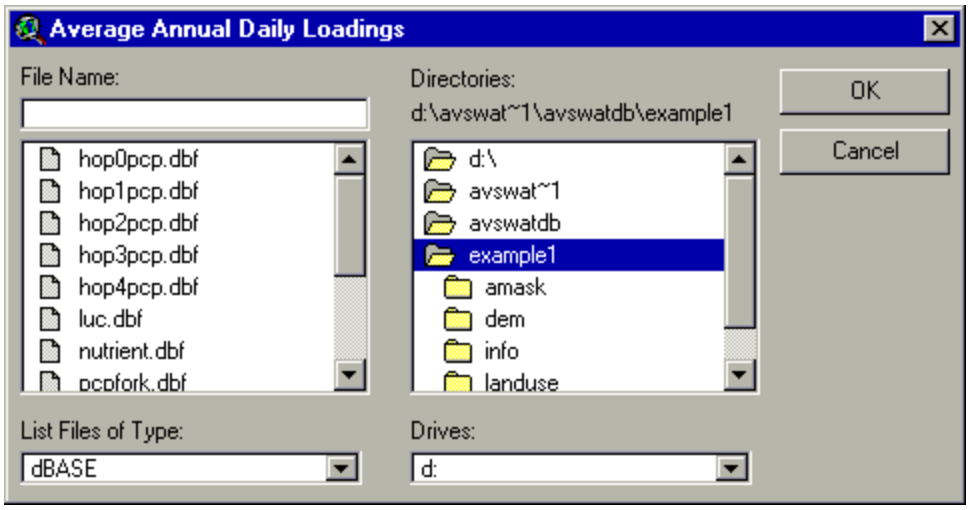


Figure 9.9

iv. Select the file and click **OK** (or double click the selection).

b. **Average monthly daily loadings.** If the point source loadings are to be input as average daily loadings summarized on a monthly basis, verify that the radio button labeled "Monthly Records"

is selected.

The Average Monthly Daily Loadings section of the dialog box is now enabled (Figure 9.10).

Figure 9.10

This option requires the user to summarize average daily loadings by month (see the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 31). Prior to creating the project, a data table should be set up following the format specified for Monthly Loadings in Section 3.3.

i. Click on the open file folder button to the left of the text box.

ii. A prompt box will appear so that the user can specify the format of the file containing the monthly point source loadings (Figure 9.11). Select dBASE (.dbf) or text file (.txt) and click **OK**.

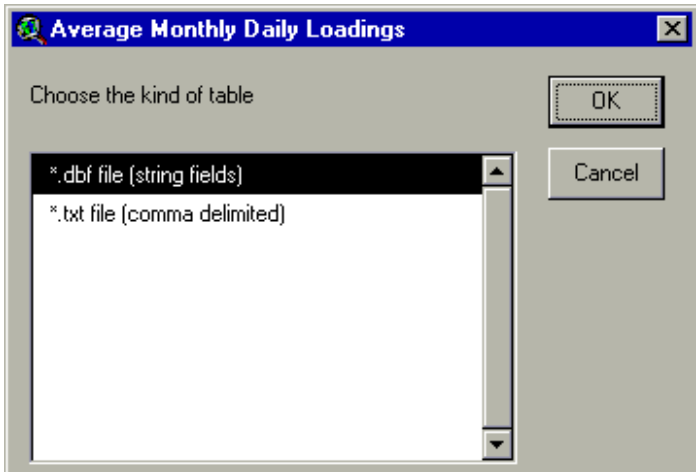


Figure 9.11

iii. A browser will be displayed, allowing the user to select the dBASE file (.dbf) or text file (.txt) file containing the point source data from disk (Figure 9.12).

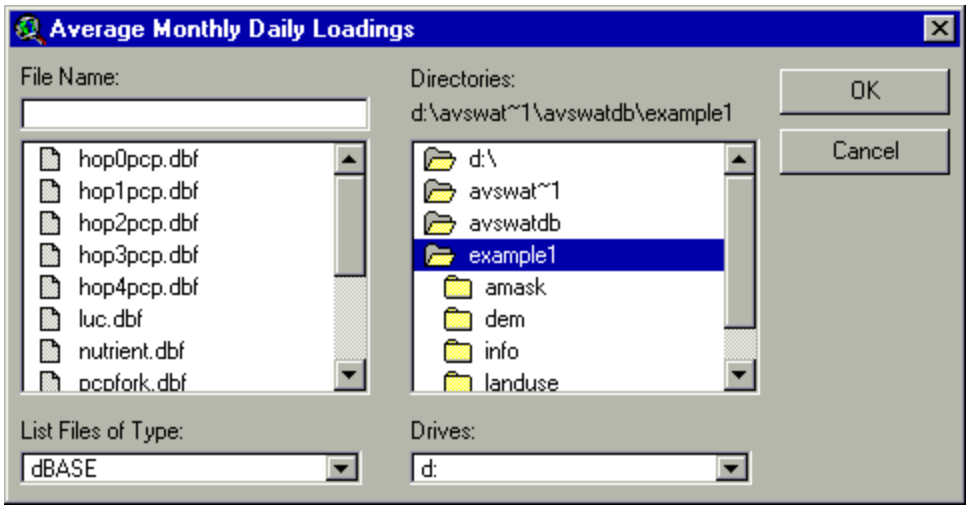


Figure 9.12

iv. Select the file and click **OK** (or double click the selection).

b. **Daily Loadings.** If the point source loadings are to be input summarized on a daily basis, verify that the radio button labeled “Daily Records”

is selected.

The Daily Loadings section of the dialog box is now enabled (Figure 9.13).

Figure 9.13

This option requires the user to summarize loadings by day (see the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 31).

Prior to creating the project, a data table should be set up following the format specified for Daily Loadings in Section 3.3.

i. Click on the open file folder button to the left of the text box.

ii. A prompt box will appear so that the user can specify the format of the file containing the average daily point source loadings (Figure 9.14). Select dBASE (.dbf) or text file (.txt) and click **OK**.

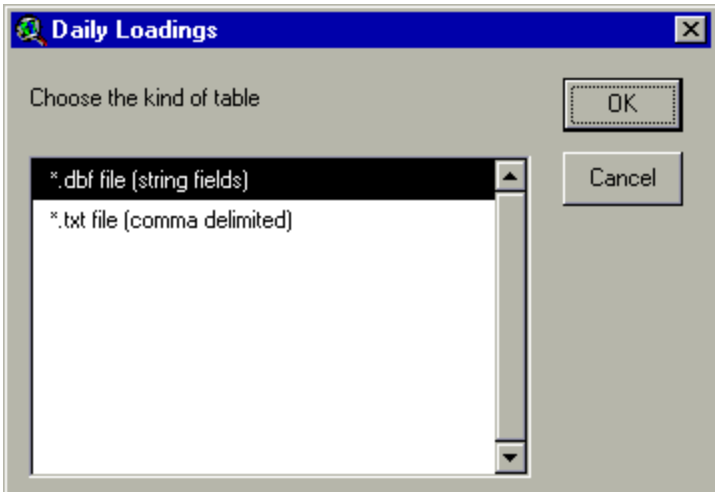


Figure 9.14

iii. A browser will be displayed, allowing the user to select the dBASE file (.dbf) or text file (.txt) file containing the point source data from disk (Figure 9.15).

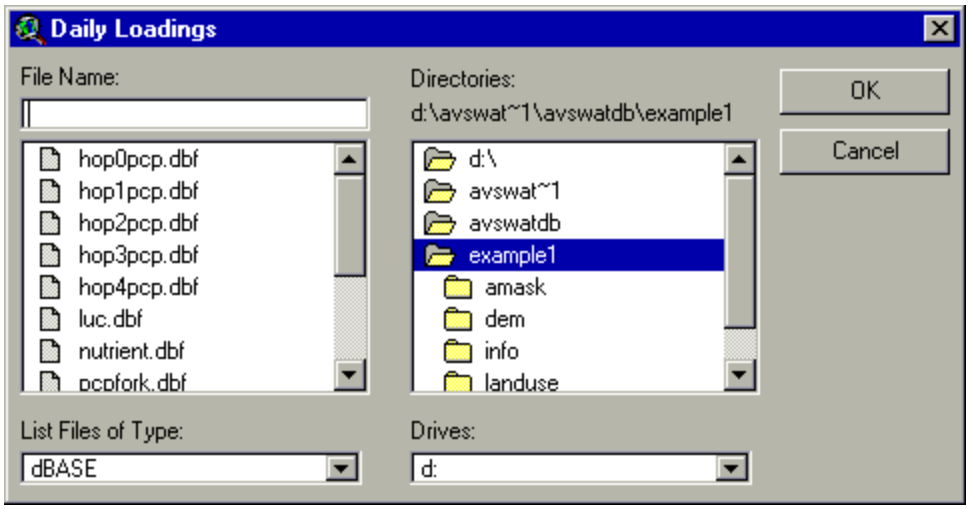


Figure 9.15

iv. Select the file and click **OK** (or double click the selection).

3. Click **Exit** in the Edit Point Discharges Input dialog box to return to the current project.

SECTION 10: INPUT MODIFICATION—INLET DISCHARGES

The **Edit Input** menu is one of four new menus introduced in the SWAT View. The **Edit Input** menu allows you to edit the SWAT model databases and the watershed database files containing the current inputs for the SWAT model. Select the **Edit Input** menu using the mouse or by pressing the ALT+D keys. Five items are listed on the **Edit Input** menu (Figure 10.1).

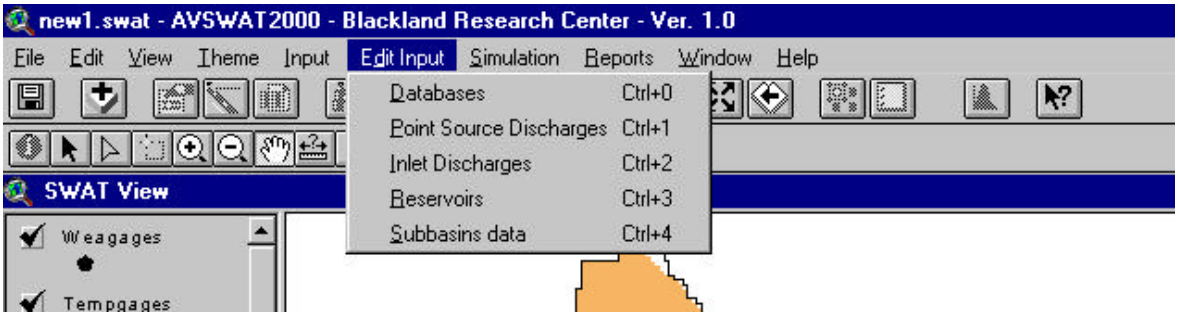


Figure 10.1

The third item of the **Edit Input** menu allows the user to edit the inlet discharge loadings. Inlet discharges are added to the watershed configuration during the watershed discretization (see Section 5).

1. Select **Inlet Discharges** from the **Edit Input** menu in the *SWAT View* (or press Ctrl+2) (Figure 10.2).

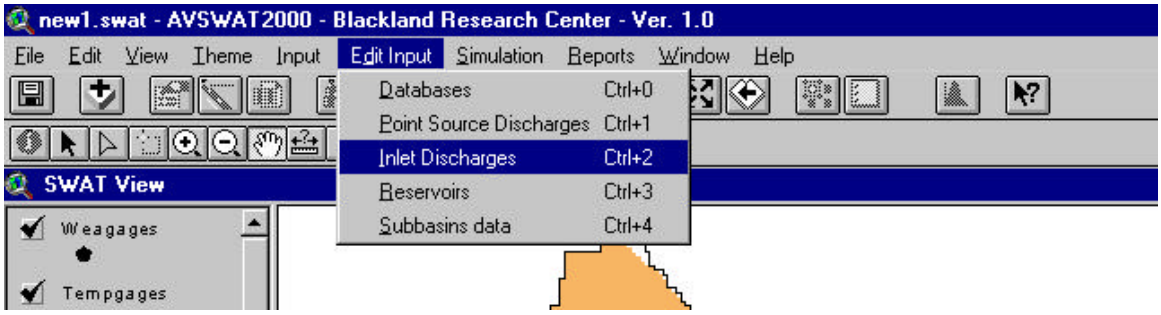


Figure 10.2

If no inlet sources are defined in the watershed (See Section 5), a dialog box notifies the user (Figure 10.3).

Figure 10.3

The Edit Inlet Discharges Inputs dialog pops up (Figure 10.4) if at least one inlet discharge was set in the watershed (See Section 5).

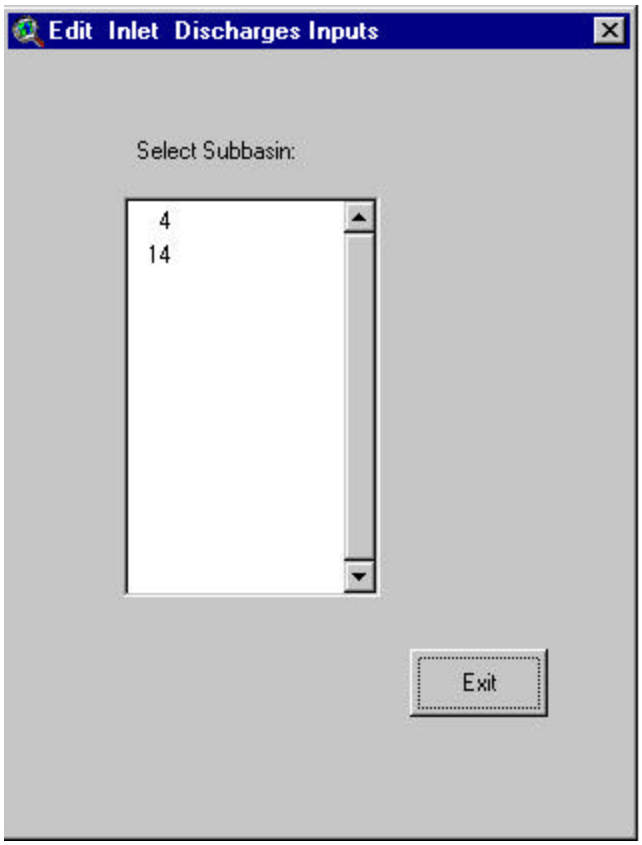


Figure 10.4

All subbasins into which inlet discharges drain will be listed.

2. To edit the inlet discharges data associated with a subbasin, click on the number of the subbasin on the list. The Inlet Discharges data dialog box will open, displaying the data for the inlet discharge associated with the selected subbasin (Figure 10.5).

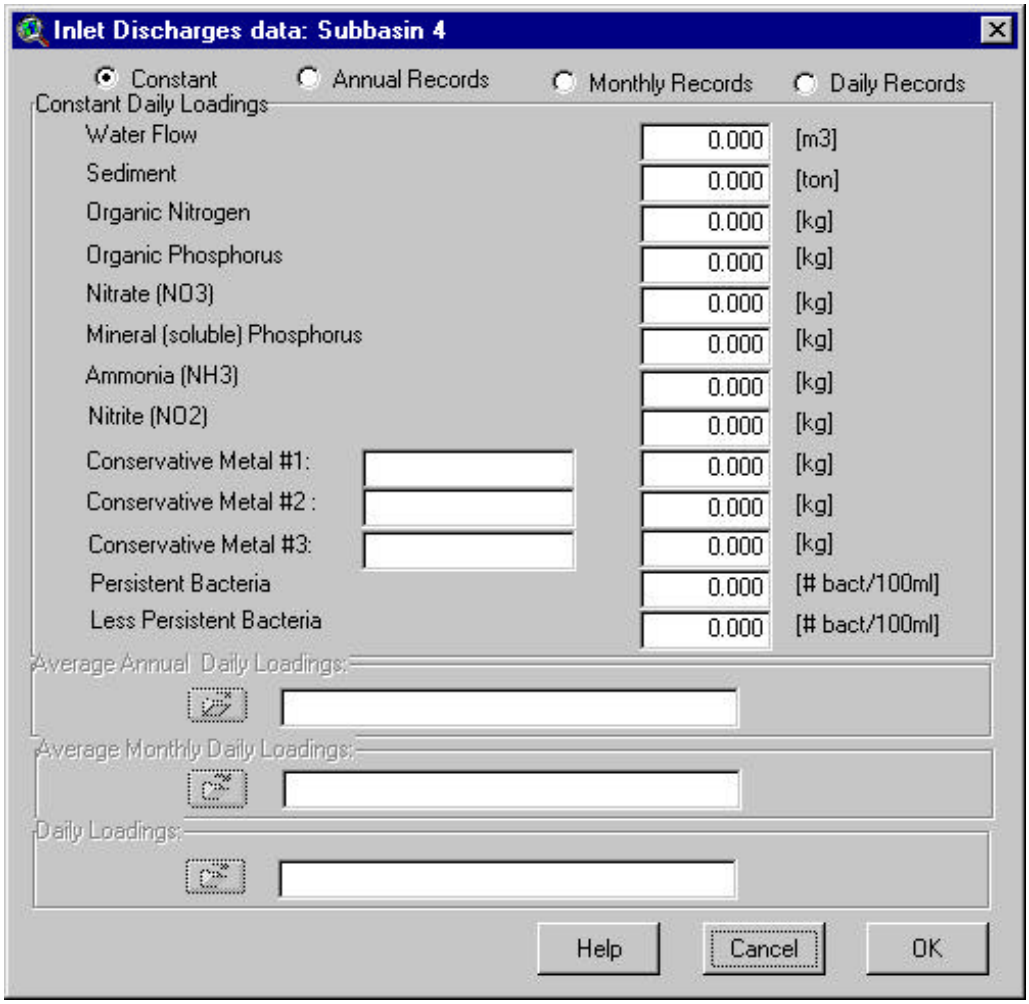


Figure 10.5

Inlet discharge data may be summarized in one of four formats: constant daily loadings, average annual loadings, average monthly loadings, and daily loadings. Four radio buttons are located at the top of the dialog box for the user to select the data format they prefer.

a. **Constant daily loadings.** By default, this radio button will be enabled. If the inlet discharge loadings are to be input as constant daily loadings, verify that the radio button labeled "Constant" is selected.

This option allows the user to enter the following data: average daily water loading [m^3], sediment loading [tons], organic Nitrogen [kg], organic Phosphorus [kg], Nitrate (NO_3) [kg], mineral (soluble) Phosphorus loading [kg], Ammonia (NH_3) [kg], Nitrite (NO_2) [kg], Conservative Metal #1 [kg], Conservative Metal #2, Conservative Metal #3 [kg], Persistent Bacteria and Less Persistent Bacteria (both in [# bacteria/100 ml]) as required by

the SWAT model (see the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 31). This data represents average daily loadings input to the stream network every day of the simulation.

To define the constant daily loadings, enter the data in the section of the dialog box labeled Constant Daily Loadings (Figure 10.6). If data for any conservative metal is input, the type of metal should be specified.

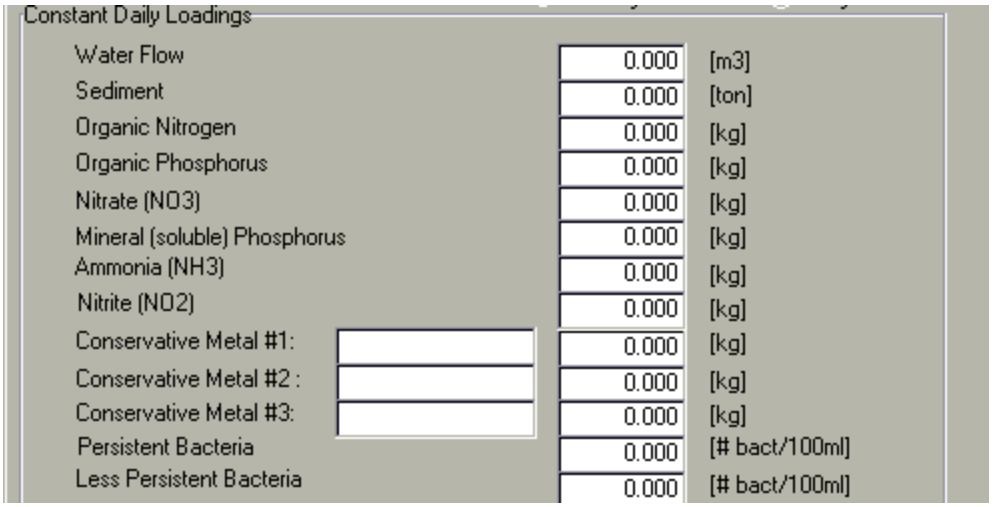
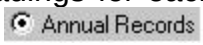


Figure 10.6

b. **Average annual daily loadings.** If the inlet discharge loadings are to be input as average daily loadings for each year, verify that the radio button labeled “Annual Records”  is selected.

The Average Annual Daily Loadings section in the dialog box is now enabled (Figure 10.7).

Figure 10.7

This option requires the user to summarize daily loadings by year (see the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 31). Prior to creating the project, a data table should be set up following the format specified for Annual Loadings in Section 3.3.

i. Click the open file folder button to the left of the text box.

ii. A prompt box will appear so that the user can specify the format of the file containing the average annual inlet discharge loadings (Figure 10.8). Select dBASE (.dbf) or text file (.txt) and click **OK**.

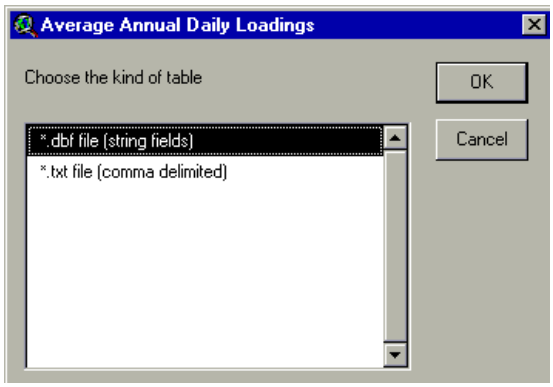


Figure 10.8

iii. A browser will be displayed, allowing the user to select the dBASE file (.dbf) or text file (.txt) file containing the inlet discharge data from disk (Figure 10.9).

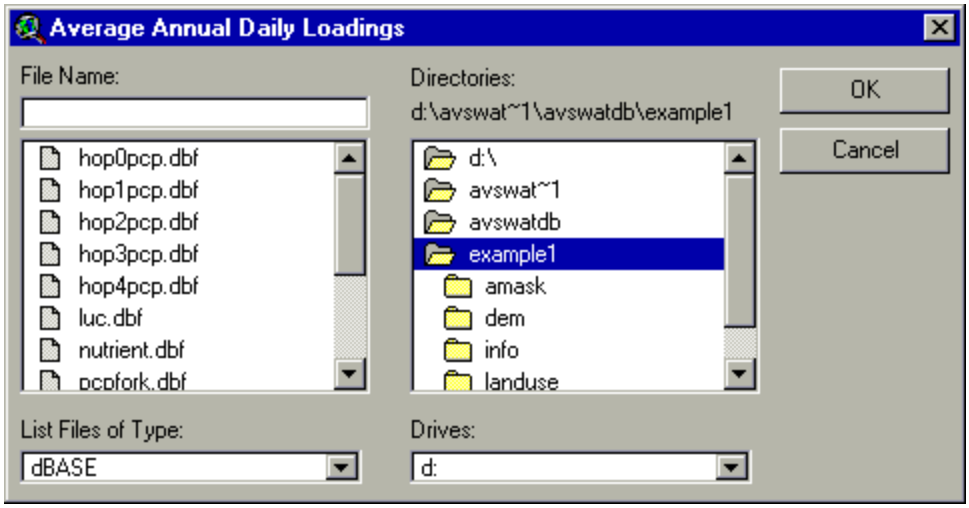


Figure 10.9

iv. Select the file and click **OK** (or double click the selection).

b. **Average monthly daily loadings.** If the inlet discharge loadings are to be input as average daily loadings summarized on a monthly basis, verify that the radio button labeled “Monthly Records”

is selected.

The Average Monthly Daily Loadings section of the dialog box is now enabled (Figure 10.10).

Figure 10.10

This option requires the user to summarize average daily loadings by month (see the *Soil and Water Assessment Tool User's Manual, Version*

2000, Chapter 31). Prior to creating the project, a data table should be set up following the format specified for Monthly Loadings in Section 3.3.

i. Click on the open file folder button to the left of the text box.

ii. A prompt box will appear so that the user can specify the format of the file containing the monthly inlet discharge loadings (Figure 10.11). Select dBASE (.dbf) or text file (.txt) and click **OK**.

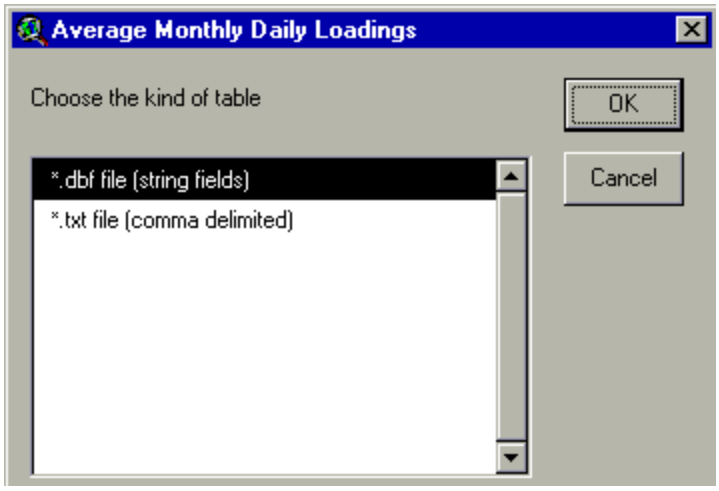


Figure 10.11

iii. A browser will be displayed, allowing the user to select the dBASE file (.dbf) or text file (.txt) file containing the inlet discharge data from disk (Figure 10.12).

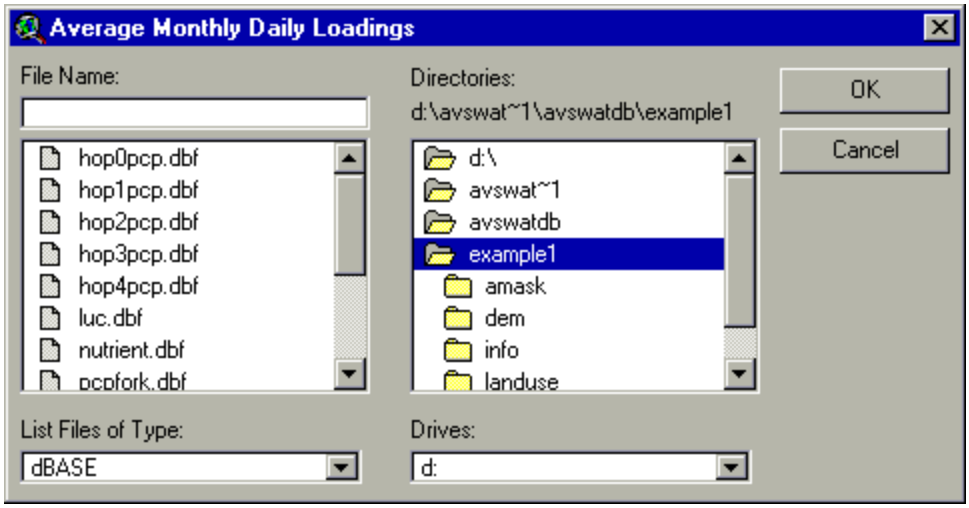


Figure 10.12

iv. Select the file and click **OK** (or double click the selection).

d. Daily Loadings. If the inlet discharge loadings are to be input summarized on a daily basis, verify that the radio button labeled “Daily Records”

is selected.

The Daily Loadings section of the dialog box is now enabled (Figure 10.13).

Figure 10.13

This option requires the user to summarize loadings by day (see the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 31). Prior to creating the project, a data table should be set up following the format specified for Daily Loadings in Section 3.3.

i. Click on the open file folder button to the left of the text box.

ii. A prompt box will appear so that the user can specify the format of the file containing the average daily inlet discharge loadings (Figure 10.14). Select dBASE (.dbf) or text file (.txt) and click **OK**.

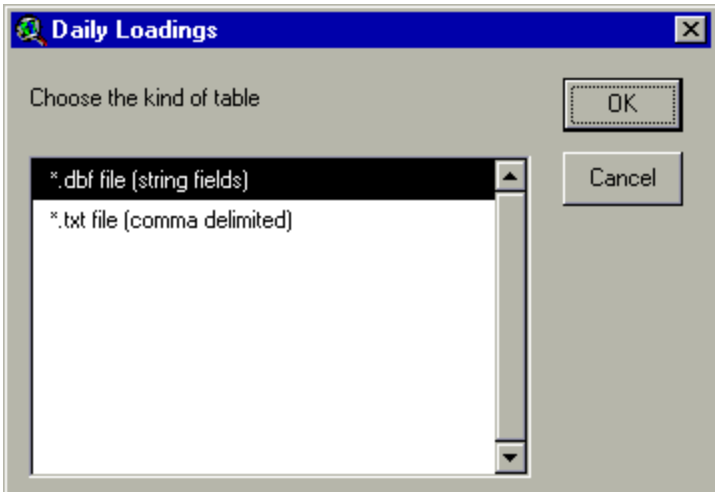


Figure 10.14

iii. A browser will be displayed, allowing the user to select the dBASE file (.dbf) or text file (.txt) file containing the inlet discharge data from disk (Figure 10.15).

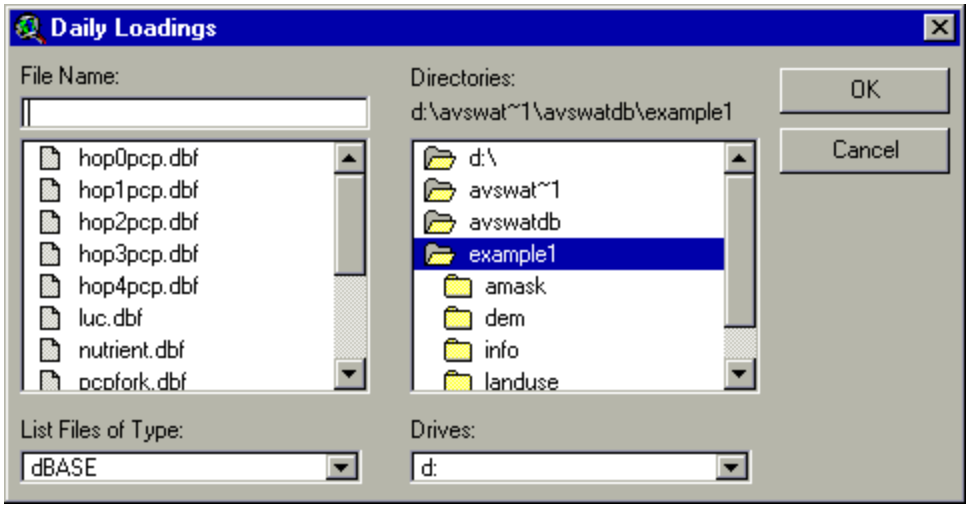


Figure 10.15

iv. Select the file and click **OK** (or double click the selection).

3. Click **Exit** in the Edit Inlet Discharges Input dialog box to return to the current project.

SECTION 11: INPUT MODIFICATION—RESERVOIRS

The **Edit Input** menu is one of four new menus introduced in the SWAT View. The **Edit Input** menu allows you to edit the SWAT model databases and the watershed database files containing the current inputs for the SWAT model. Select the **Edit Input** menu using the mouse or by pressing the ALT+D keys. Five items are listed on the **Edit Input** menu (Figure 11.1).

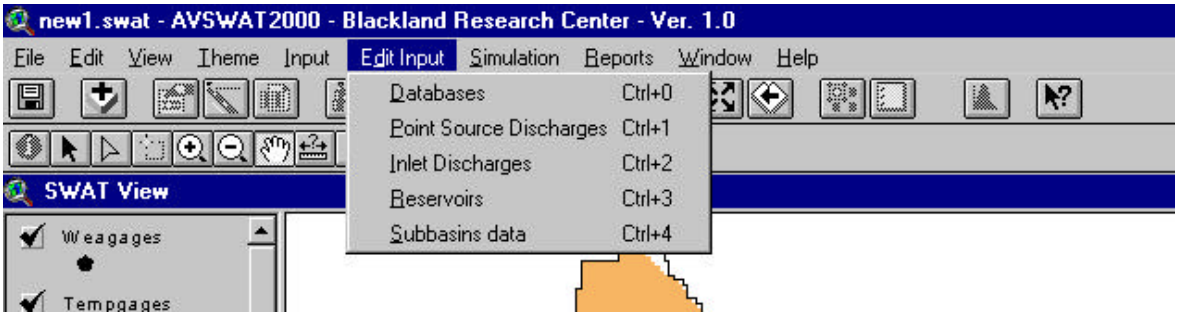


Figure 11.1

The fourth item of the **Edit Input** menu allows the user to edit reservoir parameters/inputs. Reservoirs are added to the watershed configuration during the watershed discretization (see Section 5).

1. Select **Reservoirs** from the **Edit Input** menu (or press Ctrl+3) (Figure 11.2).

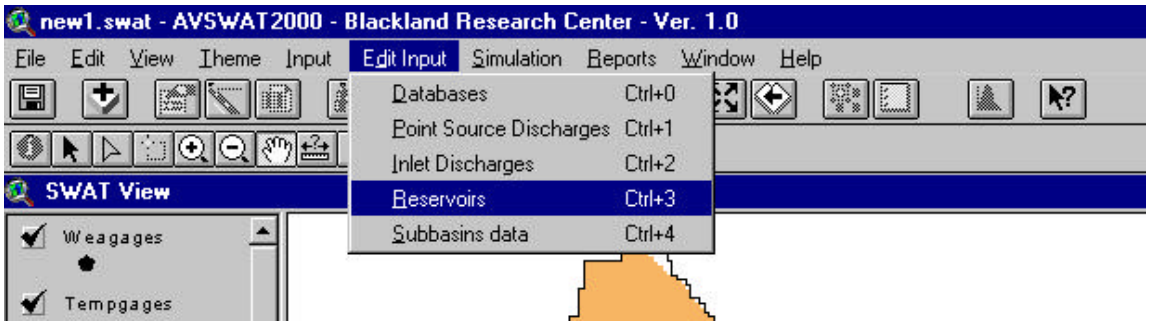


Figure 11.2

If no reservoirs are defined in the watershed (See Section 5), a dialog box notifies the user (Figure 11.3).

Figure 11.3

The Edit Reservoirs Inputs dialog is displayed (Figure 11.4) if at least one reservoir is defined in the watershed (see Section 5).

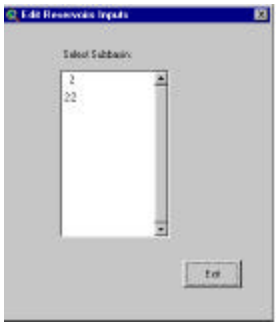


Figure 11.4

All subbasins containing reservoirs will be listed.

2. To edit the parameters for a reservoir, click on the number of the subbasin in which it is located. The Reservoir data dialog will open with the current data for the selected subbasin (Figure 11.5).

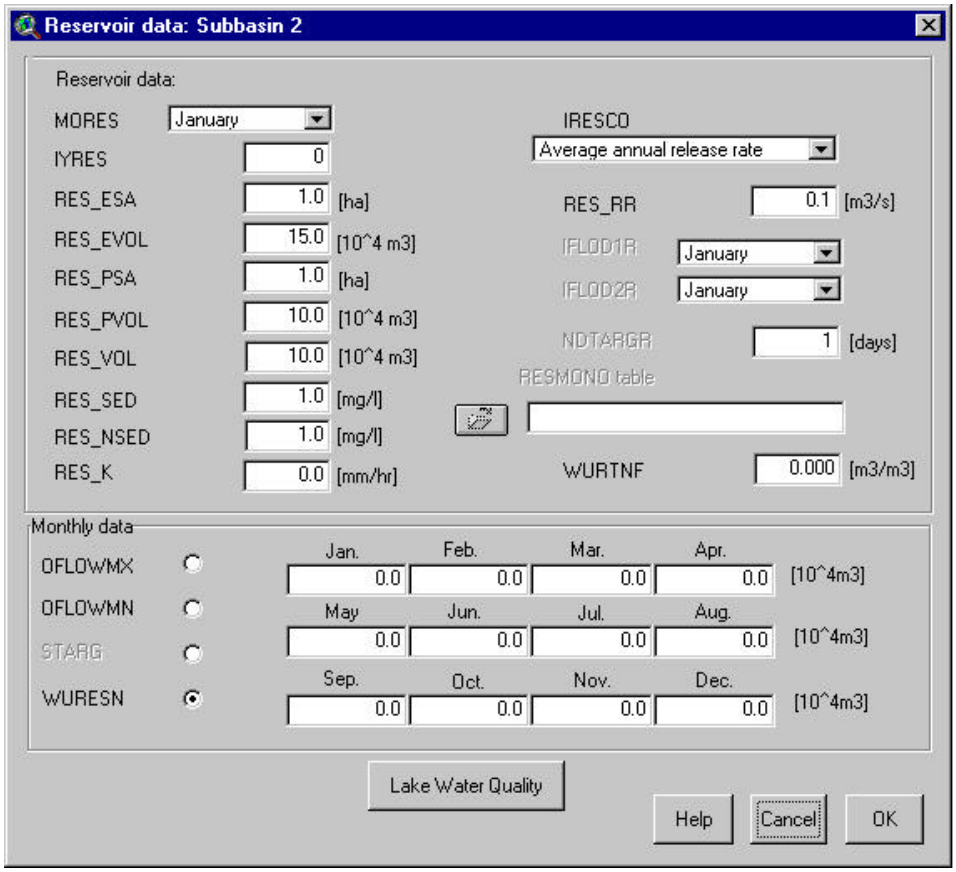


Figure 11.5

The Reservoir data dialog box allows the user to enter/edit reservoir parameters related to the water, sediment, nutrient and pesticide processes occurring in the reservoir.

a. Reservoir data. The variable names listed in the Reservoir data dialog box are defined in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 29.

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation for parameter is shown.

To edit a parameter, activate the text box adjacent to the variable name by clicking on it. Enter and/or edit the text value. You can move to the next parameter by pressing the TAB key.

If the current value of the parameter is out of range, a message box like the one in Figure 11.6 appears.

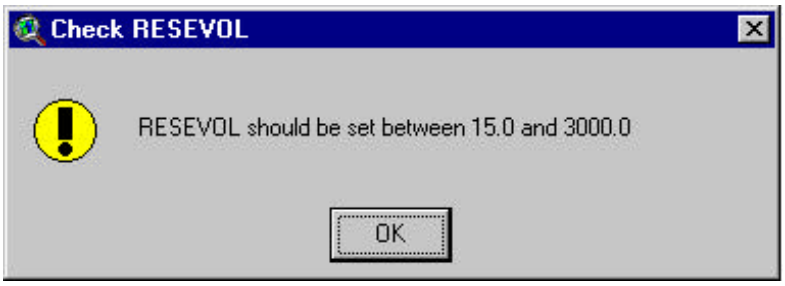


Figure 11.6

Click **OK**. The parameter is reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir/*avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

If the user selects **Measured monthly outflow** or **Measured daily outflow** as the outflow option (IRESCO), predefined formatted tables must be used to input the outflow data.

i. **Daily Reservoir Outflow data.** If **Measured daily outflow** is selected from the **IRESCO** drop-down list (Figure 11.7), the user is required to create a data table containing the daily outflow information for the reservoir following the format specified for Reservoir Daily Outflow in Section 3.3.

Figure 11.7

A section of the dialog box labeled "Resdayo table" is enabled (Figure 11.8).

Figure 11.8

Click the open file folder button .

A prompt box will appear so that the user can specify the format of the file containing the daily reservoir outflow data (Figure 11.9). Select dBASE (.dbf) or text file (.txt) and click **OK**.

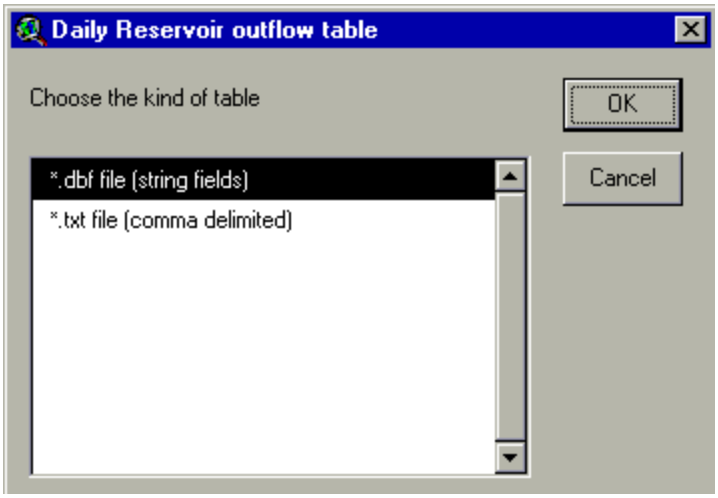


Figure 11.9

A browser will be displayed, allowing the user to select the dBASE file (.dbf) or text file (.txt) containing the daily reservoir outflow data from disk (Figure 11.10).

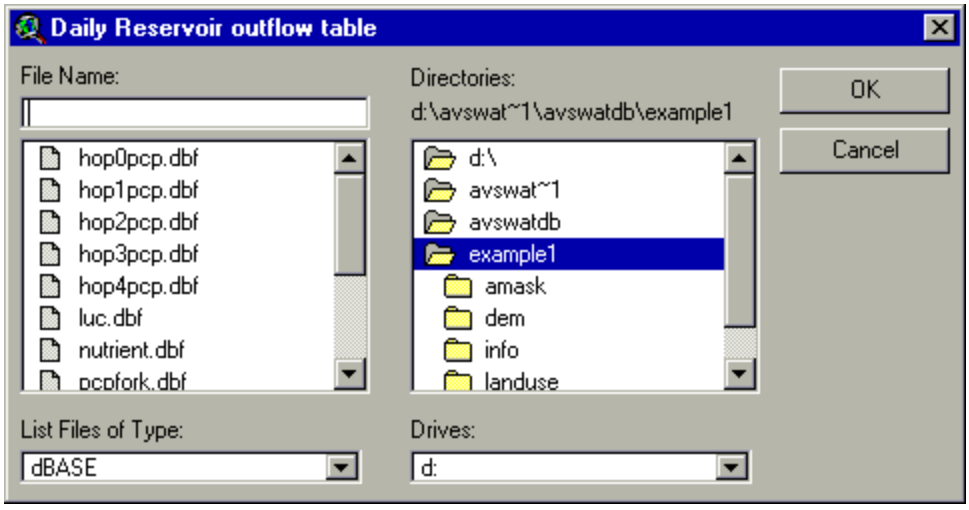


Figure 11.10

Select the file and click **OK** (or double click the selection).

ii. **Monthly Reservoir Outflow data.** If **Measured monthly outflow** is selected from the **IRESCO** drop-down list (Figure 11.11), the user is required to create a data table containing the monthly outflow

information for the reservoir following the format specified for Reservoir Monthly Outflow in Section 3.3.

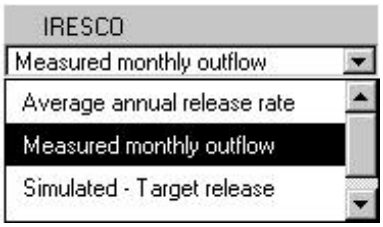


Figure 11.11

A section of the dialog box labeled "Resmono table" is enabled (Figure 11.12).

Figure 11.12

Click the open file folder button .

A prompt box will appear so that the user can specify the format of the file containing the monthly reservoir outflow data (Figure 11.13). Select dBASE (.dbf) or text file (.txt) and click **OK**.

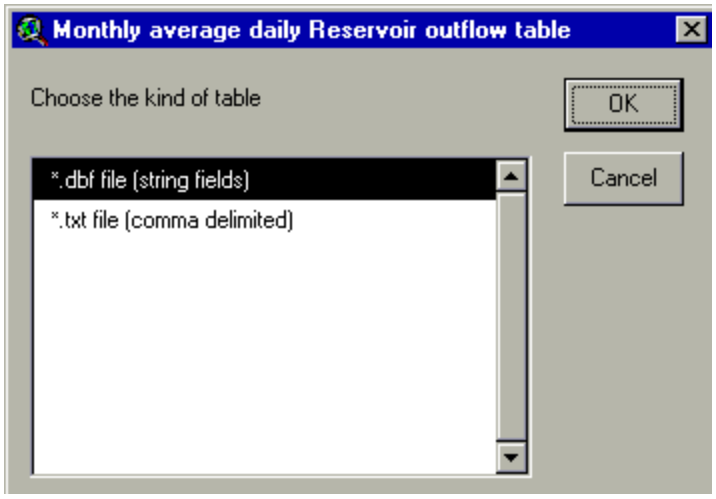


Figure 11.13

A browser will be displayed, allowing the user to select the dBASE file (.dbf) or text file (.txt) containing the data from disk (Figure 11.14).

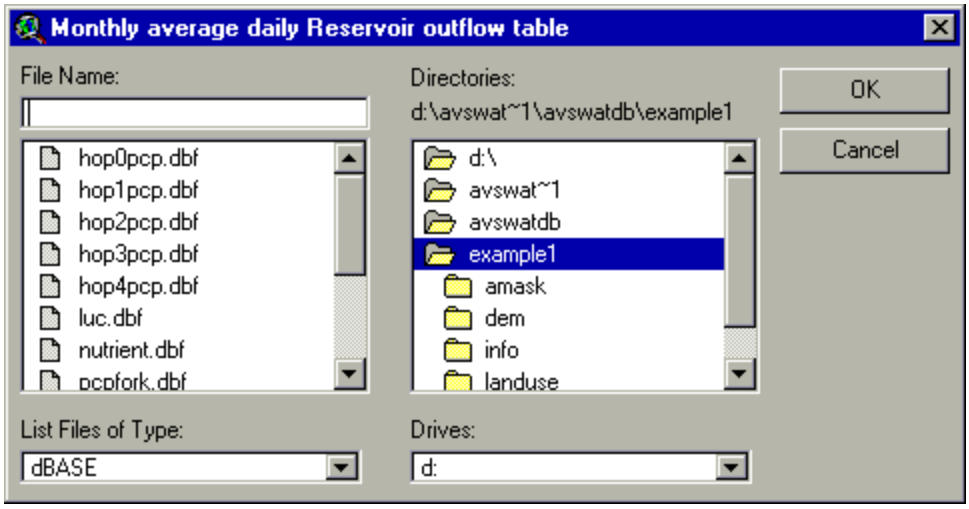
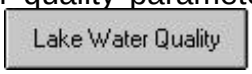


Figure 11.14

Select the file and click **OK** (or double click the selection).

b. Reservoir water quality data. To define/edit water quality parameters for the reservoir, click the **Lake Water Quality** button 

.

The Lake Water Quality dialog box will be displayed (Figure 11.15).

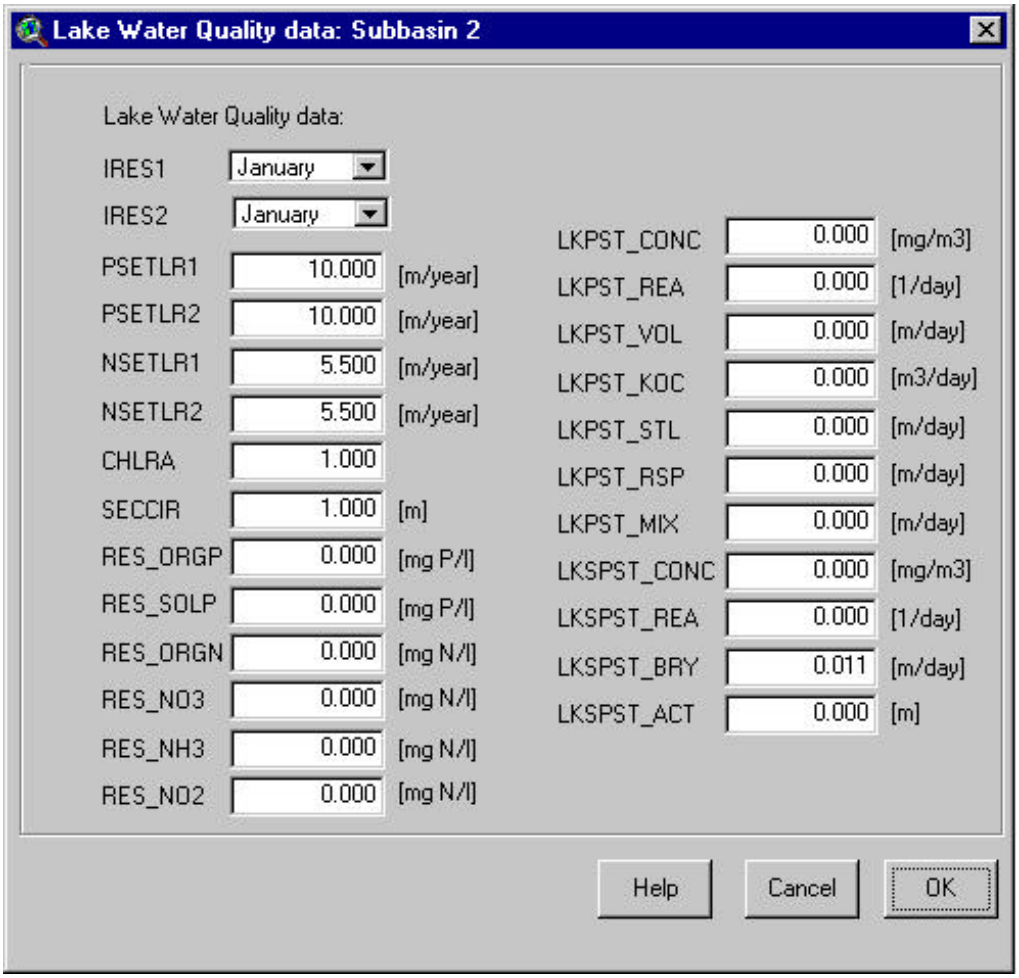


Figure 11.15

The variable names listed in the Lake Water Quality data dialog box are defined in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 30.

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation for parameters is shown.

To edit a parameter, activate the text box adjacent to the variable name by clicking on it. Enter and/or edit the text value. You can move to the next parameter by pressing the TAB key.

If the current value of the parameter is out of range, a message box like the one in Figure 11.16 appears.

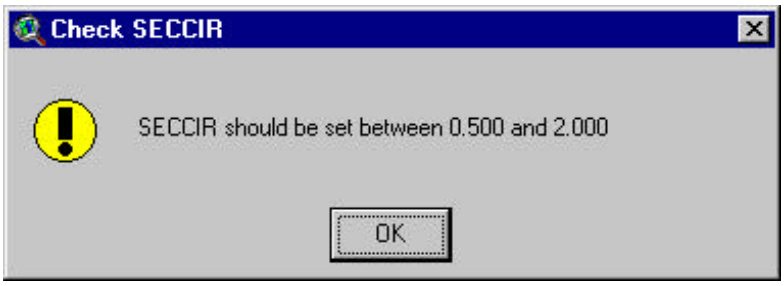


Figure 11.16

Click **OK**. The parameter is reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir/*avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

3. Once all parameters are properly set, click **OK** to exit the Lake Water Quality of Reservoir data dialog boxes and save the current settings. Click **Cancel** to exit the Lake Water Quality or Reservoir data dialog boxes without saving.

SECTION 12: INPUT MODIFICATION—SUBBASINS

The **Edit Input** menu is one of four new menus introduced in the SWAT View. The **Edit Input** menu allows you to edit the SWAT model databases and the watershed database files containing the current inputs for the SWAT model. Select the **Edit Input** menu using the mouse or by pressing the ALT+D keys. Five items are listed on the **Edit Input** menu (Figure 12.1).

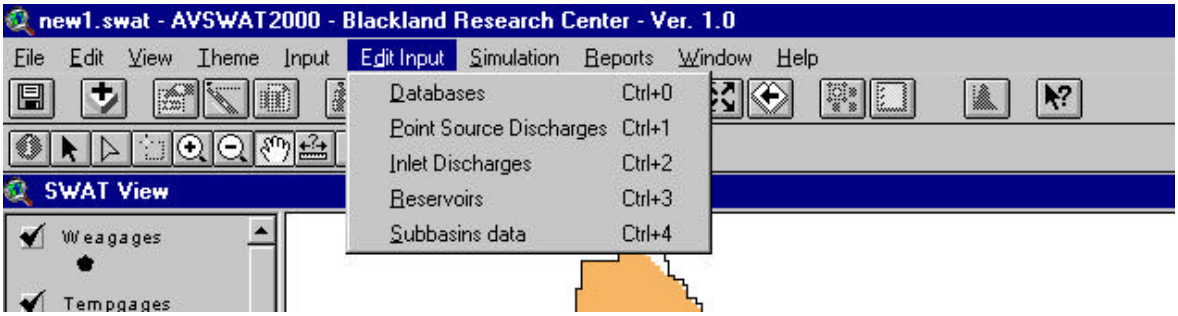


Figure 12.1

The fifth item of the **Edit Input** menu allows the user to edit land area, channel, pond/wetland, and groundwater parameters/inputs.

Select the **Subbasins data** command on the **Edit Input** menu (or press Ctrl+4) (Figure 12.2). This item is enabled only once the default Soil data is created (see Section 8).

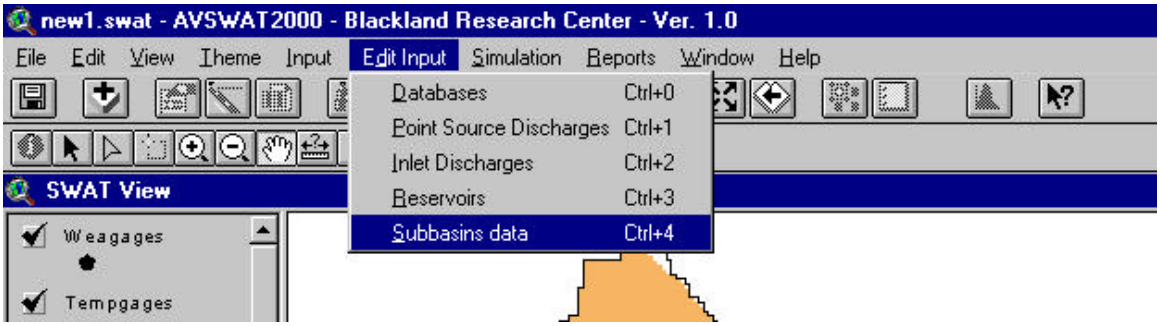


Figure 12.2

The Edit Subbasins Inputs dialog box is displayed (Figure 12.3).

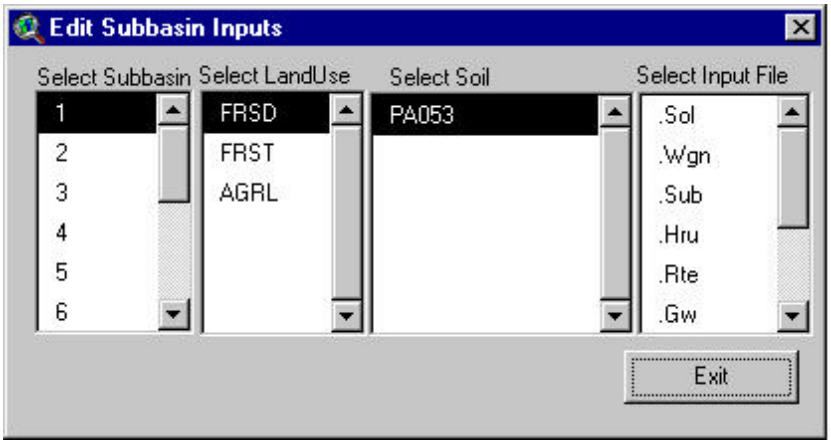


Figure 12.3

This dialog box is designed to facilitate the navigation and editing of the SWAT input data related to subbasins and HRUs. The dialog box contains four lists: subbasins, landuses, soil types, and input files. The input files extensions listed in the input file list are extensions given to the ASCII files required to run the SWAT model. Descriptions of the parameters contained in each of these files can be found in the *Soil and Water Assessment Tool User's Manual, Version 2000.*

When a subbasin is selected, the landuses and soil types present within the subbasin appear in the adjacent lists. Specify the subbasin/landuse/soil combination of interest by selecting an item from each category in the dialog box:

1. Select the subbasin number: the land use list updates to the land uses simulated in the selected subbasin.
2. Select the land use type: the soils list updates to the soils simulated with the selected land use in the selected subbasin.
3. Select the soil type.
4. Select the input file type.

The following sections review the SWAT input editors. SWAT input in the interface is organized by SWAT input file type (as described in the *Soil and Water Assessment Tool User's Manual, Version 2000*).

SECTION 12.1: EDIT SOIL PHYSICAL DATA (.SOL)

1. Click on **.Sol** in the **Select Input File** list of the Edit Subbasin Inputs dialog box. The soil physical data for the selected subbasin/landuse/soil (HRU) is displayed in a dialog box. The new dialog box is titled *Sol: Subbasin #_Landuse type_Soil name*. (Figure 12.4). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 22.

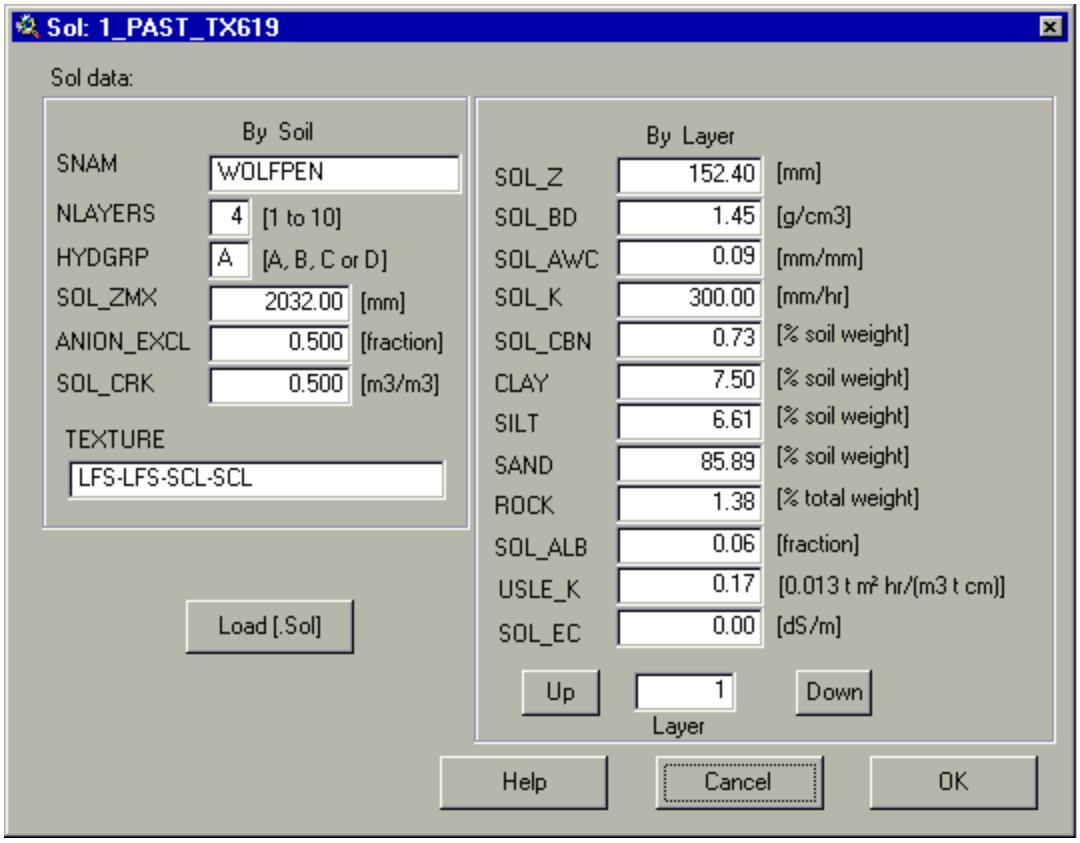
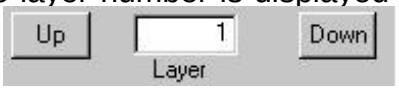


Figure 12.4

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation for the parameter is shown.

2. The data are separated into two groups, the first set of parameters pertaining to the soil profile and displayed in the box labeled 'By Soil'. The second group of parameters pertains to a particular soil layer and is displayed in the box labeled 'By Layer'. The layer number is displayed at the bottom between the **Up** and **Down** buttons 

. Data for different layers in the soil profile can be accessed by clicking these buttons.

a. Edit data: Activate a parameter text box by clicking on it. Type the desired value. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 12.5 will be displayed.

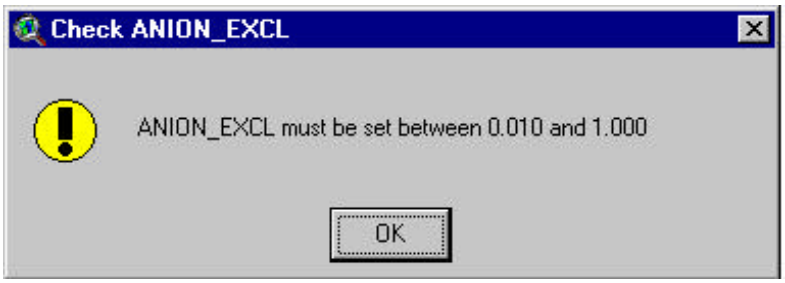


Figure 12.5

Click **OK.** The parameter will be reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *linstallation dir*/avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

b. Load data: To load a new data set from a preformatted SWAT .sol file, click the **Load (.Sol)**

button.

A prompt box containing a warning about the importance of properly formatting the file to be imported will be displayed (Figure 12.6).

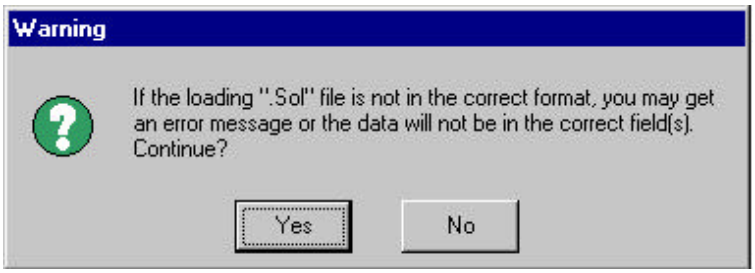


Figure 12.6

Click **No** to abort the importing procedure. To continue with the load, click **Yes**. A browser will be displayed (Figure 12.7).

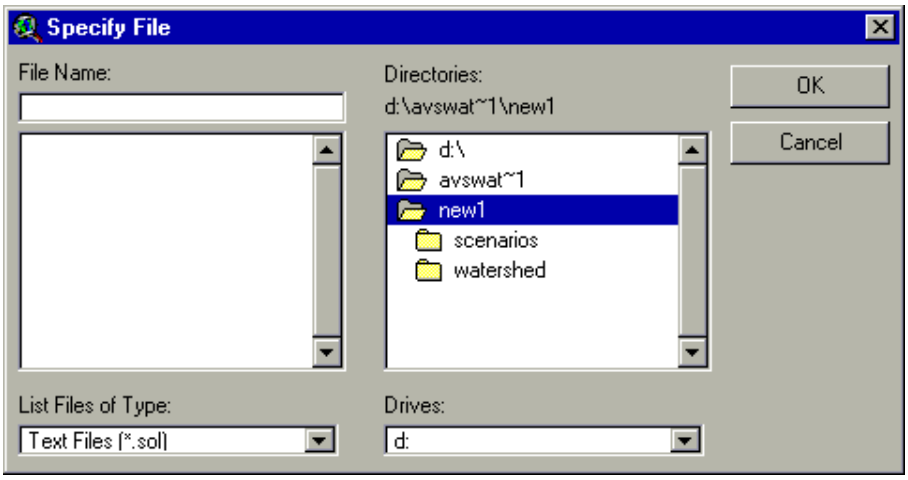


Figure 12.7

Select the soil input file (.sol) to be loaded and click **OK** (or double click the selection). The soil data will fill the fields in the dialog.

3. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

   If unacceptable parameter values were inserted, a message box may pop up like the one described above in Step 2.a. Click **OK** and return to editing and correct the values.

4. If the user saves the modified values, once the values are saved for the HRU (by clicking the **OK** button in step 3), a new dialog box will be displayed allowing the user to copy the modifications to other subbasin/land use/soil data sets (Figure 12.8).

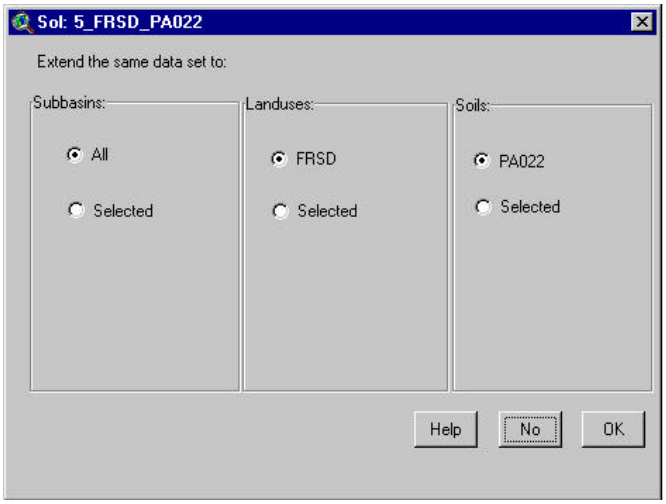


Figure 12.8

a. If you do not want to copy the edited soil data to other data sets, click **No**. The changes will have been saved for the individual HRU and no others.

b. If you wish to copy the edited soil data to other data sets, proceed as follows:

   i. Specify in the **Subbasins** section of the dialog box the subbasins to which the edited soil data will be copied. Select the **All**

button to copy to all the subbasins. If the changes are to be copied to only certain subbasins, click the **Selected**

button to specify the subbasins.

   When the **Selected** button is active, a list of subbasins will be displayed (Figure 12.9).

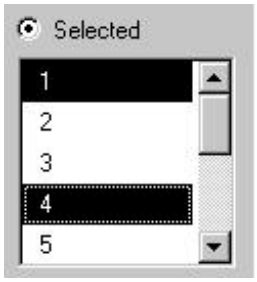


Figure 12.9

   Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

   ii. Specify in the **Landuses** section of the dialog box the landuse type(s) within the specified subbasins to which the soil data will be copied. Select the button labeled with the name of the original landuse

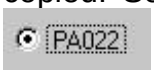
to copy the soil data only to HRUs with the same landuse type. Click the **Selected**

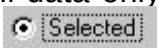
button to select other/additional landuses.

   When the **Selected** button is active, a list of landuses will be displayed (Figure 12.10).

Figure 12.10

Highlight the landuse(s) to which you would like the data to be copied from the new enabled list by clicking on the landuse code. To select more than one landuse, hold down the `Shift` key while highlighting the landuse codes.

iii. Specify in the **Soils** section of the dialog box the soil type(s) available for the specified subbasins/landuse(s) to which the soil data will be copied. Select the button labeled with the name of the original soil 

to copy the soil data only in the HRUs with this same soil type. Click the **Selected** 

button to select other/additional soil types.

When the **Selected** button is active, a list of soil types will be displayed (Figure 12.11).

Figure 12.11

Highlight the soil type(s) to which you would like the data to be copied from the new enabled list by clicking on the soil type code. To select more than one soil type, hold down the `Shift` key while highlighting the soil type codes.

> **Tip:** Although the option to copy soil data to selected soil types is provided, you are strongly discouraged from using this option. Overwriting soil data of one soil series with that from a different soil series defeats the purpose of using the interface to analyze the soil and land use distribution.

iv. Once you have made all your selections, click **OK**. The interface will save all the settings changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3).

SECTION 12.2: EDIT WEATHER GENERATOR INPUT DATA (.WGN)

1. Click on **.Wgn** in the **Select Input File** list of the Edit Subbasin Inputs dialog box. The parameters used to generate weather data for the subbasin are displayed in a new dialog box titled *Wgn: Subbasin #* (Figure 12.12) A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 13.

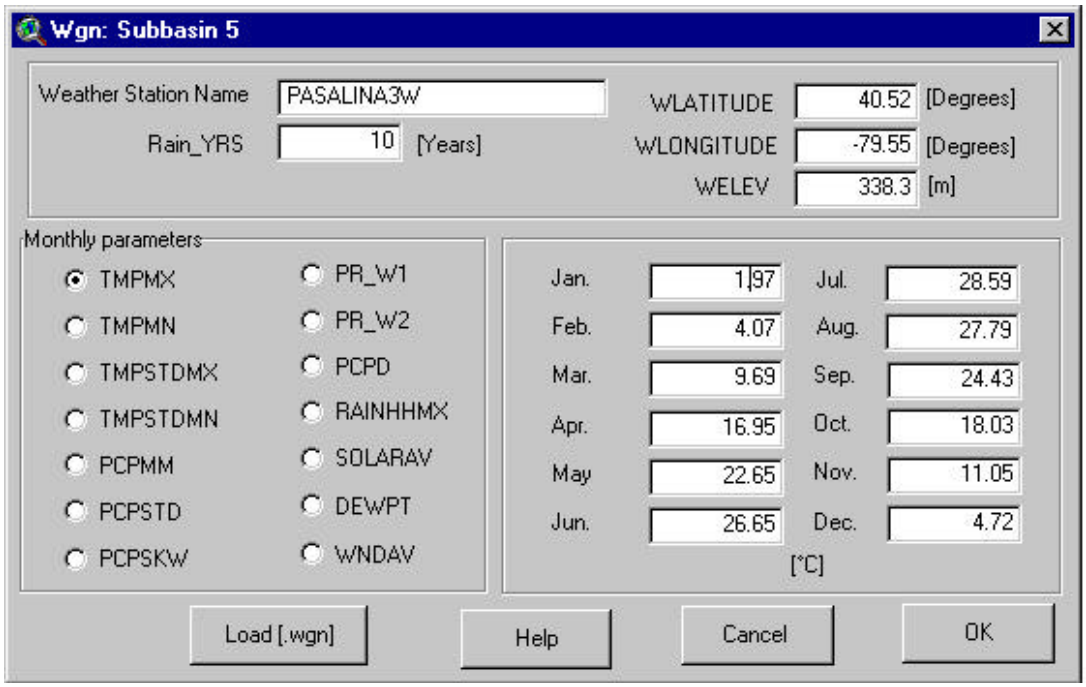


Figure 12.12

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation for parameters is shown.

2. The data are separated into two groups: the first set of parameters contains variables that require a single value. Variables that require values for each month of the year are located in the section of the dialog box labeled 'Monthly parameters'. The data fields for the monthly parameters are accessed by clicking the radio button to the left of the variable name of interest.

   a. Edit data: Activate a parameter text box by clicking on it. Type the desired value. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 12.13 will be displayed.

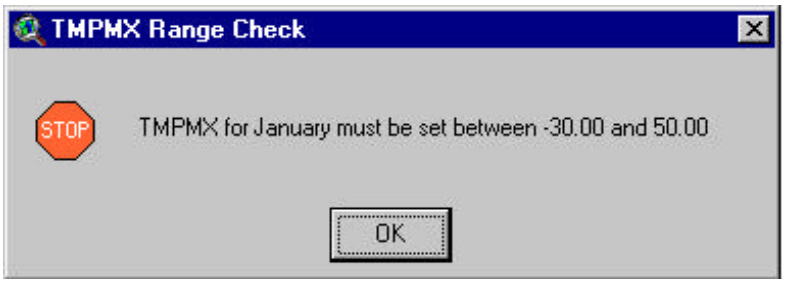


Figure 12.13

Click **OK**. The parameter is reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *linstallation dir*/avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

b. Load data: To load a new data set from a preformatted SWAT .wgn file, click the **Load (.Wgn)** Load [.wgn] button.

A prompt box containing a warning about the importance of properly formatting the file to be imported will be displayed (Figure 12.14).

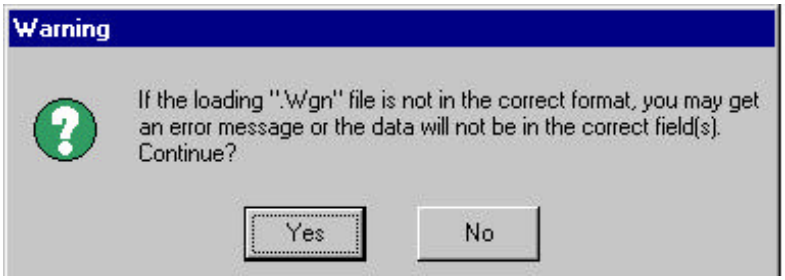


Figure 12.14

Click **No** to abort the importing procedure. To continue with the load, click **Yes**. A browser will be displayed (Figure 12.15).

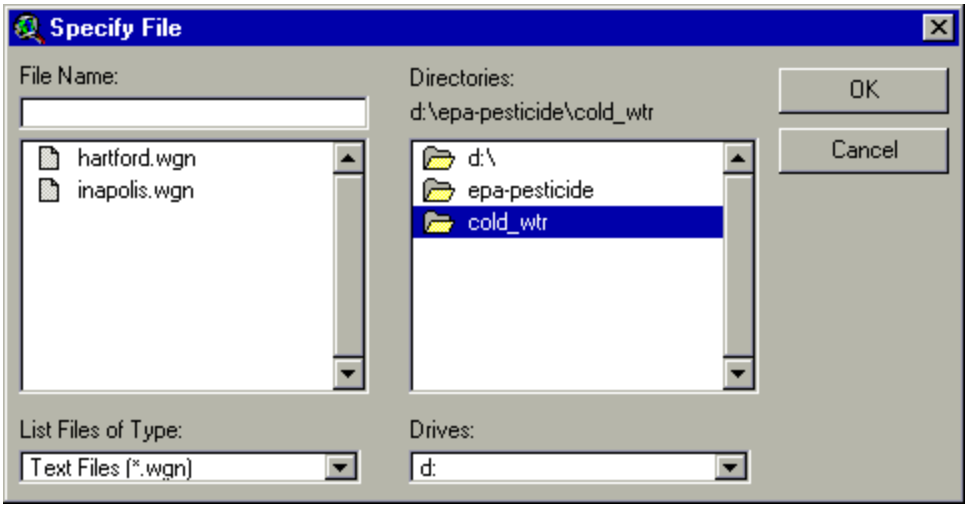


Figure 12.15

Select the weather generator input file (.wgn) to be loaded and click **OK** (or double click the selection). The data will fill the fields in the dialog.

3. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

   If unacceptable parameter values were inserted, a message box may pop up like the one described above in Step 2a. Click **OK** and return to editing and correct the values.

4. If the user saves the modified values, once the values are saved for the subbasin (by clicking the **OK** button in Step 3), a new dialog box will be displayed allowing the user to copy the modifications to other subbasin data sets (Figure 12.16).

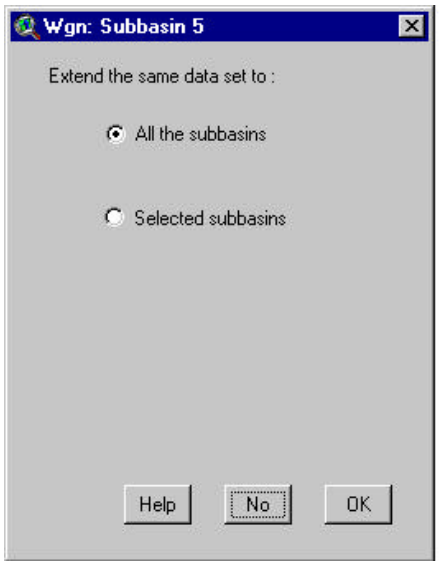
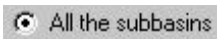
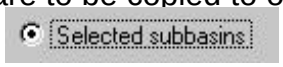


Figure 12.16

a. If you do not want to copy the edited weather generator data to other data sets, click **No**. The changes will have been saved for the individual subbasin and no others.

b. If you wish to copy the edited weather generator data to other data sets, proceed as follows:

   In the dialog box, specify the subbasins to which the data will be copied. Select the **All the subbasins** 

button to copy to all subbasins in the watershed. If the changes are to be copied to only certain subbasins, click the **Selected subbasins** 

button to choose the subbasins.

   When the **Selected subbasins** button is active, a list of subbasins will be displayed (Figure 12.17)

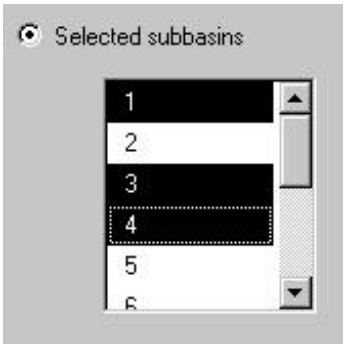


Figure 12.17

   Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

   Once you have made all selections, click **OK**. The interface will save all the changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3).

SECTION 12.3: EDIT SUBBASIN GENERAL INPUT DATA (.SUB)

1. Click **.Sub** item in the **Select Input File** list of the Edit Subbasin Inputs dialog box. General subbasin parameters dealing with elevation bands, climate change, and tributary channels are displayed in a new dialog box titled *Sub: Subbasin #* (Figure 12.18). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 6.

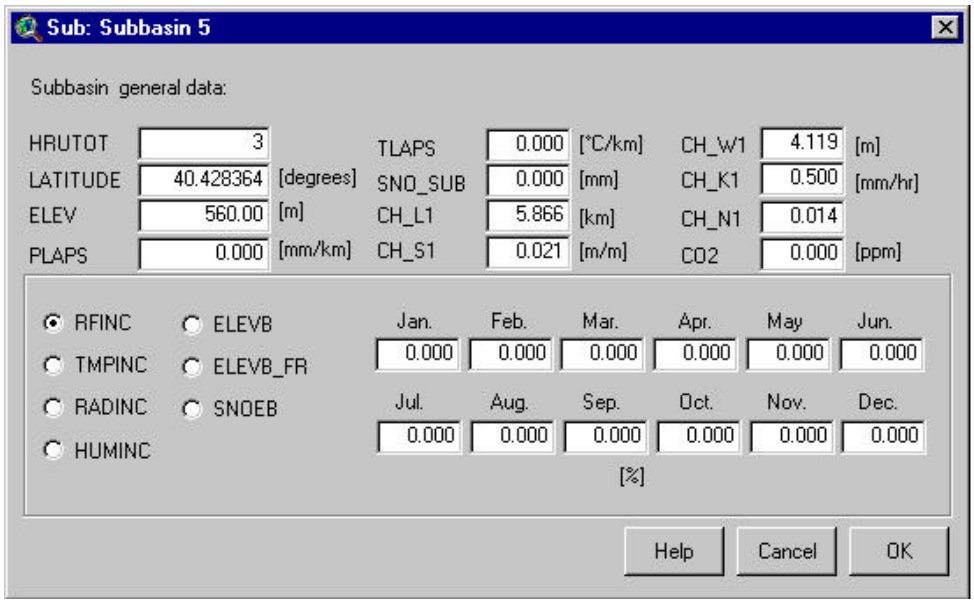


Figure 12.18

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation for parameters is shown.

2. The data are separated into two groups: the parameters listed in the upper portion of the dialog box are those that require only one value while the parameters listed in the lower portion of the dialog box next to radio buttons are those that require values for every month or elevation band. The data fields for the parameters in the lower portion of the dialog box are accessed by clicking the radio button to the left of the variable name.

   Edit data: Activate a parameter text box by clicking on it. Type the desired value. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 12.19 will be displayed.

> **Note:** The parameter HRUTOT cannot be edited.

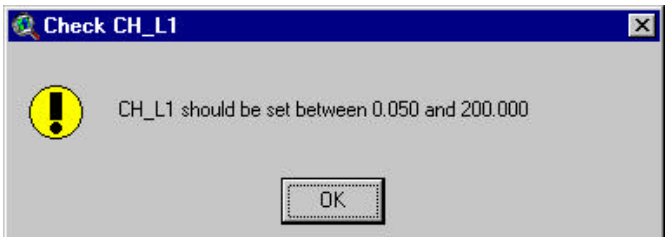


Figure 12.19

Click **OK.** The parameter is reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir/*avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

3. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

   If unacceptable parameter values were inserted, a message box may pop up like the one described above in Step 2. Click **OK** and return to editing and correct the values.

4. If the user saves the modified values, once the values are saved for the subbasin (by clicking **OK** in Step 3), a new dialog box is displayed allowing the user to copy the modifications to other subbasin data sets (Figure 12.20).

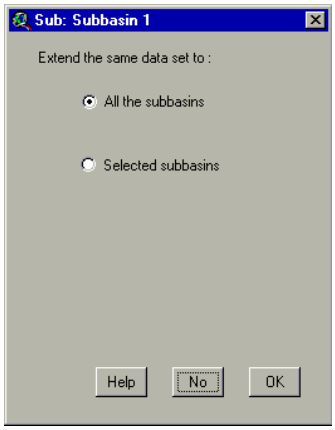
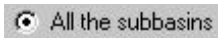


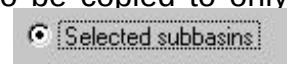
Figure 12.20

Note: Values for the parameter HRUTOT are not allowed to overwrite those in a different subbasin. Values for all other variables will be updated.

a. If you do not want to copy the edited general subbasin data to other data sets, click **No**. The changes will have been saved for the individual subbasin and no others.

b. If you wish to copy the edited general subbasin data to other data sets, proceed as follows:

   In the dialog box, specify the subbasins to which the edited soil data will be copied. Select the **All the subbasins** 

button to copy to all subbasins in the watershed. If the changes are to be copied to only certain subbasins, click the **Selected subbasins** 

button to choose the subbasins.

   When the **Selected subbasins** button is active, a list of subbasins will be displayed (Figure 12.21)

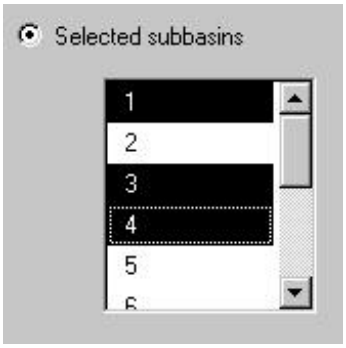


Figure 12.21

   Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

   Once you have made all selections, click **OK**. The interface will save all the changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3).

SECTION 12.4: EDIT HRU GENERAL INPUT DATA (.HRU)

1. Click **.Hru** in the **Select Input File** list of the Edit Subbasin Inputs dialog box. General HRU parameters dealing with surface and subsurface water flow, erosion, and management inputs related to the simulation of urban areas, irrigation, tile drains and potholes are displayed in a new dialog box titled *Hru: Subbasin #_Landuse type_Soil name* (Figure 12.22). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 19.

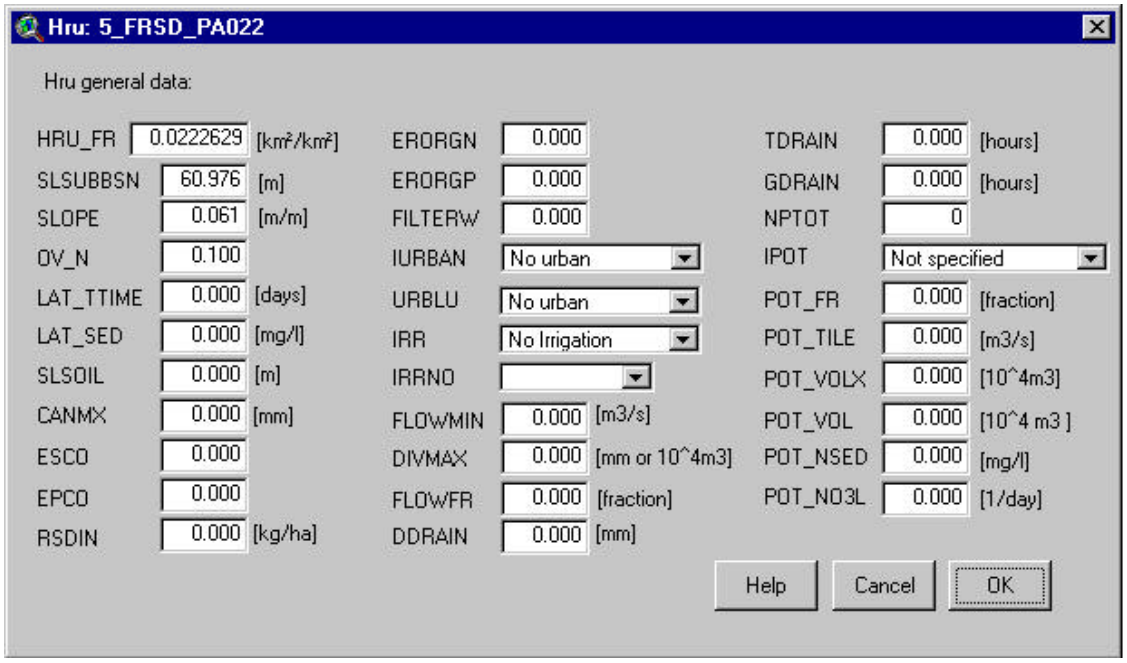


Figure 12.22

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears, and the range of variation for parameters is shown.

> **Note:** If the HRU was defined for an urban land type, the variable URBLU will be set to that land type.

2. Edit data: Activate a parameter text box by clicking on it. Type the desired value. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 12.23 pops up.

> **Note:** The parameter HRU_FR cannot be edited.

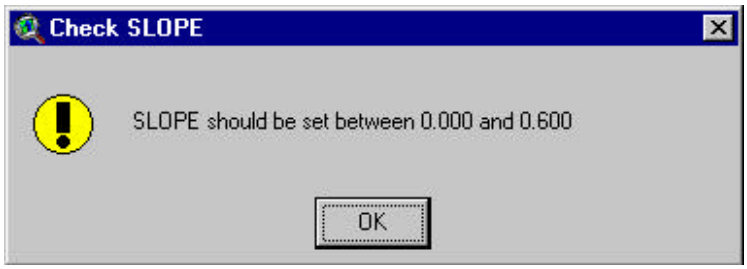


Figure 12.23

Click **OK.** The parameter is reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir/*avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

3. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

   If unacceptable parameter values were inserted, a message box may pop up like the one described above in Step 2. Click **OK** and return to editing and correct the values.

4. If the user saves the modified values, once the values are saved for the HRU (by clicking **OK** in Step 3), a new dialog box is displayed allowing the user to copy the modifications to other HRU data sets (Figure 12.24).

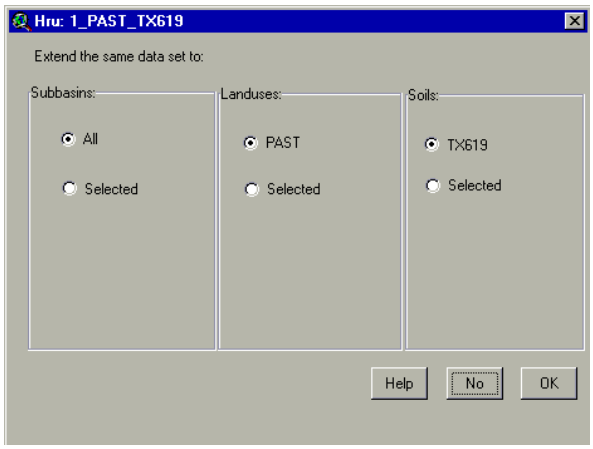


Figure12.24

Note: Values for the parameters HRU_FR and IPOT are not allowed to overwrite those in a different HRU. Values for all other variables will be updated.

a. If you do not want to copy the edited general HRU data to other data sets, click **No**. The changes will have been saved for the individual HRU and no others.

b. If you wish to copy the edited general HRU data to other data sets, proceed as follows:

   i. Specify in the **Subbasins** section of the dialog box the subbasins to which the edited general HRU data will be copied. Select the **All** button to copy to all the subbasins. If the changes are to be copied to only certain subbasins, click the **Selected** button to specify the subbasins.

   When the **Selected** button is active, a list of subbasins will be displayed (Figure 12.25).

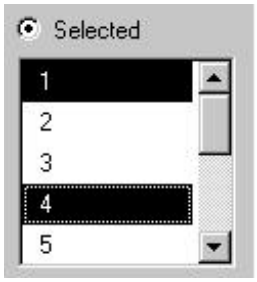


Figure 12.25

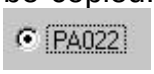
   Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

   ii. Specify in the **Landuses** section of the dialog box the landuse type(s) within the specified subbasins to which the general HRU data will be copied. Select the button labeled with the name of the original landuse to copy the general HRU data only to HRUs with the same landuse type. Click the **Selected** button to select other/additional landuses.

   When the **Selected** button is active, a list of landuses will be displayed (Figure 12.26).

Figure 12.26

Highlight the landuse(s) to which you would like the data to be copied from the new enabled list by clicking on the landuse code. To select more than one landuse, hold down the `Shift` key while highlighting the landuse codes.

iii. Specify in the **Soils** section of the dialog box the soil type(s) available for the specified subbasins/landuse(s) to which the general HRU data will be copied. Select the button labeled with the name of the original soil 

to copy the general HRU data only in the HRUs with this same soil type. Click the **Selected**

button to select other/additional soil types.

When the **Selected** button is active, a list of soil types will be displayed (Figure 12.27).

Figure 12.27

Highlight the soil type(s) to which you would like the data to be copied from the new enabled list by clicking on the soil type code. To select more than one soil type, hold down the `Shift` key while highlighting the soil type codes.

iv. Once you have made all your selections, click **OK**. The interface will save all the changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3).

SECTION 12.5: EDIT MAIN CHANNEL INPUT DATA (.RTE)

1. Click **.Rte** in the **Select Input File** list of the Edit Subbasin Inputs dialog box. Main channel parameters related to water and sediment transport are displayed in a new dialog box titled *Rte: Subbasin #* (Figure 12.28). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 25.

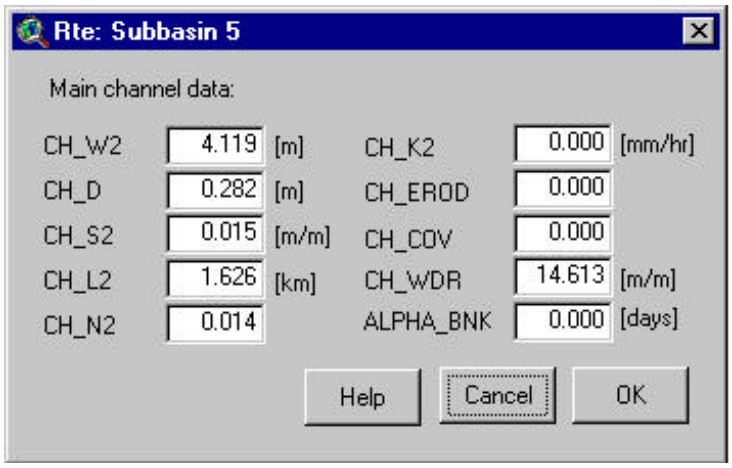


Figure 12.28

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation for parameters is shown.

2. Edit data: Activate a parameter text box by clicking on it. Type the desired value. You can move to the next parameter text box by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 12.29 pops up.

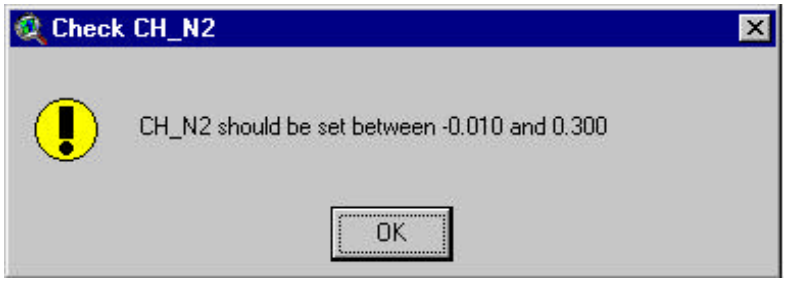


Figure 12.29

Click **OK.** The parameter is reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not

be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir/*avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

3. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

   If unacceptable parameter values were inserted, a message box similar to the one described in Step 2 may be displayed. Click **OK** and return to editing and correct the values.

4. If the user saves the modified values, once the values are saved for the subbasin (by clicking **OK** in Step 3), a new dialog box is displayed allowing the user to copy the modifications to other subbasin data sets (Figure 12.30).

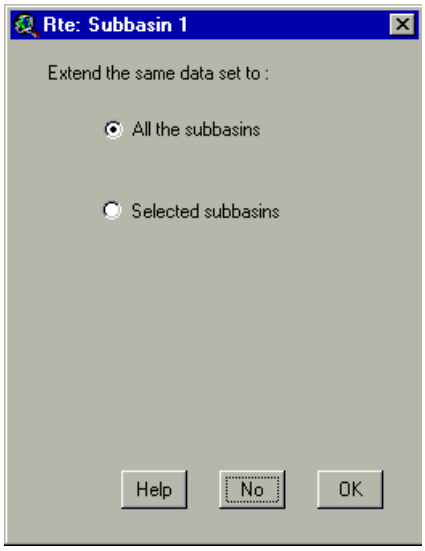
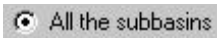


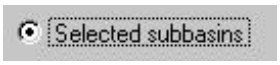
Figure 12.30

   a. If you do not want to copy the edited main channel data to other data sets, click **No**. The changes will have been saved for the individual subbasin and no others.

   b. If you wish to copy the edited main channel data to other data sets, proceed as follows:

      In the dialog box, specify the subbasins to which the edited main channel data will be copied. Select the **All the subbasins** 

button to copy to all subbasins in the watershed. If the changes are to be copied to

only certain subbasins, click the **Selected subbasins** 

button to choose the subbasins.

When the **Selected subbasins** button is active, a list of subbasins will be displayed (Figure 12.31)

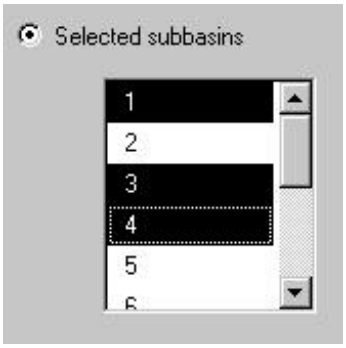


Figure 12.31

Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

Once you have made all selections, click **OK**. The interface will save all the changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3).

SECTION 12.6: EDIT GROUNDWATER INPUT DATA (.GW)

1. Click **.Gw** in the **Select Input File** list of the Edit Subbasin Inputs dialog box. Groundwater parameters are displayed in a new dialog box titled *Gw: Subbasin #_Landuse type_Soil name* (Figure 12.32). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 24.

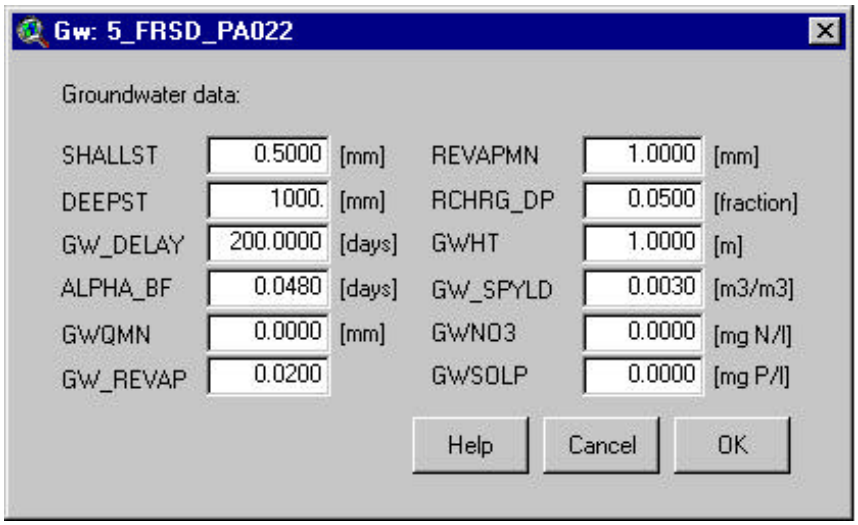


Figure 12.32

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation for parameters is shown.

2. Edit data: Activate a parameter text box by clicking on it. Type the desired value in the text box. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 12.33 pops up.

Figure 12.33

Click **OK.** The parameter is reset to the original value or to the closest range limit.

Note: Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir*/avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

3. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

   If unacceptable parameter values were inserted, a message box similar to the one described in Step 2 may be displayed. Click **OK** and return to editing and correct the values.

4. If the user saves the modified values, once the values are saved for the HRU (by clicking **OK** in Step 3), a new dialog box is displayed allowing the user to copy the modifications to other HRU data sets (Figure 12.34).

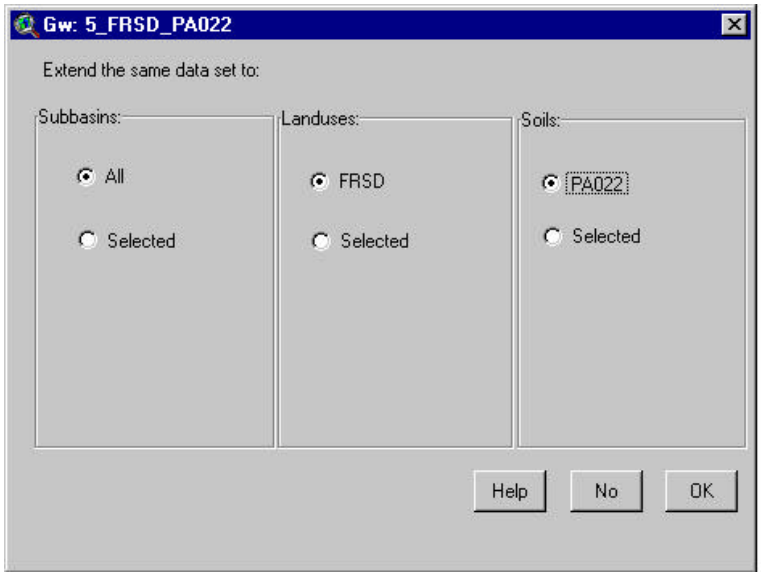
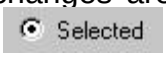


Figure 12.34

   a. If you do not want to copy the edited groundwater data to other data sets, click **No**. The changes will have been saved for the individual HRU and no others.

   b. If you wish to copy the edited groundwater data to other data sets, proceed as follows:

iii. Specify in the **Subbasins** section of the dialog box the subbasins to which the edited groundwater data will be copied. Select the **All**

button to copy to all the subbasins. If the changes are to be copied to only certain subbasins, click the **Selected** 

button to specify the subbasins.

When the **Selected** button is active, a list of subbasins will be displayed (Figure 12.35).

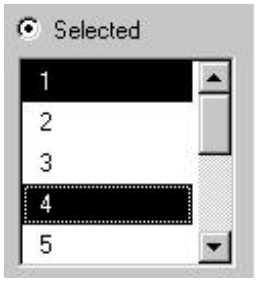
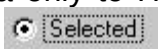


Figure 12.35

Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

iv. Specify in the **Landuses** section of the dialog box the landuse type(s) within the specified subbasins to which the groundwater data will be copied. Select the button labeled with the name of the original landuse

to copy the groundwater data only to HRUs with the same landuse type. Click the **Selected** 

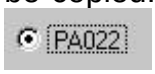
button to select other/additional landuses.

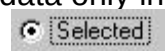
When the **Selected** button is active, a list of landuses will be displayed (Figure 12.36).

Figure 12.36

Highlight the landuse(s) to which you would like the data to be copied from the new enabled list by clicking on the landuse code. To select

more than one landuse, hold down the `Shift` key while highlighting the landuse codes.

v. Specify in the **Soils** section of the dialog box the soil type(s) available for the specified subbasins/landuse(s) to which the groundwater data will be copied. Select the button labeled with the name of the original soil 

to copy the groundwater data only in the HRUs with this same soil type. Click the **Selected** 

button to select other/additional soil types.

When the **Selected** button is active, a list of soil types will be displayed (Figure 12.37).

Figure 12.37

Highlight the soil type(s) to which you would like the data to be copied from the new enabled list by clicking on the soil type code. To select more than one soil type, hold down the `Shift` key while highlighting the soil type codes.

iv. Once you have made all your selections, click **OK**. The interface will save all the changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3).

SECTION 12.7: EDIT WATER USE INPUT DATA (.WUS)

1. Click **.Wus** in the **Select Input File** list of the Edit Subbasin Inputs dialog box. Consumptive water use parameters are displayed in a new dialog box titled *Wus: Subbasin #* (Figure 12.38). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 21.

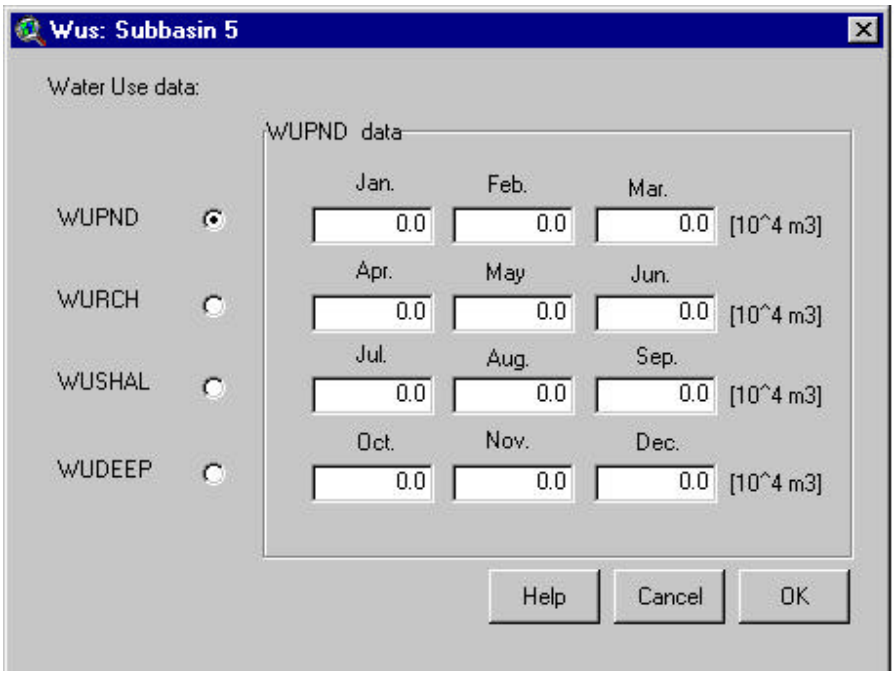


Figure 12.38

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears, and the range of variation for parameters is shown.

2. Edit data: Activate a parameter text box by clicking on it. Type the desired value. You can move to the next parameter text box by pressing the TAB key. If the value of the parameter is out of range a message box like the one in Figure 12.39 pops up.

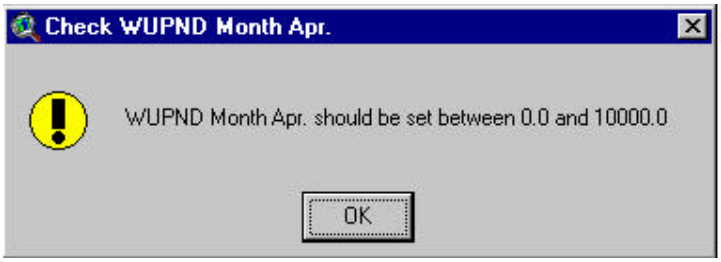


Figure 12.39

Click **OK.** The parameter is reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir/*avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

3. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

   If unacceptable parameter values were inserted, a message box similar to the one described in Step 2 may be displayed. Click **OK** and return to editing and correct the values.

4. If the user saves the modified values, once the values are saved for the subbasin (by clicking **OK** in Step 3), a new dialog box is displayed allowing the user to copy the modifications to other subbasin data sets (Figure 12.40).

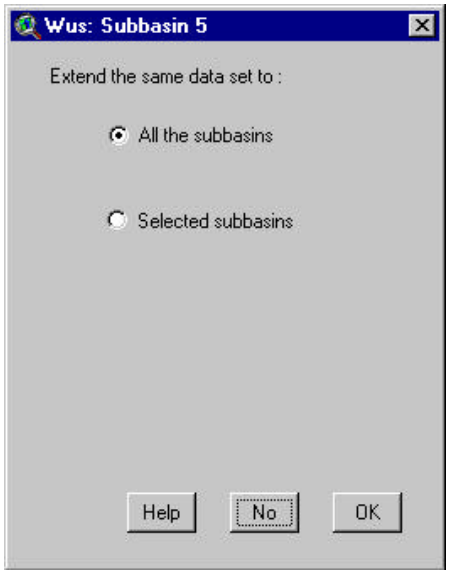
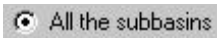
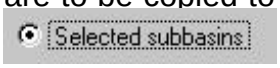


Figure 12.40

   a. If you do not want to copy the edited water use data to other data sets, click **No**. The changes will have been saved for the individual subbasin and no others.

b. If you wish to copy the edited water use data to other data sets, proceed as follows:

In the dialog box, specify the subbasins to which the edited water use data will be copied. Select the **All the subbasins** 

button to copy to all subbasins in the watershed. If the changes are to be copied to only certain subbasins, click the **Selected subbasins** 

button to choose the subbasins.

When the **Selected subbasins** button is active, a list of subbasins will be displayed (Figure 12.41)

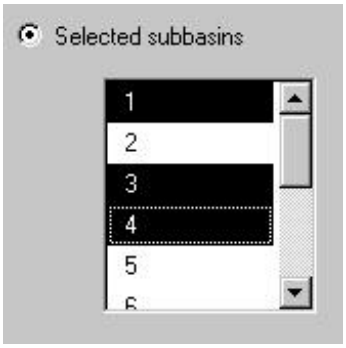


Figure 12.41

Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

Once you have made all selections, click **OK**. The interface will save all the changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3).

SECTION 12.8: EDIT MANAGEMENT INPUT DATA (.MGT)

1. Click **.Mgt** in the **Select Input File** list of the Edit Subbasin Inputs dialog box. Management parameters are displayed in a new dialog box titled *Mgt: Subbasin #_Landuse type_Soil name* (Figure 12.42). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 20.

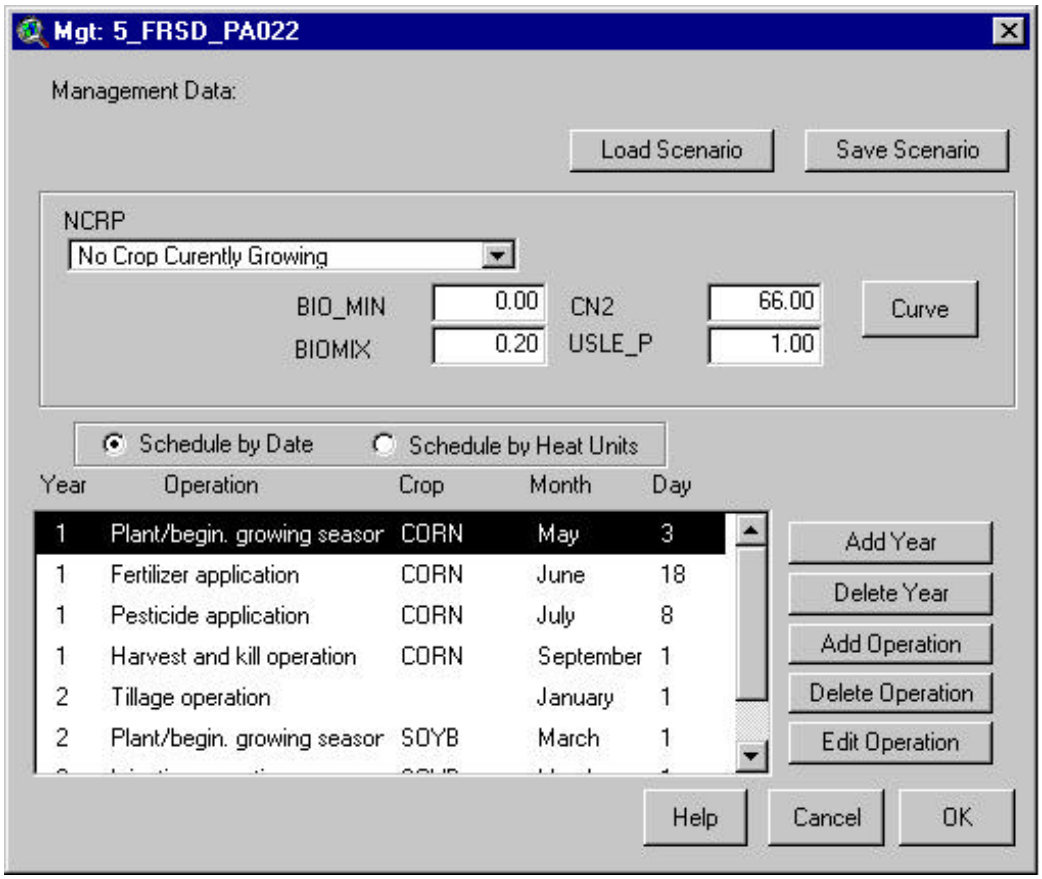


Figure 12.42

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears, and the range of variation for parameters is shown.

2. Two types of management data are displayed in the dialog box: general information and operation-specific information. Figure 12.42 displays the first page of the management editor, which contains general management information, the current operations schedule and controls to modify the management operation scenario.

   a. General management information is displayed in the middle section of the dialog box. To edit this information, activate a parameter text box by

clicking on it. Type the desired value in the text box. You can move to the next parameter by pressing the TAB key.

The initial curve number for the management scenario may be entered by typing in the text box or by selecting a value from the curve number table. To access the curve number table, click the **Curve**

button. A dialog box will be displayed (Figure 12.43).

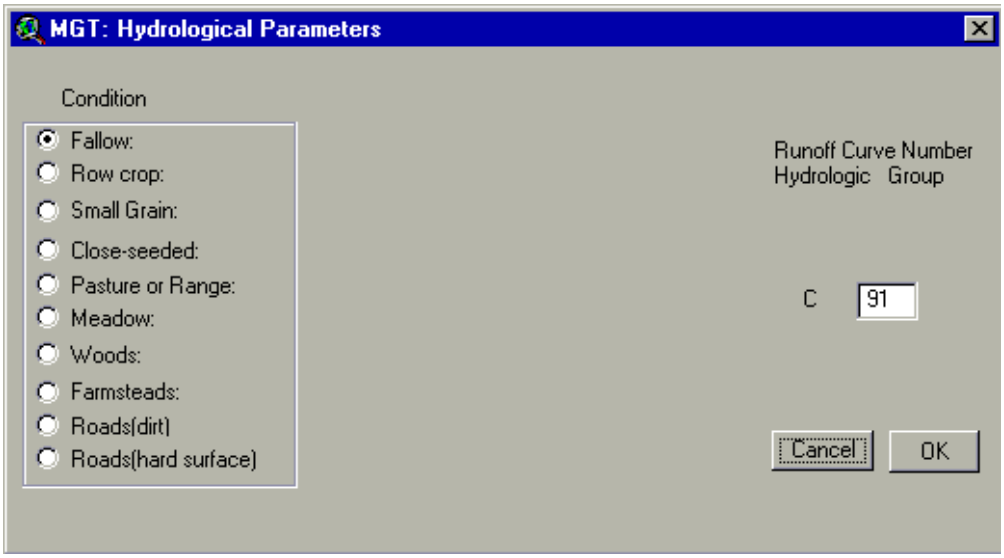


Figure 12.43

Curve number values for different management conditions are accessed by clicking the radio buttons for the condition and, if applicable, the cover. Curve numbers are displayed for the hydrologic group of the soil being simulated in the HRU (Figure 12.44).

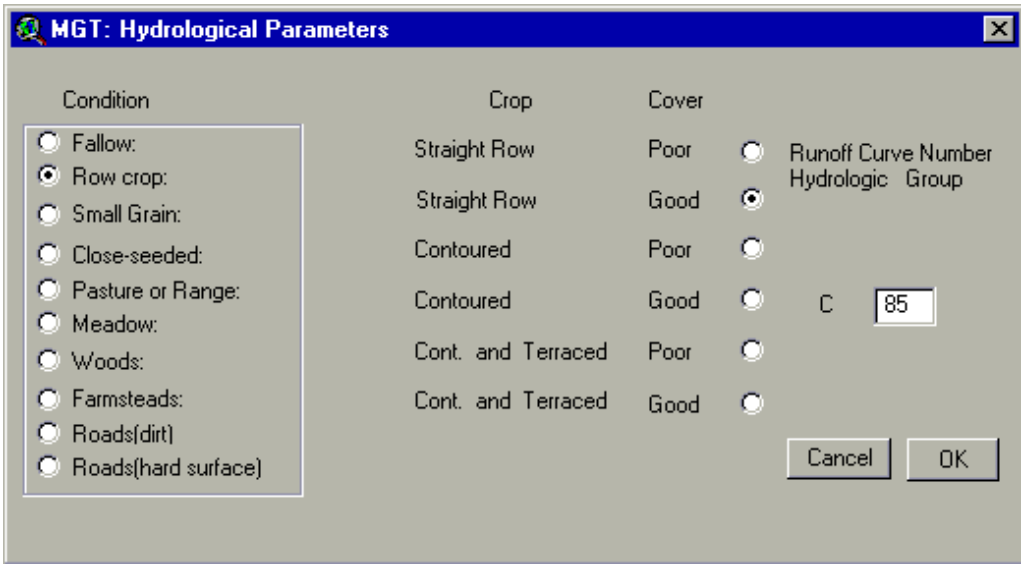


Figure 12.44

To use the curve number displayed in the dialog box, click **OK**. To exit the curve number lookup dialog box without modifying the initial curve number value, click **Cancel**.

If the value of a parameter is out of range, a message box like the one in Figure 12.45 pops up.

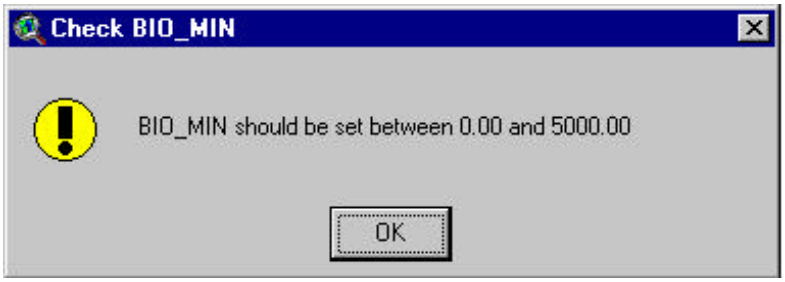


Figure 12.45

Click **OK**. The parameter is reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir/*avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

b. Operation-specific information must be scheduled by date or by plant growth stage (i.e. heat units). The interface will allow different types of scheduling to be used in different HRUs within the watershed, but for a given HRU, all operations must be scheduled exclusively by date or by plant growth stage (i.e., heat units). Click the respective radio button

to select the desired option.

When switching from one method to the other, a warning is displayed (Figure 12.46).

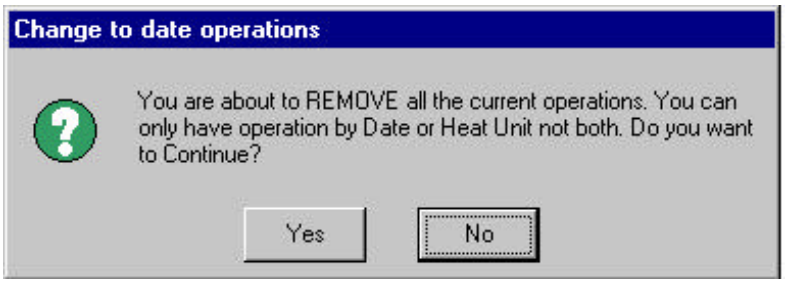


Figure 12.46

To abort the change in scheduling method, click **No**. Click **Yes** to continue and switch the method. When the scheduling method is changed, all operations in the list are removed.

Note: The default management operations are scheduled by heat units.

To move up and down the management operation list, use the mouse to click on an operation or use the up and down arrow keys on the keyboard.

The user can modify the operations scenario using the buttons located to the right of the list of operations:

Add Year: This command will add a new rotation year to the operation schedule.
Delete Year: This command will delete the highlighted year from the operation schedule.
Add Operation: This command will add a new operation to the rotation year currently active.
Delete Operation: This command will delete the highlighted operation.
Edit Operation: This command will display the parameter values for the highlighted operation, allowing the user to modify the values.

In addition to these commands, the two buttons at the top of the dialog box may be used to modify or save the displayed scenario.

Load Scenario: This command will replace existing management items with those from a predefined scenario.
Save Scenario: This command will store the management information for the displayed management data set as a predefined scenario. The data set is then available for loading in the current project and other projects.

To add a rotation year:

a. Click the **Add Year** button.

b. A new year number will appear in the operation schedule list.

c. Select the line with the new year number to add operations in the new rotation year.

To delete a rotation year:

a. Highlight one operation line belonging to the year to be deleted.

b. Click the **Delete Year** button.

c. A warning prompt dialog pops up (Figure 12.47).

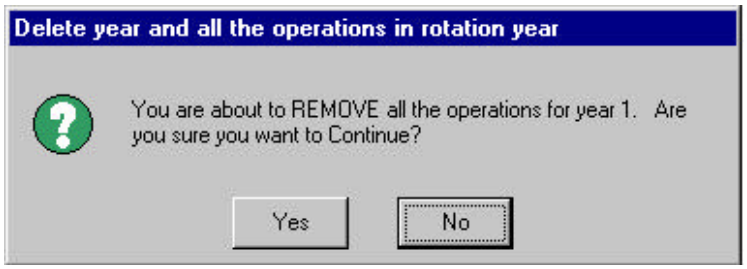
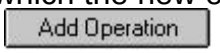


Figure 12.47

d. Click **No** to abort. Click **Yes** to continue and remove all the operations in the specified rotation year.

e. If **Yes** is selected, the remaining rotation years will be renumbered so that all rotation years in the operation list are consecutively numbered.

To add an operation:

a. Highlight the operation in the list, after which the new operation is to be added. Click the **Add Operation**  button. If the highlighted operation is the first one listed in the operation schedule, a prompt box will appear requesting the user to specify whether the operation will be added before or after the highlighted operation.

b. A prompt box will be displayed (Figure 12.48).

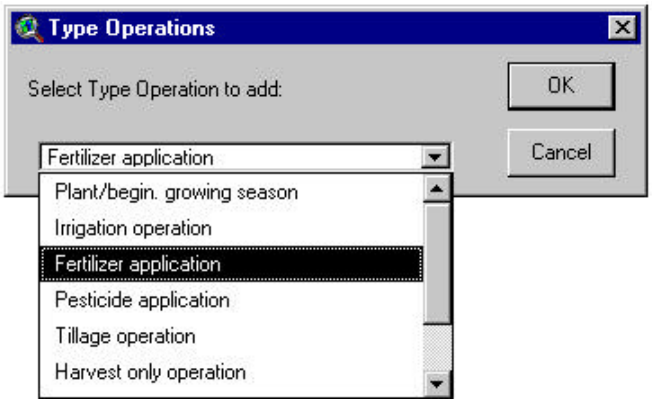


Figure 12.48

c. To abort adding an operation, click **Cancel**. Otherwise, display the drop-down list by clicking the arrow to the right of the text box. The drop-down list will display allowed operations. Select the desired operation and click **OK**.

d. The Add Operation dialog box will be displayed (Figure 12.49).

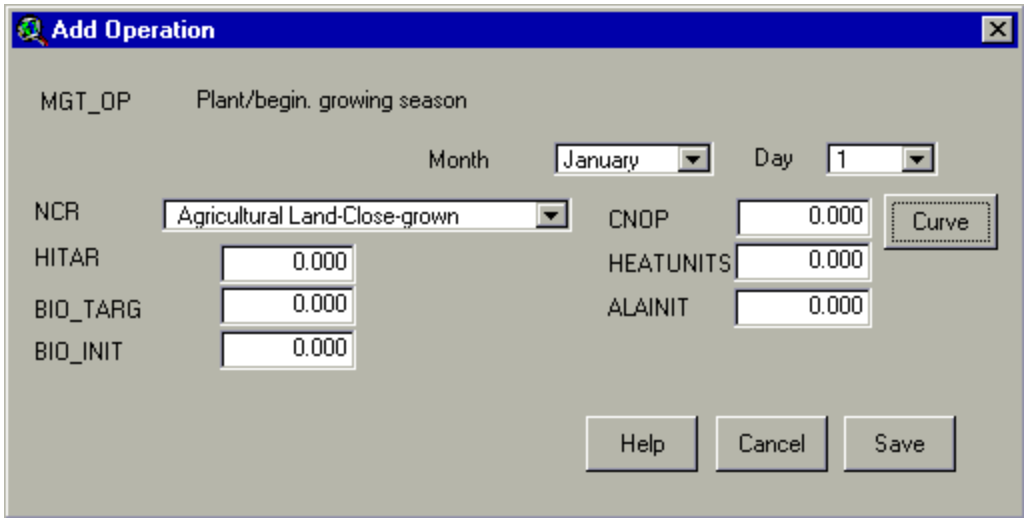


Figure 12.49

e. The fields in the dialog box will vary depending on the type of operation added and the scheduling method used in the HRU. Figure 12.49 shows the input parameters for a plant operation. Figure 12.50 displays the input parameters for an irrigation operation; Figure 12.51 displays the input parameters for a fertilizer application; Figure 12.52 shows the input parameters for a pesticide application; Figure 12.53 shows the input parameters for a tillage operation; Figure 12.54 displays the input parameters for a harvest only operation; Figure 12.55 displays the input parameters for a grazing operation; Figure 12.56 shows the input parameters for an auto-irrigation operation; Figure 12.57 shows the input parameters for an auto-fertilization operation; Figure 12.58 shows the input parameters for a harvest and kill operation; Figure 12.59 shows the input parameters for a kill operation; Figure 12.60 displays the input parameters for a street sweeping operation; and Figure 12.61 displays the input parameters for a release/impound operation.

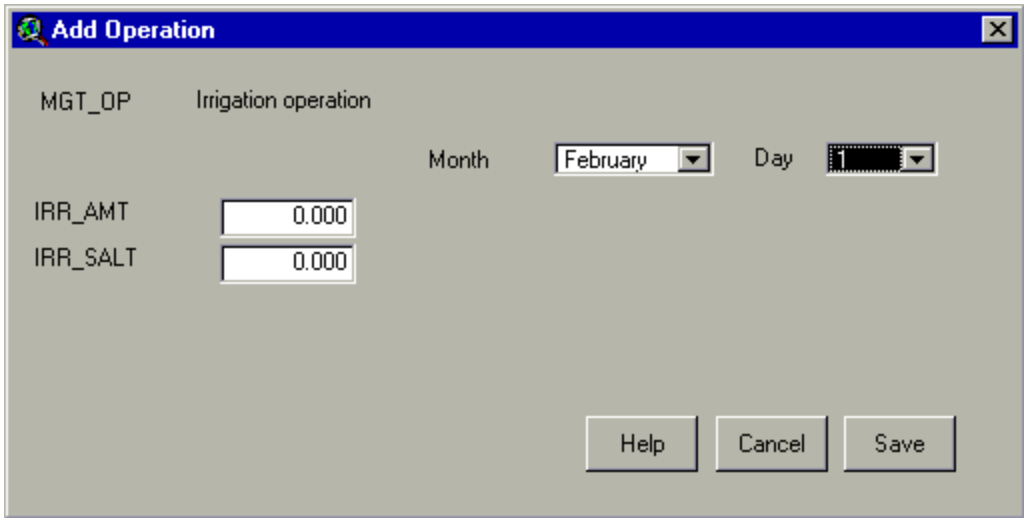


Figure 12.50

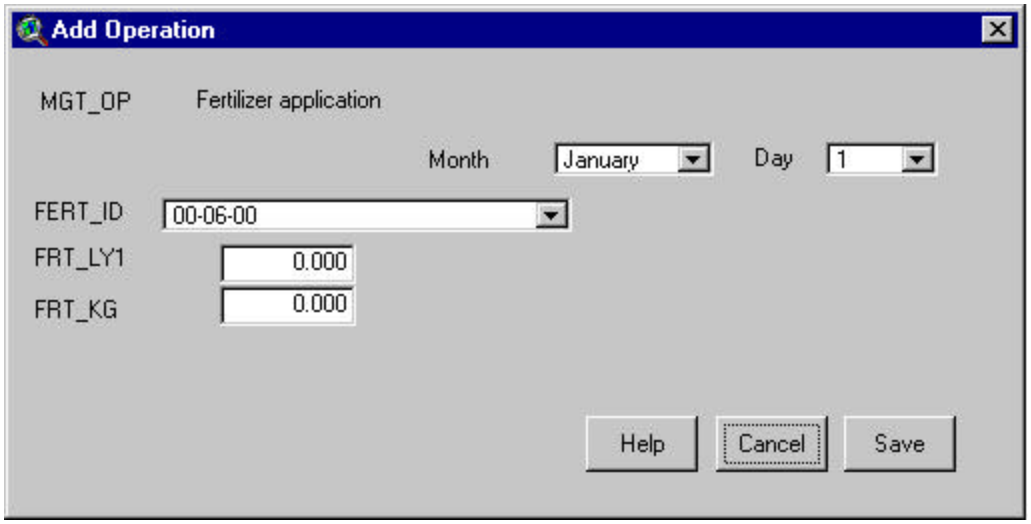


Figure 12.51

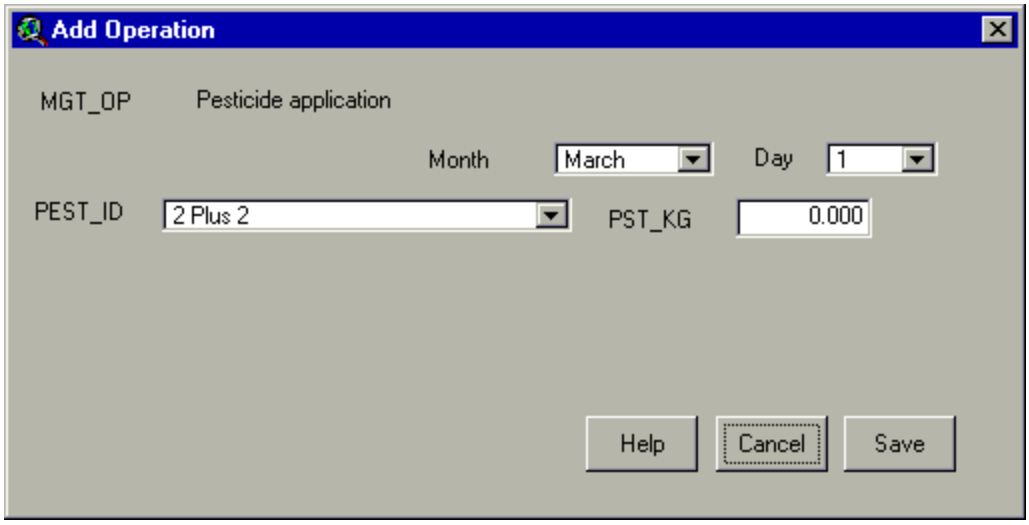


Figure 12.52

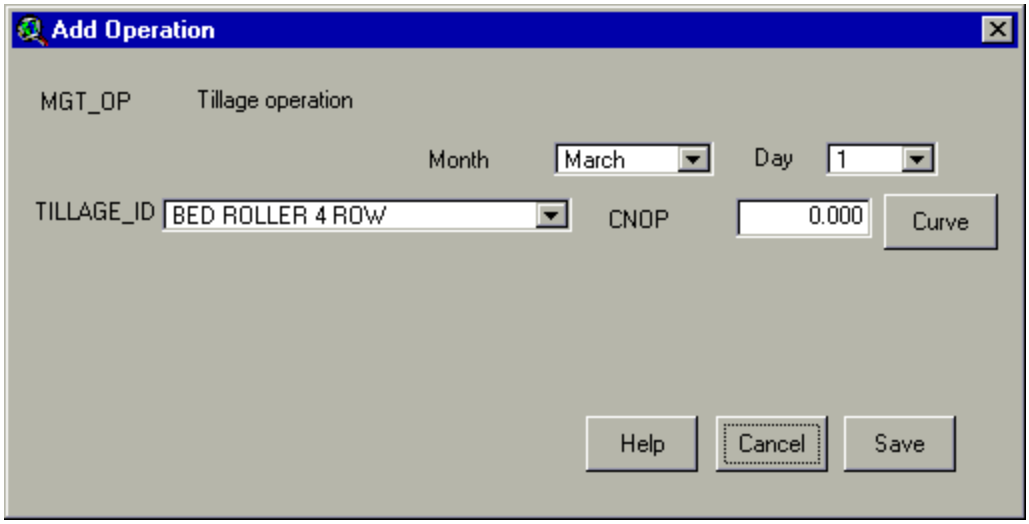


Figure 12.53

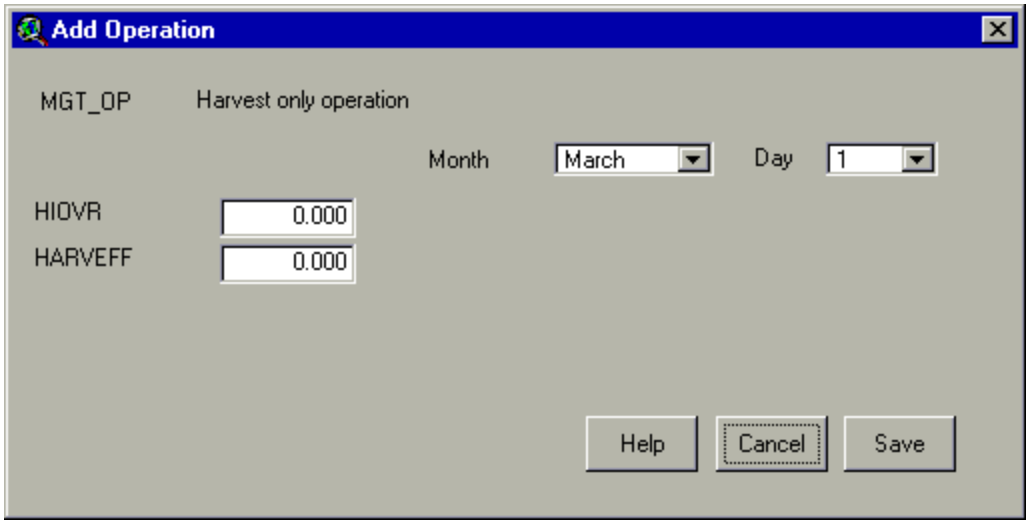


Figure 12.54

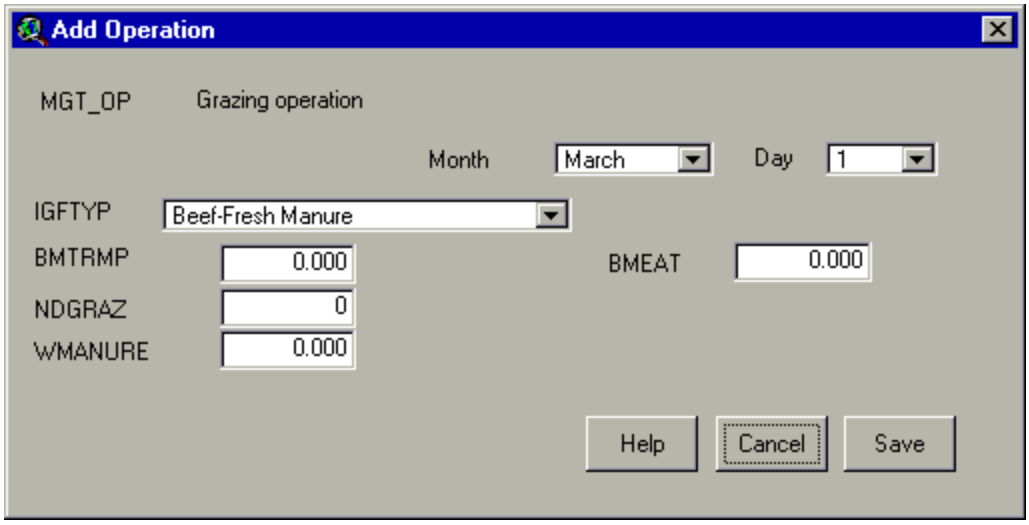


Figure 12.55

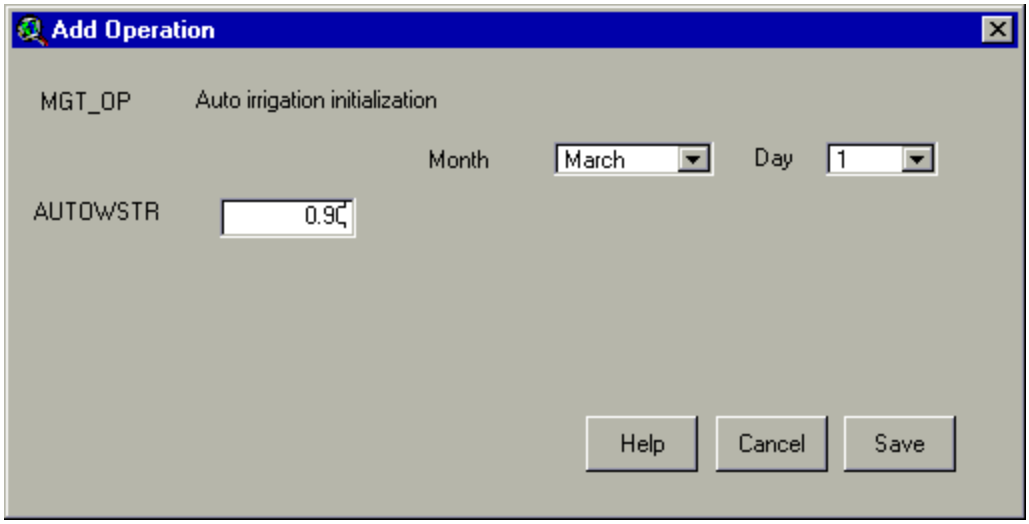


Figure 12.56

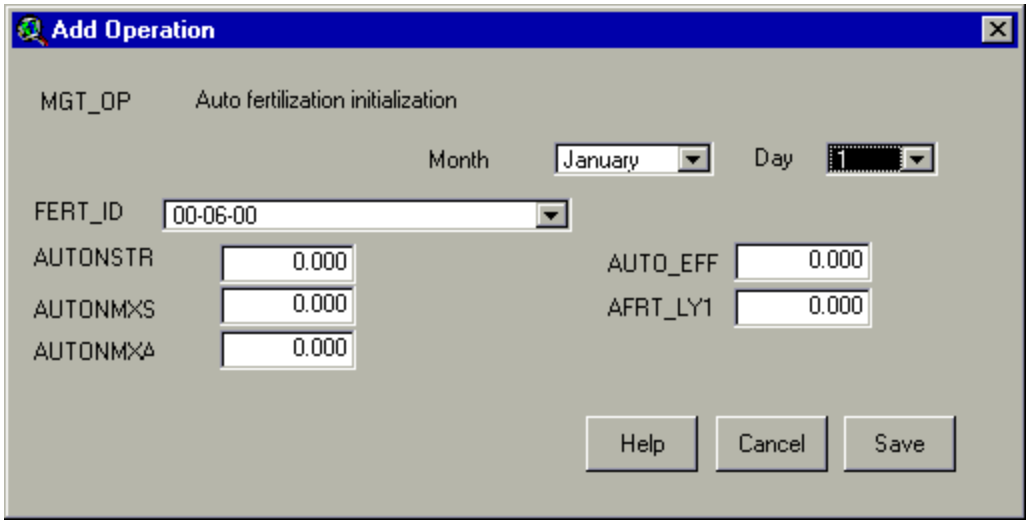


Figure 12.57

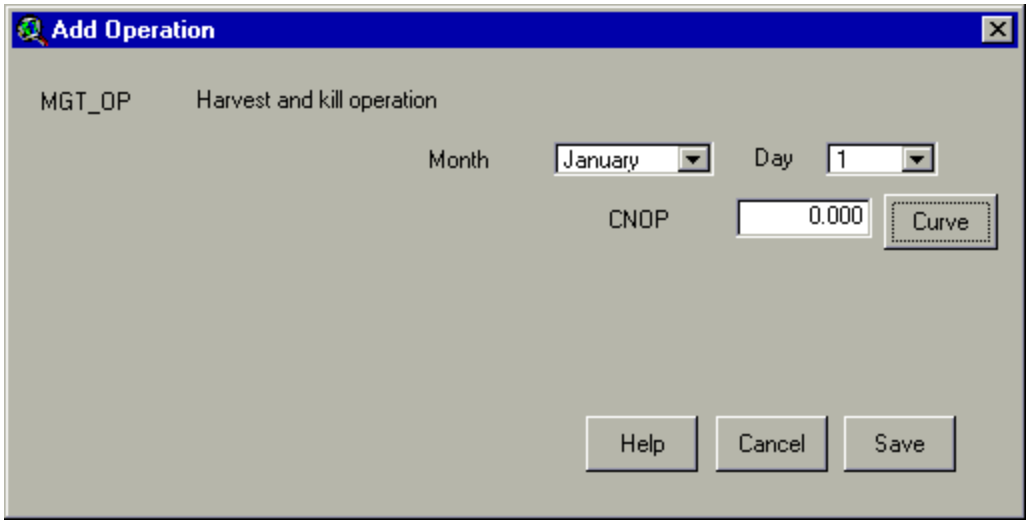


Figure 12.58

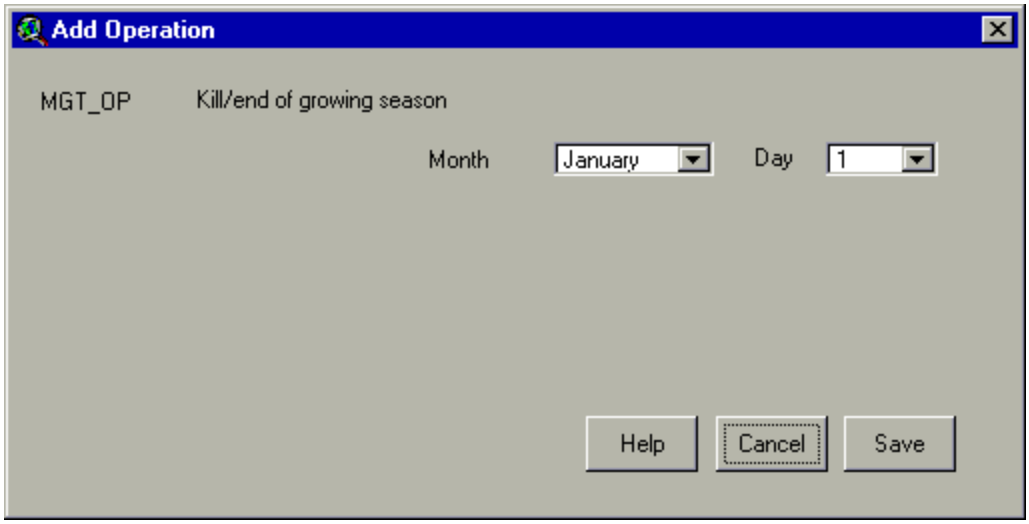


Figure 12.59

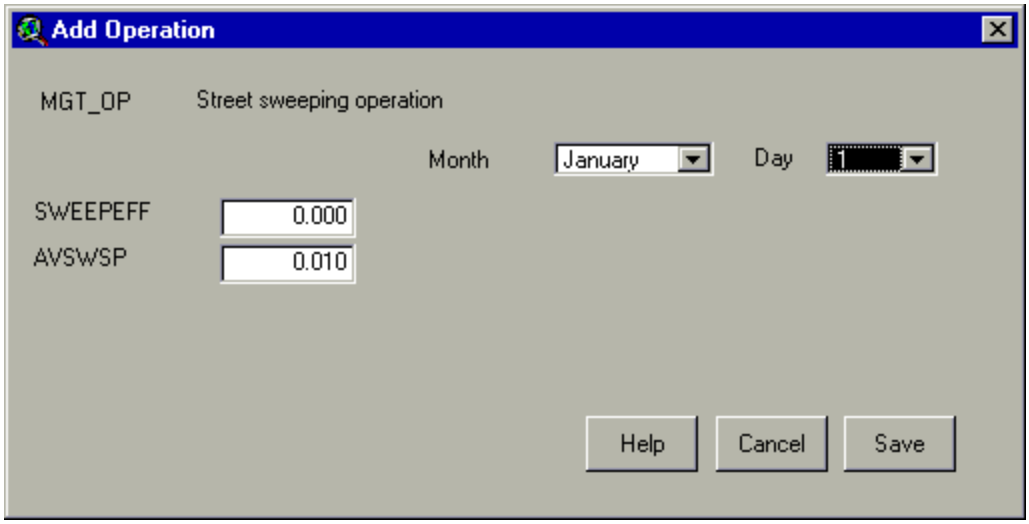


Figure 12.60

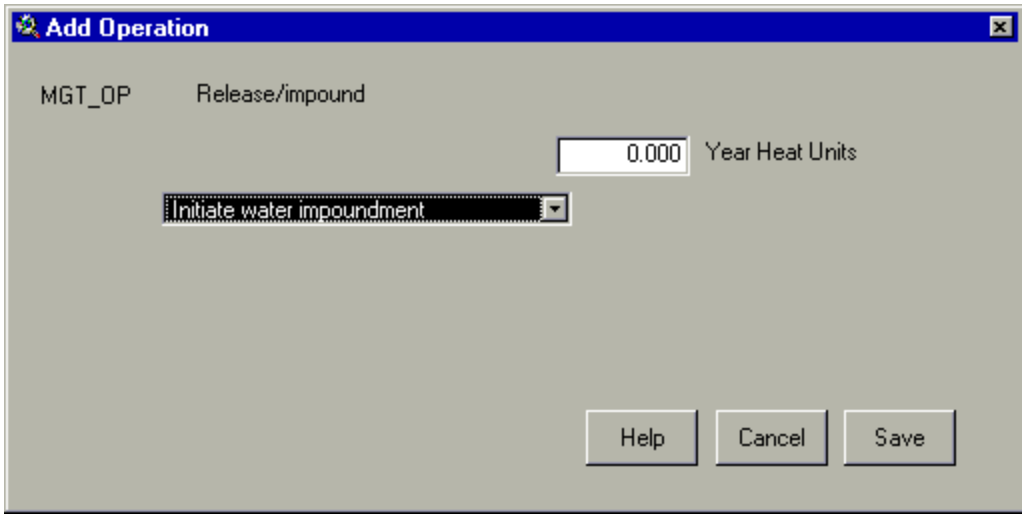


Figure 12.61

Note: When adding/editing a plant operation, a drop-down list allows the user to select a land cover/plant from the SWAT plant growth database. When adding/editing a fertilizer, auto-fertilization or grazing operation, a drop-down list allows the user to select a fertilizer/manure from the SWAT fertilizer database. When adding/editing a pesticide operation, a drop-down list allows the user to select a pesticide from the SWAT pesticide database. When adding/editing a tillage operation, a drop-down list allows the user to select a tillage implement from the SWAT tillage database.

Figure 12.51 displays the input parameters for a fertilizer application where the application is scheduled by date. Figure 12.62 displays the same screen when scheduling by heat units.

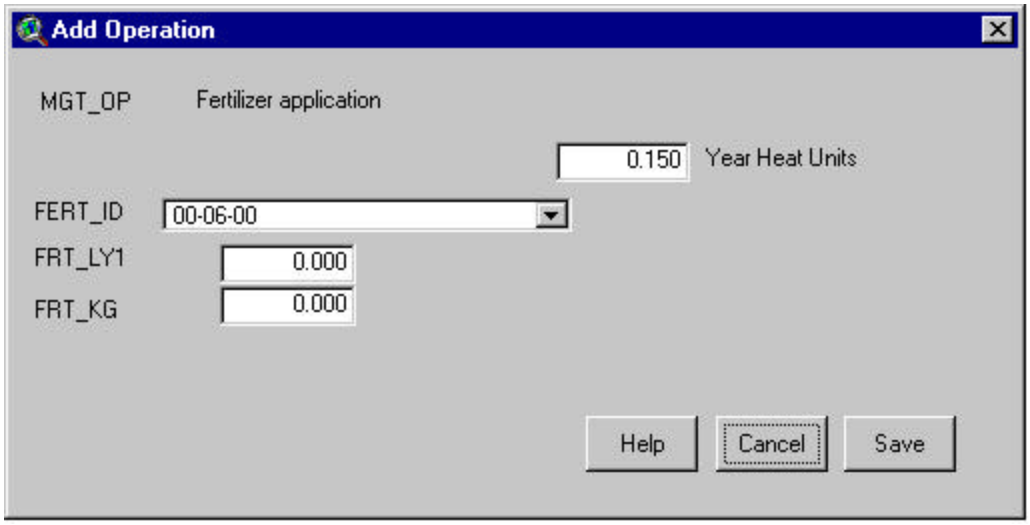


Figure 12.62

Note: When scheduling with heat units, the values used to schedule the operations between the time of planting and the time at which the land cover is killed are fractions of total accumulated plant heat units. Outside this growing period, the values used to schedule operations are fractions of annual, base-zero, heat units. Chapter 17 in the *SWAT 2000 Theoretical Documentation* explains heat unit scheduling in detail.

f. When all the parameters have been properly defined for the operation, click **Save**. To abort the add operation process, click **Cancel**.

g. If the new operation was saved, it will be added to the management operation schedule.

To delete an operation:

a. Highlight the operation to be deleted by clicking on it.

b. Click the **Delete Operation**

button.

c. A prompt box will appear (Figure 12.63).

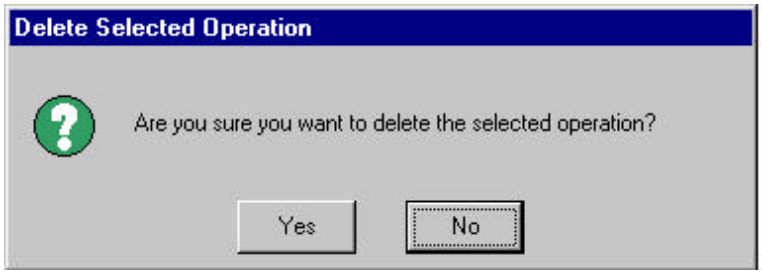


Figure 12.63

d. Click **No** to abort the deletion. Click **Yes** to remove the operation from the operation schedule.

To edit an operation:

a. Highlight the operation to be edited by clicking on it.

b. Click the **Edit Operation**

button.

> **Tip:** The operation editor can also be activated by double clicking the operation in the management operation schedule list.

c. An Edit Operation dialog box will be displayed that contains the parameters and defined values for the operation (e.g. Figure 12.64). This dialog box has the same format as the corresponding Add Operation dialog box.

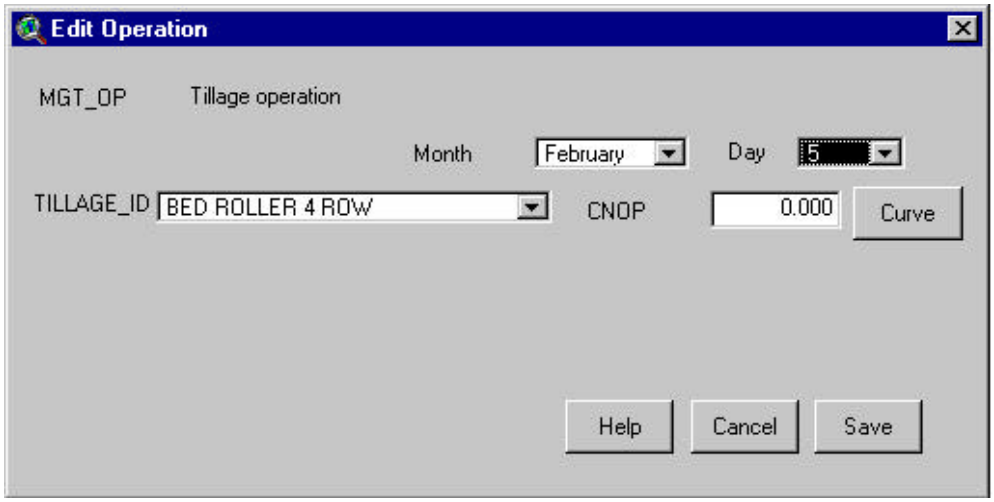


Figure 12.64

d. Make desired changes. Once all editing changes have been made, click **Save** to save changes and close the dialog box. To close the dialog box without saving changes, click **Cancel**.

To add a scenario:

Once the user has set up a management operation schedule, the list of operations may be saved as a scenario. As a scenario, the operation scheduling and management settings are available pre-built for future projects.

a. Click the **Save Scenario**

button.

b. A prompt box will appear (Figure 12.65).

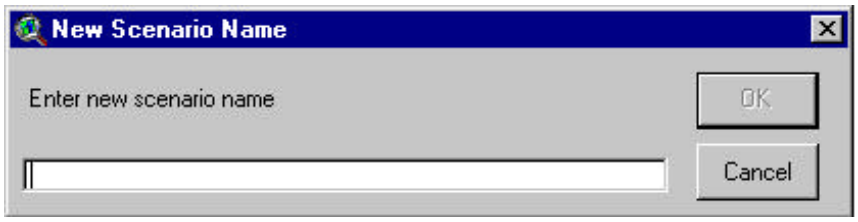


Figure 12.65

c. Enter a name for the scenario and click **OK** or click **Cancel** to terminate the save.

d. If the current management operations were loaded from a scenario, a message box is displayed (Figure 12.66) that gives the user the option to update the original scenario or to save the scenario under another name.

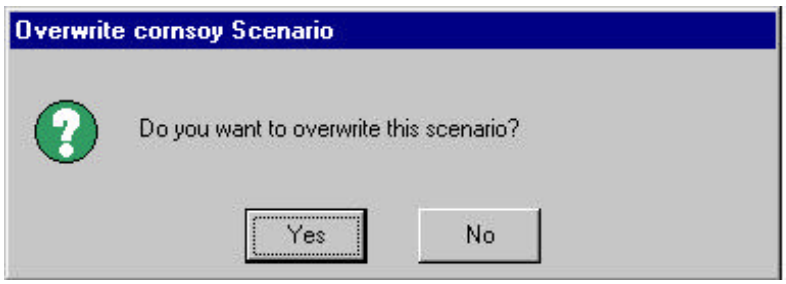


Figure 12.66

e. Click **Yes** to update the original scenario or **No** to create a new scenario.

Note: The scenarios saved by the user are available for use in the current project as well as other projects.

To load a scenario:

a. Click the **Load Scenario**

button.

b. A prompt box will appear (Figure 12.67).

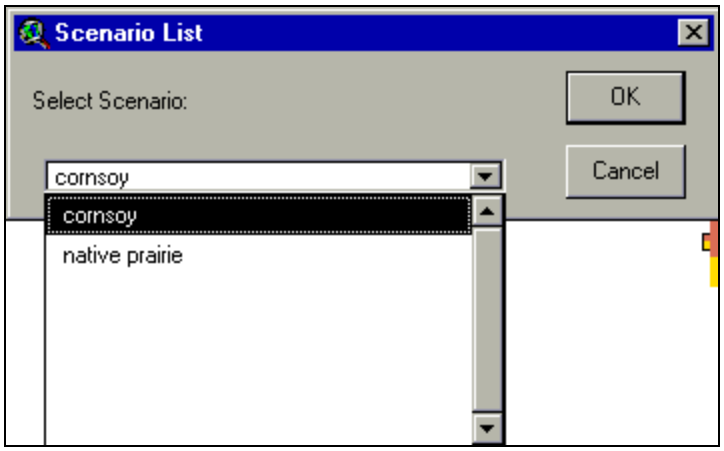


Figure 12.67

c. All scenarios stored in the database are listed. Select one scenario by clicking on it. The scenario name will be displayed in the text box. Click **OK** to continue with the load or click **Cancel** to terminate the load.

d. The existing management operation list will be deleted and replaced with the list of operations for the scenario.

To delete a scenario:

a. Click the **Delete Scenario**

button.

b. A message box will appear (Figure 12.68).

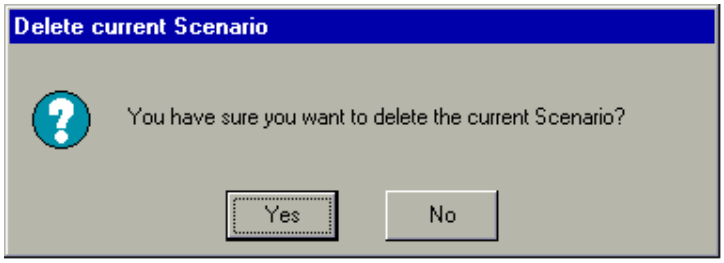


Figure 12.68

c. Click **Yes** to proceed with the deletion. Click **No** to abandon the delete procedure.

d. If **Yes** was clicked, the scenario file will be removed from the database.

Note: When a scenario is deleted, the management schedule is reset to the last schedule saved for the HRU.

3. Exit the management editor dialog box: Once all desired changes have been made to the management parameters, click the **OK** button located in the lower right corner of the Management Data dialog box (Figure 12.69), or click **Cancel** to exit the dialog without saving the changes.

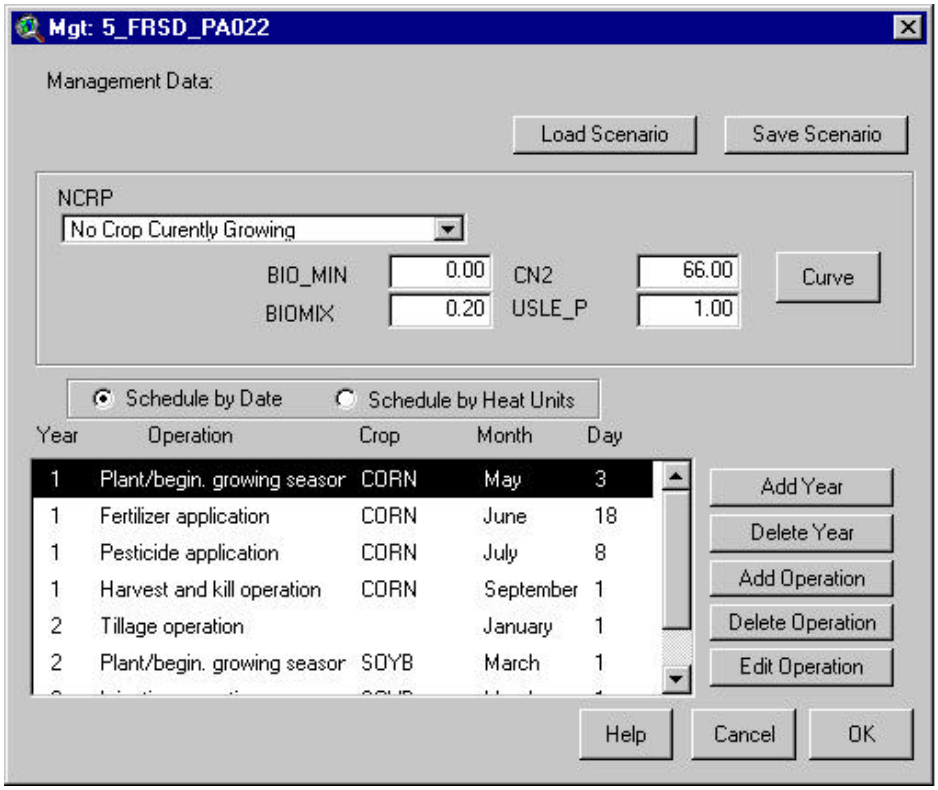


Figure 12.69

4. If the user saves the modified values, once the values are saved for the HRU (by clicking **OK** in Step 3), a new dialog box is displayed allowing the user to copy the modifications to other HRU data sets (Figure 12.70).

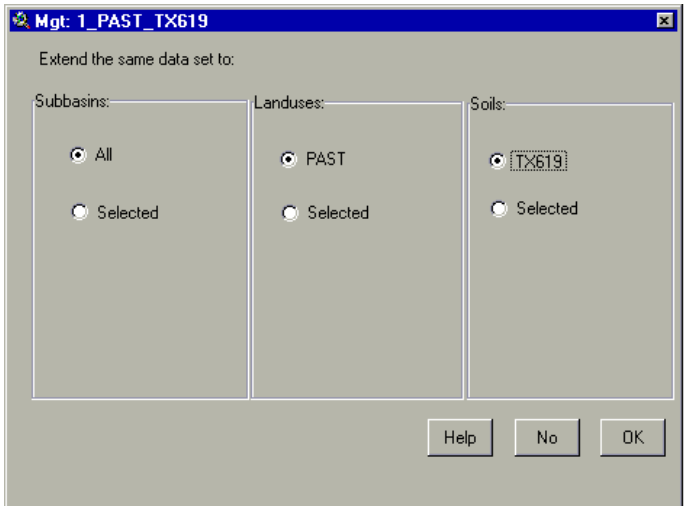


Figure 12.70

a. If you do not want to copy the edited management data to other data sets, click **No**. The changes will have been saved for the individual HRU and no others.

b. If you wish to copy the edited management data to other data sets, proceed as follows:

   i. Specify in the **Subbasins** section of the dialog box the subbasins to which the edited management data will be copied. Select the **All**

button to copy to all the subbasins. If the changes are to be copied to only certain subbasins, click the **Selected**

button to specify the subbasins.

   When the **Selected** button is active, a list of subbasins will be displayed (Figure 12.71).

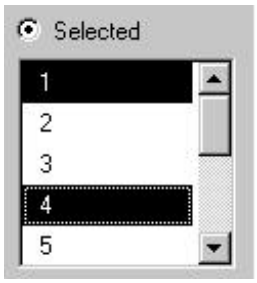


Figure 12.71

   Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

   ii. Specify in the **Landuses** section of the dialog box the landuse type(s) within the specified subbasins to which the management data will be copied. Select the button labeled with the name of the original landuse

to copy the management data only to HRUs with the same landuse type. Click the **Selected**

button to select other/additional landuses.

   When the **Selected** button is active, a list of landuses will be displayed (Figure 12.72).

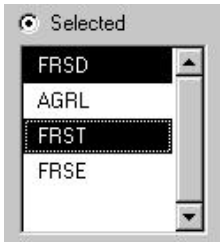
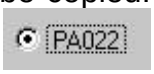
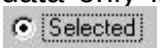


Figure 12.72

Highlight the landuse(s) to which you would like the data to be copied from the new enabled list by clicking on the landuse code. To select more than one landuse, hold down the `Shift` key while highlighting the landuse codes.

iii. Specify in the **Soils** section of the dialog box the soil type(s) available for the specified subbasins/landuse(s) to which the management data will be copied. Select the button labeled with the name of the original soil 

to copy the management data only in the HRUs with this same soil type. Click the **Selected** 

button to select other/additional soil types.

When the **Selected** button is active, a list of soil types will be displayed (Figure 12.73).

Figure 12.73

Highlight the soil type(s) to which you would like the data to be copied from the new enabled list by clicking on the soil type code. To select more than one soil type, hold down the `Shift` key while highlighting the soil type codes.

v. Once you have made all your selections, click **OK**. The interface will save all the changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3).

SECTION 12.9: EDIT SOIL CHEMICAL INPUT DATA (.CHM)

1. Click **.Chm** in the **Select Input File** list of the Edit Subbasin Inputs dialog box. Soil chemical parameters are displayed in a new dialog box titled *Chm: Subbasin #_Landuse type_Soil name* (Figure 12.74). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 23.

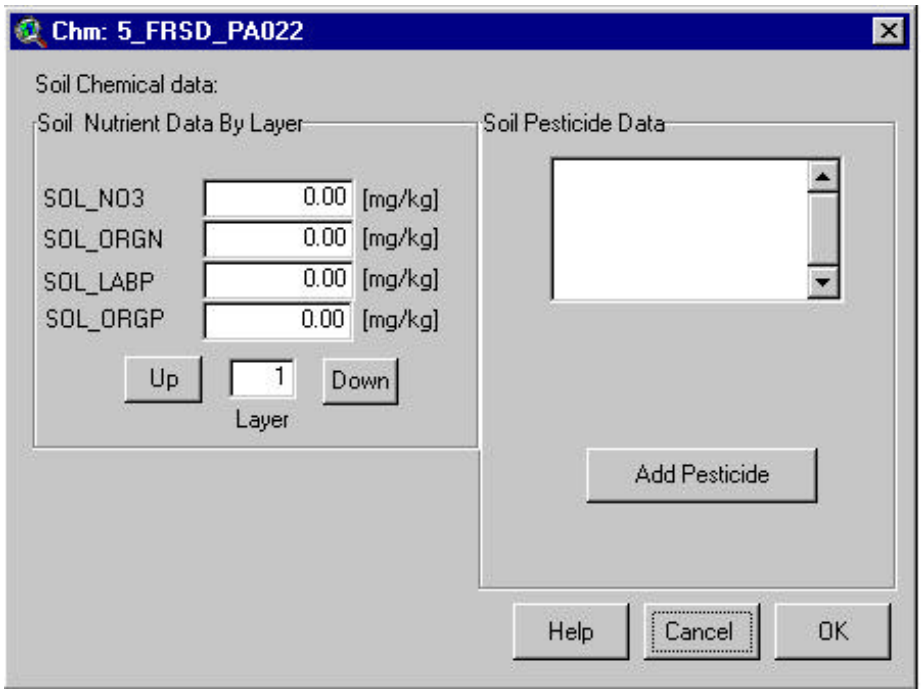
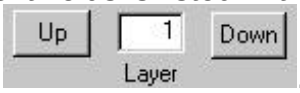


Figure 12.74

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation for parameters is shown.

2. The soil chemical data is divided into two groups. The group of parameters in the section of the dialog box labeled 'Soil Nutrient Data By Layer' define levels of nutrients in the soil layers. The parameters in the portion of the dialog box labeled 'Soil Pesticide Data' allow users to specify pesticide levels in the soil.

3. Edit nutrient data:

   a. The layer number of the data displayed in the text fields is listed in the text box between the **Up** and **Down** buttons 

. The

different layers in the soil profile can be accessed by clicking these buttons.

b. Once the desired layer number is displayed, activate a parameter text box by clicking on it. Type the desired value in the text box. You can move to the next parameter by pressing the Tab key.

c. If the parameter is out of range, a message box like the one in Figure 12.75 pops up.

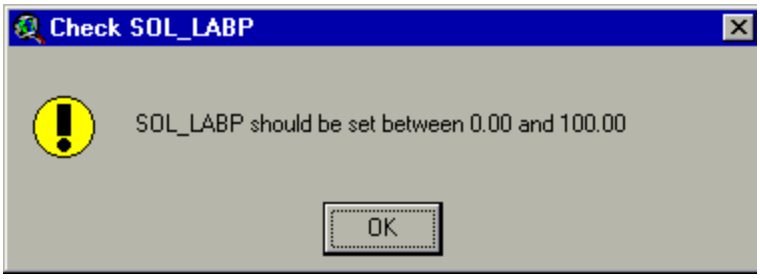


Figure 12.75

Click **OK**. The parameter is reset to the original value or to the closest range limit.

4. Edit pesticide data:

To add a pesticide:

a. Click the **Add Pesticide** button.

b. A prompt box with a drop-down list is displayed (Figure 12.76). All pesticides in the SWAT pesticide database (see Section 14) are listed.

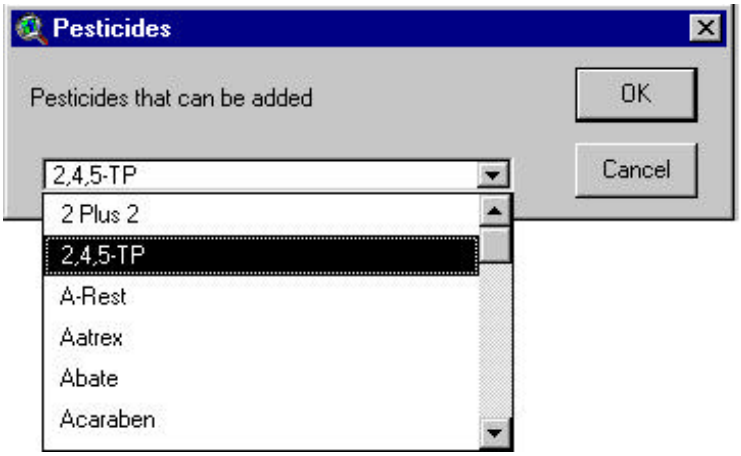


Figure 12.76

c. Select the desired pesticide by clicking on the name. Click **OK** to add the pesticide to the list of pesticides in the soil or click **Cancel** to close the prompt box without adding a pesticide to the list.

d. If the user added a pesticide, the pesticide name will appear in the list of pesticides in the Soil Pesticide Data section (Figure 12.77).

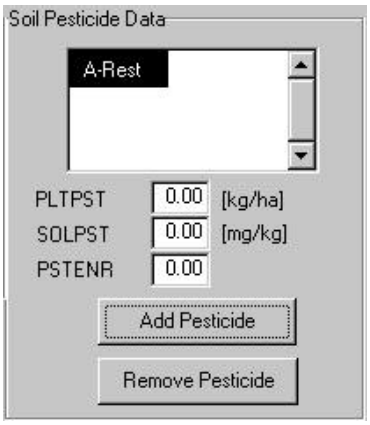


Figure 12.77

Note: Up to 10 different pesticides may be simulated in the land areas of the watershed. Pesticides are input in the soil chemical (.chm) and management (.mgt) files.

e. To edit the parameter values for a particular pesticide, highlight the pesticide name. Activate a parameter text box by clicking on it. Type the desired value in the text box. You can move to the next parameter by pressing the Tab key. If the value of the parameter is out of range, a message box like the one in Figure 12.78 pops up.

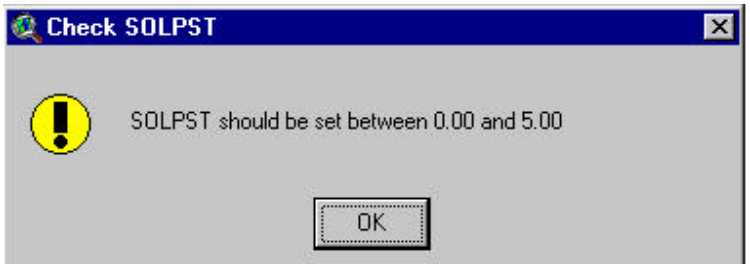


Figure 12.78

Click **OK.** The parameter is reset to the original value or to the closest range limit.

Note: Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir*/avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by

"rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

To remove a pesticide:

a. Highlight the name of the pesticide in the list by clicking on it.

b. Click the **Remove Pesticide**

button.

c. A prompt box appears (Figure 12.79).

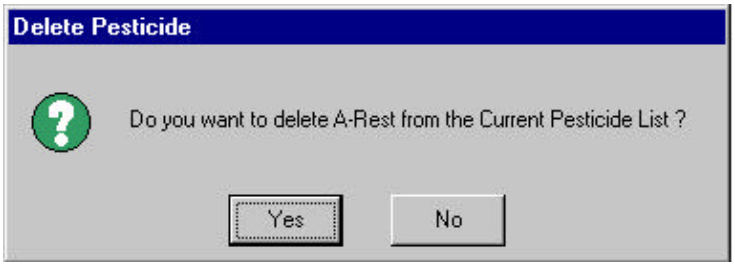


Figure 12.79

d. Click **Yes** to remove the pesticide or click **No** to abandon the delete process.

5. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

   If unacceptable parameter values were inserted, a message box similar to the one described in Step 2 may be displayed. Click **OK** and return to editing and correct the values.

6. If the user saves the modified values, once the values are saved for the HRU (by clicking **OK** in Step 3), a new dialog box is displayed allowing the user to copy the modifications to other HRU data sets (Figure 12.80).

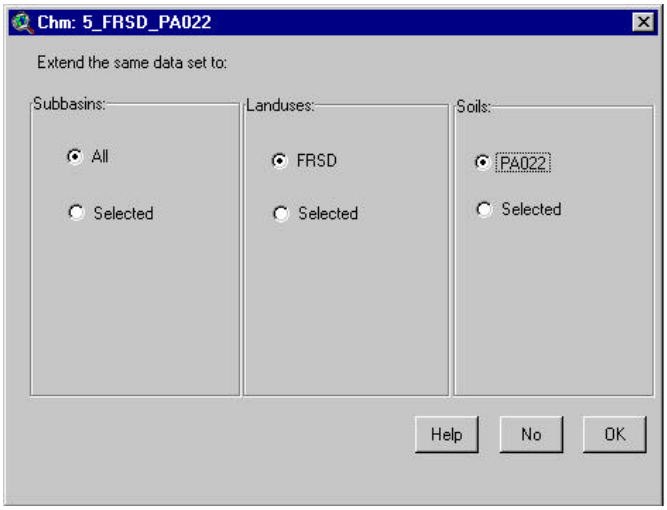


Figure 12.80

a. If you do not want to copy the edited soil chemical data to other data sets, click **No**. The changes will have been saved for the individual HRU and no others.

b. If you wish to copy the edited soil chemical data to other data sets, proceed as follows:

   i. Specify in the **Subbasins** section of the dialog box the subbasins to which the edited soil chemical data will be copied. Select the **All**

button to copy to all the subbasins. If the changes are to be copied to only certain subbasins, click the **Selected**

button to specify the subbasins.

   When the **Selected** button is active, a list of subbasins will be displayed (Figure 12.81).

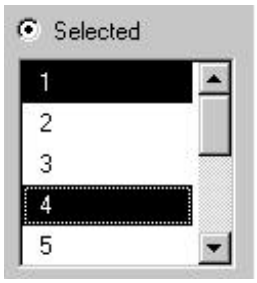


Figure 12.81

   Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

   ii. Specify in the **Landuses** section of the dialog box the landuse type(s) within the specified subbasins to which the soil chemical data will be copied. Select the button labeled with the name of the original landuse

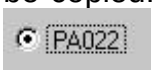
to copy the soil chemical data only to HRUs with the same landuse type. Click the **Selected**

button to select other/additional landuses.

   When the **Selected** button is active, a list of landuses will be displayed (Figure 12.82).

Figure 12.82

Highlight the landuse(s) to which you would like the data to be copied from the new enabled list by clicking on the landuse code. To select more than one landuse, hold down the `Shift` key while highlighting the landuse codes.

iii. Specify in the **Soils** section of the dialog box the soil type(s) available for the specified subbasins/landuse(s) to which the soil chemical data will be copied. Select the button labeled with the name of the original soil 

to copy the soil chemical data only in the HRUs with this same soil type. Click the **Selected**

button to select other/additional soil types.

When the **Selected** button is active, a list of soil types will be displayed (Figure 12.83).

Figure 12.83

Highlight the soil type(s) to which you would like the data to be copied from the new enabled list by clicking on the soil type code. To select more than one soil type, hold down the `Shift` key while highlighting the soil type codes.

iv. Once you have made all your selections, click **OK**. The interface will save all the changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3).

SECTION 12.10: EDIT POND/WETLAND INPUT DATA (.PND)

1. Click **.Pnd** in the **Select Input File** list of the Edit Subbasin Inputs dialog box. Subbasin impoundment parameters are displayed in a new dialog box titled *Pnd: Subbasin #* (Figure 12.84). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 28.

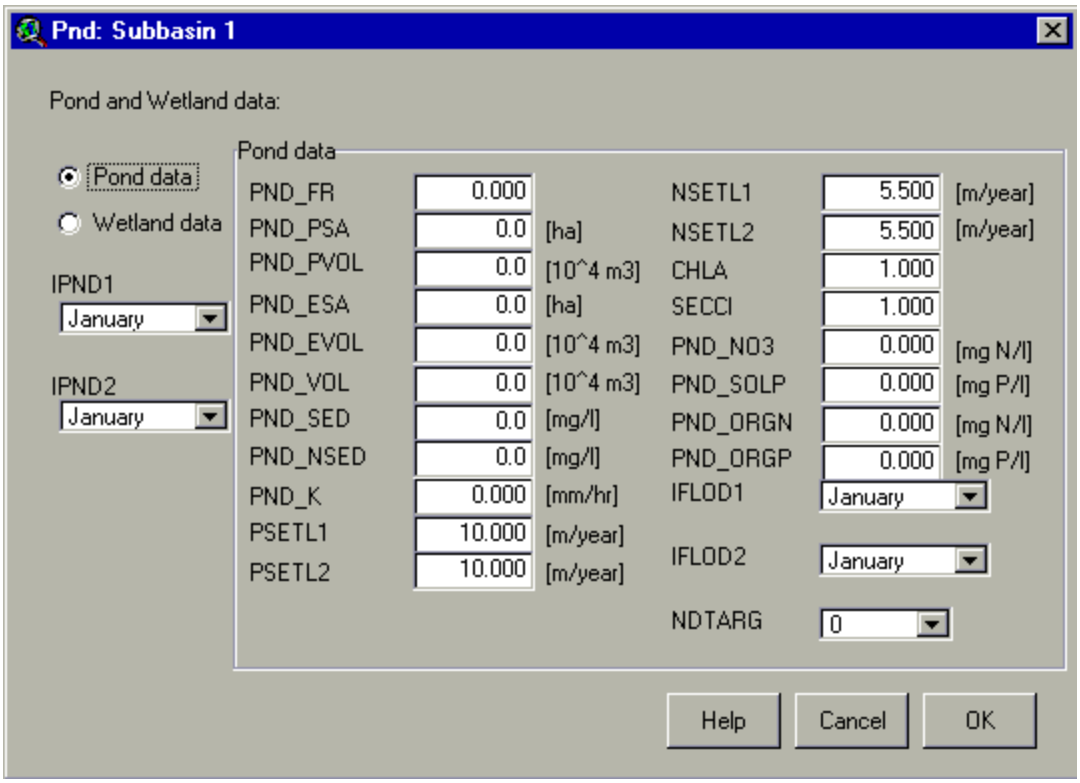
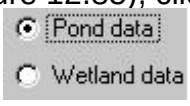


Figure 12.84

2. When the dialog box appears, the subbasin pond parameters are shown. To display the subbasin wetland parameters (Figure 12.85), click the radio button adjacent to **Wetland Data**. The radio buttons 

allow the user to switch back and forth between the two data types.

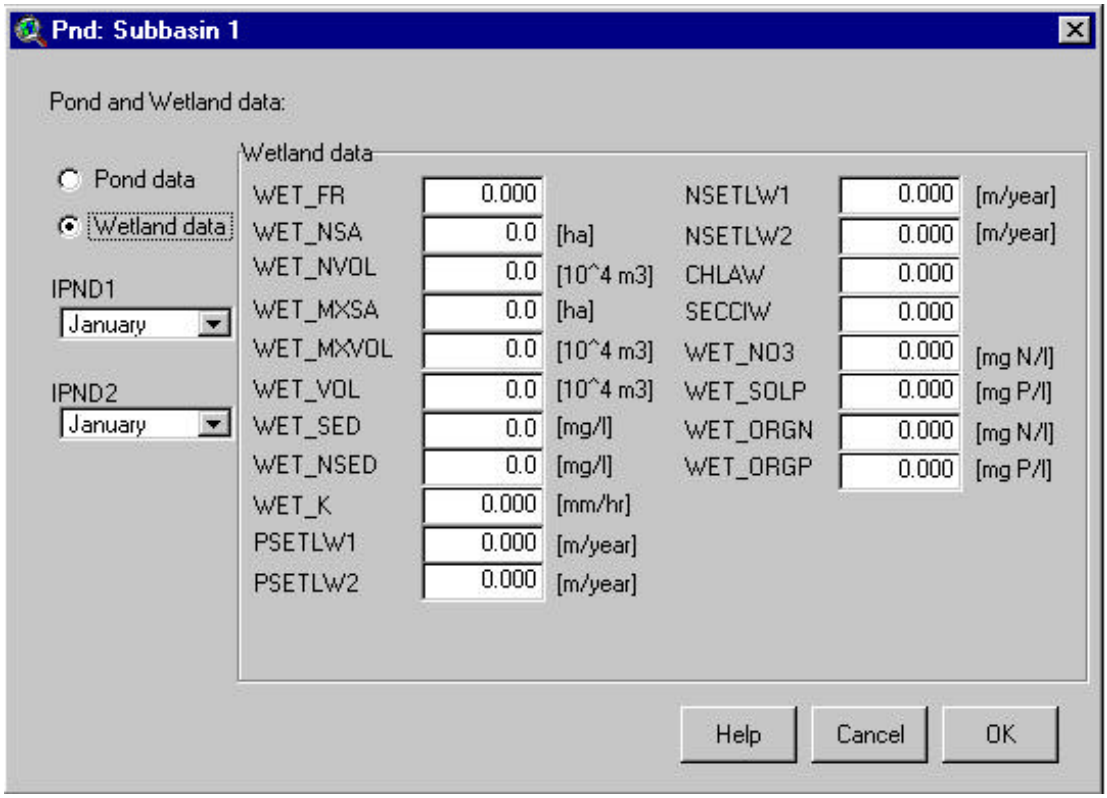


Figure 12.85

Note: In both pages if the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation for parameters is shown.

3. Edit data: Activate a parameter text box by clicking on it. Type the desired value. You can move to the next parameter text box by pressing the Tab key. If the value of the parameter is out of range, a message box like the one in Figure 12.86 pops up.

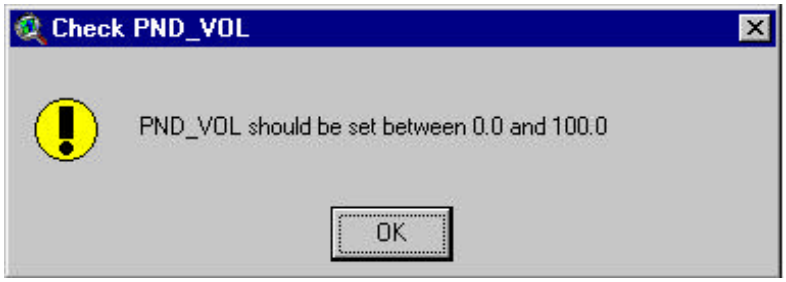


Figure 12.86

Click **OK**. The parameter is reset to the original value or to the closest range limit.

Note: Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir*/avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

4. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

   If unacceptable parameter values were inserted, a message box similar to the one described in step 3 may be displayed. Click **OK** and return to editing and correct the values.

5. If the user saves the modified values, once the values are saved for the subbasin (by clicking **OK** in step 4) a new dialog box is displayed allowing the user to copy the modifications to other subbasin data sets (Figure 12.87).

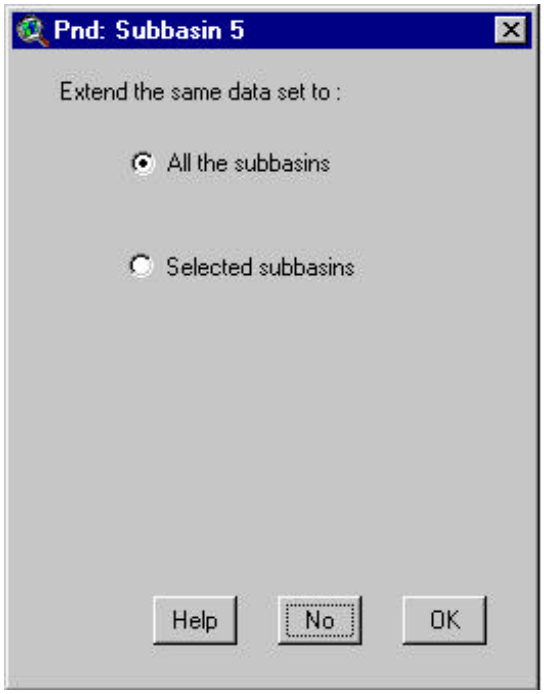
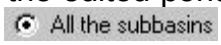
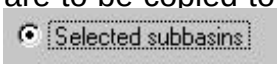


Figure 12.87

   a. If you do not want to copy the edited pond/wetland data to other data sets, click **No**. The changes will have been saved for the individual subbasin and no others.

b. If you wish to copy the edited pond/wetland data to other data sets, proceed as follows:

In the dialog box, specify the subbasins to which the edited pond/wetland data will be copied. Select the **All the subbasins** 

button to copy to all subbasins in the watershed. If the changes are to be copied to only certain subbasins, click the **Selected subbasins** 

button to choose the subbasins.

When the **Selected subbasins** button is active, a list of subbasins will be displayed (Figure 12.88)

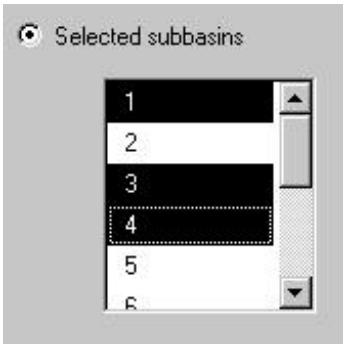


Figure 12.88

Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

Once you have made all selections, click **OK**. The interface will save all the changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3).

SECTION 12.11: EDIT STREAM WATER QUALITY INPUT DATA (.SWQ)

1. Click **.Swq** in the **Select Input File** list of the Edit Subbasin Inputs dialog box. Stream water quality parameters are displayed in a new dialog box titled *Swq: Subbasin #* (Figure 12.89). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 27.

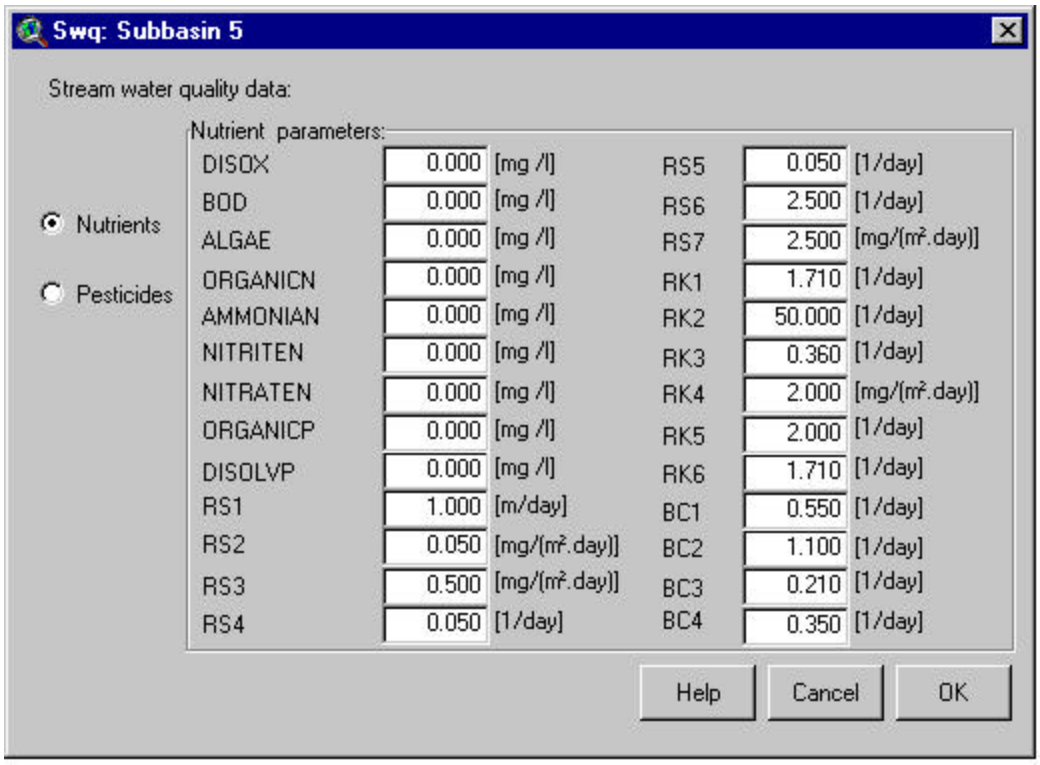
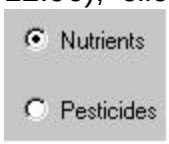


Figure 12.89

2. When the dialog box appears, the stream nutrient water quality parameters are shown. To display the stream pesticide water quality parameters (Figure 12.90), click the radio button adjacent to **Pesticides**. The radio buttons



allow the user to switch back and forth between the two data types.

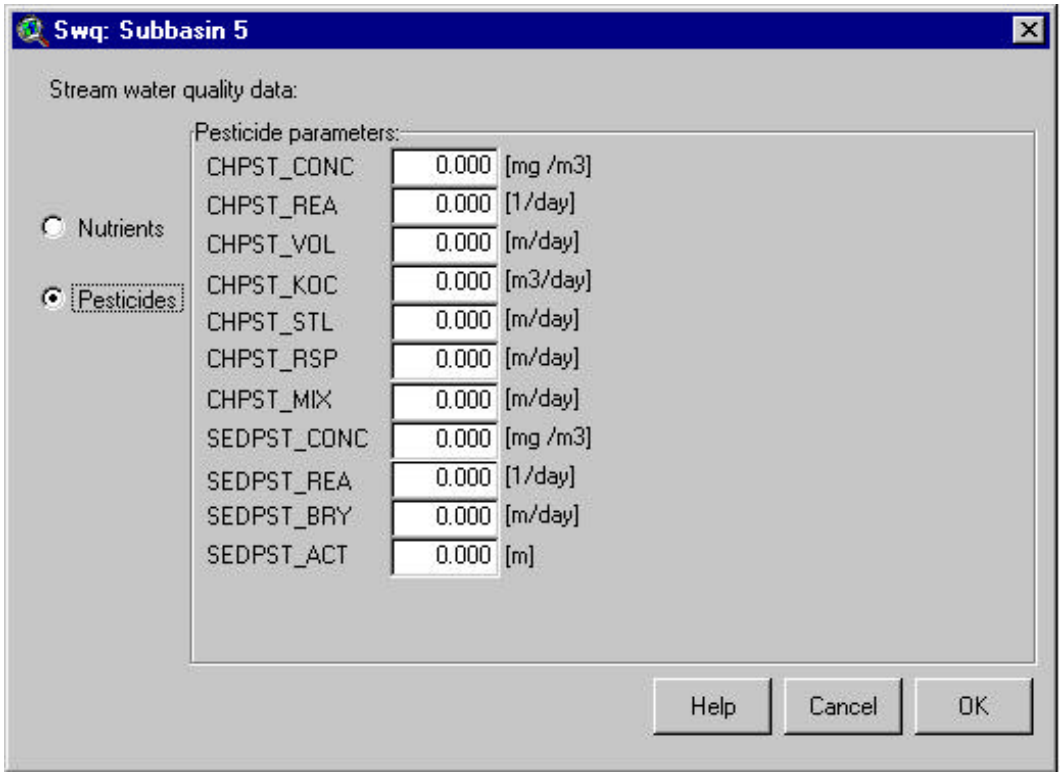


Figure 12.90

Note: In both pages if the cursor is placed on top of any text box or button, a short help description (yellow label) appears, and the range of variation for parameters is shown.

3. Edit data: Activate a parameter text box by clicking on it. Type the desired value. You can move to the next parameter text box by pressing the Tab key. If the value of the parameter is out of range a message box like the one in Figure 12.91 pops up.

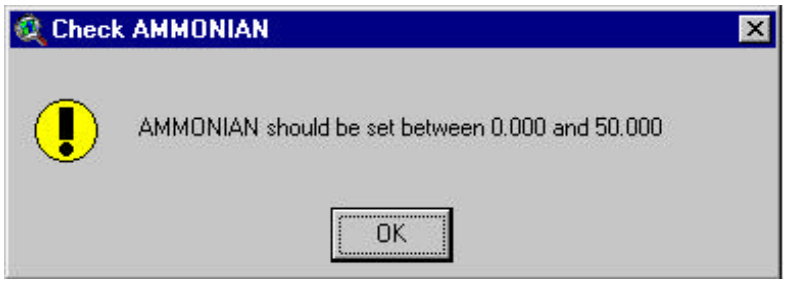


Figure 12.91

Click **OK**. The parameter is reset to the original value or to the closest range limit.

Note: Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not

be reasonable for your watershed. All tables containing the variable ranges are located in the */installation dir/*avswatdb directory. The names of the files are the three letter extensions of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

4. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

   If unacceptable parameter values were inserted, a message box similar to the one described in step 3 may be displayed. Click **OK** and return to editing and correct the values.

5. If the user saves the modified values, once the values are saved for the subbasin (by clicking **OK** in step 4) a new dialog box is displayed allowing the user to copy the modifications to other subbasin data sets (Figure 12.92).

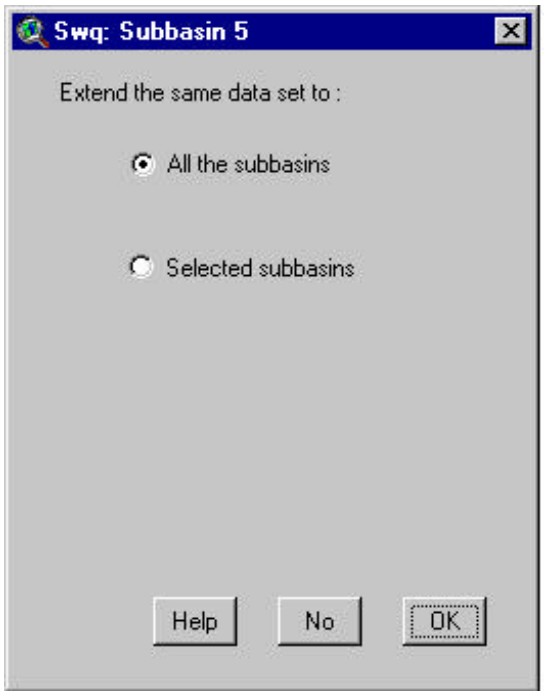
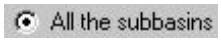
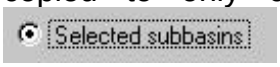


Figure 12.92

   a. If you do not want to copy the edited stream water quality data to other data sets, click **No**. The changes will have been saved for the individual subbasin and no others.

   b. If you wish to copy the edited stream water quality data to other data sets, proceed as follows:

In the dialog box, specify the subbasins to which the edited stream water quality data will be copied. Select the **All the subbasins** 

button to copy to all subbasins in the watershed. If the changes are to be copied to only certain subbasins, click the **Selected subbasins** 

button to choose the subbasins.

When the **Selected subbasins** button is active, a list of subbasins will be displayed (Figure 12.93)

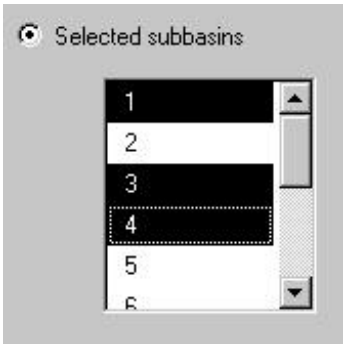


Figure 12.93

Highlight the subbasin(s) to which you would like the data to be copied from the new enabled list by clicking on the subbasin number. To select more than one subbasin, hold down the `Shift` key while highlighting the subbasin numbers.

6. Once you have made all selections, click **OK**. The interface will save all the changes and return to the Edit Subbasins Inputs dialog box (Figure 12.3)

SECTION 13: SWAT OUTPUT

The **Simulation** menu is one of four new menus introduced in the SWAT View. The **Simulation** menu allows you to finalize the set up of input for the SWAT model and run the SWAT model, "read" the results of the simulation and build dBASE tables, apply a calibration tool and perform load calculations. Select the **Simulation** menu using the mouse or by pressing the ALT+S keys. Four items are listed on the **Simulation** menu (Figure 13.1).

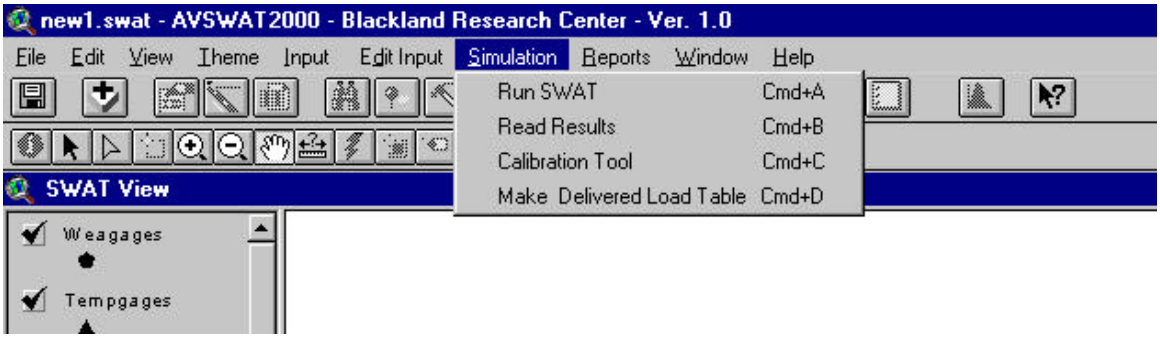


Figure 13.1

SECTION 13.1: RUN SWAT

The first command in the **Simulation** menu allows the user to set up and run the SWAT model.

1. Select **Run SWAT** from the **Simulation** menu on the SWAT View (or press Cmd+A) (Figure 13.2).

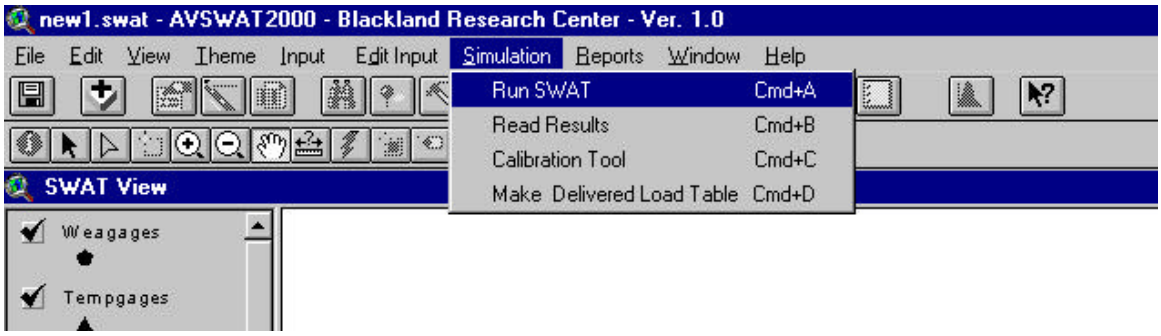


Figure 13.2

2. The Set Up and Run SWAT model simulation dialog box is displayed. (Figure 13.3).

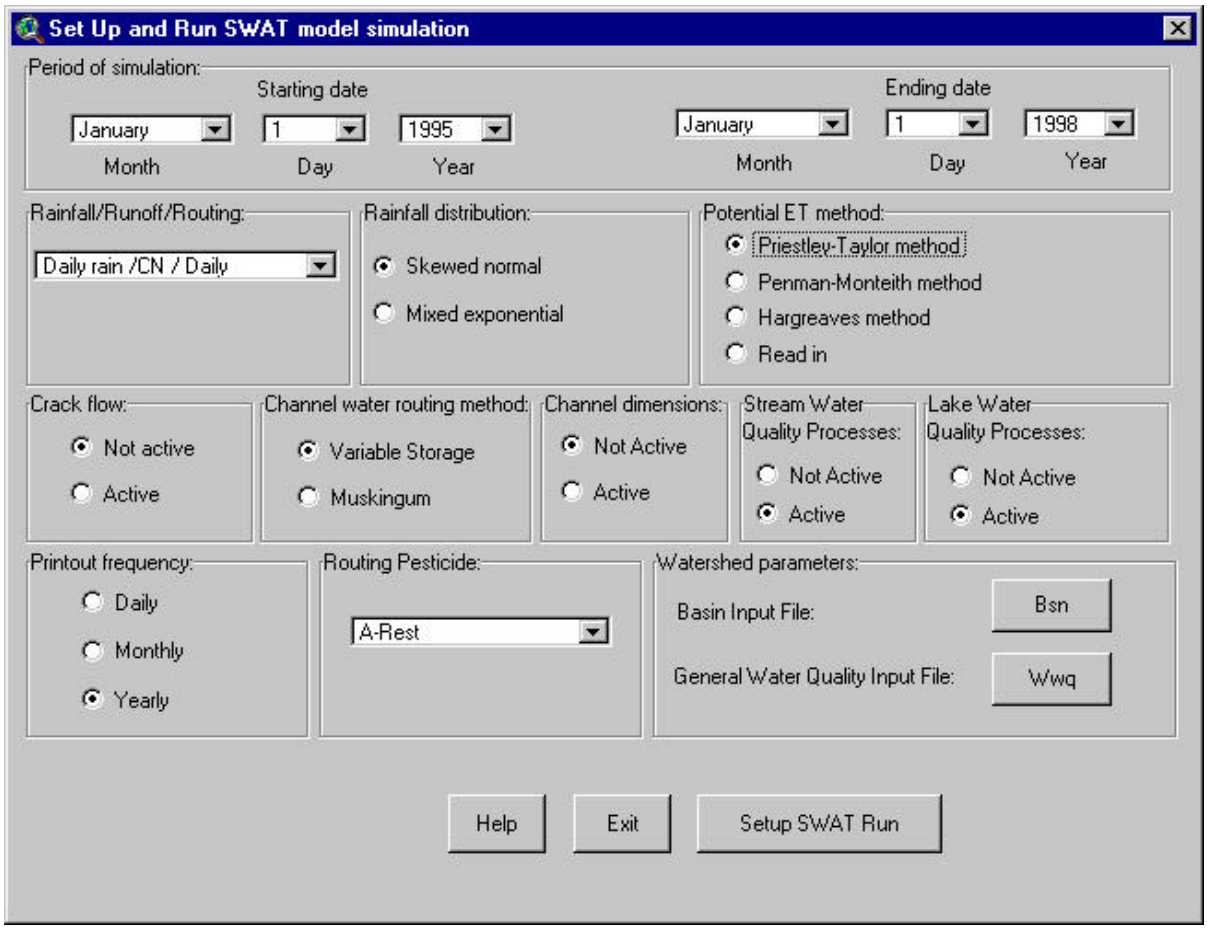
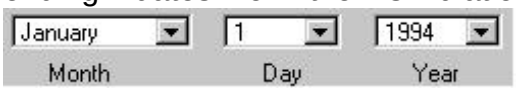


Figure 13.3

3. The dialog box contains several sections in which the user defines the option to be used in the simulation of various processes.

   a. Period of Simulation. In this section, the user specifies the starting and ending dates of the simulation using the drop-down controls 

(click on the drop down list or use the up and down arrow keys to change the value).

   b. Rainfall/Runoff/Routing. In this section, the user specifies the options to be used for precipitation time step, runoff calculation method, and routing time step. The user may choose between three options (Figure 13.4):

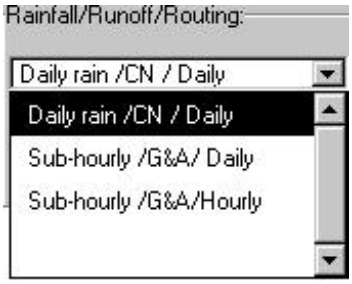


Figure 13.4

i. **Daily rain/CN/ Daily** refers to daily rainfall/curve number runoff/daily routing.

ii. **Sub-hourly/G&A/Daily** refers to sub-hourly rainfall/Green & Ampt runoff/daily routing.

iii. **Sub-hourly/G&A/Hourly** refers to sub-hourly rainfall/Green & Ampt runoff/hourly routing.

See *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 4 for more information about these options.

> **Note:** The second and third options require sub-daily precipitation. See *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 7 for the required formats. The user must create a file called pcp.pcp with the sub-hourly precipitation to replace the pcp.pcp file generated by the interface when the button labeled **Set up Swat Run** (see 3 below) is clicked. The file should be copied to the directory: *project name*\scenarios\default\txtinout before running the model.

c. Rainfall distribution. In this section, the user selects the distribution used to generate precipitation data. The user may choose between two options, **Skewed normal** or **Mixed exponential,** using the respective radio button. When **Mixed exponential** is selected, a text box is enabled for the user to specify the exponent (Figure 13.5)

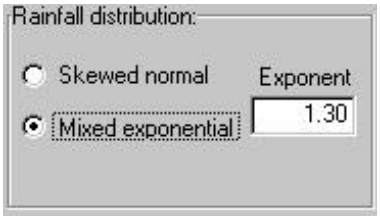


Figure 13.5

See *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 4 for more information about these options.

d. Potential ET method. In this section, the user selects the method used to estimate potential evapotranspiration (PET). The available options are **Priestly-Taylor**, **Penman-Monteith**, **Hargreaves,** and **Read in**. When **Read in** is selected, the user must provide a file, either in dBASE or txt format, containing the daily value of PET (Figure 13.6). The format of the PET file is given in Section 3.

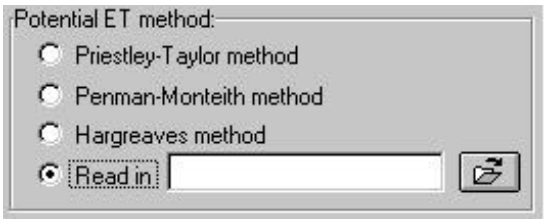


Figure 13.6

To import PET file:

i. Click the open file folder button.

ii. A prompt box is displayed (Figure 13.7).

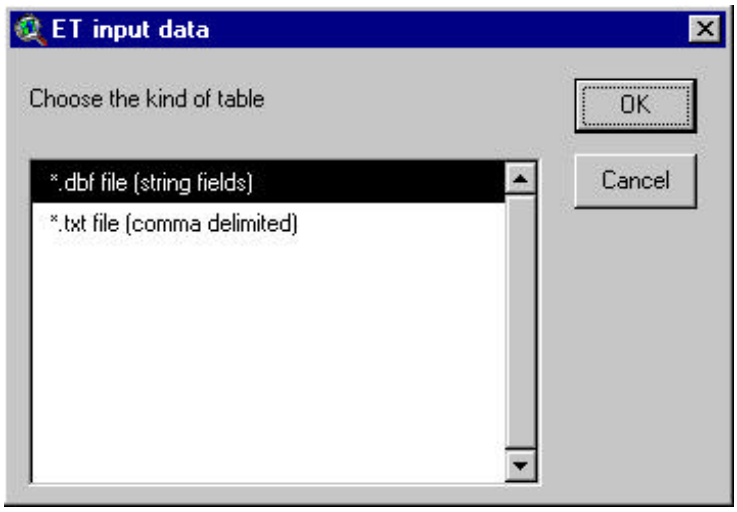


Figure 13.7

iii. Select the proper format and click **OK** (or double click selection).

iv. A browser will appear, allowing the user to locate and select the file containing the daily PET values (Figure 13.8)

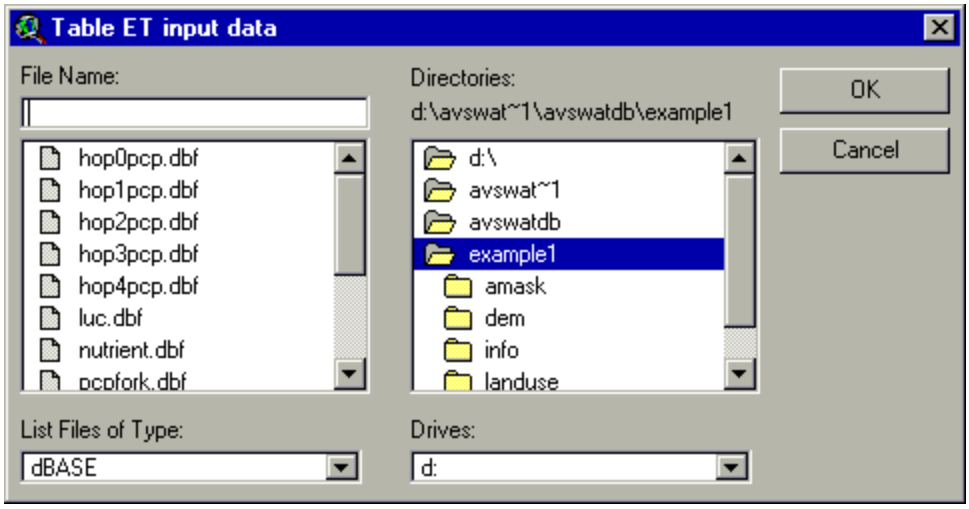


Figure 13.8

v. Highlight the name of the file and click **OK** (or double click selection).

vi. The name of the table will be displayed in the text box beside the **Read in** radio button.

See *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 4 for more information about these options.

e. Crack flow. In this section, the user specifies whether crack or by-pass flow will or will not be simulated. The user has two options, **Not active** or **Active** (Figure 13.9). To select an option, click the radio button adjacent to the option.

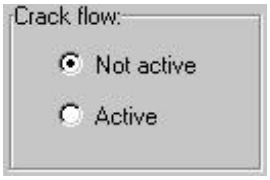


Figure 13.9

See *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 4 for more information about these options.

f. Channel water routing method. In this section, the user selects the method used to route water in the channel network. Two options are available, the **Variable storage** and **Muskingum** methods (Figure 13.10). To select an option, click the radio button adjacent to the option.

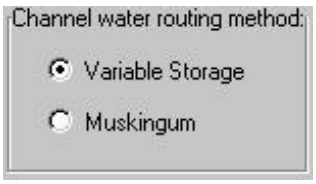


Figure 13.10

See *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 4 for more information about these options.

g. Channel dimensions. In this section, the user defines whether or not channel dimensions are allowed to change during the course of the simulation due to channel degradation (Figure 13.11).

Figure 13.11

See *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 4 for more information about these options.

h. Stream Water Quality processes. In this section, the user decides whether or not in-stream nutrient transformations are calculated (Figure 13.12).

Figure 13.12

See *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 4 for more information about these options.

i. Lake Water Quality processes. This section controls the printing of lake water quality information. During the reformatting of output files, this variable became obsolete, and the selection of a radio button has no impact on model results (Figure 13.13).

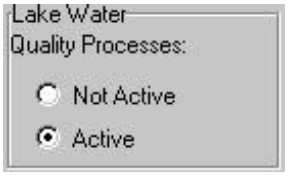


Figure 13.13

j. Print out frequency. This section controls the frequency of print output. The available options are: **Daily**, **Monthly,** and **Yearly** (Figure 13.14).

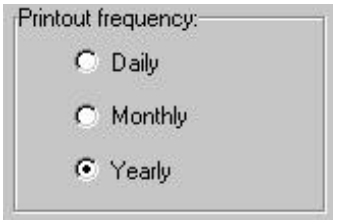


Figure 13.14

k. Routing Pesticide. In this section, the user can select the pesticide that SWAT will route through the channel network during the simulation. A drop-down control provides a list of all pesticides simulated in the HRUs (Figure 13.15).

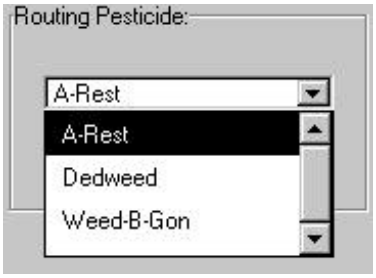


Figure 13.15

> **Note:** This list reports pesticides that eventually have been set within the MGT (Management Input Data, see Section 12.8) and/or CHM (Soil Chemical Input Data, see Section 12.9) inputs.

l. Watershed Parameters. Within this section, the user can access parameters in two additional input files (Figure 13.16).

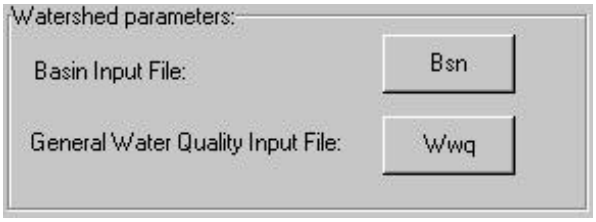


Figure 13.16

Basin Input File.

i. To access parameters stored in the .bsn file, click the

button. A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 5.

ii. The Bsn Inputs dialog box is displayed. The dialog box contains two pages of parameters (Figure 13.17 and 13.18).

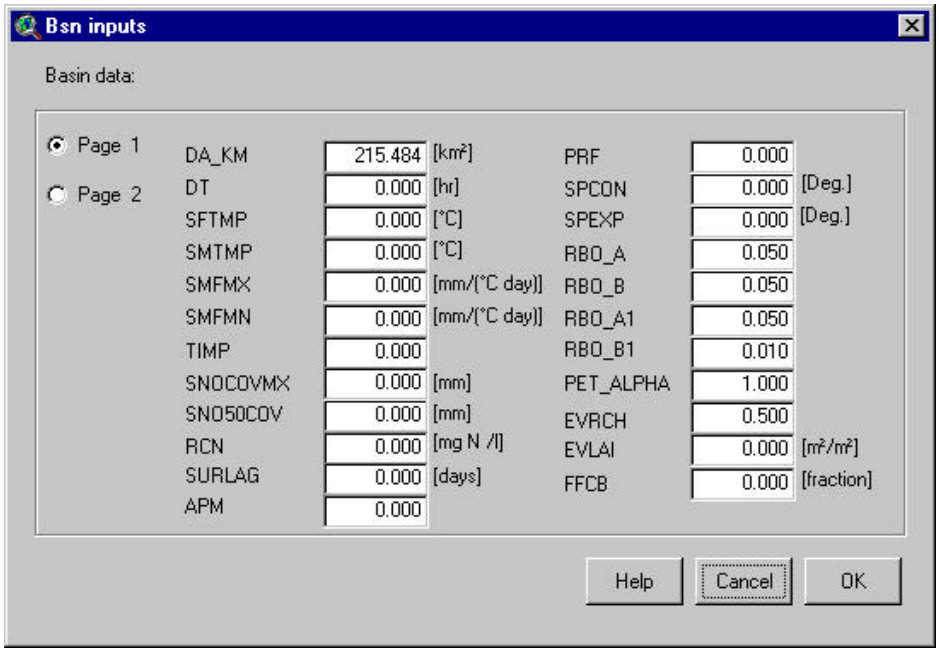


Figure 13.17

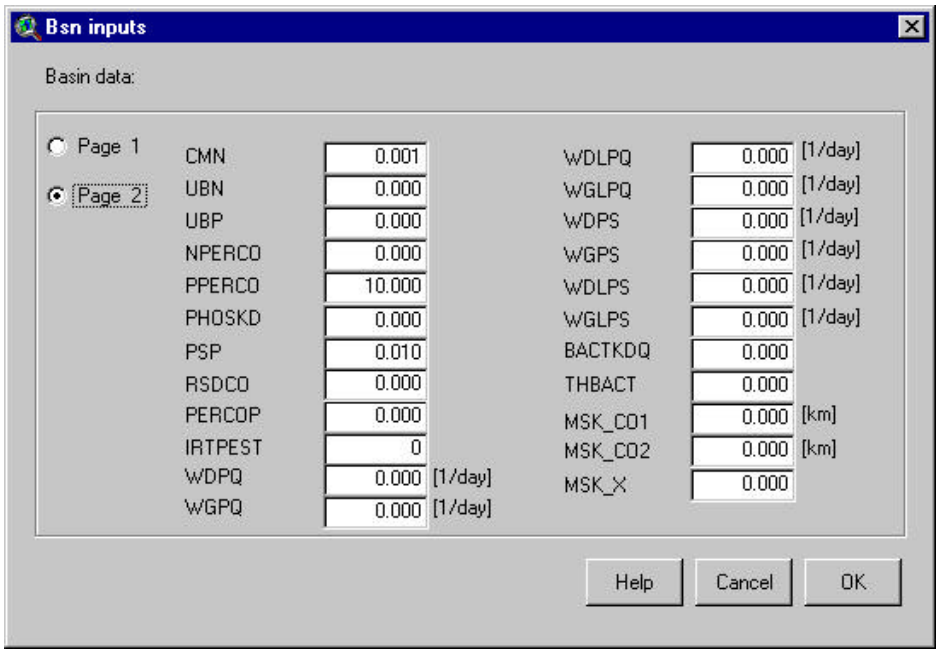


Figure 13.18

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears, and the range of variation for the parameter is shown.

iii. To switch from one page to the other, click the radio buttons next to the page number of interest

.

iv. Edit data: Activate a parameter text box by clicking on it. Type the desired value. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 13.19 will be displayed.

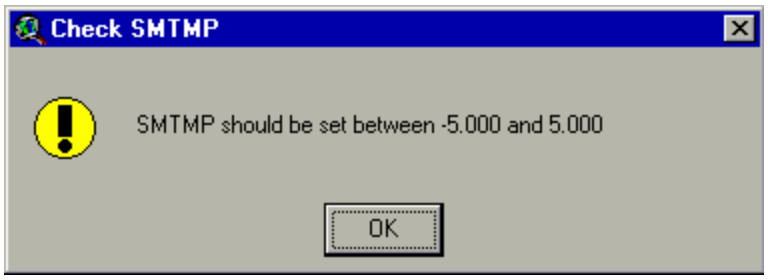


Figure 13.19

Click **OK.** The parameter will be reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *linstallation dir*/avswatdb directory. The names of the files are the three letter extension of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

v. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

General Water Quality Input File.

i. To access parameters stored in the .wwq file, click the

button. A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 26.

ii. The Wwq Inputs dialog box is displayed (Figure 13.20).

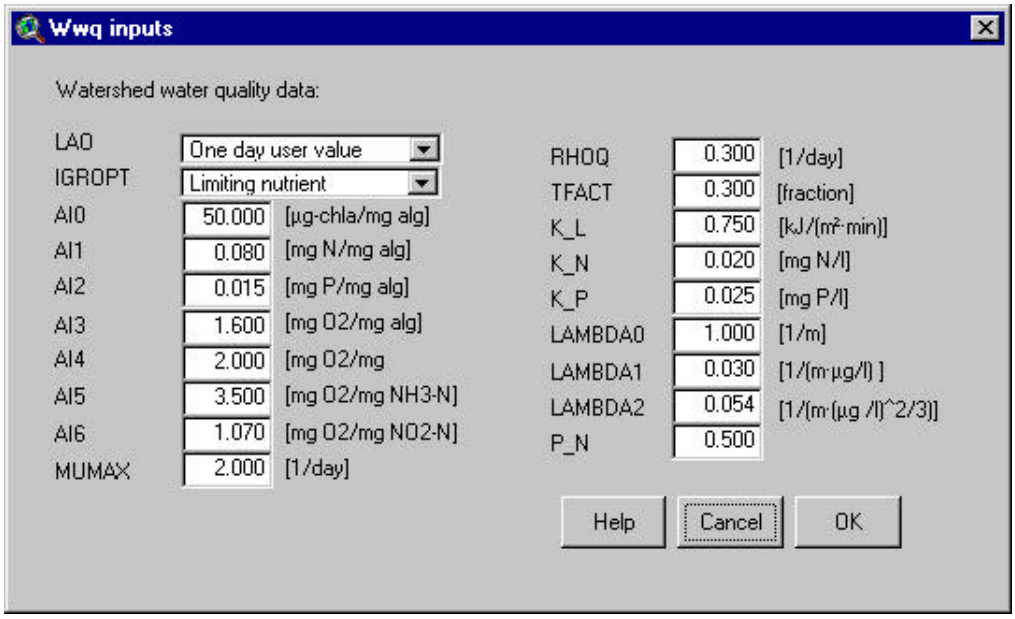


Figure 13.20

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation for the parameter is shown.

iii. Edit data: Activate a parameter text box by clicking on it. Type the desired value. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 13.21 will be displayed.

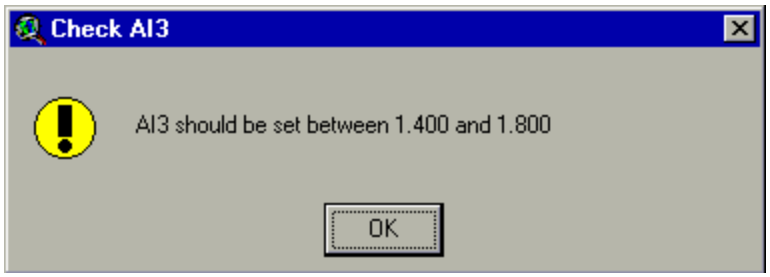


Figure 13.21

Click **OK.** The parameter will be reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *linstallation dir*/avswatdb directory. The names of the files are the three letter extension of the SWAT ASCII input files listed in the

Soil and Water Assessment Tool User's Manual, Version 2000, followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

iv. Close the dialog box: Click **Cancel** to exit without saving the changes. Click **OK** to exit and save any changes made.

4. Once all options and parameters are defined. Click the **Setup SWAT Run**

button. This button generates the final input files based on the settings defined in the Set Up and Run SWAT model Simulation dialog box. The main tasks performed during this process include preparing the weather input files and verifying that the provided data (reservoir outflow, point source and inlet discharge, evapotranspiration) is complete and consistent with the set period of simulation. An error massage will notify the user of any problems.

5. When the preprocessing is successfully completed, a message box is displayed (Figure 13.22).

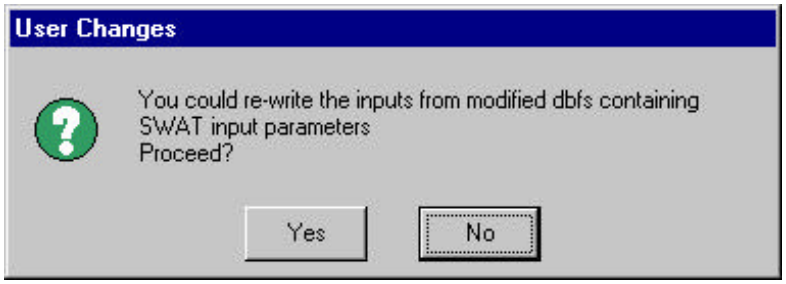


Figure 13.22

6. If some dBASE files were modified without using the built-in editing tools described in Section 12, click **Yes** to re-write the ASCII input files used to run SWAT from the modified dBASE files. If no dBASE files were modified outside the editing tools, click **No**.

   If the user clicked **Yes**, a list of input file types is displayed (Figure 13.23).

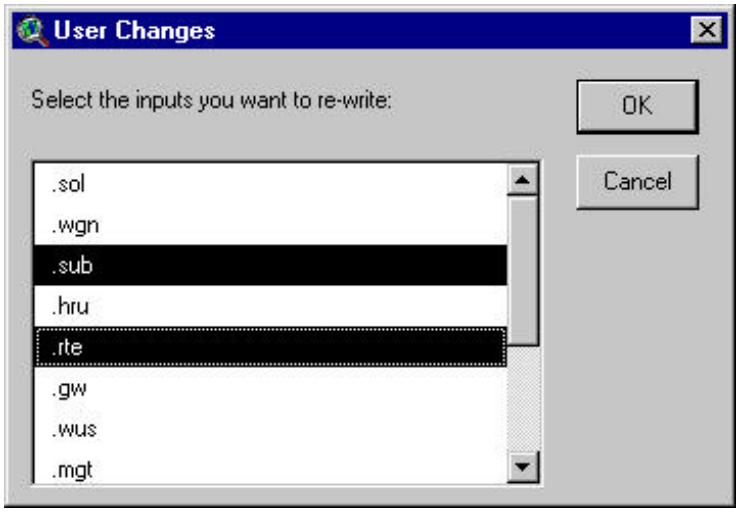


Figure 13.23

Select the input file type(s) that needs to be re-written, and click **OK** or click **Cancel** to exit the list box. If files were re-written, a message will notify the user when all input files have been re-written (Figure 13.24).

Figure 13.24

Click **OK**.

7. Once the Setup of the SWAT input files is complete, two additional buttons are displayed on the Set Up and Run SWAT model Simulation dialog box (Figure 13.25).

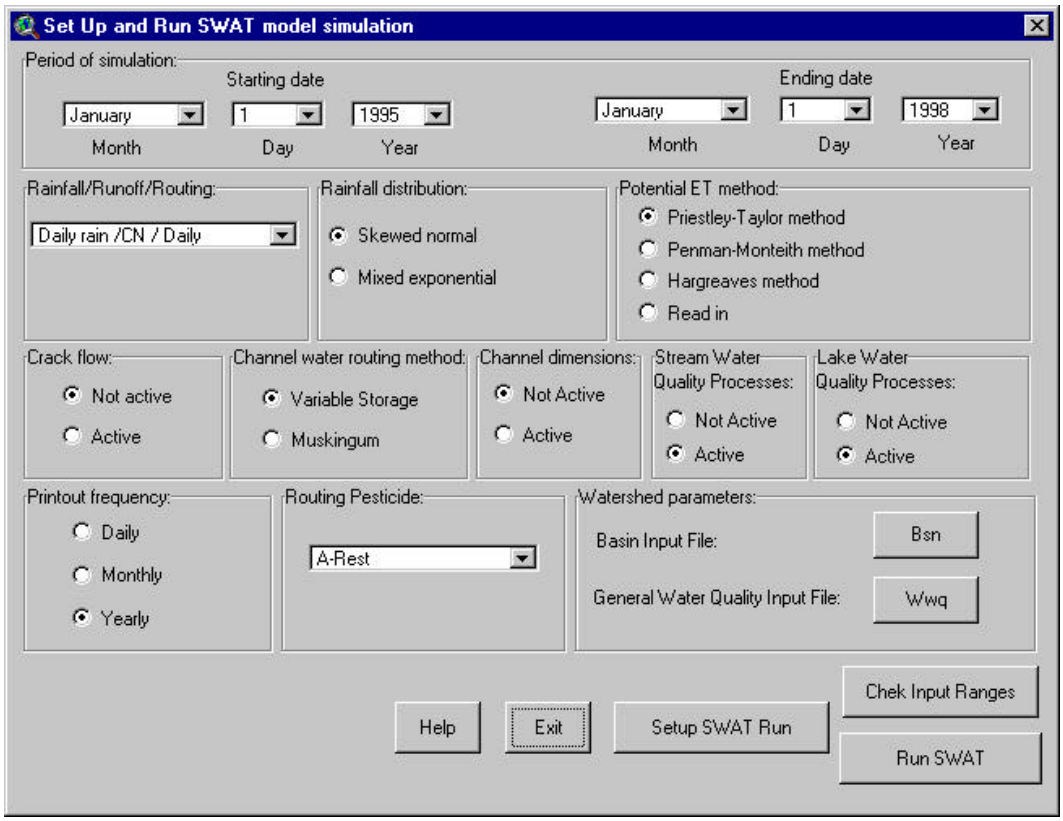


Figure 13.25

8. A tool is provided to check the ranges of all input values. The user is not required to check the ranges of input values. If the user wishes to check input values, the following steps are performed:

   a. Click the **Check Input Ranges** button.

   b. A warning is displayed (the process could be time consuming) (Figure 13.26)

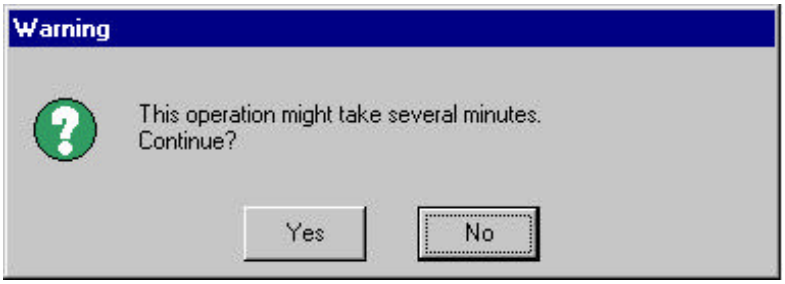


Figure 13.26

c. Click **No** to abort the checking process, or click **Yes** to continue. The checking process searches for parameters in the current watershed input database and the SWAT model databases that are out of range. When finished, a message notifies the user (Figure 13.27).

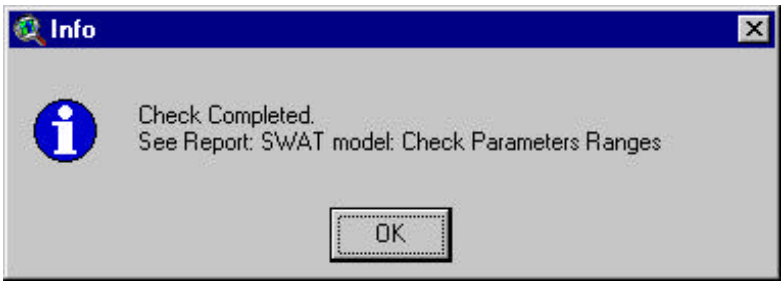
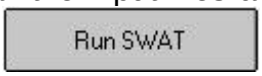


Figure 13.27

d. The tool has generated a report titled: **SWAT model: check parameter ranges**. This report is accessed by selecting **Show List** from the **Reports** menu and selecting the report name from the list.

e. Click **OK**.

Tip:	The user should correct the input parameters that are out of range before proceeding with the SWAT model run.

9. Once all the input files are built, the user can run the model. Click the **Run SWAT** 

button.

10. When the SWAT simulation terminates, one of two message boxes will be displayed:

a. If the simulation terminates before the entire simulation period is completed, a message box will notify the user of a run failure (Figure 13.28).

Figure 13.28

Click **OK**. Review inputs before attempting a new run.

> **Note:** To debug a simulation:
>
> 1. Copy swat2000.exe from the avswatpr directory to the txtinout directory for the project.
> 2. Open a DOS command prompt window. Change to the project txtinout directory and run swat2000.exe
> 3. The error message will remain visible and will specify the line of code where the model is crashing. The user can use the source code to identify the problem or send the error message to technical support for assistance in locating the problem.

b. If the simulation terminates properly, a message box will notify the user of a successful run (Figure 13.29).

Figure 13.29

Click **OK**. Another prompt box will be displayed (Figure 13.30).

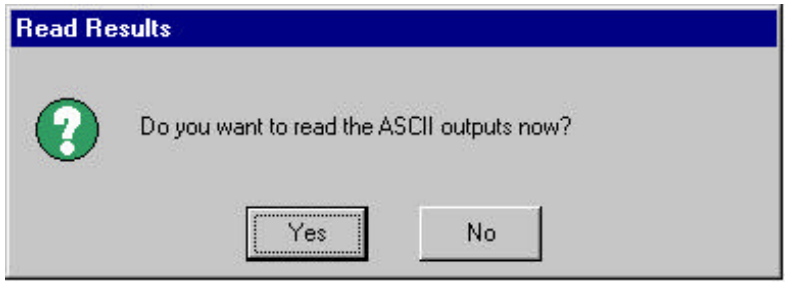


Figure 13.30

The SWAT simulation generates ASCII output files whose data must be transferred to dBASE tables for access in the interface. Click **Yes** to load the output data to dBASE tables. Click **No** to defer the generation of dBASE output tables to a later time. Clicking **Yes** performs the same operations performed by the **Read Results** command in the **Simulation** menu.

SECTION 13.2: READ RESULTS

The second command in the **Simulation** menu allows the user to process the ASCII outputs of SWAT in order to make ArcView standard dBASE tables.

Only the ASCII output files in spreadsheet format are loading into dBASE tables (see the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 32):

a. HRU Output File (.sbs): loaded only when yearly print frequency is selected

b. Subbasin Output File (.bsb)

c. Main Channel Output File (.rch)

d. HRU impoundment output file (.wtr): (optional)

e. Reservoir Output File (.rsv): (optional)

The other SWAT output files can be viewed using the **Show list** command under the **Reports** menu.

1. Select **Read Results** from the **Simulation** menu (or press `Cmd+B`) (Figure 13.31).

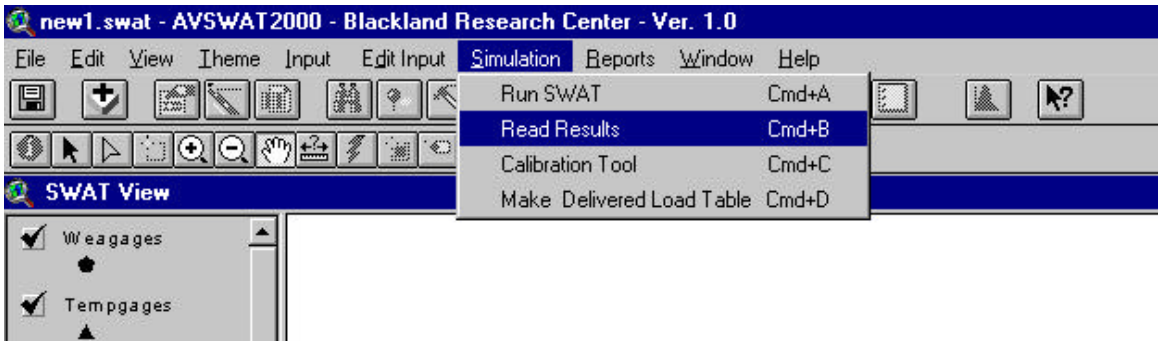


Figure 13.31

   a. If the Calibration Tool was not used and the model did not run a calibration scenario (see next section), the reading routine builds and displays the output tables (Figure 13.32).

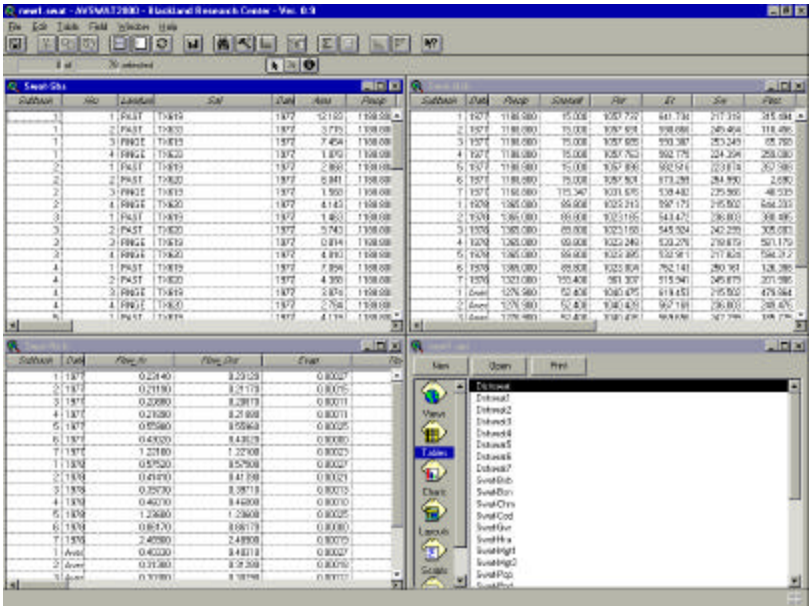


Figure 13.32

b. If the Calibration Tool was used and the model did run a calibration scenario (see next section), a dialog pops up (Figure 13.33).

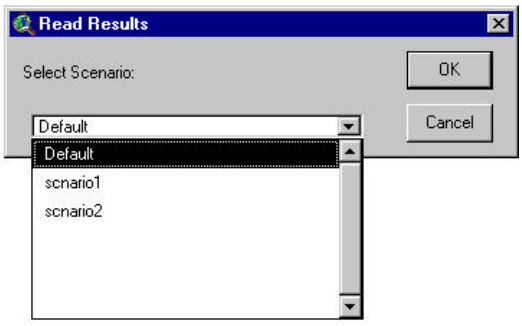


Figure 13.33

A drop-down control lists the current default output item and any calibration scenario.

Click **Cancel** to abort the reading. To continue, select one of the scenarios, and click **OK** to load the SWAT output for the selected simulation (Figure 13.34).

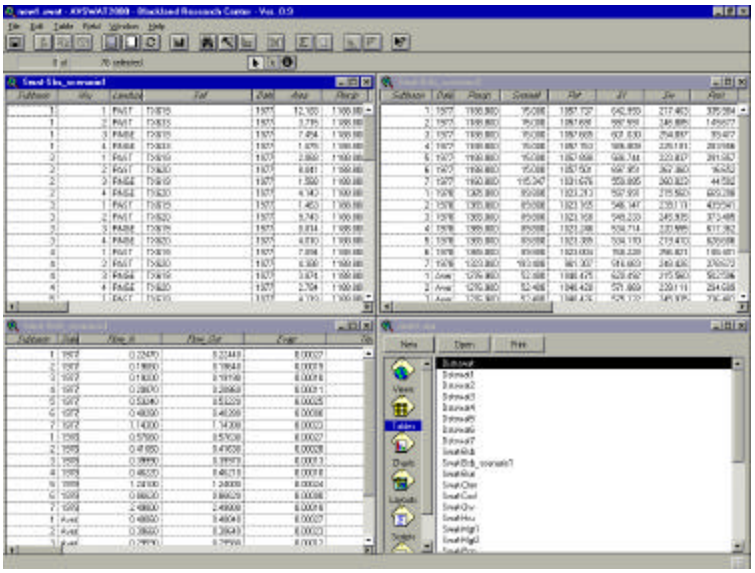


Figure 13.34

2. The output tables can be exported or analyzed in the interface with the **Map-Chart** tool under the **Reports** menu (see Section 13.5) or outside of the interface using standard ArcView mapping/graphing features.

SECTION 13.3: CALIBRATION TOOL

The third command in the **Simulation** menu allows the user to perform global changes on input parameters that are commonly modified during the calibration process.

1. Select **Calibration Tool** from the **Simulation** menu (or press `Cmd+C`) (Figure 13.35).

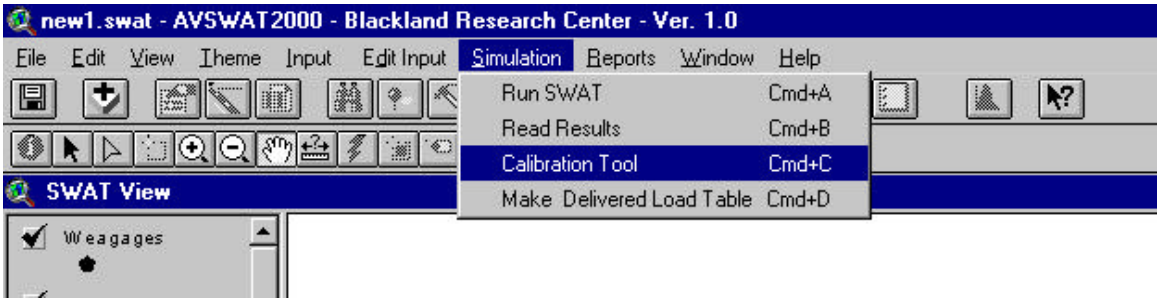


Figure 13.35

2. The Calibration Set Up dialog box is displayed (Figure 13.36).

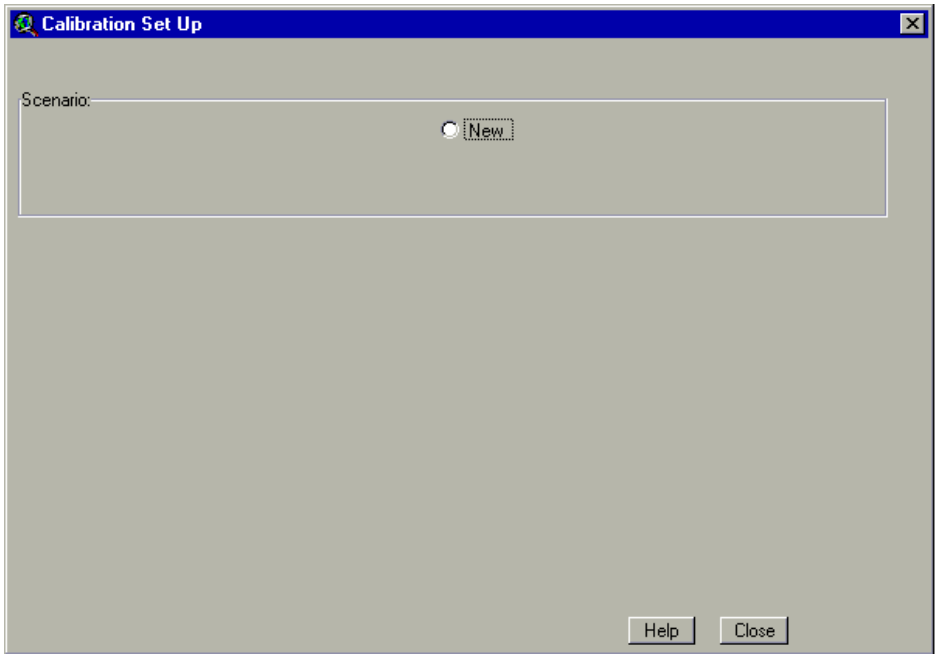


Figure 13.36

3. The user may create a new calibration scenario or load a previously saved scenario.

   Create a new scenario:

   a. Click the **New** button in the section of the dialog box labeled 'Scenario'.

   b. Type a name for the scenario in the text box labeled **New Name** .

   c. Press the `Enter` key on the keyboard.

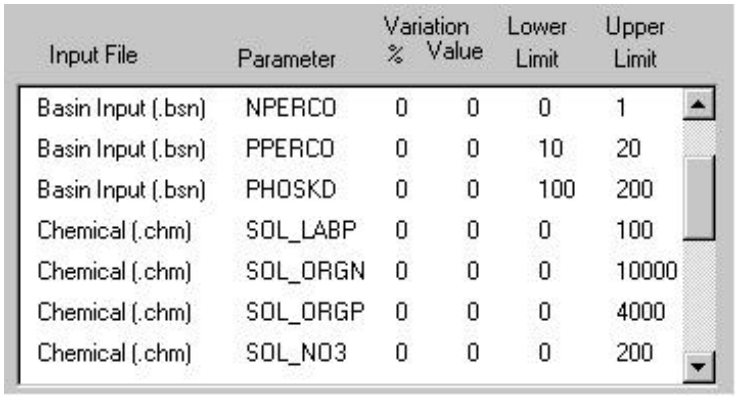
   Load an existing scenario:

   a. Click the **Load** button in the section of the dialog box labeled 'Scenario'.

   b. A list will be displayed showing all available scenarios .

   c. Select the desired scenario from the list.

4. Once a scenario is created or loaded, a table of SWAT input parameters will be enabled (Figure 13.37).



Input File	Parameter	Variation %	Value	Lower Limit	Upper Limit
Basin Input (.bsn)	NPERCO	0	0	0	1
Basin Input (.bsn)	PPERCO	0	0	10	20
Basin Input (.bsn)	PHOSKD	0	0	100	200
Chemical (.chm)	SOL_LABP	0	0	0	100
Chemical (.chm)	SOL_ORGN	0	0	0	10000
Chemical (.chm)	SOL_ORGP	0	0	0	4000
Chemical (.chm)	SOL_NO3	0	0	0	200

Figure 13.37

5. This table lists 27 SWAT input parameters. The name of the parameter and the related SWAT input file are listed in the table. The user can change the value of any of these parameters (alone or in combination), run SWAT for the scenario, and compare the scenario results to those in the original 'default' simulation or to other scenarios.
6. Select one parameter in the table by clicking on the parameter name. Another section of the dialog box is enabled (Figure 13.38).

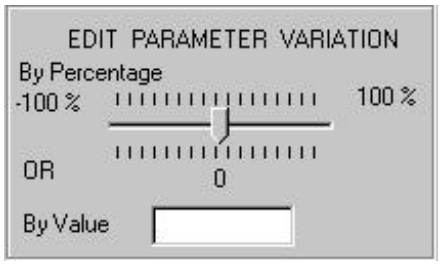
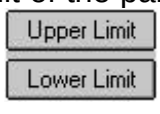


Figure 13.38

7. The Calibration Tool provides the user with two options for altering the value of a parameter from the value defined in the original or default scenario.
   a. **By Percentage**. Using the slider, select the percent change in value. (**Note:** the lower and upper limits of the percent variation are parameter dependent.)
   b. **By Value**. Type the change in value in the text box.

> **Note:** CN2 and SOL_AWC parameter variation are allowed to vary only by value.

8. The user can change the current upper limit and lower limit of the parameters using the respective **Upper limit** and **Lower limit** buttons 

.

When one of these buttons is clicked, a prompt box is displayed (Figure 13.39).

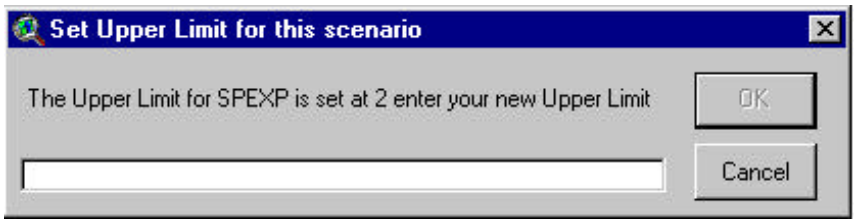
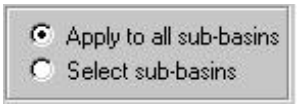


Figure 13.39

Type the new value in the text box. Click **OK** to close the prompt box and save the change, or click **Cancel** to close the prompt box without saving the change.

9. The change in parameter value may be applied to the entire watershed or to only a portion of the watershed.

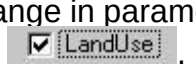
   a. To apply the change to all subbasins, select the radio button labeled **Apply to all sub-basins**. To apply the change to only a portion of the subbasins, select the radio button **Select sub-basins**

   .

   If the user decides to apply the change to only a few subbasins, a list of all subbasins in the watershed is displayed when the **Select sub-basins** radio button is chosen. Scroll the list and highlight the number of all subbasins where the change in parameter value is to be made. Hold down the `Shift` key to select multiple subbasins.

   Note: USLE_C is a SWAT database parameter and cannot be changed for particular subbasins.

   b. If the selected parameter is an HRU-level parameter, the user can limit the change in parameter value to specific landuses in the selected subbasins. To isolate the change in parameter value to certain landuses, click the box labeled **Landuse** 

   .

   If the box is checked, a list of landuses is displayed. Scroll the list and highlight the landuses where the change in parameter value is to be made. Hold down the `Shift` key to select multiple landuses.

   Note: For USLE_C, you must select the target landuses.

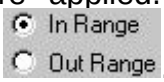
10. Once the settings for the parameter are complete, the user has three options: save the settings for the parameter, exit the parameter editor tool without saving the settings, or erase the current settings and define other settings.

a. **To save settings**. Click the **Apply** button below the table of parameters. This closes the editing tools for the specific parameter and saves the settings.

b. **To exit without saving**. Click the **Cancel Apply** button below the table of parameters. This closes the editing tools for the specific parameter without saving the settings.

c. **To erase current settings**. Click the **Reset** button below the table of parameters. This resets all values back to the original settings.

11. Repeat steps 6 through 10 for as many input parameters as desired.

12. Once all parameter modifications have been made for the scenario, the user can choose whether or not parameter values in the watershed are allowed to exceed range limits after parameter modifications are applied. This is controlled by the **In Range** and **Out Range** radio buttons  .

    When the **In Range** button is selected, the interface will apply all parameter modifications defined in the scenario and then verify that the new values are within range limits for the parameters. If the new values exceed range limits, they will be set to the closest allowed value.

    When the **Out Range** button is selected, the interface will apply all parameter modifications defined in the scenario without checking that the new values are within range limits for the parameters.

13. Click **OK** to save the scenario.

    A subdirectory will be created in the *project name*\scenarios\default\scen directory that contains the dbf tables and SWAT input files for the scenario. The subdirectory is given the name of the scenario defined by the user.

14. A prompt box will be displayed during the save (Figure 13.40).

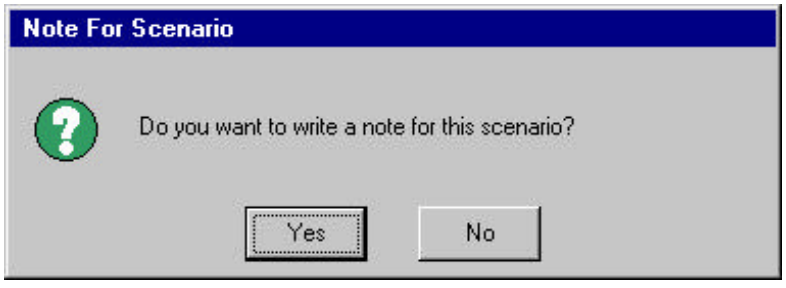


Figure 13.40

15. Click **No** to skip this option. Click **Yes** to write a note. If the user clicks **Yes,** a text box is displayed (Figure 13.41).

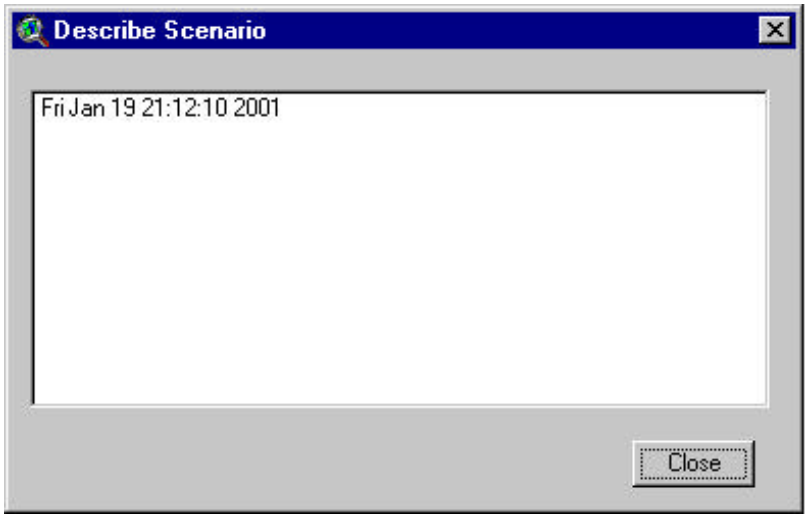


Figure 13.41

Enter a note and click **Close**.

16. Once all changes are made, a prompt box is displayed (Figure 13.42).

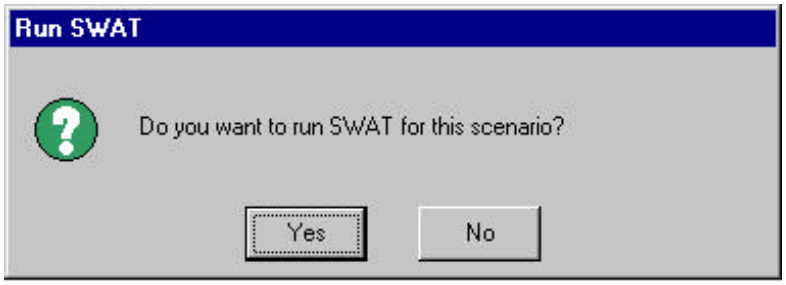


Figure 13.42

17. Click **Yes** to run a SWAT simulation for the scenario or click **No** to skip this option. If the user runs SWAT, one of two message boxes will be displayed when the SWAT simulation terminates.

a. If the simulation terminated before the entire simulation period was completed, a message box will notify the user of a run failure (Figure 13.43).

Figure 13.43

Click **OK**. Review inputs before attempting a new run.

b. If the simulation terminates properly, a message box will notify the user of a successful run (Figure 13.44).

Figure 13.44

Click **OK**. Another prompt box will be displayed (Figure 13.45).

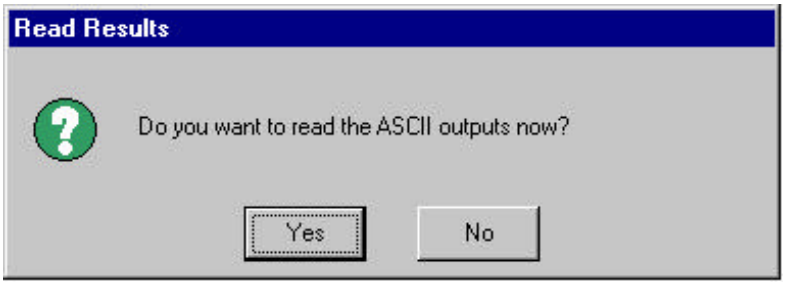


Figure 13.45

The SWAT simulation generates ASCII output files whose data that must be transferred to dBASE tables for access in the interface. Click **Yes** to load the output data to dBASE tables. Click **No** to defer the generation of dBASE output tables to a later time. (Clicking **Yes** performs the same operations performed by the **Read Results** command in the **Simulation** menu.

18. Click **Close** to exit the Calibration Tool.

SECTION 13.4: CREATE DELIVERED LOAD TABLE

The fourth command in the **Simulation** menu allows the user to generate a table itemizing the amount of load at any point on the channel network originating from each of the upstream subbasins. This table can only be generated when the yearly print frequency is selected. Average annual data are analyzed to produce this table.

1. Select **Make Delivered Load Table** from the **Simulation** menu (or press `Cmd+D`) (Figure 13.46).

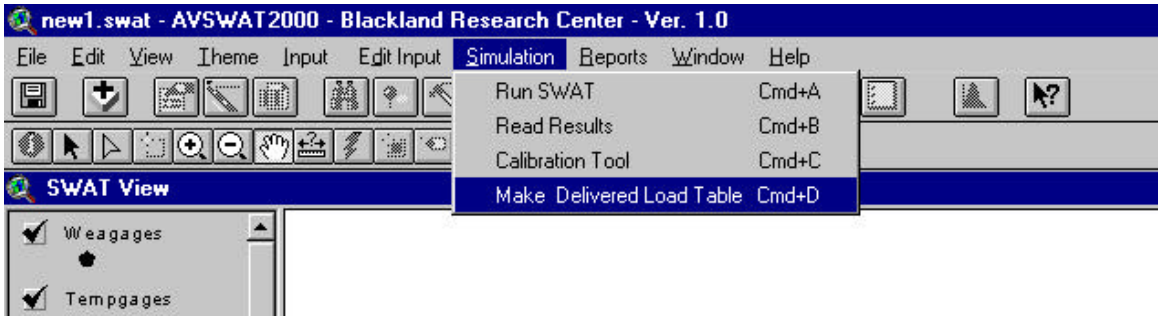


Figure 13.46

2. If a calibration scenario has been created (see previous section), a prompt box is displayed (Figure 13.47).

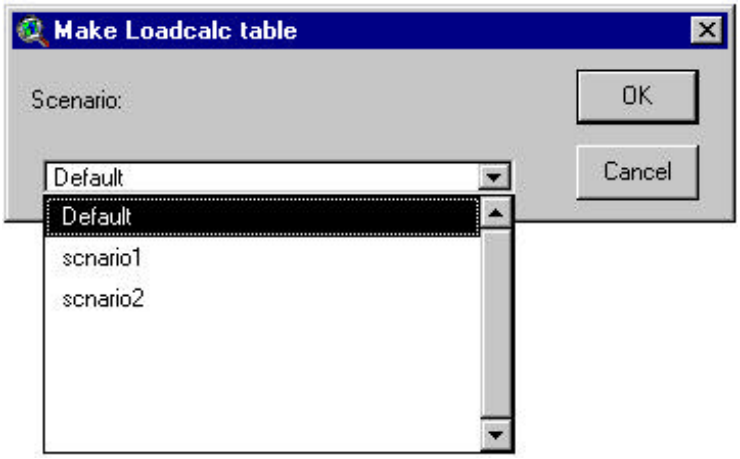


Figure 13.47

A drop-down control lists the original (default) run and any calibration scenarios for the project. Select one of the runs and click **OK,** or click **Cancel** to abort the process.

3. A prompt box is displayed for the user to select the location in the channel network at which the calculation is made (Figure 13.48).

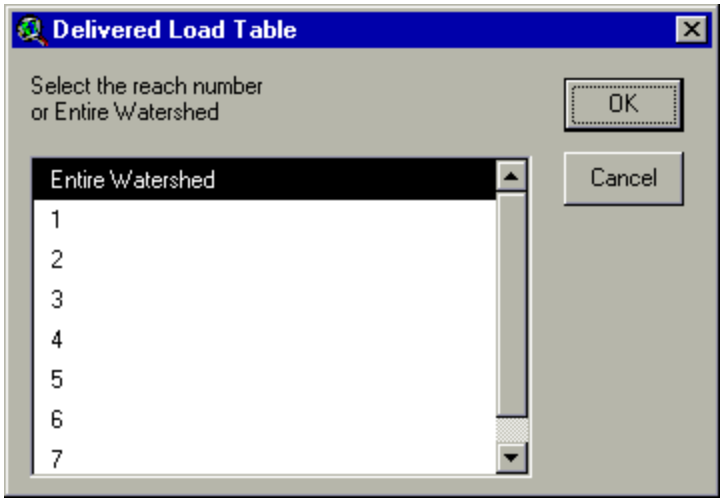


Figure 13.48

4. Select the location of interest and click **OK,** or click **Cancel** to exit the tool.
5. A message box is displayed when the table is made (Figure 13.49).

Figure 13.49

6. The table can be viewed by selecting **Delivered Load Table** in the **Reports** menu.

> **Note:** The Delivery Load Table reports the loads regarding the same variables contained in the Subbasin Output File (.bsb). In addition, the respective Delivery Ratios (variable names are prefixed by "R") are reported.

SECTION 13.5 REPORTS: MAP-CHART

The fourth command in the **Reports** menu allows the user to graph and map output from a SWAT simulation.

1. Select **Map-Chart** from the **Reports** menu on the SWAT View (or press `Ctr+F4`) (Figure 13.50).

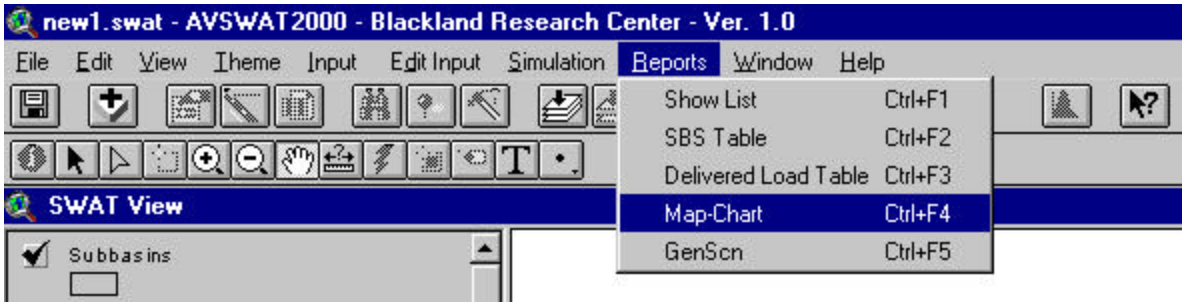


Figure 13.50

2. If a calibration scenario has been created (see Section 13.3 previous section), and the respective simulation output imported into ArcView (see Section 13.2), a prompt box is displayed (Figure 13.51).

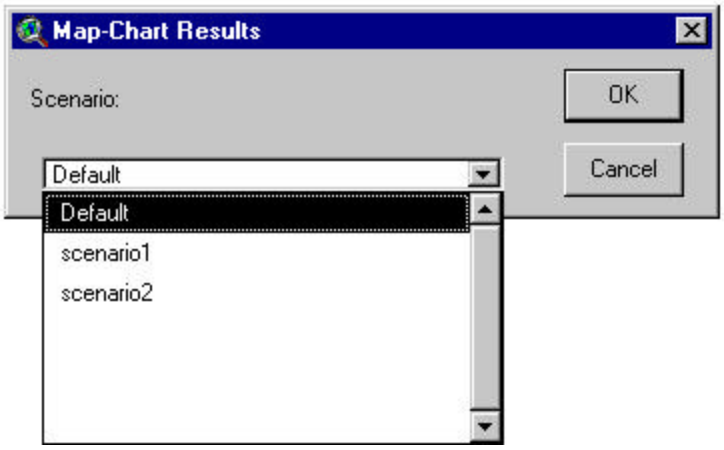


Figure 13.51

A drop-down control lists the original (default) run and any calibration scenarios for the project. Select one of the runs and click **OK,** or click **Cancel** to abort the process.

3. A dialog box will be displayed (Figure 13.52) that provides options for mapping/graphing data from two SWAT output files, the reach (.rch) output file and the subbasin (.bsb) output file, and the Delivered Load Table (this

table is only available when the yearly print frequency is selected and the calculations are made for the Entire Watershed; see Section 13.4). Data from the reach output file will be graphed while data from the subbasin output file and the Delivered Load Table will be mapped and graphed.

Figure 13.52

4. Depending on the output file selected, the list of variables available for plotting will vary (Figures 13.53, 13.54 and 13.55).

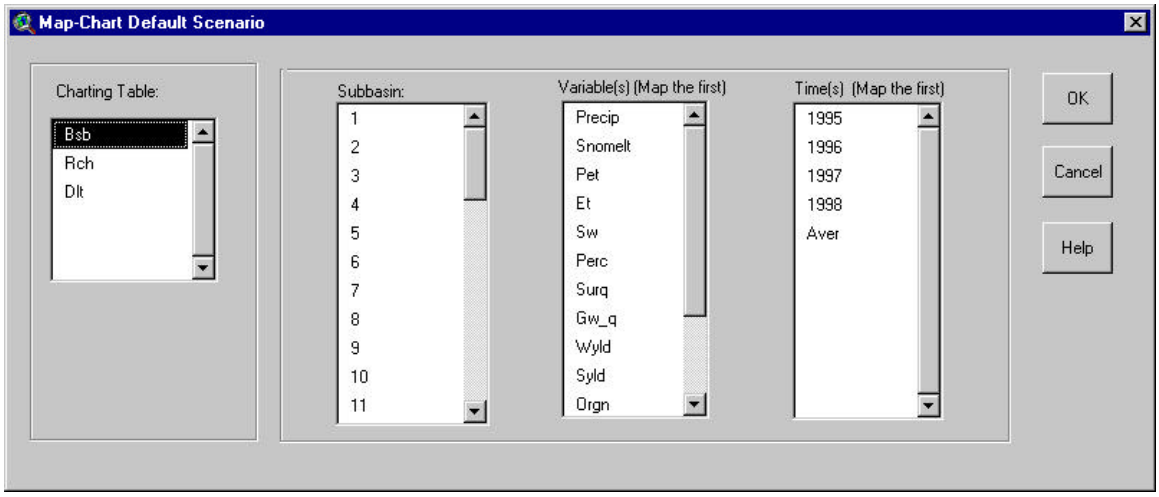


Figure 13.53

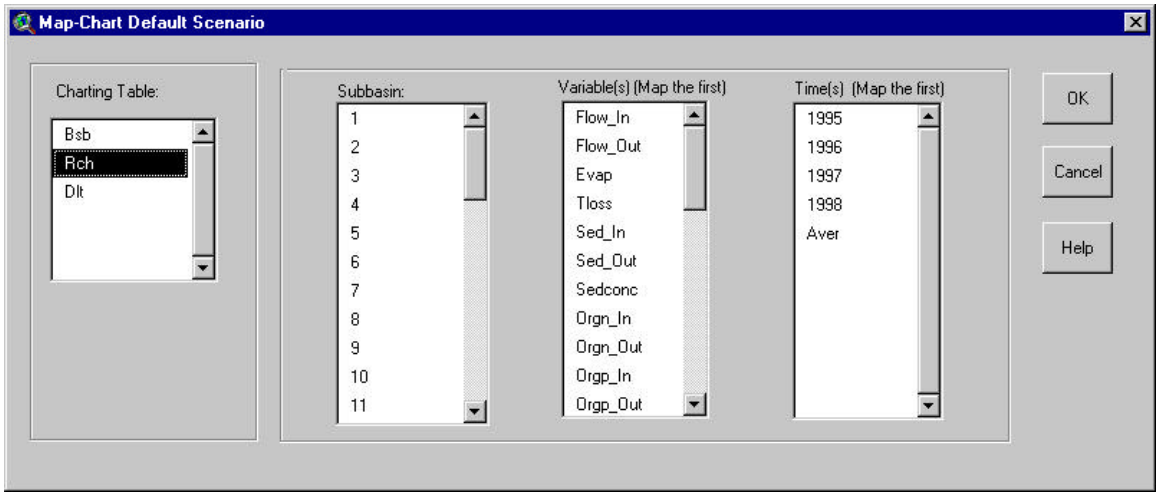


Figure 13.54

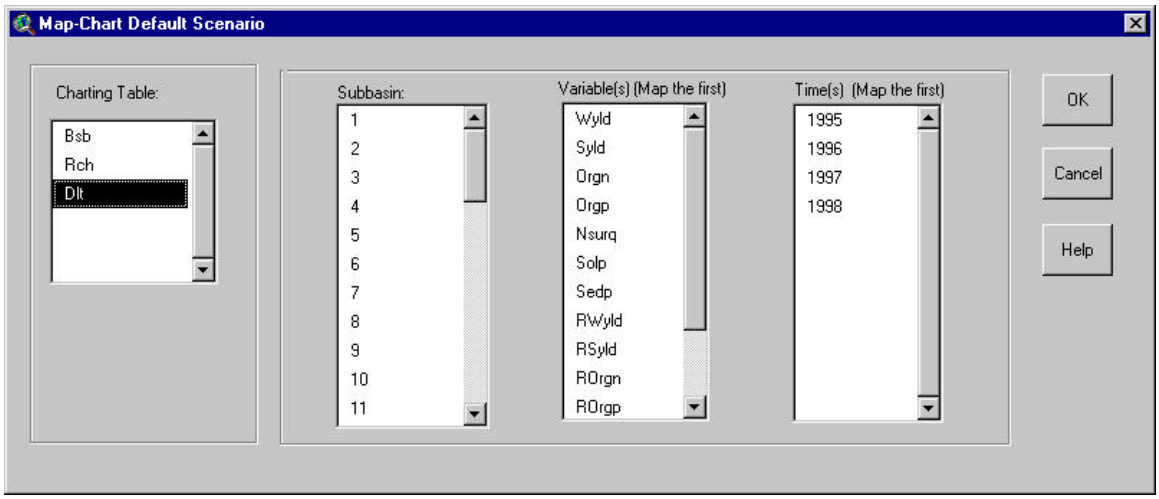


Figure 13.55

5. For each output file, there are three plotting parameters that must be defined: Subbasin, Variable(s), and Time(s). Only one subbasin may be selected. Multiple variables and/or times may be selected by holding down the `Shift` key while clicking the parameters of choice.
6. Once all plotting parameters are set, click **OK**.
7. A graph will be displayed. If subbasin output or the Delivery Output Table are being plotted, a map of the watershed will be produced showing values for the first variable type selected in the list of variables and for the first time period selected in the list of output times (Figures 13.56).

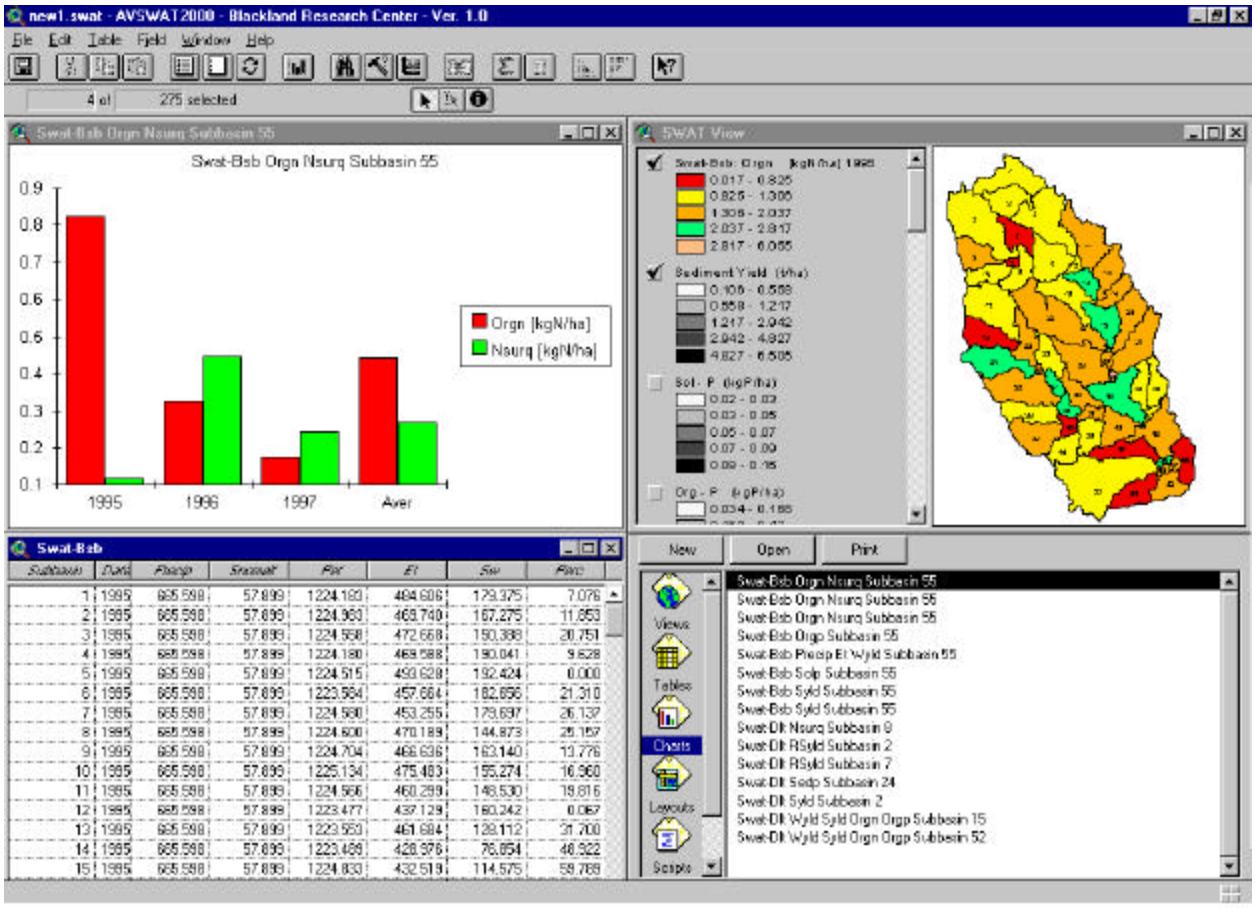


Figure 13.55

Note: The graphic capabilities of ArcView are adequate for basic data presentation. For more sophisticated graphs, the SWAT output should be imported into a separate graphics package.

SECTION 14: SWAT DATABASE EDITORS

Purpose

The SWAT model uses five databases to store required information about plant growth and urban land uses, tillage, fertilizer components, and pesticide properties (See *Soil and Water Assessment Tool User's Manual, Version 2000*). The interface provides dialog-based editors to access and edit these five databases, as well as two additional databases that store custom soils and custom weather station parameters.

Application

The SWAT Database Editors are available once the *AvSWAT 2000* extension is loaded.

1. On the main interface, click the button beside **Edit SWAT Databases,** or when the SWAT View is active, select **Databases** from the **Edit Input** menu.
2. A dialog box will appear with a list of the databases (Figure 14.1).

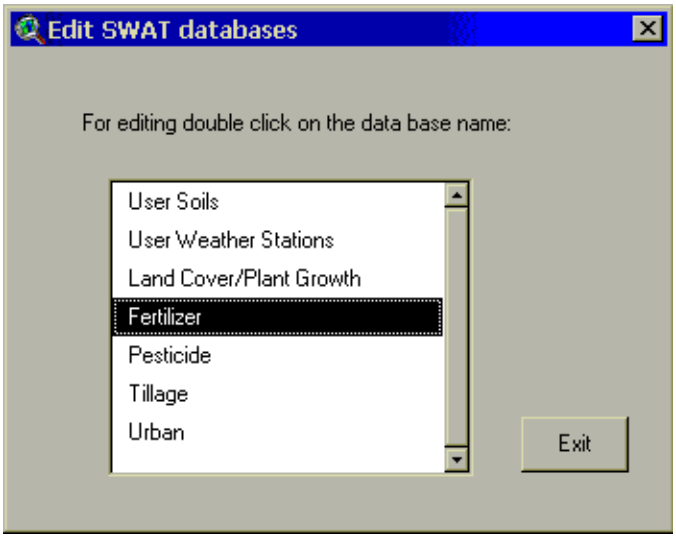


Figure 14.1

Seven databases are available for editing:

User Soils

This database is used to store custom soil data. Data are entered into this database for soil maps that do not use the US STATSGO soil database included with the interface.

User Weather Stations

This database is used to store custom weather generator

station data. Statistical weather data is entered into this database for weather simulation stations not included in the provided database.

Land Cover/Plant Growth

This database contains SWAT plant growth parameters. While users are given the option of modifying existing land cover/plant parameters or adding additional plant species to the database, we would like to emphasize that changes to the plant database should be based on experimental data. The typical user will not need to make changes to this database. Information about the plant growth parameters provided with the interface is provided in *Soil and Water Assessment Tool User's Manual, Version 2000*, Appendix A.

Fertilizer

This database contains SWAT fertilizer/manure parameters. Both inorganic and organic (manure) fertilizer data are stored in this database. Information about the fertilizer parameters provided with the interface is provided in *Soil and Water Assessment Tool User's Manual, Version 2000*, Appendix A.

Pesticide

This database contains SWAT pesticide parameters. Information about the pesticide parameters provided with the interface is provided in *Soil and Water Assessment Tool User's Manual, Version 2000*, Appendix A.

Tillage

This database contains SWAT tillage parameters. Information about the tillage parameters provided with the interface is provided in *Soil and Water Assessment Tool User's Manual, Version 2000*, Appendix A.

Urban

This database contains SWAT urban land type parameters. Information about the urban land type parameters provided with the interface is provided in *Soil and Water Assessment Tool User's Manual, Version 2000*, Appendix A.

SECTION 14.1: USER SOILS DATABASE

To edit the User Soils Database:

1. Double-click **User Soils** on the list of databases (Figure 14.2).

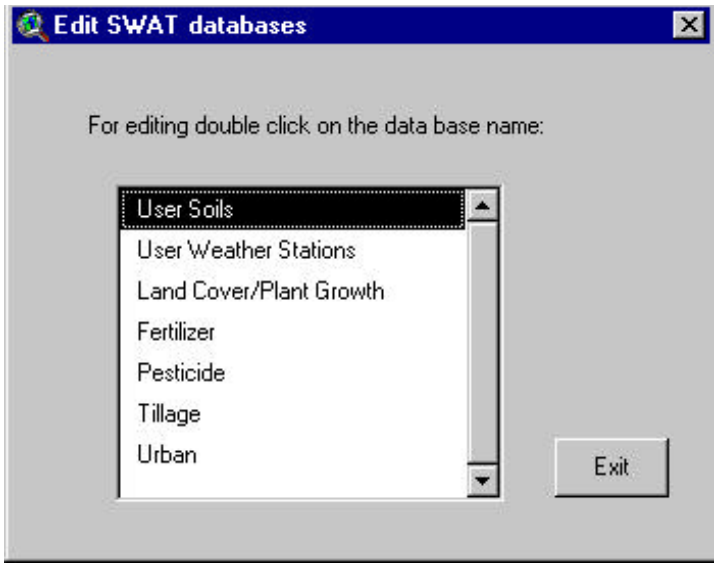


Figure 14.2

2. The User Soils dialog box will be displayed (Figure 14.3)

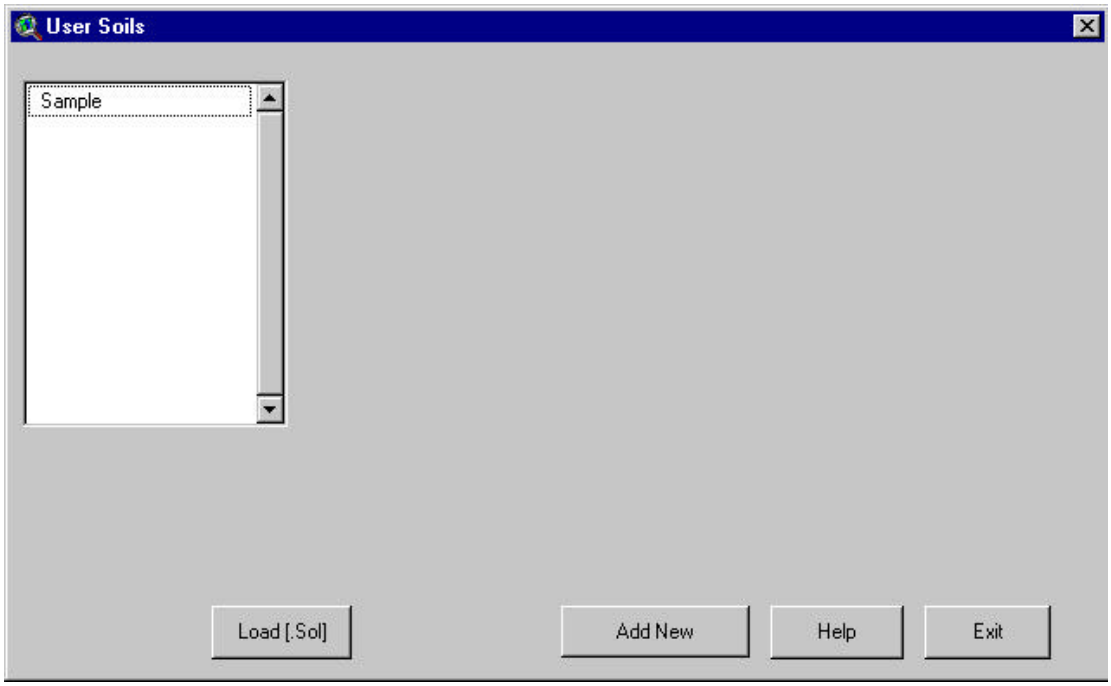


Figure 14.3

A list of soils in the custom database is displayed on the left side of the dialog box.

3. The user has four options: edit an existing soil dataset, add a new soil dataset, delete an existing soil dataset, or exit the database. A list of soils in the custom database is also shown.

   To edit an existing soil dataset:

   a. Click the name of the soil to be edited.

   b. The data for the soil will be displayed (Figure 14.4). A complete description of the variables is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 22.

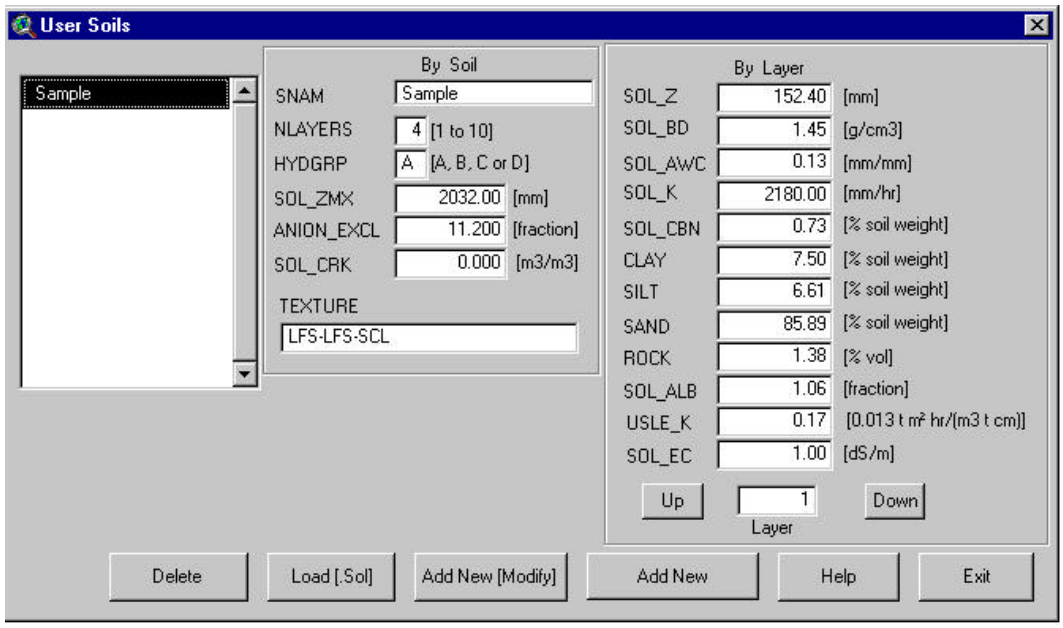
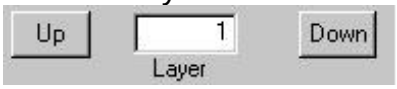


Figure 14.4

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears, and the range of variation is shown.

   c. The soil parameters are separated into two groups. The parameters that are applicable to the entire soil profile are displayed in the box titled **By Soil,** and the parameters that pertain to a particular soil layer are displayed in the box titled **By Layer**. The layer number is displayed at the bottom of the **By Layer** box between the **Up** and **Down** buttons. The different layers in the soil profile can be accessed via these buttons



   .

   d. Edit data: Activate a cursor by clicking in the text box. Type the desired value. You can move to the next parameter by pressing the TAB key. If the

value of the parameter is out of range, a message box like the one in Figure 14.5 will be displayed.

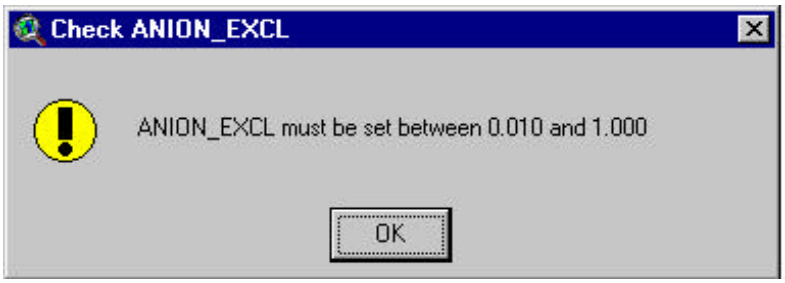


Figure 14.5

Click **OK.** The parameter will be reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *\installation dir*avswatdb directory. The names of the files are the three letter extension of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

e. Once all editing changes have been made, click **Exit**

.

f. A prompt box may appear, notifying the user of out-of-range parameter values (Figure 14.6).

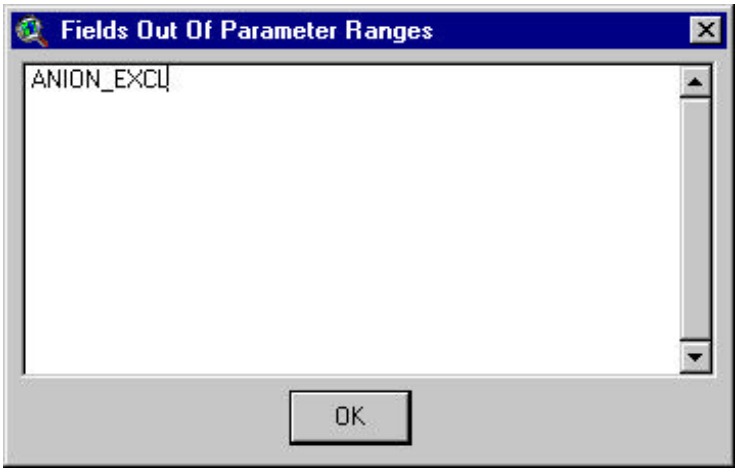


Figure 14.6

Click **OK**. A prompt box will then be displayed (Figure 14.7).

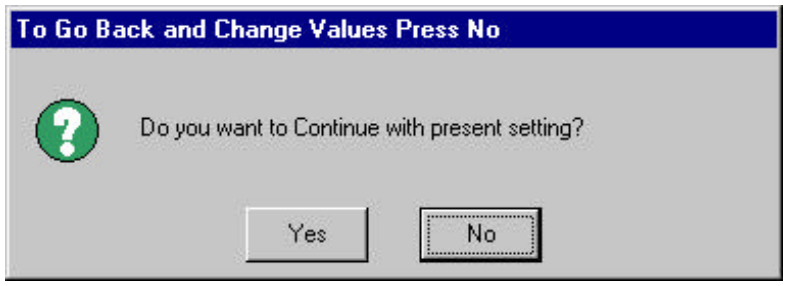


Figure 14.7

Click **No** to return to editing and correct the values, or click **Yes** to continue with the exit.

g. A prompt box appears (Figure 14.8).

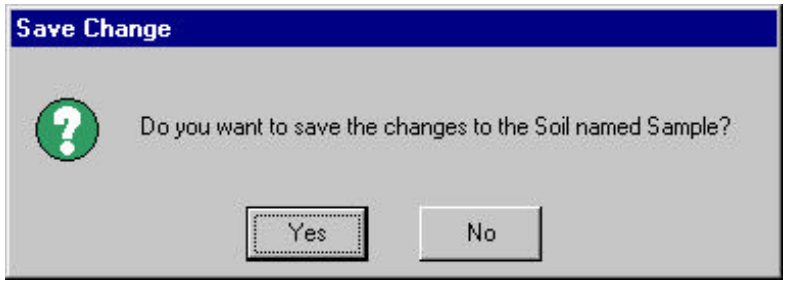


Figure 14.8

h. To save the editing changes, click **Yes**. To discard the editing changes, click **No**.

To add a new soil dataset:

A new soil may be added to the database by setting parameters on a blank form, by loading data from a SWAT Soil Input File or by copying data from an existing soil record to a new record.

To add a new soil using a blank form:

a. Click the button labeled **Add New** on the User Soils dialog box.

b. A modified dialog box will appear which contains input fields for the soil profile data (Figure 14.9). If the minimum value in a field is greater than 0.0, the interface will preset the value in the blank form to that minimum.

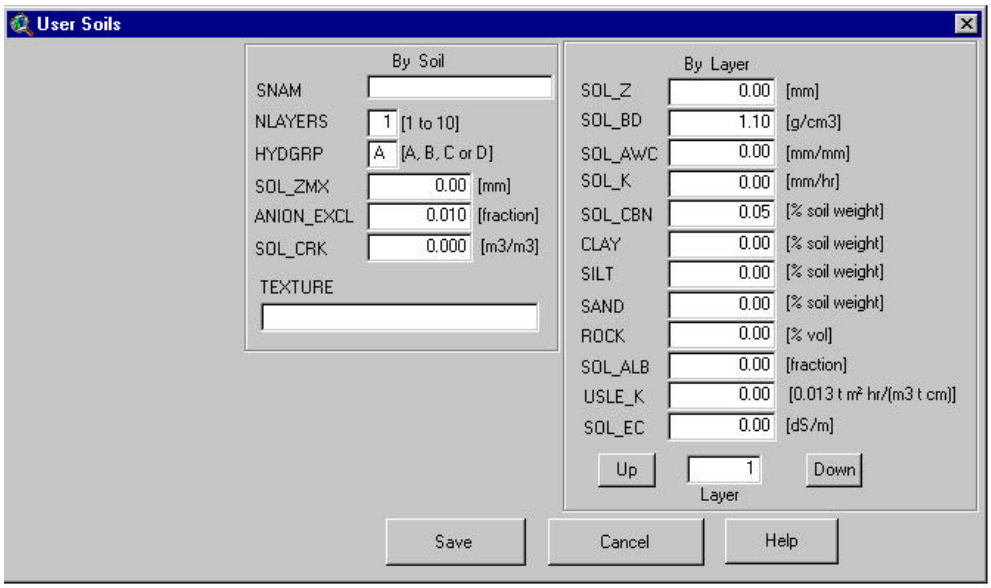


Figure 14.9

c. At this point, the user may type the necessary data into the different fields. A unique name must be given to the soil (the name can include numbers, but the name must begin with a letter).

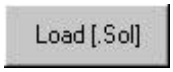
d. Once the soil data is entered, click the button labeled **Save**

to store the data, or click the **Cancel**

button to exit the editor without saving.

e. The user is returned to the original User Soils dialog box. If the soil data was saved, the list of soils in the database will include the name of the soil just added.

To load soil data from a file:

a. Click the button labeled **Load (.Sol)** 

on the User Soils dialog box.

b. A prompt box will appear warning the user of potential problems if the .sol file to be loaded is not in the correct format (Figure 14.10).

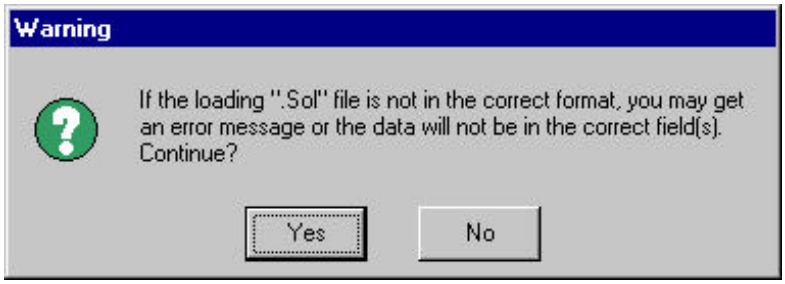


Figure 14.10

To proceed with the load, click **Yes**. To abandon the load operation, click **No.**

c. A browser will be displayed if the user clicks **Yes** (Figure 14.11).

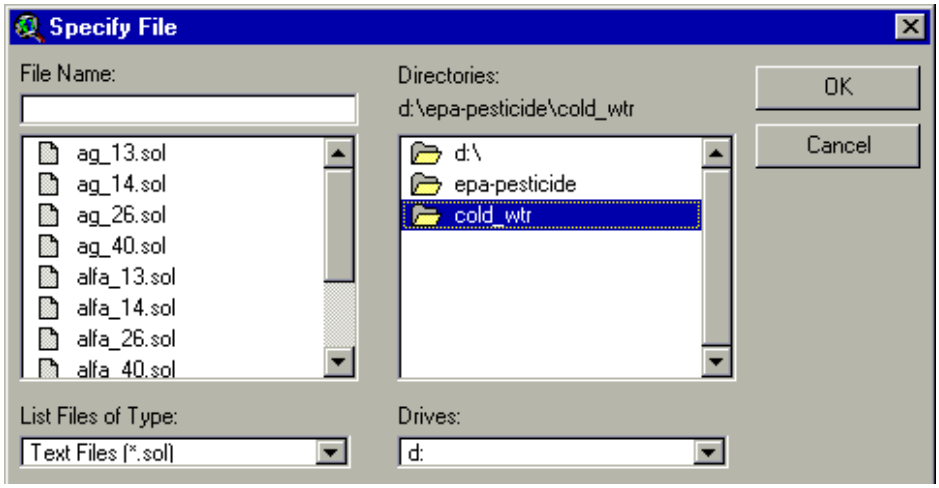


Figure 14.11

d. Select the soil input file (.sol) to be loaded and click **OK** (or double click the selection). The soil data will fill the fields in the User Soils dialog.

e. Look over the data to verify it was imported correctly.

f. Click the **Save** button to store the new data set, or click **Cancel** to exit the load procedure without saving.

To add a new soil by modifying an existing record:

a. From the User Soils dialog box, click the name of the soil to be copied.

b. Once the soil data is displayed, click the button labeled **Add New (Modify)**

.

c. A modified dialog box will appear with all the data fields filled (copied from the previously selected soil data) except the soil name (Figure 14.12).

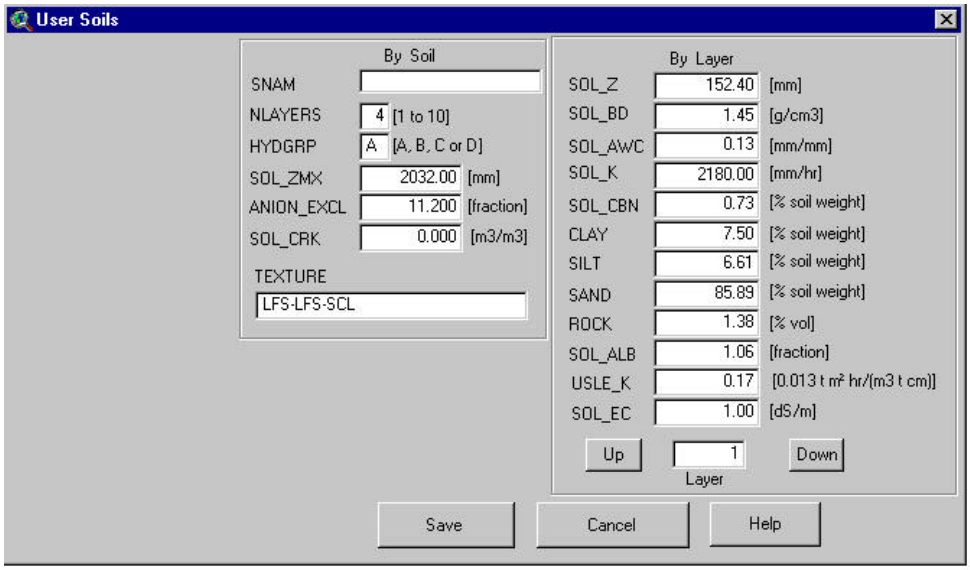


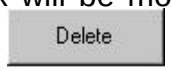
Figure 14.12

d. Enter a new soil name and modify the other data fields.

e. Click **Save** to store the modified soil data or **Cancel** to exit the editor without saving.

To delete a soil dataset:

a. From the User Soils dialog box, click the name of the soil to be deleted.

b. The dialog box will be modified to display the soil data. Click the button labeled **Delete**  at the bottom of the dialog box.

c. A prompt box will appear requesting the user to verify the delete (Figure 14.13).

Figure 14.13

To continue with the delete process, click **Yes**. To abandon the delete process, click **No**.

To exit the User Soils database:

a. From the User Soils dialog box, click **Exit** .

SECTION 14.2: USER WEATHER STATIONS DATABASE

To edit the User Weather Station Database:

1. Double-click **User Weather Stations** on the list of databases (Figure 14.14).

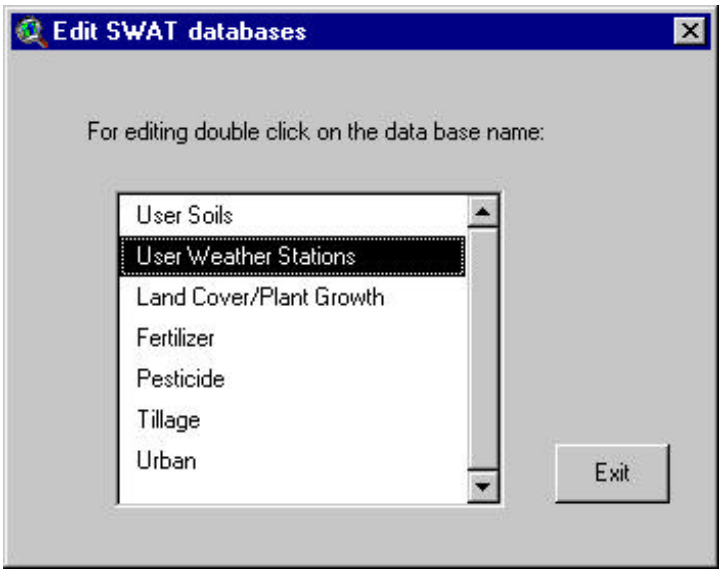


Figure 14.14

2. The User Weather Stations dialog box will be displayed (Figure 14.15).

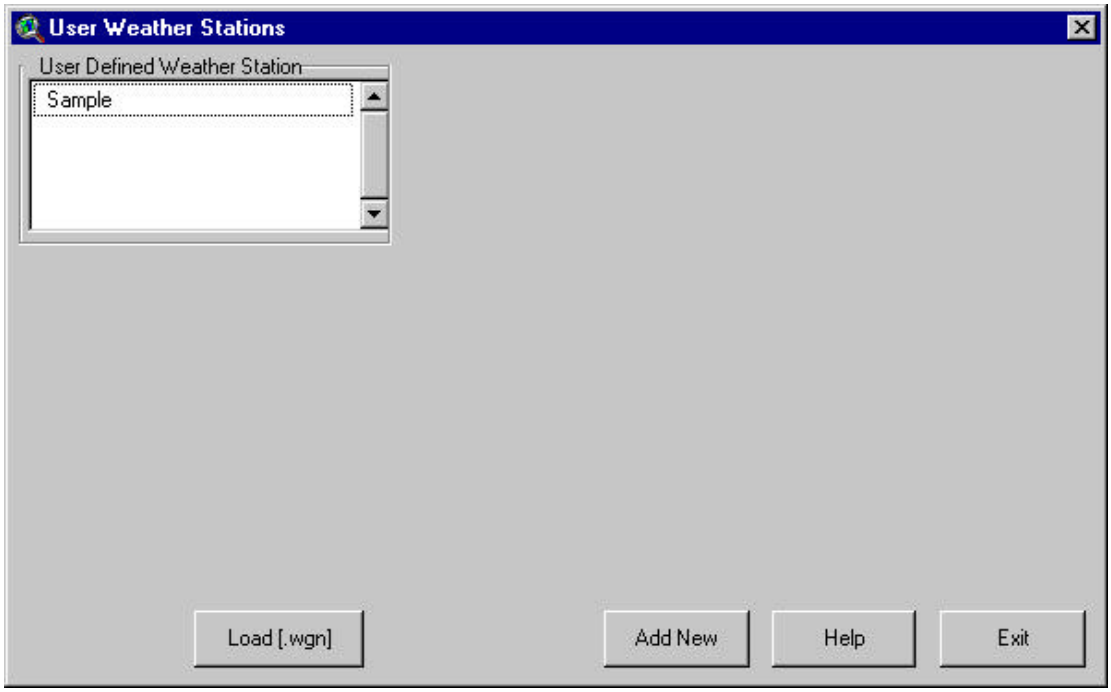


Figure 14.15

A list of stations in the custom database is displayed on the left side of the dialog box.

3. The user has four options: edit an existing weather station data set, add a new weather station data set, delete an existing weather station data set, or exit the database.

   To edit an existing weather station dataset:

   a. Click the name of the weather station to be edited.

   b. The data for the weather station will be displayed (Figure 14.16). A complete description of the parameters is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 13).

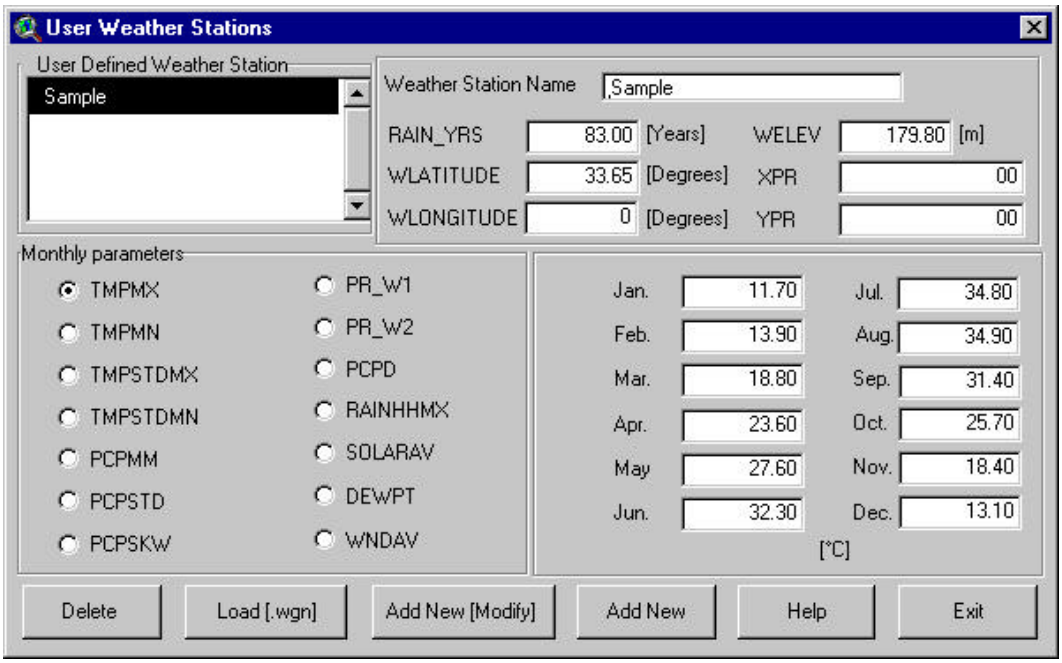


Figure 14.16

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation is shown.

   c. The weather data are separated into two groups. The statistical data that is summarized over the entire period of record is listed in one box, while variables summarized by month are listed in the box labeled **Monthly parameters**.

   d. The data fields for the monthly parameters can be accessed by clicking the circle to the left of the variable names (Figure 14.17).

Figure 14.17

e. Edit data: Activate a cursor by clicking in the text box. Type the desired value. You can move to the next parameter by pressing the Tab key. If the value of the parameter is out of range a message box like the one in Figure 14.18 will be displayed

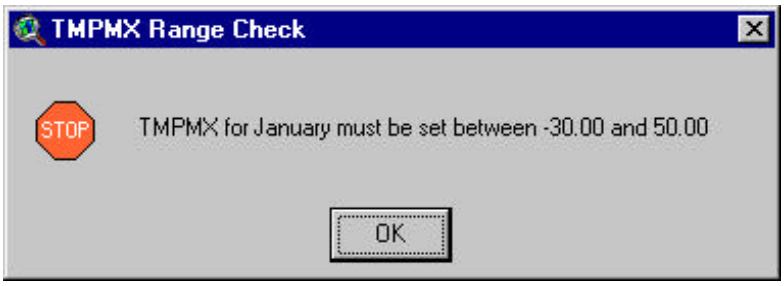


Figure 14.18

Click **OK.** The parameter will be reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *linstallation dir*/avswatdb directory. The names of the files are the three letter extension of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

f. Once all editing changes have been made, click **Exit**

.

g. A prompt box may appear notifying the user of out-of-range parameter values (Figure 14.19).

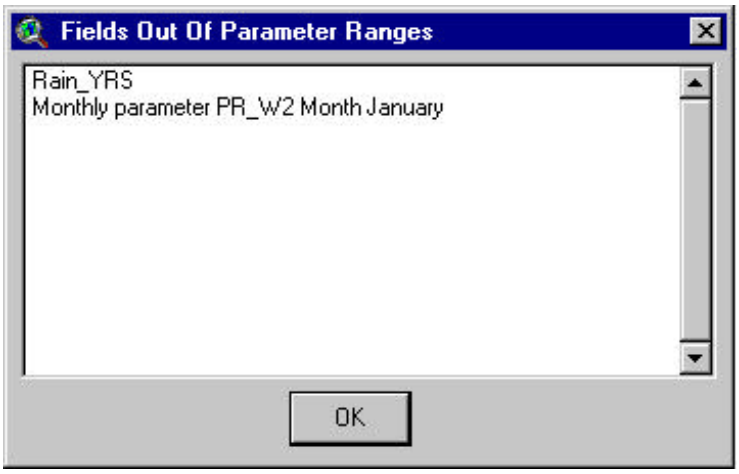


Figure 14.19

Click **OK**. A prompt box will then be displayed (Figure 14.20).

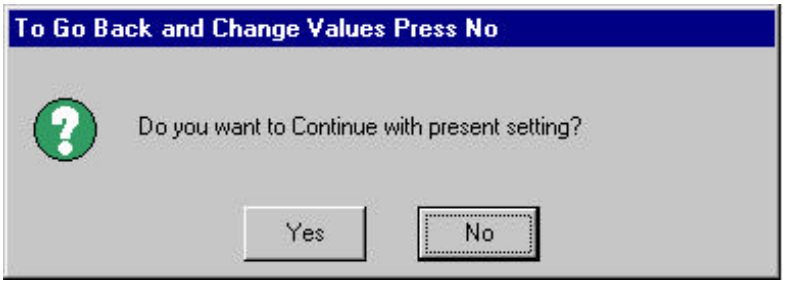


Figure 14.20

Click **No** to return to editing and correct the values or click **Yes** to continue with the exit.

h. A prompt box appears (Figure 14.21).

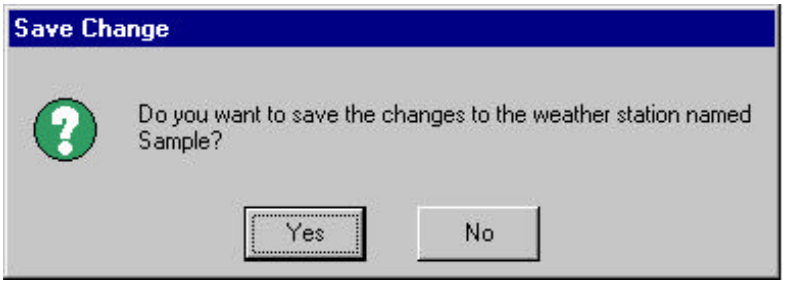


Figure 14.21

i. To save the editing changes, click **Yes**. To discard the editing changes, click **No.**

To add a new weather data set:

A new weather station may be added to the database by setting parameters on a blank form, by loading data from a SWAT Weather Generator Input File, or by copying data from an existing weather station record to a new record.

To add a new weather station using a blank form:

a. Click the button labeled **Add New**

on the User Weather Station dialog box.

b. A modified dialog box will appear which contains input fields for the weather station data (Figure 14.22). The interface will preset the values to the minimum value allowed for the parameters.

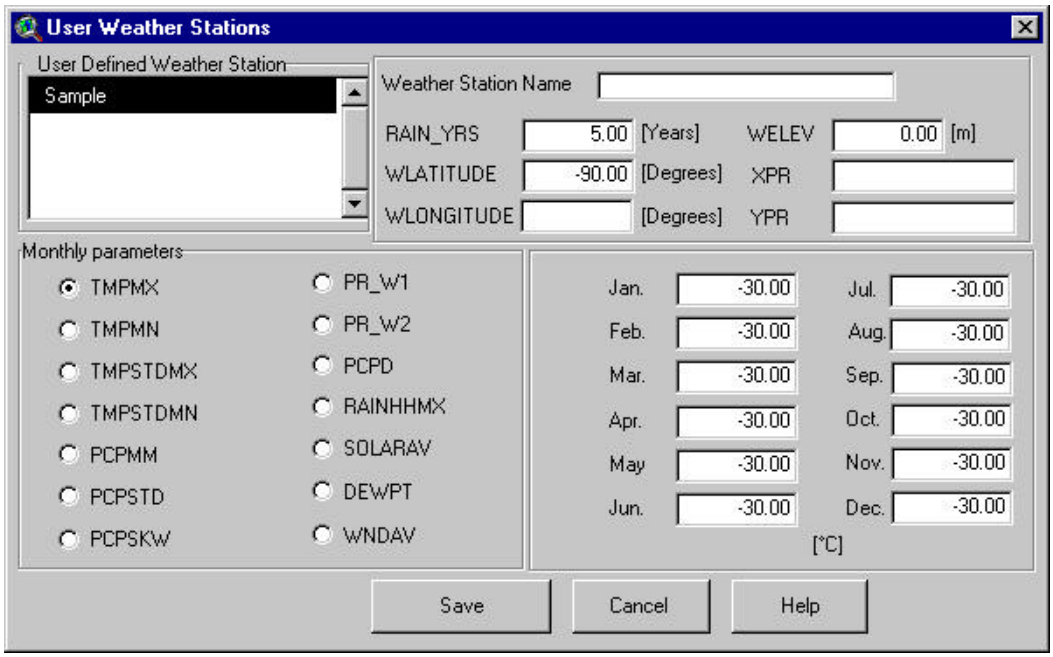


Figure 14.22

c. At this point, the user may type the necessary data into the different fields. The fields for monthly parameters are not displayed until the radio button to the left of a parameter name is clicked. A unique name must be given to the station (the name can include numbers but must begin with a letter).

d. Once the weather station data is entered, click the button labeled **Save**

to store the data, or click the **Cancel**

button to exit the editor without saving.

e. The user is returned to the original User Weather Station dialog box. If the station data was saved, the list of weather stations in the database will include the name of the weather station just added.

To load weather station data from a file:

a. Click the button labeled **Load (.wgn)**

on the User Weather Station dialog box.

b. A prompt box will appear, warning the user of potential problems if the .wgn file to be loaded is not in the correct format (Figure 14.23).

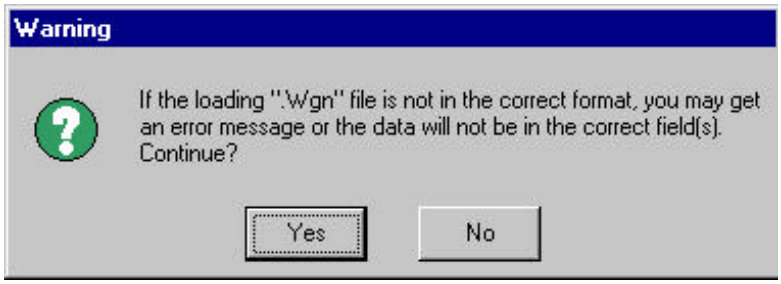


Figure 14.23

To proceed with the load, click **Yes**. To abandon the load operation, click **No.**

c. A browser will be displayed if the user selects **Yes** (Figure 14.24).

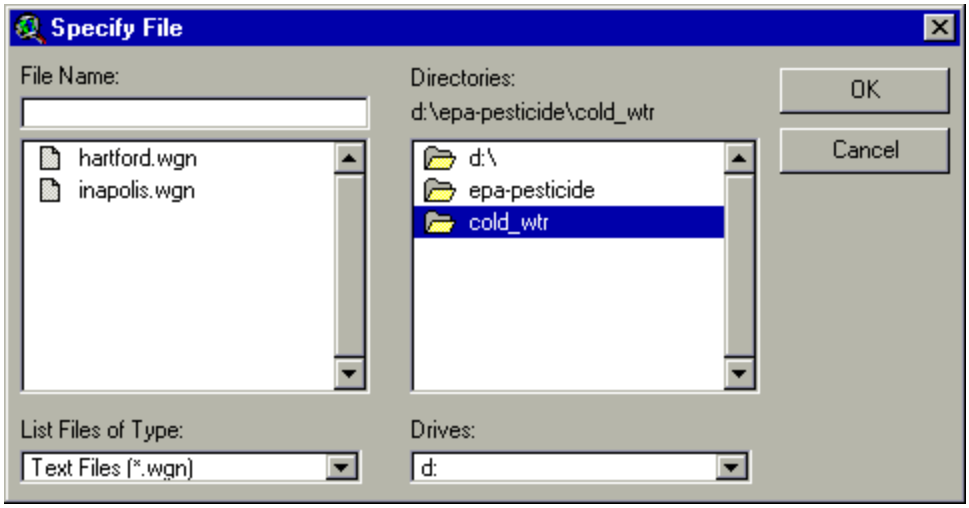


Figure 14.24

d. Select the weather generator input file (.wgn) to be loaded and click **OK** (or double click the selection). The weather data will fill the fields in the User Weather Station dialog box

e. Look over the data to verify it was imported correctly.

f. Click the **Save** button to store the data set or click **Cancel** to exit the load procedure without saving.

To add a new weather station by modifying an existing record:

a. From the User Weather Station dialog box, click the name of the weather station to be copied.

b. Once the weather data is displayed, click the button labeled **Add New (Modify)**

.

c. A modified dialog box will appear with all the data fields filled (copied from the previously selected weather station data) except the weather station name (Figure 14.25).

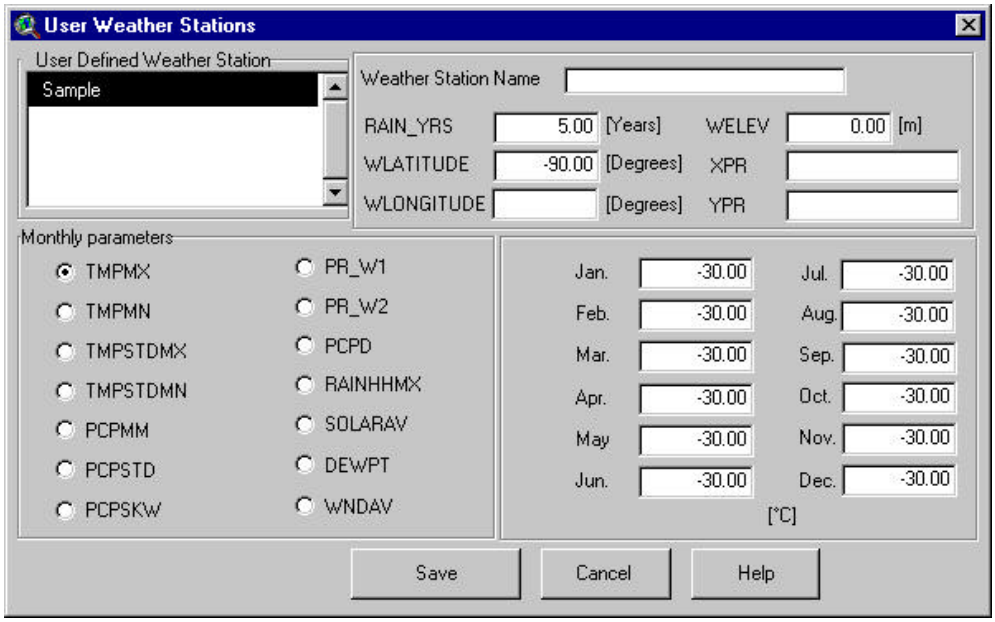


Figure 14.25

d. Enter a new weather station name, and modify the other data fields.

e. Click **Save** to store the modified weather station data, or click **Cancel** to exit the editor without saving.

To delete a weather station dataset:

a. From the User Weather Station dialog box, click the name of the weather station to be deleted.

b. The dialog box will be modified to display the weather data. Click the button labeled **Delete**

at the bottom of the dialog box.

c. A prompt box will appear requesting the user to verify the delete (Figure 14.26).

Figure 14.26

To continue with the delete process, click **Yes**. To abandon the delete process, click **No**.

To exit the User Weather Station database:

a. From the User Weather Station dialog box, click **Exit**

.

SECTION 14.3: LAND COVER/PLANT GROWTH DATABASE

To edit the Land Cover/Plant Growth Database:

1. Double-click **Land Cover/Plant Growth** on the list of databases (Figure 14.27).

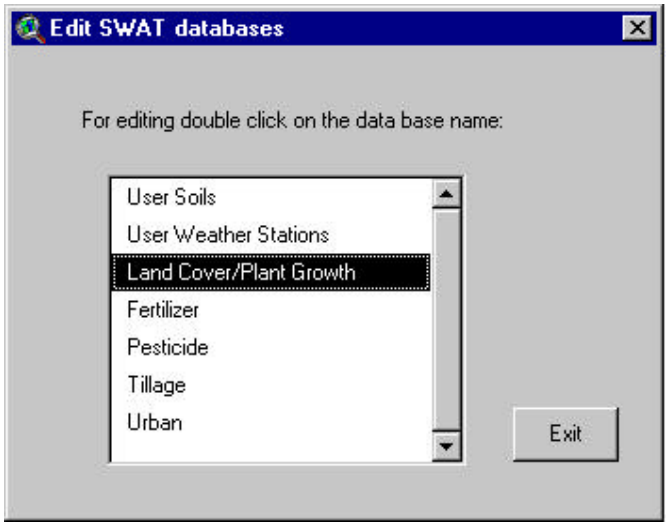


Figure 14.27

2. The Land Cover/Plant Growth database dialog box will be displayed (Figure 14.28).

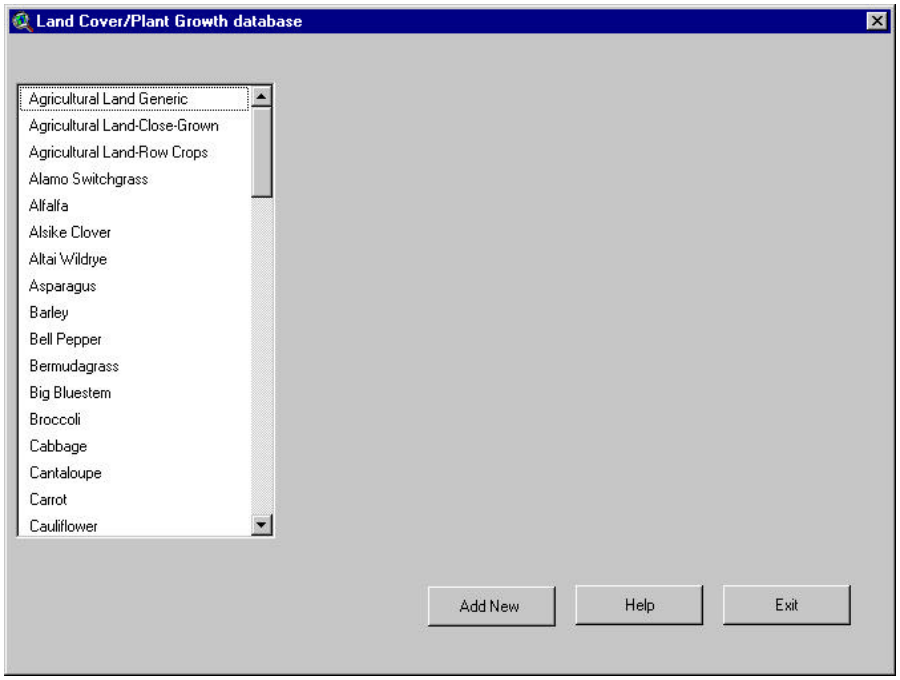


Figure 14.28

A list of land cover/plants in the database is displayed on the left side of the dialog box.

3. The user has four options: edit an existing land cover data set, add a new land cover data set, delete an existing land cover data set, or exit the database.

   To edit an existing land cover dataset:

   a. Click the name of the land cover/plant to be edited.

   b. The data for the land cover will be displayed (Figures 14.29). A complete description of the parameters is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 14.

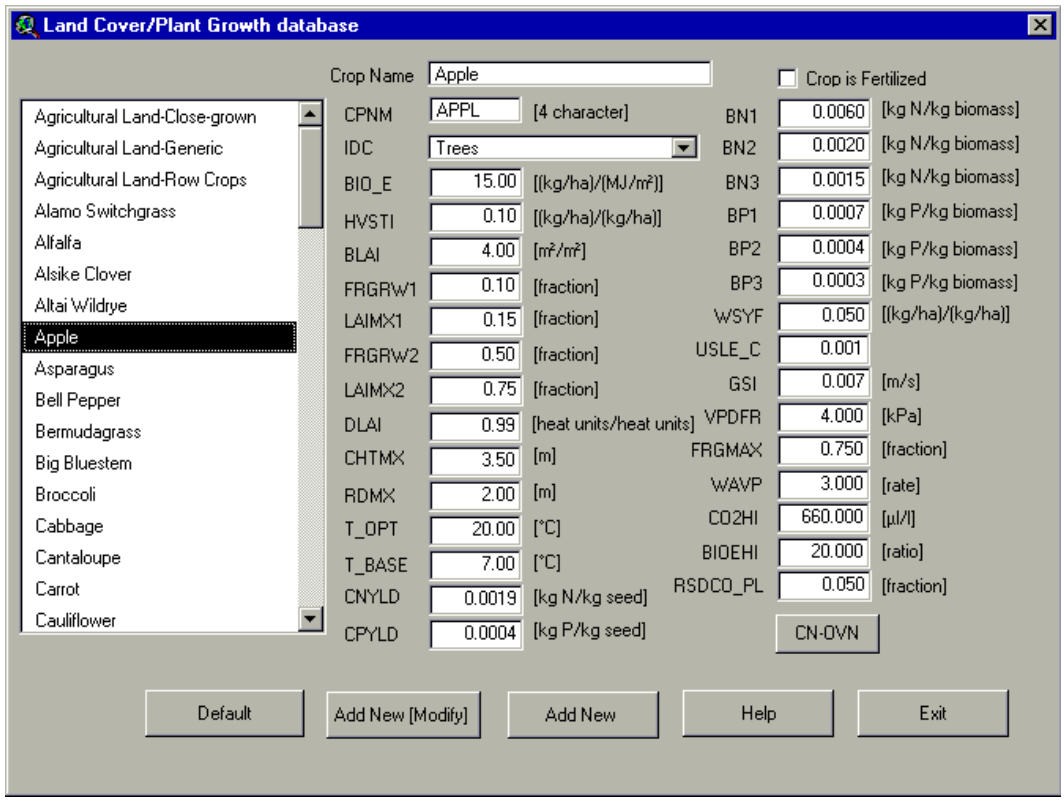


Figure 14.29

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation is shown.

   c. Edit data: Activate a cursor by clicking in the text box. Type the desired value. You can move to the next parameter by pressing the Tab key. If the value of the parameter is out of range, a message box like the one in Figure 14.30 will be displayed.

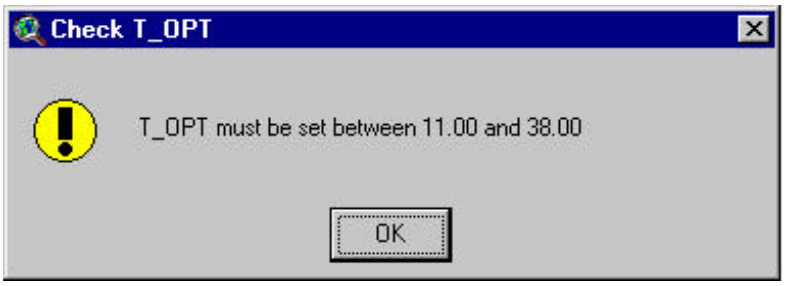


Figure 14.30

Click **OK.** The parameter will be reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *linstallation dir*/avswatdb directory. The names of the files are the three letter extension of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

d. Once all editing changes have been made, click **Exit**

.

e. A prompt box appears (Figure 14.31).

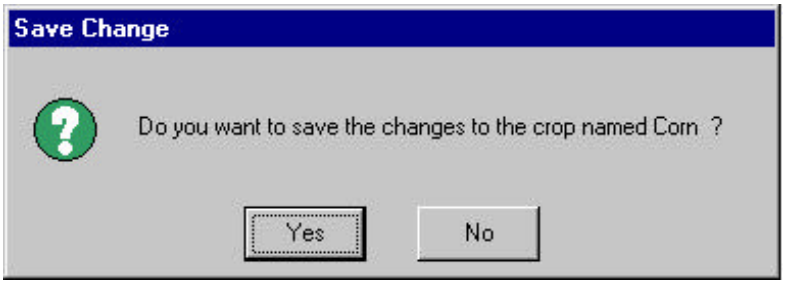


Figure 14.31

f. To save the editing changes, click **Yes**. To discard the editing changes, click **No.**

To recover default values for an edited land cover dataset:

The default growth parameters for the land cover types provided with the interface are stored and can be recovered at any time. This operation may only be performed for land covers provided with the interface. This operation will eliminate all changes made to the growth parameters by the user.

a. Click the name of the land cover/plant.

b. Click the button labeled **Default** at the bottom of the dialog box.

c. A prompt box will appear (Figure 14.32).

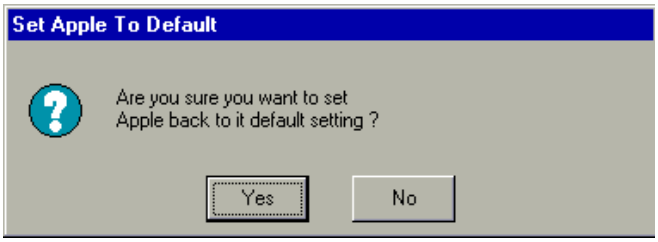


Figure 14.32

d. To recover default values, click **Yes**. To abort the recovery of default values, click **No**.

To add a new land cover dataset:

A new land cover may be added to the database by setting parameters on a blank form or by copying data from an existing land cover record to a new record.

To add a new land cover using a blank form:

a. Click the button labeled **Add New** on the Land Cover/Plant Growth dialog box.

b. A modified dialog box will appear which contains input fields for the growth parameters (Figure 14.33). The interface will preset the values to the minimum allowed for the parameters.

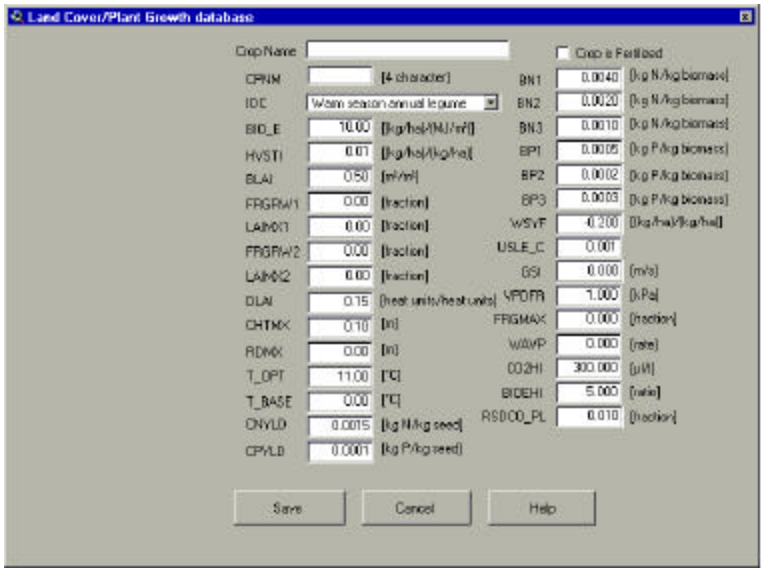


Figure 14.33

c. At this point, the user may type the necessary data into the different fields.

d. Once the land cover data is entered, click the button labeled **Save**

to store the data, or click the **Cancel**

button to exit the editor without saving.

e. The CROP: Hydrological Parameters dialog box is displayed when data for a new land cover/plant is saved (Figure 14.34).

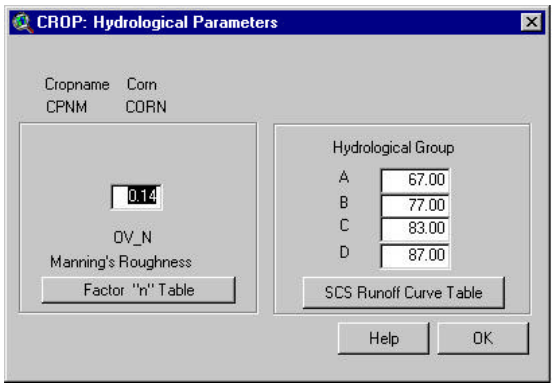


Figure 14.34

The user must define a default Manning's overland n value and default curve numbers that will be used by the interface to initialize these parameters when setting up projects where the new land cover/plant is present in the watershed.

Values may be typed into the text boxes or the user can access tables to decide what value should be assigned.

To access the SCS Runoff Curve Table, click the **SCS Runoff Curve Table**

button. The table will be displayed (Figure 14.35).

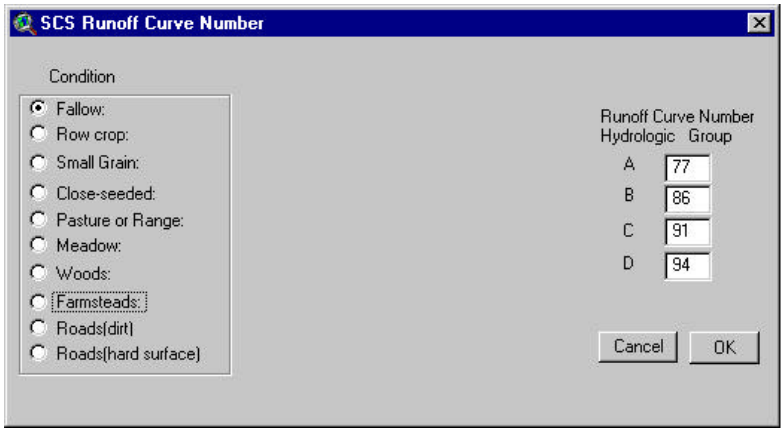


Figure 14.35

Curve number values for different management conditions are accessed by clicking the radio buttons for the condition and, if applicable, the cover. To exit the table and copy the displayed curve numbers, click **OK**. To exit the table without copying the displayed curve numbers, click **Cancel**.

To access the Manning's n Table, click the **Factor 'n' Table** Factor "n" Table button. The table will be displayed (Figure 14.36).

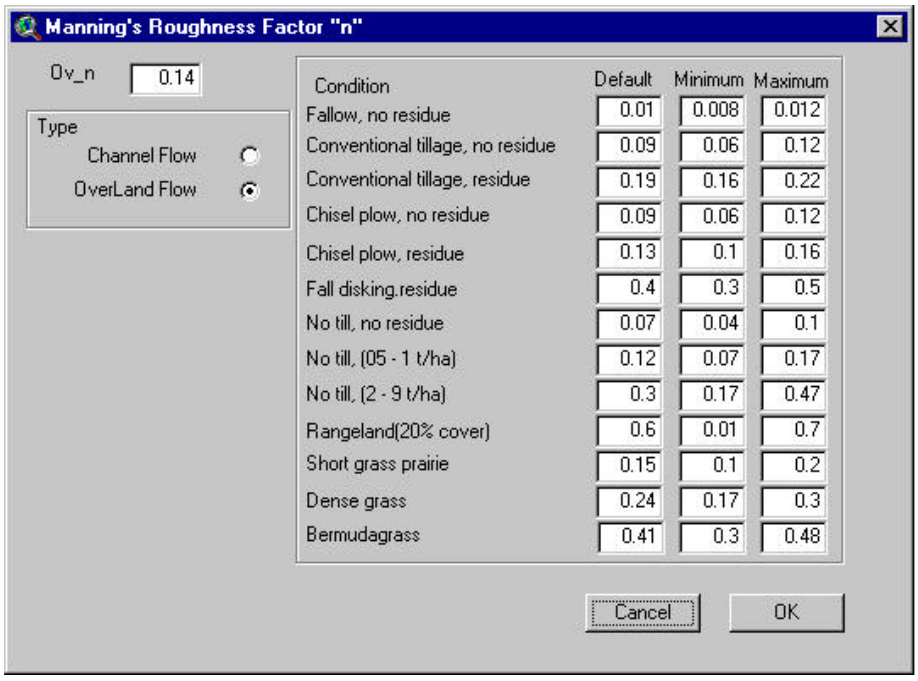


Figure 14.36

To use a value from the table, click on the desired value and then click **OK**. To exit the table without copying a value, click **Cancel**.

Once the default values for curve number and Manning's n value for overland flow are set correctly, click **OK**.

f. The user is returned to the original Land Cover/Plant Growth dialog box. The list of land covers in the database will include the name of the land cover just added.

To add a new land cover by modifying an existing record:

a. From the Land Cover/Plant Growth dialog box, click the name of the land cover to be copied.

b. Once the land cover data is displayed, click the button labeled **Add New (Modify)** Add New [Modify].

c. A modified dialog box will appear with all the data fields filled (copied from the previously selected land cover), except the land cover name (Figure 14.37).

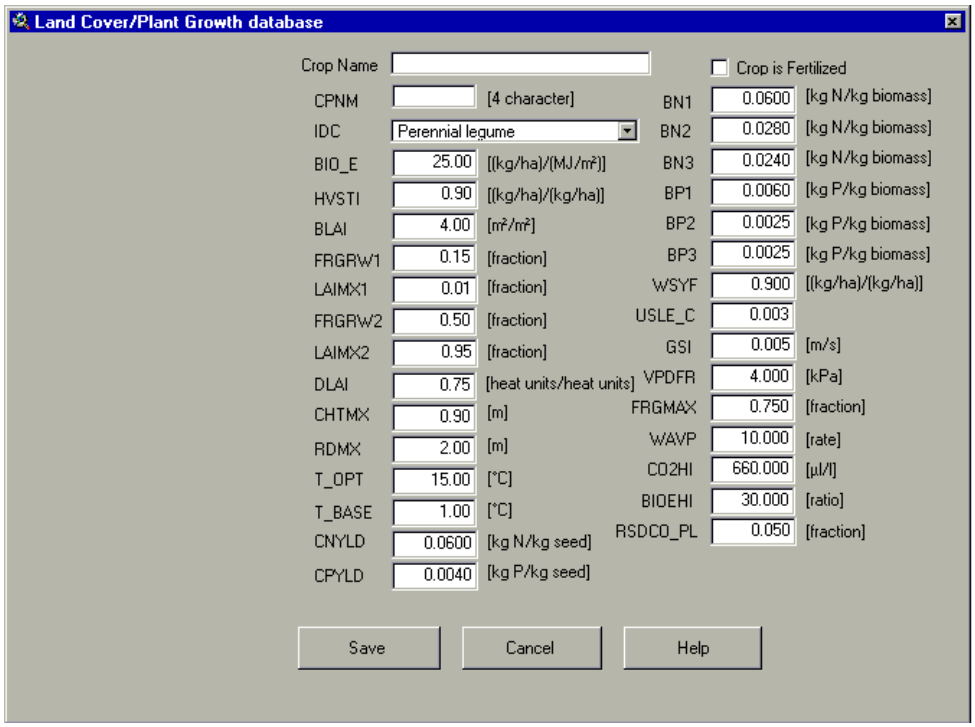


Figure 14.37

d. Enter a new land cover name and a four character code, and modify the other data fields.

e. Click **Save** to store the modified land cover data, or click **Cancel** to exit the editor without saving.

f. The CROP: Hydrological Parameters dialog box is displayed when data for a new land cover/plant is saved (Figure 14.38)

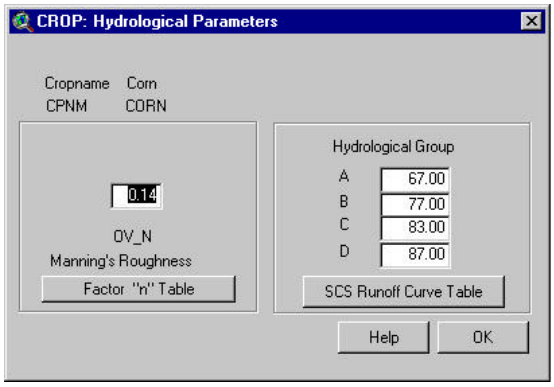


Figure 14.38

The user must define a default Manning's overland n value and default curve numbers that will be used by the interface to initialize these parameters when setting up projects where the new land cover/plant is present in the watershed.

Values may be typed into the text boxes or the user can access tables to decide what value should be assigned.

To access the SCS Runoff Curve Table, click the **SCS Runoff Curve Table** SCS Runoff Curve Table button. The table will be displayed (Figure 14.39).

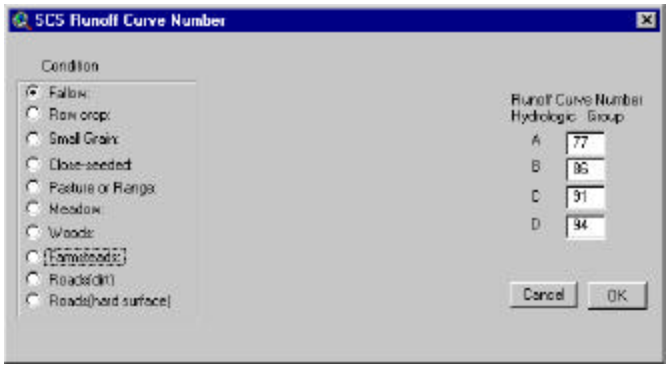


Figure 14.39

Curve number values for different management conditions are accessed by clicking the radio buttons for the condition and, if applicable, the cover. To exit the table and copy the displayed curve numbers, click **OK**. To exit the table without copying the displayed curve numbers, click **Cancel**.

To access the Manning's n Table, click the **Factor 'n' Table** Factor "n" Table button. The table will be displayed (Figure 14.40).

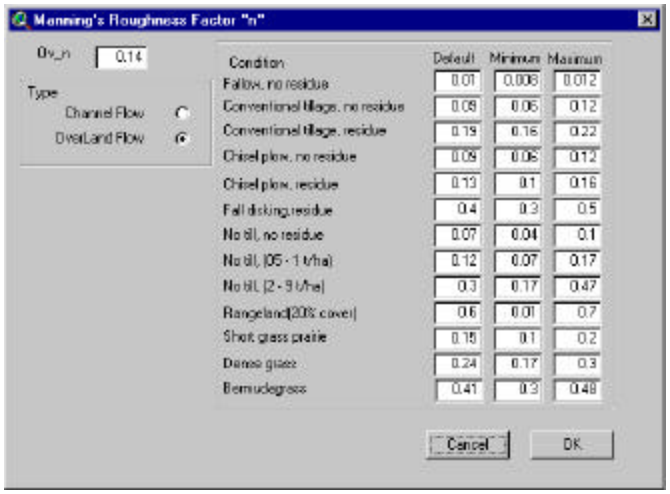


Figure 14.40

To use a value from the table, click on the desired value and then click **OK**. To exit the table without copying a value, click **Cancel**.

Once the default values for curve number and Manning's n value for overland flow are set correctly, click **OK**.

g. The user is returned to the original Land Cover/Plant Growth dialog box. The list of land covers in the database will include the name of the land cover just added.

To delete a user defined land cover dataset:

a. From the Land Cover/Plant Growth dialog box, click the name of the land cover to be deleted.

b. The dialog box will be modified to display the land cover data. Click the button labeled **Delete** at the bottom of the dialog box.

c. A prompt box will appear requesting the user to verify the delete (Figure 14.41).

Figure 14.41

To continue with the delete process, click **Yes**. To abandon the delete process, click **No**.

> **Note:** Only the new Land Cover/Plant Growth data sets entered by the User can be removed from the database. The user can not remove the data sets that are included with the interface but can modify the parameters and/or restore the default values.

To exit the Land Cover/Plant Growth database:

a. From the Land Cover/Plant Growth dialog box, click **Exit** .

SECTION 14.4: FERTILIZER DATABASE

To edit the Fertilizer Database:

1. Double-click **Fertilizer** on the list of databases (Figure 14.42).

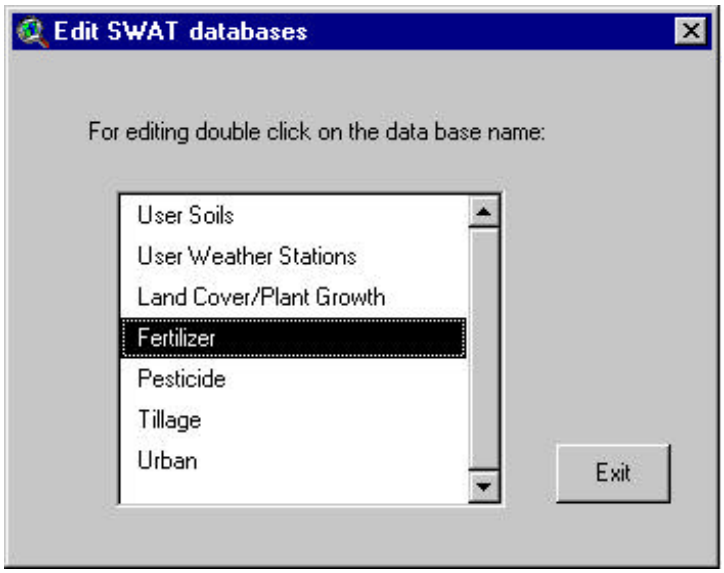


Figure 14.42

2. The Fertilizer database dialog box will be displayed (Figure 14.43)

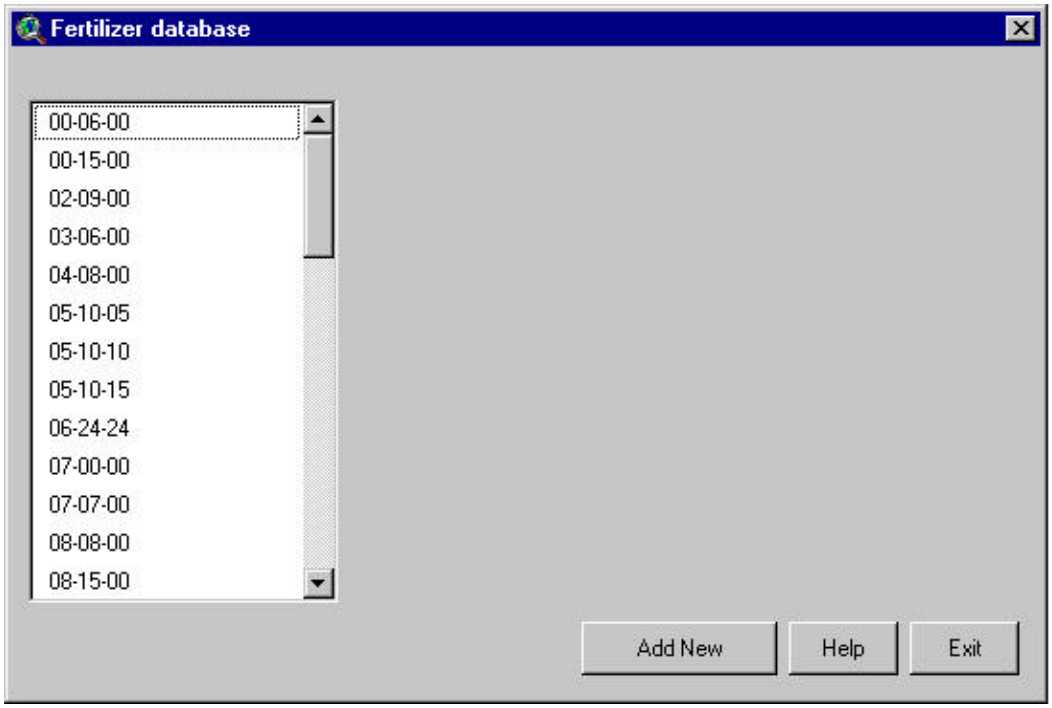


Figure 14.43

A list of fertilizers/manure in the database is displayed on the left side of the dialog box.

3. The user has four options: edit an existing fertilizer, add a new fertilizer, delete an existing fertilizer, or exit the database.

   To edit an existing fertilizer:

   a. Click the name of the fertilizer to be edited.

   b. The data for the fertilizer will be displayed (Figure 14.44). A complete description of the parameters is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 17.

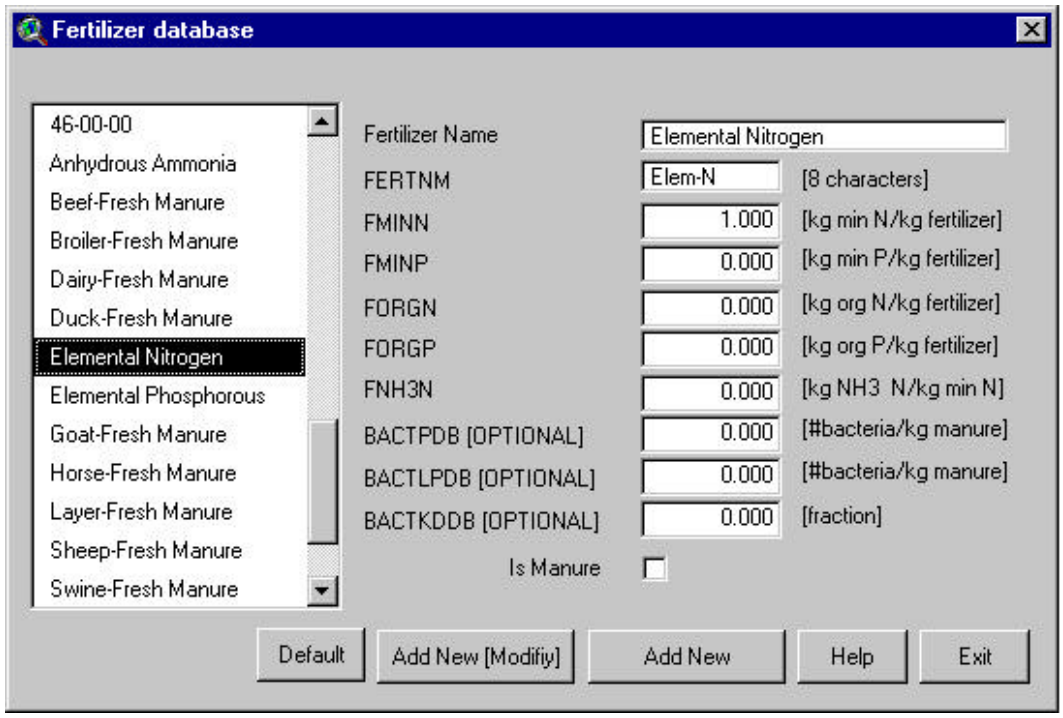


Figure 14.44

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation is shown.

   c. Edit data: Activate a cursor by clicking in the text box. Type the desired value. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 14.45 will be displayed.

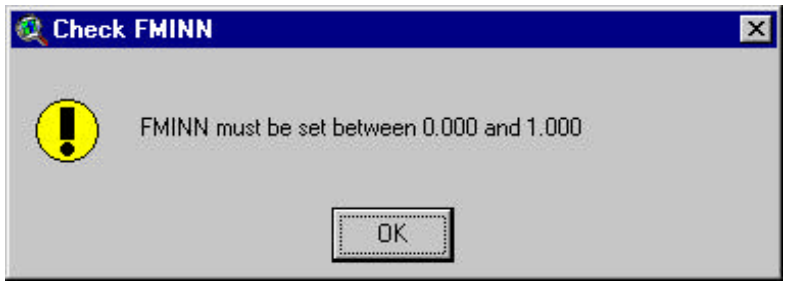


Figure 14.45

Click **OK.** The parameter will be reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *installation dir*/avswatdb directory. The names of the files are the three letter extension of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

d. Once all editing changes have been made, click **Exit**

.

e. A prompt box appears (Figure 14.46).

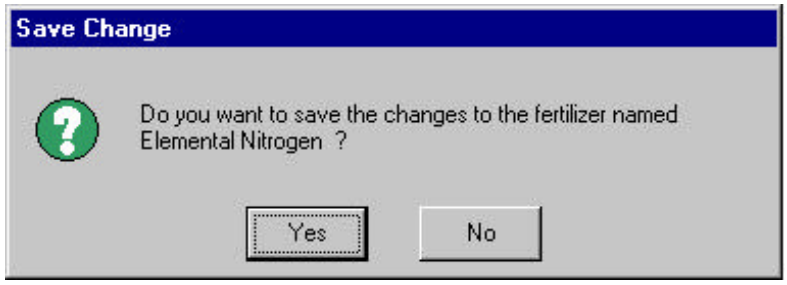


Figure 14.46

f. To save the editing changes, click **Yes**. To discard the editing changes, click **No**.

To recover default values for an edited fertilizer:

The default values for the fertilizers provided with the interface are stored and can be recovered at any time. This operation may only be performed for fertilizers provided with the interface. This operation will eliminate all changes made to the fertilizer parameters by the user.

a. Click the name of the fertilizer.

b. Click the button labeled **Default** at the bottom of the dialog box.

c. A prompt box will appear (Figure 14.47).

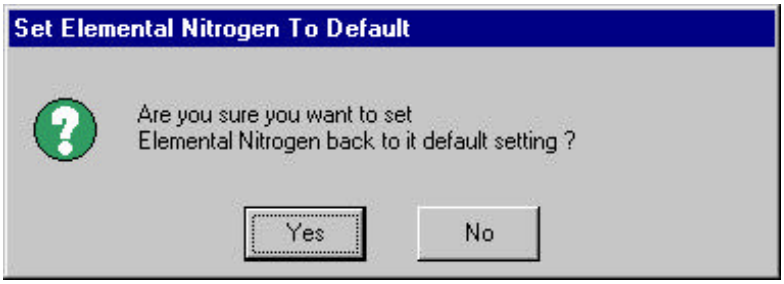


Figure 14.47

d. To recover default values, click **Yes**. To abort the recovery of default values, click **No**.

To add a new fertilizer:

A new fertilizer may be added to the database by setting parameters on a blank form or by copying data from an existing fertilizer record to a new record.

To add a new fertilizer using a blank form:

a. Click the button labeled **Add New** on the Fertilizer Database dialog box.

b. A modified dialog box will appear which contains input fields for the fertilizer data (Figure 14.48).

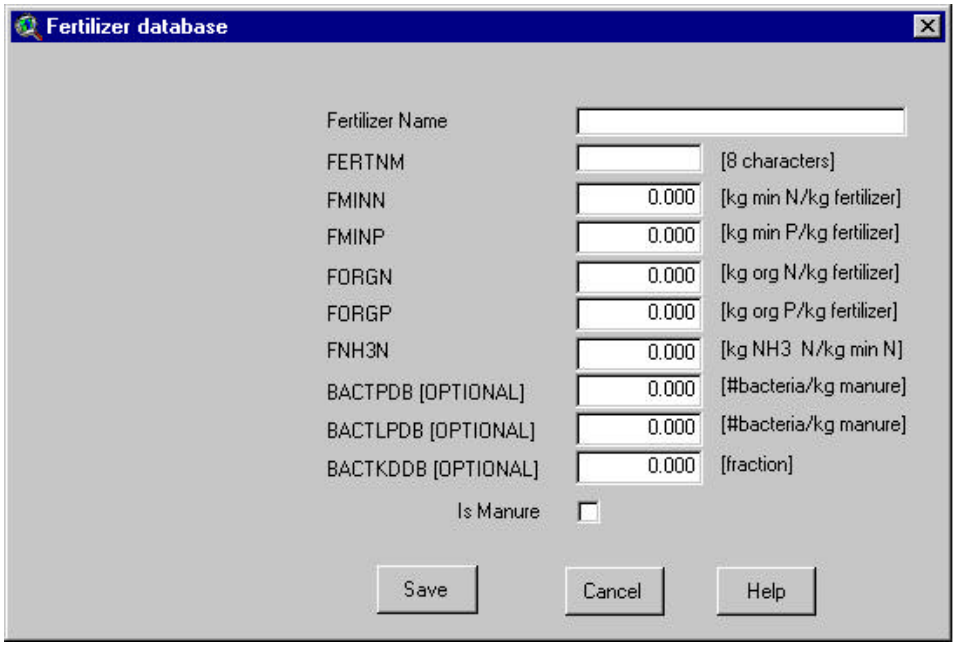


Figure 14.48

c. At this point, the user may type the necessary data into the different fields.

d. Once the fertilizer data is entered, click the button labeled **Save** to store the data, or click **Cancel** to exit the editor without saving.

e. The user is returned to the original Fertilizer Database dialog box. The list of fertilizers in the database will include the name of the fertilizer just added.

To add a new fertilizer by modifying an existing record:

a. From the Fertilizer Database dialog box, click the name of the fertilizer to be copied.

b. Once the fertilizer data is displayed, click the button labeled **Add New (Modify)** .

c. A modified dialog box will appear with all the data fields filled except the fertilizer name (Figure 14.49).

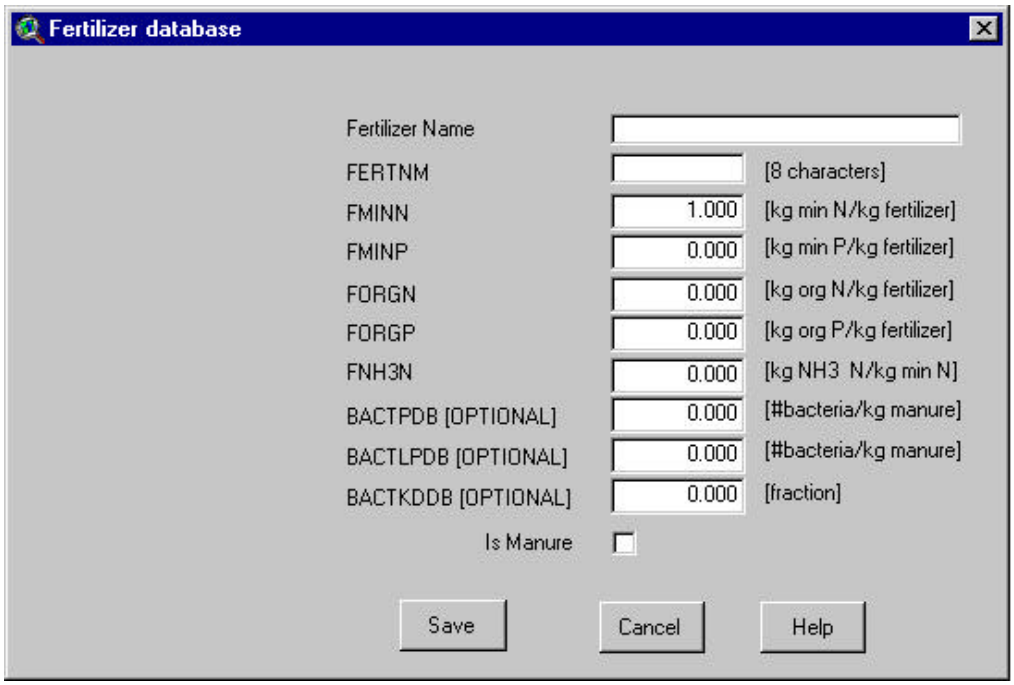


Figure 14.49

d. Enter a new fertilizer name and character code and modify the other data fields.

e. Click **Save** to store the modified fertilizer data, or click **Cancel** to exit the editor without saving.

f. The user is returned to the original Fertilizer Database dialog box. The list of fertilizers in the database will include the name of the fertilizer just added.

To delete a user defined fertilizer:

a. From the Fertilizer Database dialog box, click the name of the fertilizer to be deleted.

b. The dialog box will be modified to display the fertilizer data. Click the button labeled **Delete**

at the bottom of the dialog box.

c. A prompt box will appear requesting the user to verify the delete (Figure 14.50).

Figure 14.50

d. To continue with the delete process, click **Yes**. To abandon the delete process, click **No**.

> **Note:** Only new Fertilizer data sets entered by the user can be removed from the database. The user cannot remove the data sets that are included with the interface but can modify the parameters and/or restore the default values.

To exit the Fertilizer Database:

a. From the Fertilizer Database dialog box, click **Exit**

.

SECTION 14.5: PESTICIDE DATABASE

To edit the Pesticide Database:

1. Double-click **Pesticide** on the list of databases (Figure 14.51).

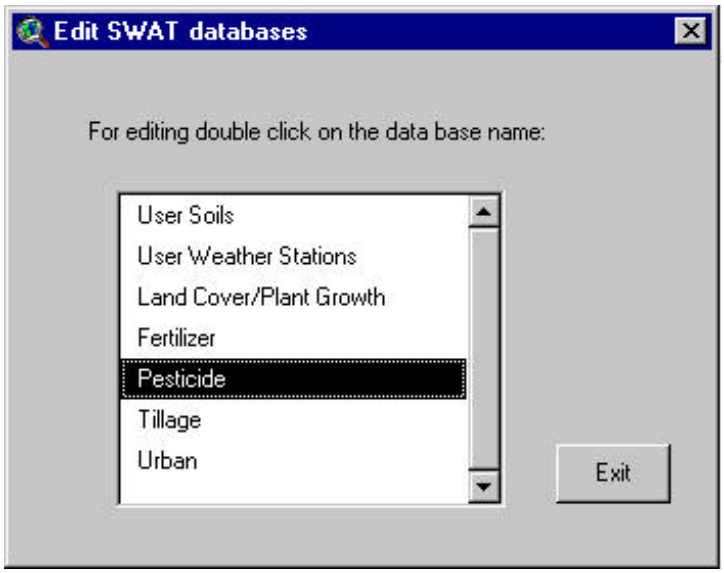


Figure 14.51

2. The Pesticide database dialog box will be displayed (Figure 14.52).

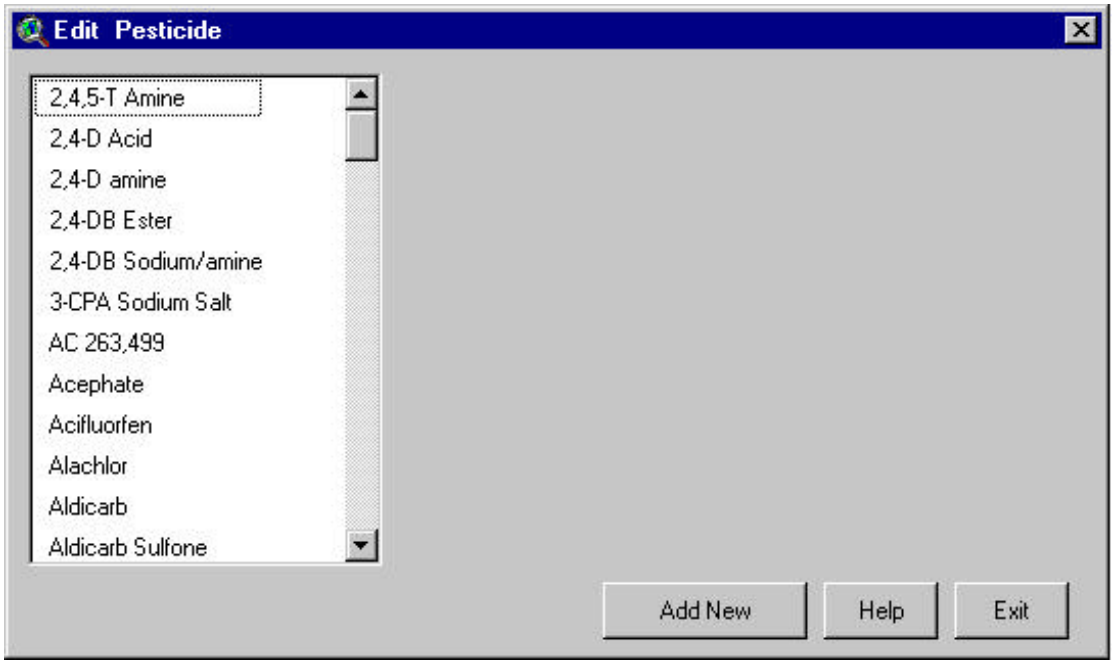


Figure 14.52

A list of pesticides in the database is displayed on the left side of the dialog box.

3. The user has four options: edit an existing pesticide, add a new pesticide, delete an existing pesticide, or exit the database.

   To edit an existing pesticide:

   a. Click the name of the pesticide to be edited.

   b. The data for the pesticide will be displayed (Figure 14.53). A complete description of the parameters is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 16.

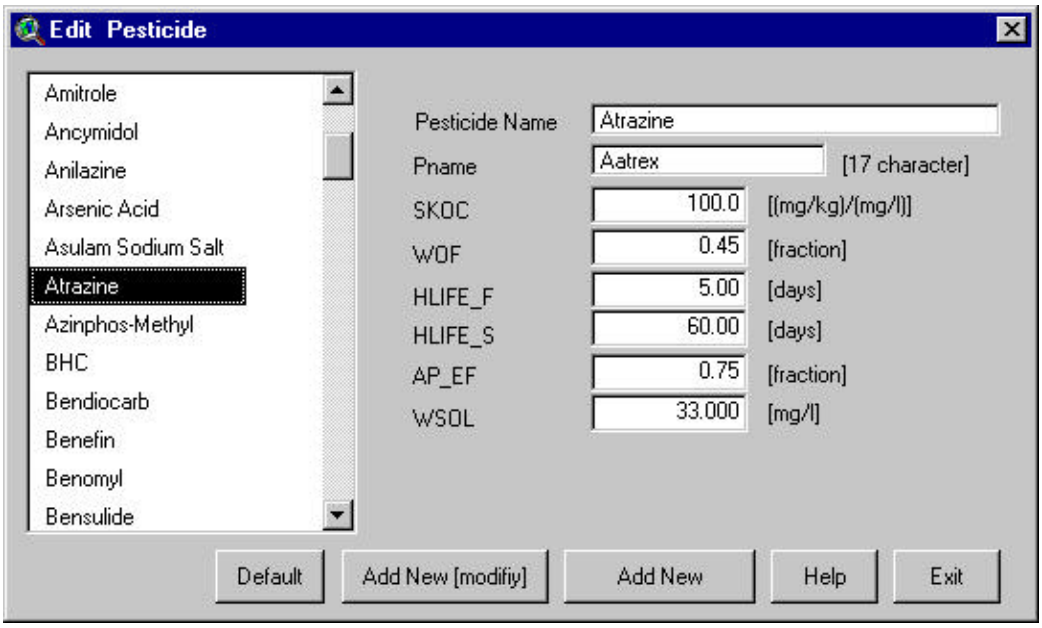


Figure 14.53

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation is shown.

   c. Edit data: Activate a cursor by clicking in the text box. Type the desired value. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 14.54 will be displayed.

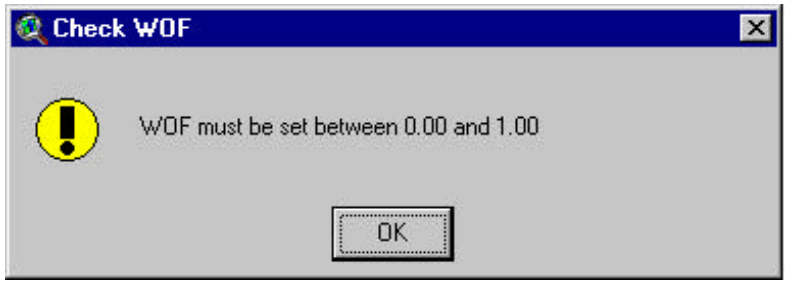


Figure 14.54

Click **OK.** The parameter will be reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *linstallation dir*/avswatdb directory. The names of the files are the three letter extension of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

d. Once all editing changes have been made, click **Exit**

.

e. A prompt box appears (Figure 14.55).

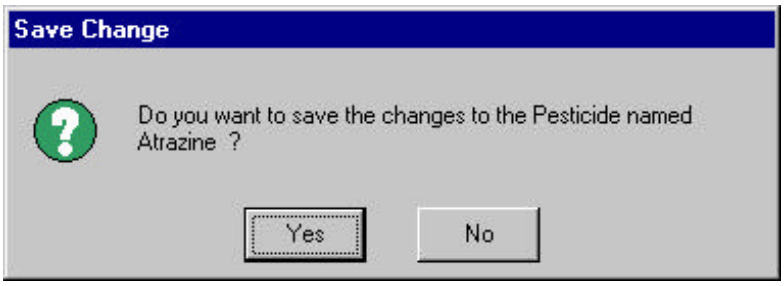


Figure 14.55

f. To save the editing changes, click **Yes**. To discard the editing changes, click **No.**

To recover default values for an edited pesticide:

The default values for the pesticides provided with the interface are stored and can be recovered at any time. This operation may only be performed for pesticides provided with the interface. This operation will eliminate all changes made to the pesticide parameters by the user.

a. Click the name of the pesticide.

b. Click the button labeled **Default** at the bottom of the dialog box.

c. A prompt box will appear (Figure 14.56).

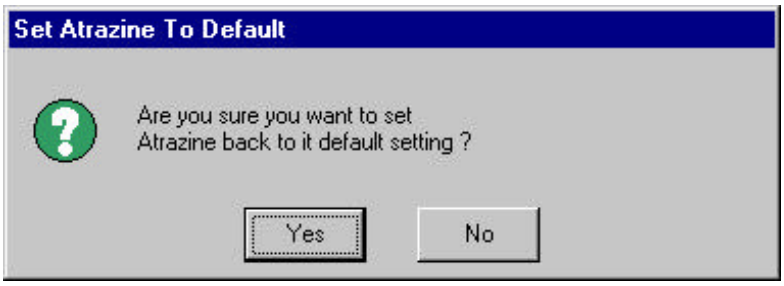


Figure 14.56

d. To recover default values, click **Yes**. To abort the recovery of default values, click **No**.

To add a new pesticide:

A new pesticide may be added to the database by setting parameters on a blank form or by copying data from an existing pesticide record to a new record.

To add a new pesticide using a blank form:

a. Click the button labeled **Add New** on the Pesticide Database dialog box.

b. A modified dialog box will appear which contains input fields for the pesticide data (Figure 14.57).

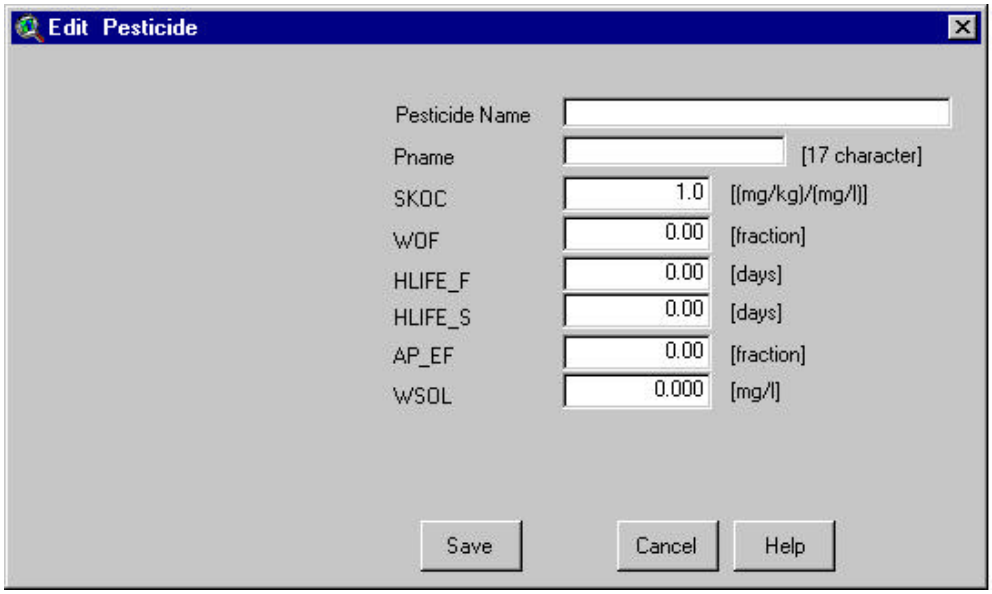


Figure 14.57

c. At this point, the user may type the necessary data into the different fields.

d. Once the pesticide data is entered, click the button labeled **Save**

to store the data, or click **Cancel**

to exit the editor without saving.

e. The user is returned to the original Pesticide Database dialog box. The list of pesticides in the database will include the name of the pesticide just added.

To add a new pesticide by modifying an existing record:

a. From the Pesticide Database dialog box, click the name of the pesticide to be copied.

b. Once the pesticide data is displayed, click the button labeled **Add New (Modify)**

.

c. A modified dialog box will appear with all the data fields filled except the pesticide name (Figure 14.58).

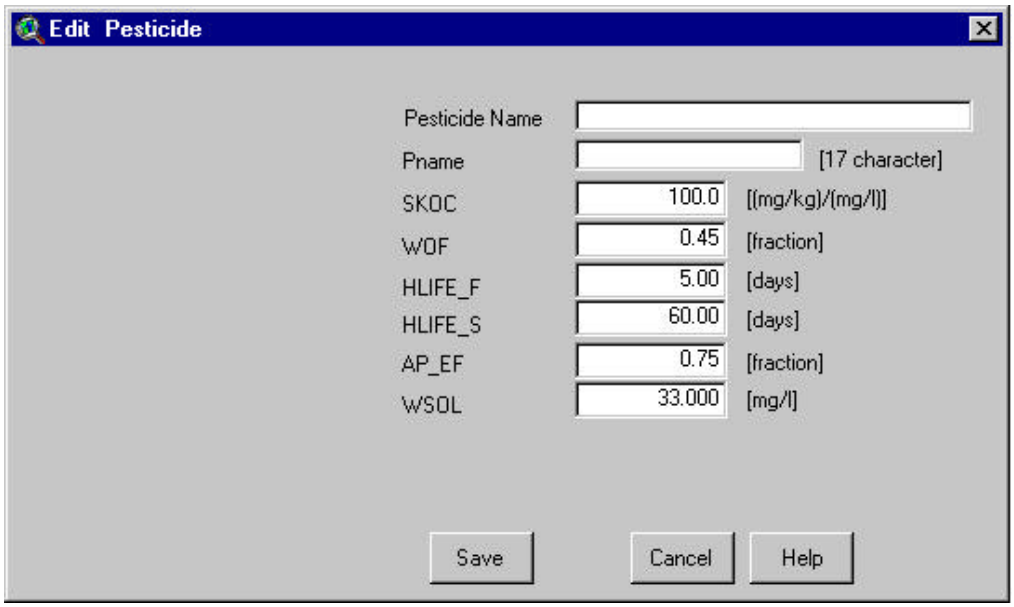


Figure 14.58

d. Enter a new pesticide name, and modify the other data fields.

e. Click **Save** to store the modified pesticide data, or click **Cancel** to exit the editor without saving.

f. The user is returned to the original Pesticide Database dialog box. The list of pesticides in the database will include the name of the pesticide just added.

To delete a user defined pesticide:

a. From the Pesticide Database dialog box, click the name of the pesticide to be deleted.

b. The dialog box will be modified to display the pesticide data. Click the button labeled **Delete**

at the bottom of the dialog box.

c. A prompt box will appear, requesting the user to verify the delete (Figure 14.59).

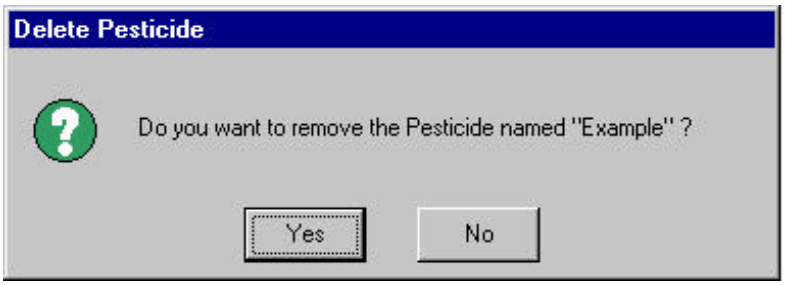


Figure 14.59

d. To continue with the delete process, click **Yes**. To abandon the delete process, click **No**.

> **Note:** Only the new Pesticide data sets entered by the User can be removed from the database. The user cannot remove the data sets that are included with the interface but can modify the parameters and/or restore the default values.

To exit the Pesticide Database:

a. From the Pesticide Database dialog box, click **Exit**

.

SECTION 14.6: TILLAGE DATABASE

To edit the Tillage Database:

1. Double-click **Tillage** on the list of databases (Figure 14.60).

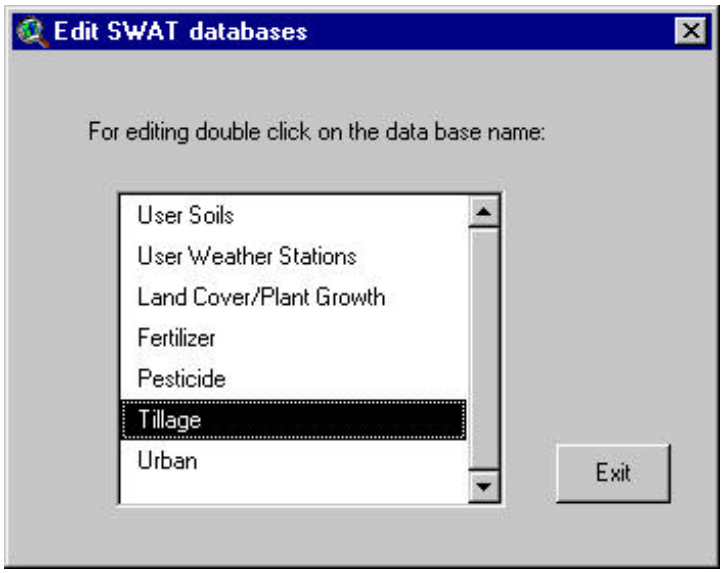


Figure 14.60

2. The Tillage database dialog box will be displayed (Figure 14.61).

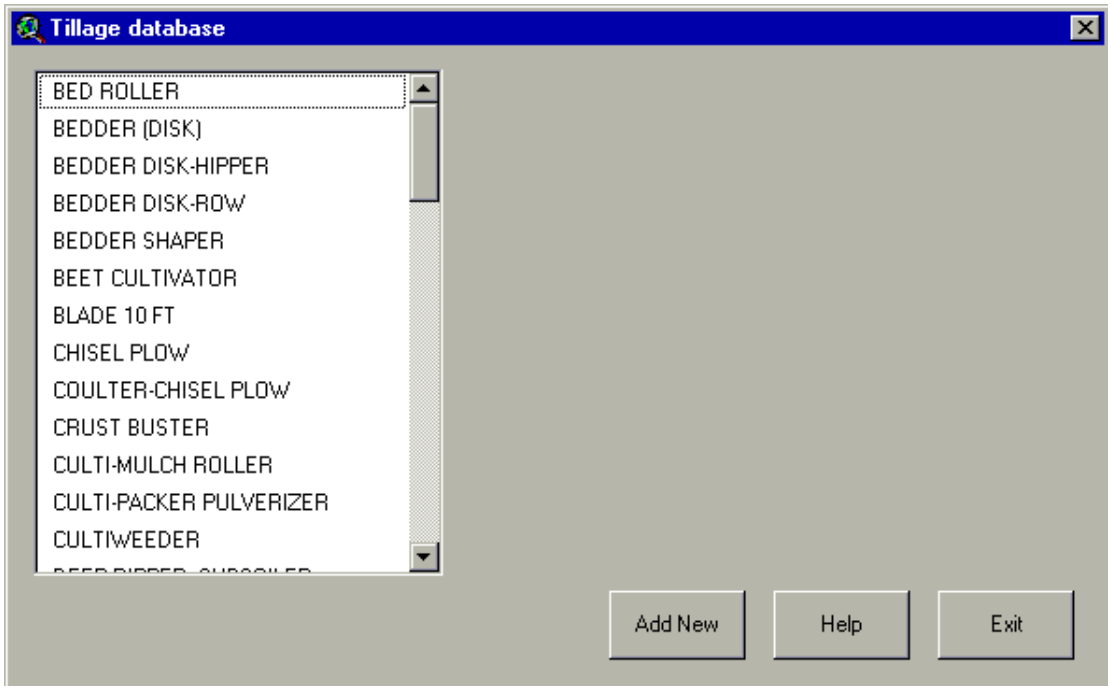


Figure 14.61

A list of tillage implements in the database is displayed on the left side of the dialog box.

3. The user has four options: edit an existing tillage data set, add a new tillage data set, delete an existing tillage data set, or exit the database.

   To edit an existing tillage operation:

   a. Click the name of the tillage data set to be edited.

   b. The data for the tillage operation will be displayed (Figure 14.62). A complete description of the parameters is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 15.

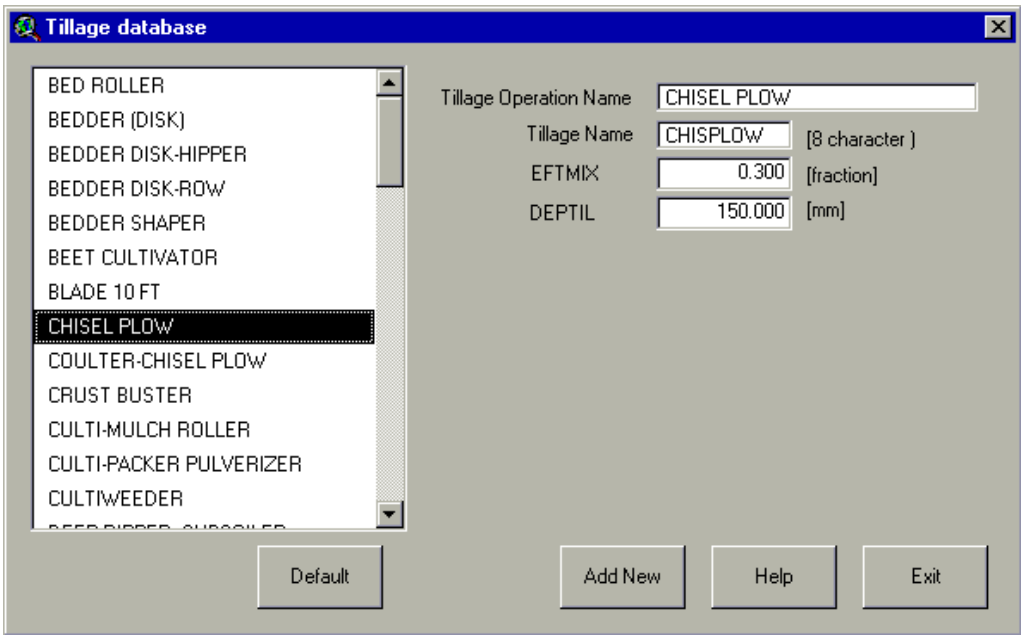


Figure 14.62

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation is shown.

   c. Edit data: Activate a cursor by clicking the text box. Type the desired value. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 14.63 will be displayed.

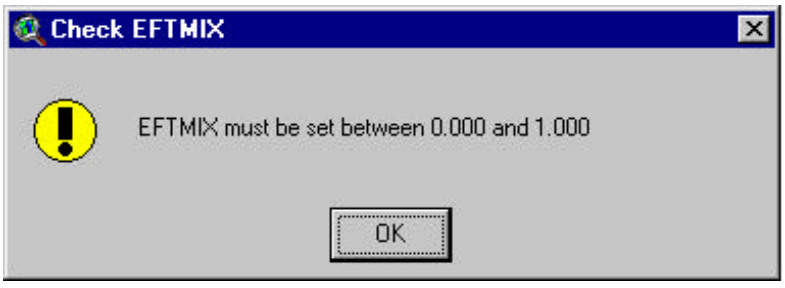


Figure 14.63

Click **OK.** The parameter will be reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *linstallation dir*/avswatdb directory. The names of the files are the three letter extension of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

d. Once all editing changes have been made, click **Exit**

.

e. A prompt box appears (Figure 14.64).

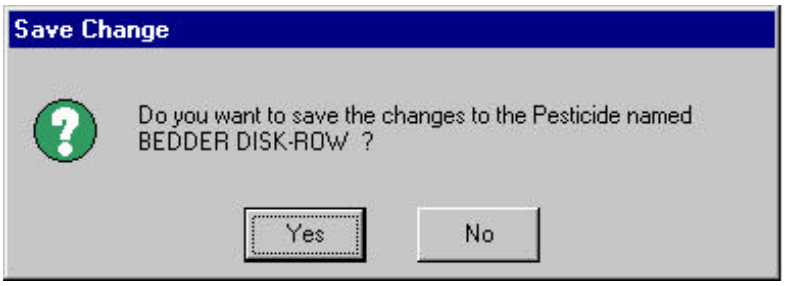


Figure 14.64

f. To save the editing changes, click **Yes**. To discard the editing changes, click **No.**

To recover default values for an edited tillage operation:

The default values for the tillage operations provided with the interface are stored and can be recovered at any time. This procedure may only be performed for tillage operations provided with the interface. This procedure will eliminate all changes made to the tillage parameters by the user.

a. Click the name of the tillage operation.

b. Click the button labeled **Default** at the bottom of the dialog box.

c. A prompt box will appear (Figure 14.65).

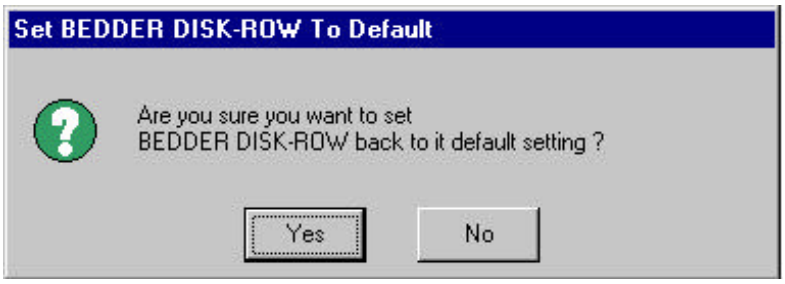


Figure 14.65

d. To recover default values, click **Yes**. To abort the recovery of default values, click **No**.

To add a new tillage operation:

a. Click the button labeled **Add New** on the Tillage Database dialog box.

b. A modified dialog box will appear which contains input fields for the tillage data (Figure 14.66).

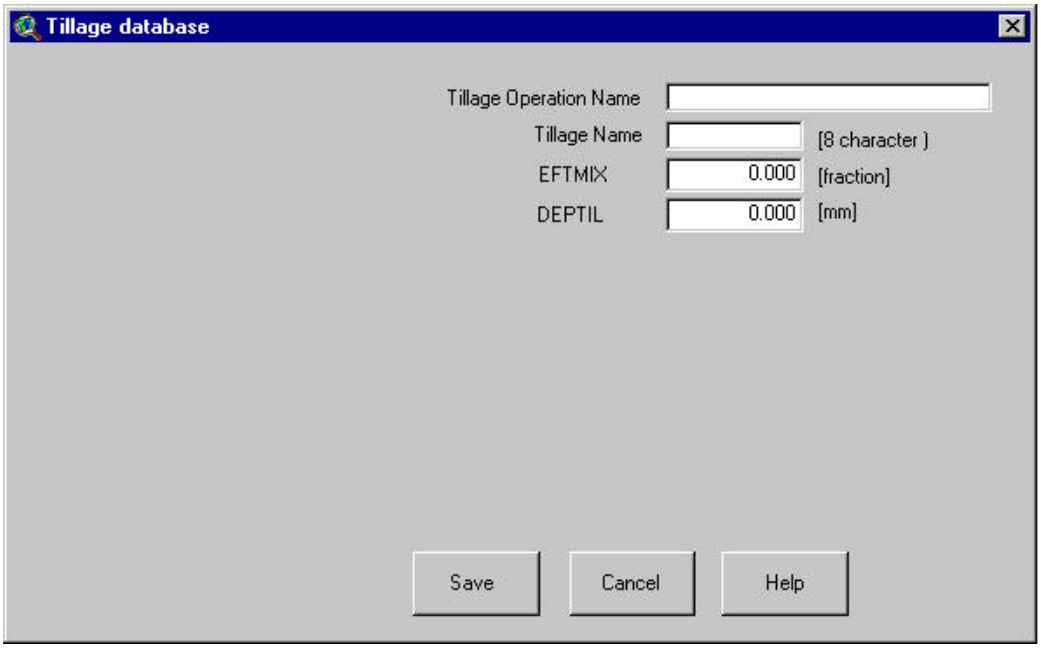


Figure 14.66

c. At this point, the user may type the necessary data into the different fields.

d. Once the tillage data is entered, click the button labeled **Save** to store the data, or click **Cancel** to exit the editor without saving.

e. The user is returned to the original Tillage Database dialog box. The list of tillage operations in the database will include the name of the tillage operation just added.

To delete a tillage operation:

a. From the Tillage Database dialog box, click the name of the tillage operation to be deleted.

b. The dialog box will be modified to display the tillage data. Click the button labeled **Delete**

at the bottom of the dialog box.

c. A prompt box will appear, requesting the user to verify the delete (Figure 14.67).

Figure 14.67

d. To continue with the delete process, click **Yes**. To abandon the delete process, click **No**.

> **Note:** Only the new Tillage data sets entered by the User can be removed from the database. The user cannot remove the data sets that are included with interface but can modify the parameters and/or restore the default values.

To exit the Tillage Database:

a. From the Tillage Database dialog box, click **Exit**

.

SECTION 14.7: URBAN DATABASE

To edit the Urban Database:

1. Double-click **Urban** on the list of databases (Figure 14.68).

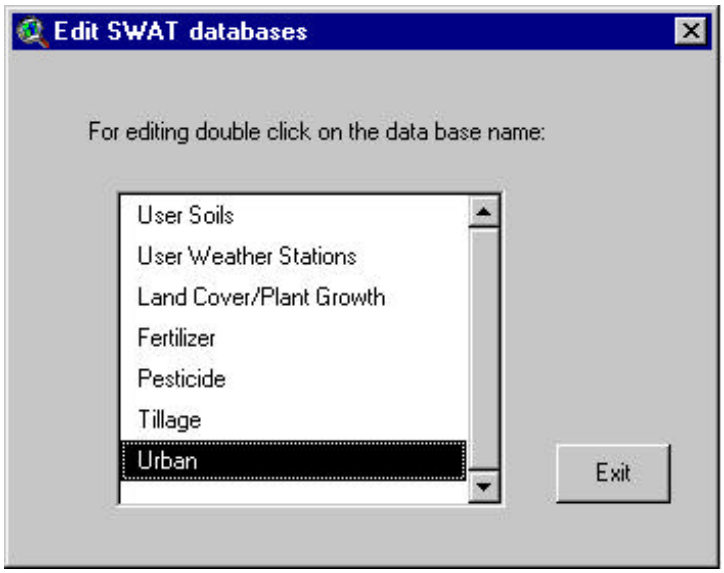


Figure 14.68

2. The Urban database dialog box will be displayed (Figure 14.69).

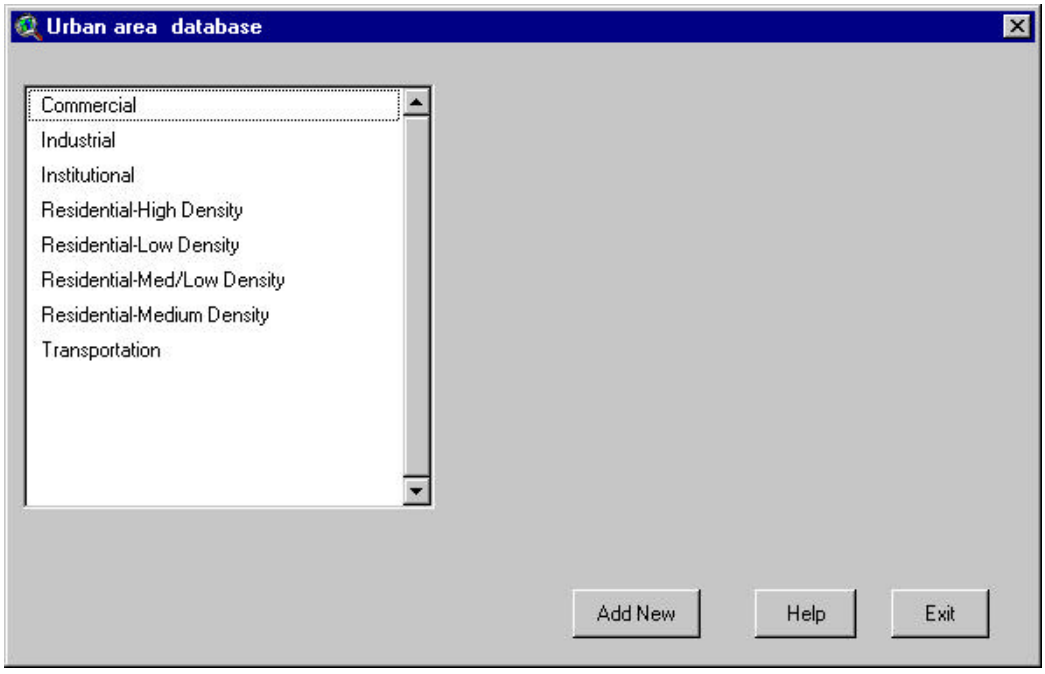


Figure 14.69

A list of urban land types in the database is listed on the left side of the dialog box.

3. The user has four options: edit an existing urban data set, add a new urban data set, delete an urban data set, or exit the database.

   To edit an existing urban land type:

   a. Click the name of the urban land type to be edited.

   b. The data for the urban land type will be displayed (Figure 14.70). A complete description of the parameters is provided in the *Soil and Water Assessment Tool User's Manual, Version 2000*, Chapter 18.

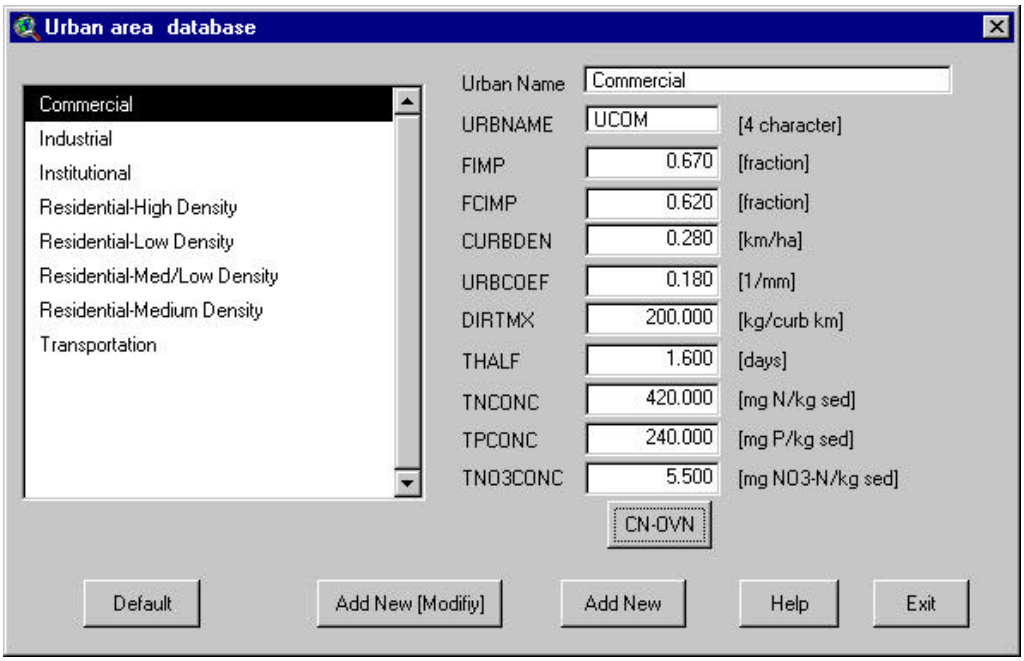


Figure 14.70

> **Note:** If the cursor is placed on top of any text box or button, a short help description (yellow label) appears and the range of variation is shown.

   c. Edit data: Activate a cursor by clicking in the text box. Type the desired value. You can move to the next parameter by pressing the TAB key. If the value of the parameter is out of range, a message box like the one in Figure 14.71 will be displayed.

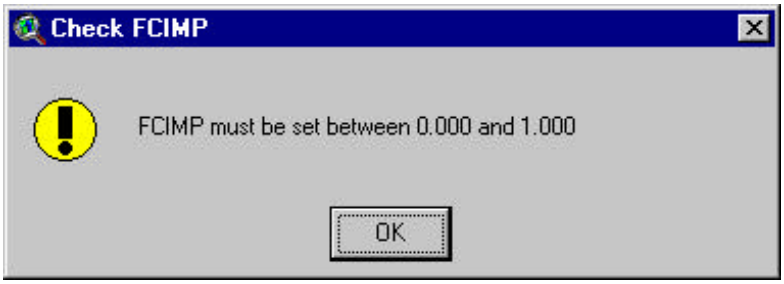


Figure 14.71

Click **OK.** The parameter will be reset to the original value or to the closest range limit.

> **Note:** Value ranges for parameters are editable. Some of the default minimum and maximum values assigned to variables may not be reasonable for your watershed. All tables containing the variable ranges are located in the *linstallation dir*/avswatdb directory. The names of the files are the three letter extension of the SWAT ASCII input files listed in the *Soil and Water Assessment Tool User's Manual, Version 2000,* followed by "rng.dbf". As an illustration, the reservoir parameter ranges are stored in "resrng.dbf"

d. Once all editing changes have been made, click **Exit**

.

e. A prompt box appears (Figure 14.72).

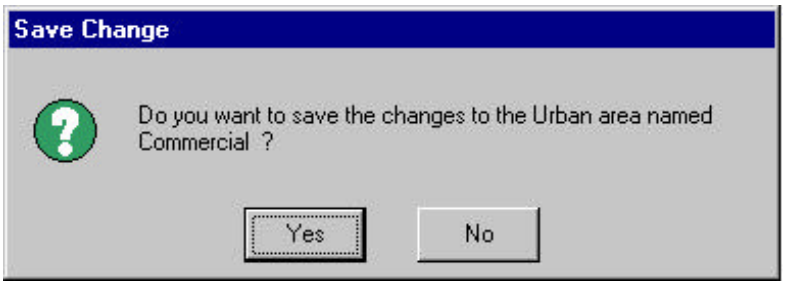


Figure 14.72

f. To save the editing changes, click **Yes**. To discard the editing changes, click **No**.

To recover default values for an edited urban land type:

The default values for the urban land types provided with the interface are stored and can be recovered at any time. This procedure may only be performed for urban land types provided with the interface. This procedure will eliminate all changes made to the urban parameters by the user.

a. Click the name of the urban land type.

b. Click the button labeled **Default** at the bottom of the dialog box.

c. A prompt box will appear (Figure 14.73).

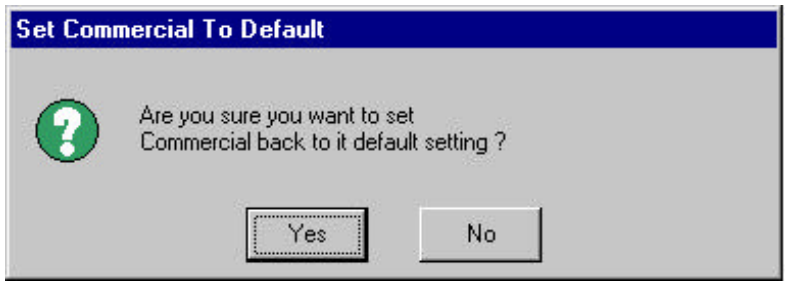


Figure 14.73

d. To recover default values, click **Yes**. To abort the recovery of default values, click **No**.

To add a new urban land type:

A new urban land type may be added to the database by setting parameters on a blank form or by copying data from an existing urban land type record to a new record.

To add a new urban land type using a blank form:

a. Click the button labeled **Add New** on the Urban Database dialog box.

b. A modified dialog box will appear which contains input fields for the urban data (Figure 14.74).

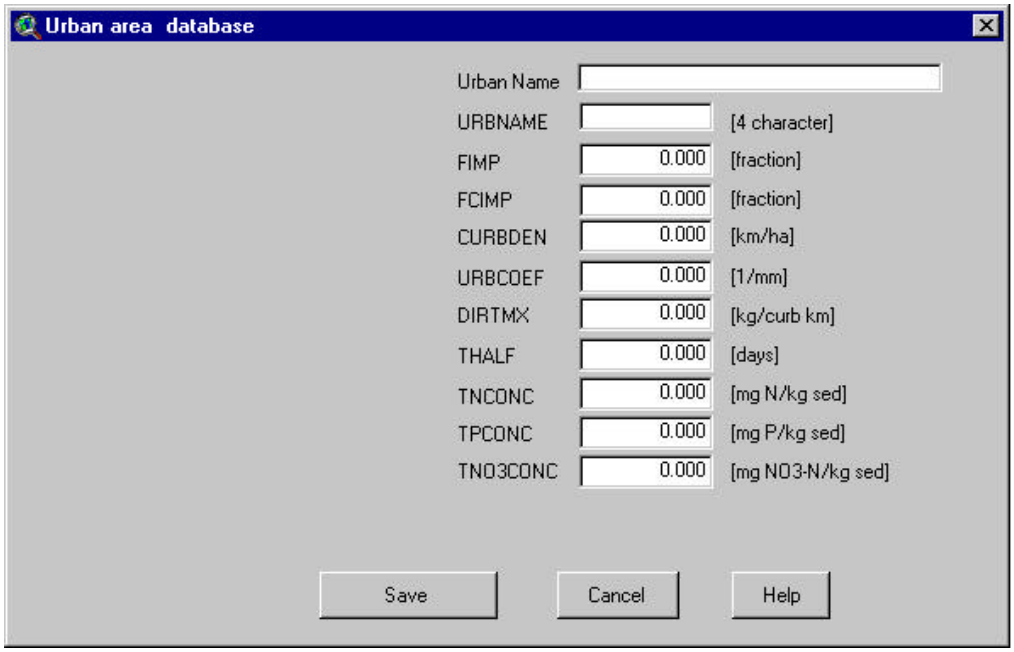


Figure 14.74

c. At this point, the user may type the necessary data into the different fields.

d. Once the urban data is entered, click the button labeled **Save**

to store the data, or click **Cancel**

to exit the editor without saving.

e. The CROP: Hydrological Parameters dialog box is displayed when data for a new urban land type is saved (Figure 14.75)

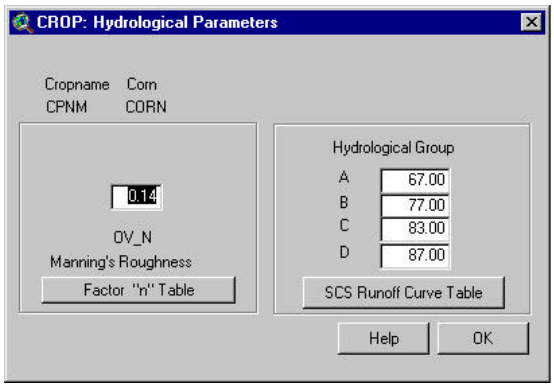


Figure 14.75

The user must define a default Manning's overland n value and default curve numbers that will be used by the interface to initialize these parameters when setting up projects where the new urban land type is present in the watershed.

Values may be typed into the text boxes, or the user can access tables to decide what value should be assigned.

To access the SCS Runoff Curve Table, click the **SCS Runoff Curve Table**

button. The table will be displayed (Figure 14.76).

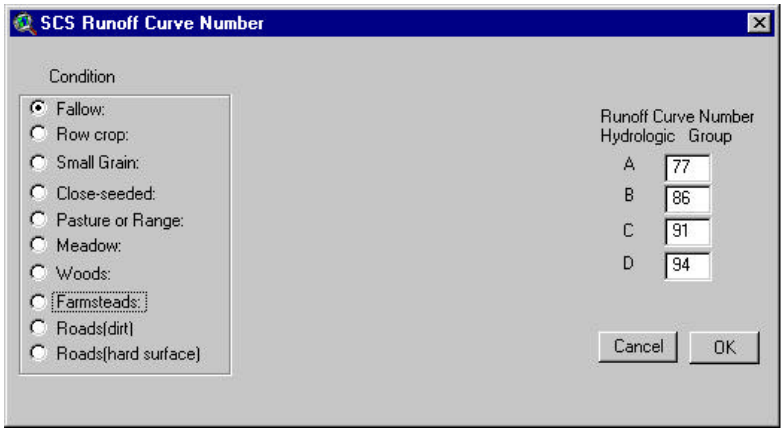


Figure 14.76

Curve number values for different management conditions are accessed by clicking the radio buttons for the condition and, if

applicable, the cover. To exit the table and copy the displayed curve numbers, click **OK**. To exit the table without copying the displayed curve numbers, click **Cancel**.

To access the Manning's n Table, click the **Factor 'n' Table** button. The table will be displayed (Figure 14.77).

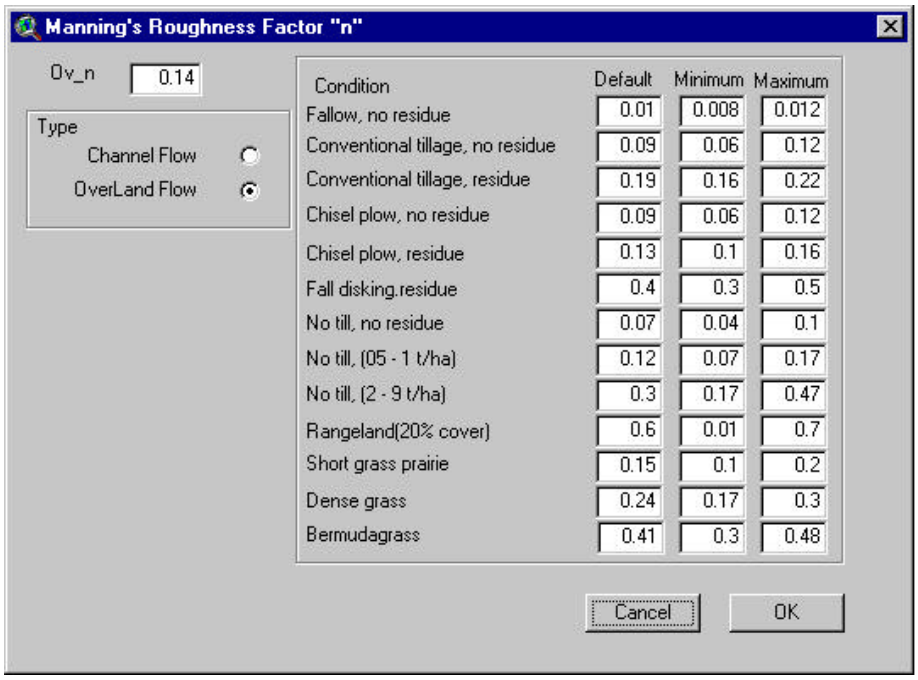


Figure 14.77

To use a value from the table, click on the desired value and then click **OK**. To exit the table without copying a value, click **Cancel**.

Once the default values for curve number and Manning's n value for overland flow are set correctly, click **OK**.

f. The user is returned to the original Urban Database dialog box. The list of urban land types in the database will include the name of the urban land type just added.

To add a new urban land type by modifying an existing record:

a. From the Urban Database dialog box, click the name of the urban land type to be copied.

b. Once the urban data is displayed, click the button labeled **Add New (Modify)** Add New [Modify].

c. A modified dialog box will appear with all the data fields filled (copied from the previously selected urban land type), except the urban land type name (Figure 14.78).

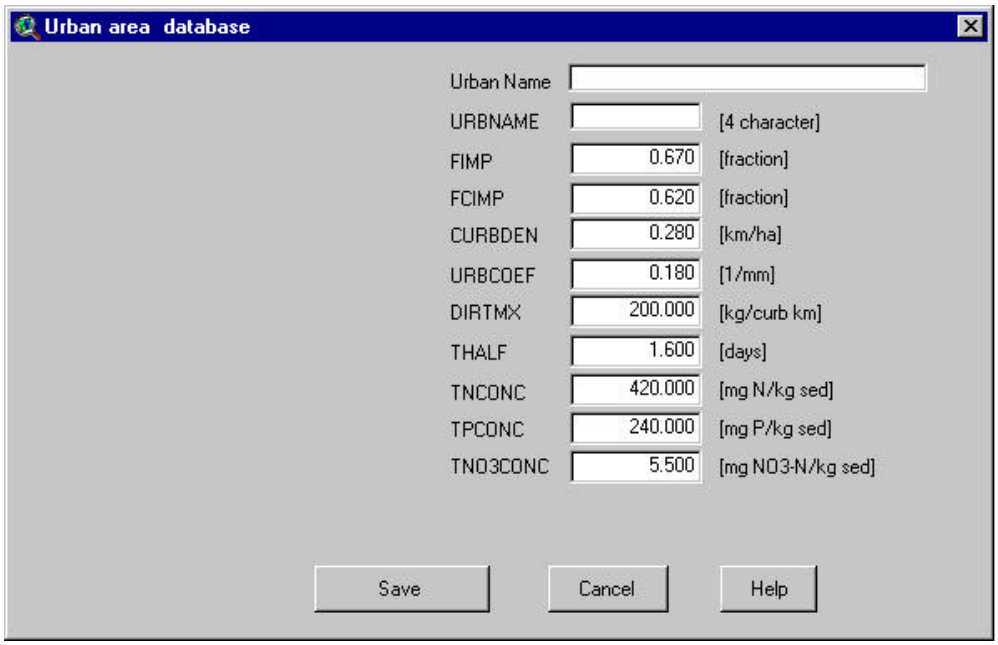


Figure 14.78

d. Enter a new urban land type name and modify the other data fields.

e. Click **Save** to store the modified urban data, or click **Cancel** to exit the editor without saving.

f. The CROP: Hydrological Parameters dialog box is displayed when data for a new urban land type is saved (Figure 14.79)

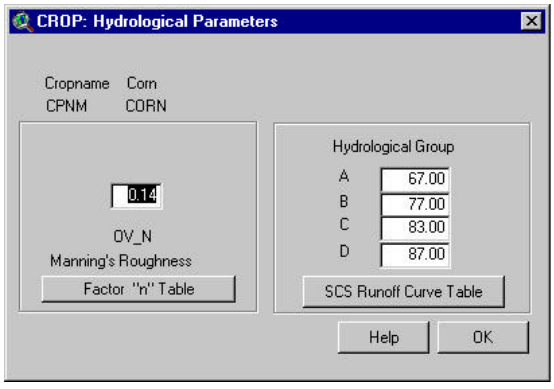


Figure 14.79

The user must define a default Manning's overland n value and default curve numbers that will be used by the interface to initialize these parameters when setting up projects where the new urban land type is present in the watershed.

Values may be typed into the text boxes, or the user can access tables to decide what value should be assigned.

To access the SCS Runoff Curve Table, click the **SCS Runoff Curve Table** SCS Runoff Curve Table button. The table will be displayed (Figure 14.80).

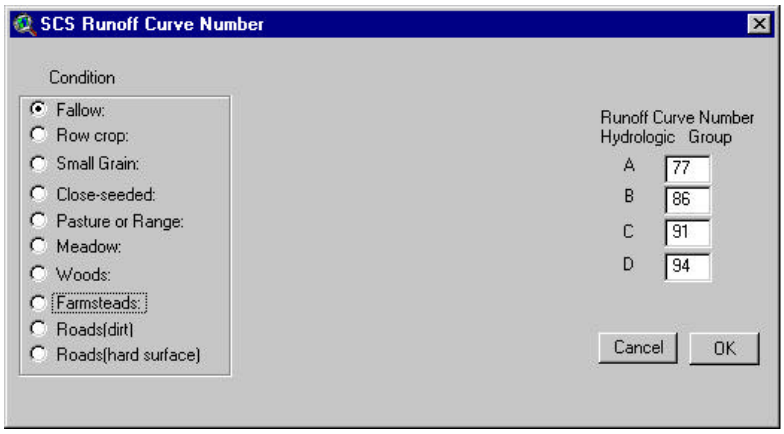


Figure 14.80

Curve number values for different management conditions are accessed by clicking the radio buttons for the condition and, if applicable, the cover. To exit the table and copy the displayed curve numbers, click **OK**. To exit the table without copying the displayed curve numbers, click **Cancel**.

To access the Manning's n Table, click the **Factor 'n' Table** Factor "n" Table button. The table will be displayed (Figure 14.81).

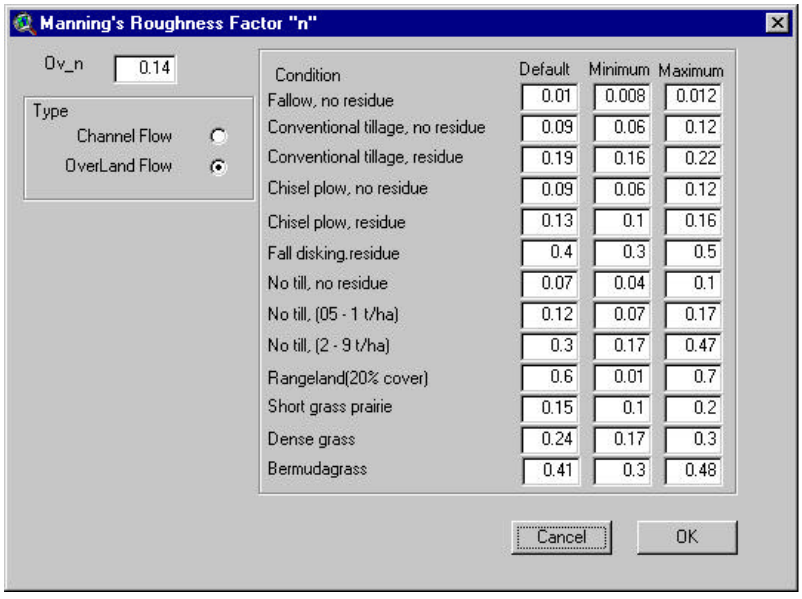


Figure 14.81

To use a value from the table, click on the desired value and then click **OK**. To exit the table without copying a value, click **Cancel**.

Once the default values for curve number and Manning's n value for overland flow are set correctly, click **OK**.

g. The user is returned to the original Urban Database dialog box. The list of urban land types in the database will include the name of the urban land type just added.

To delete a user defined urban land type:

a. From the Urban Database dialog box, click the name of the urban land type to be deleted.

b. The dialog box will be modified to display the urban data. Click the button labeled **Delete** at the bottom of the dialog box.

c. A prompt box will appear requesting the user to verify the delete (Figure 14.82).

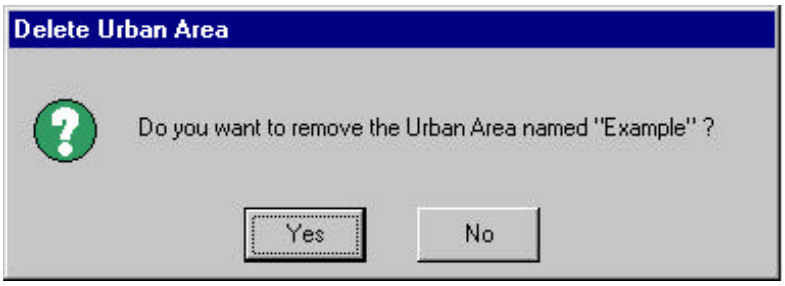


Figure 14.82

d. To continue with the delete process, click **Yes**. To abandon the delete process, click **No**.

Note:	Only the new Urban data sets entered by the user can be removed from the database. The user cannot remove the data sets that are included with the interface but can modify the parameters and/or restore the default values.

To exit the Urban Database:

a. From the Urban Database dialog box (Figure 42), click **Exit**

.

SECTION 15: THE EXAMPLE DATA SET

Data for the Lake Fork Watershed in Northeast Texas has been included in the installation package as a demonstration data set. The example data set is stored in the directory :*Installation dir*\AvSwatDB\Example1, which can be found on the drive that the SWAT ArcView interface is installed (Figure 15.1).

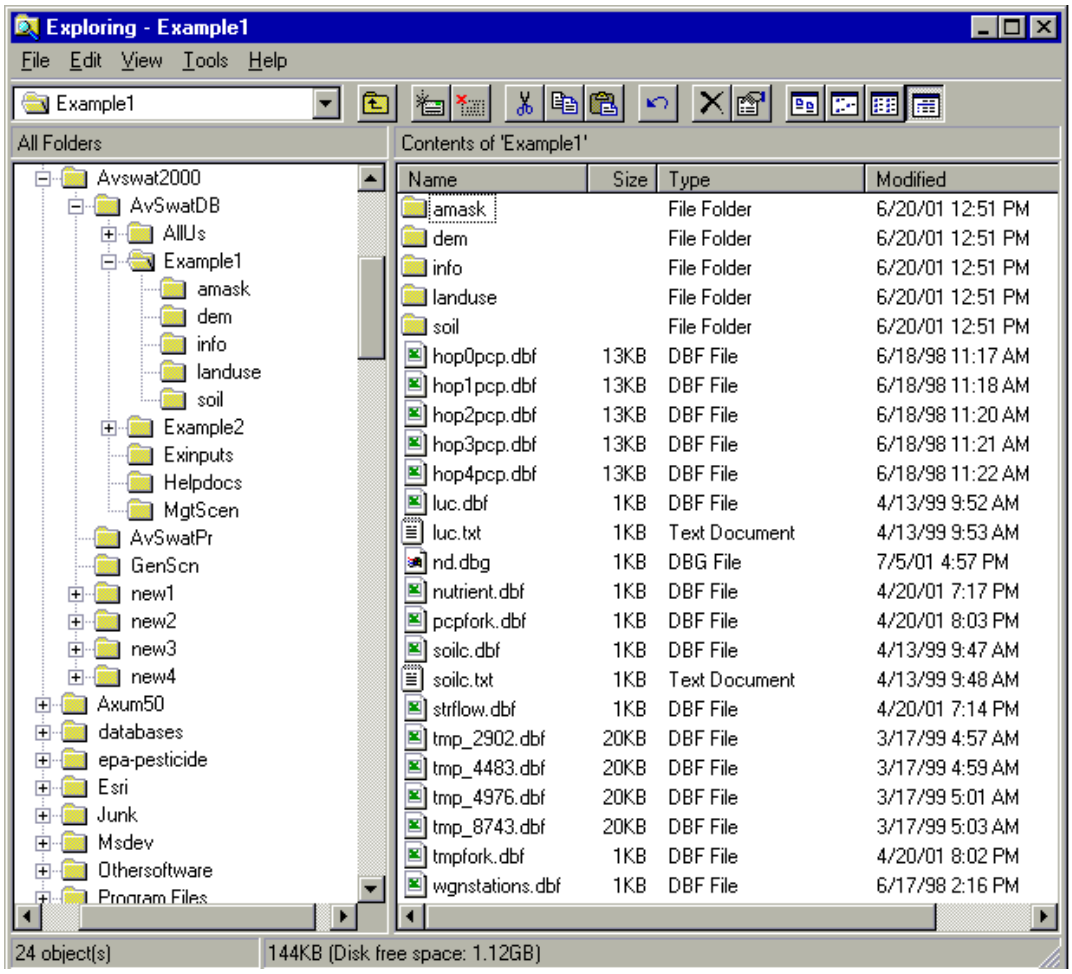


Figure 15.1

The example data set includes 4 ArcView grid themes, 16 DBF tables, and two text files. The 4 ArcView grid themes are:

- ***dem***: A Digital Elevation Model (DEM) ArcView grid for the Lake Fork Watershed. The map was created in the Albers Equal Area projection with the resolution in meters and the elevation in meters.
- ***amask***: A DEM Mask ArcView grid. The map was created in the Albers Equal Area projection with the resolution in meters.

- ***landuse***: A Land Cover/Land Use ArcView grid for the Lake Fork Watershed. The map was created in the Albers Equal Area projection with the resolution in meters.
- ***soil***: A Soil ArcView grid for the Lake Fork Watershed. The map was created in the Albers Equal Area projection with the resolution in meters. The Soil grid is a STATSGO soil map.

The DBF tables and text files are:

- Location table for USGS stream flow gages: ***strflow.dbf***
- Location table for in-stream nutrient monitoring points: ***nutrient.dbf***
- Location table for rain gages: ***pcpfork.dbf***
- Precipitation data tables: ***pcp_8743.dbf***, ***pcp_2902.dbf***, ***pcp_9836.dbf***, ***pcp_4483.dbf***, ***pcp_4976.dbf***
- Location table for temperature gages: ***tmpfork.dbf***
- Temperature data tables: ***tmp_2902.dbf***, ***tmp_4483.dbf***, ***tmp_4976.dbf***, ***tmp_8743.dbf***
- Location table for weather stations whose data was used to create custom weather generator data sets: ***wgnstations.dbf***
- Land Use look up table: ***luc.dbf***
- Land Use look up file: ***luc.txt***
- Soil look up table, STMUID option: ***soilc.dbf***
- Soil look up file, STMUID option: ***soilc.txt***

For more information on the different types of tables and maps required to run the interface, please see Section 3.

SECTION 15.1: CREATE SWAT RUN WITH EXAMPLE DATASET

1. Start ArcView 3.2 by double-clicking the icon. If an icon for the program is not present, click the **Start** button, then highlight **Programs**. From the software list displayed, highlight **ESRI**. The program name **ArcView 3.2** will be displayed. Click the program name to start ArcView.
2. On the **File** menu, click **Extensions**.
3. Scroll the list of available extensions until you locate **AVSWAT2000**. Check the box beside **AVSWAT2000** and click **OK**.

4. The main interface will be displayed (Figure 15.2).

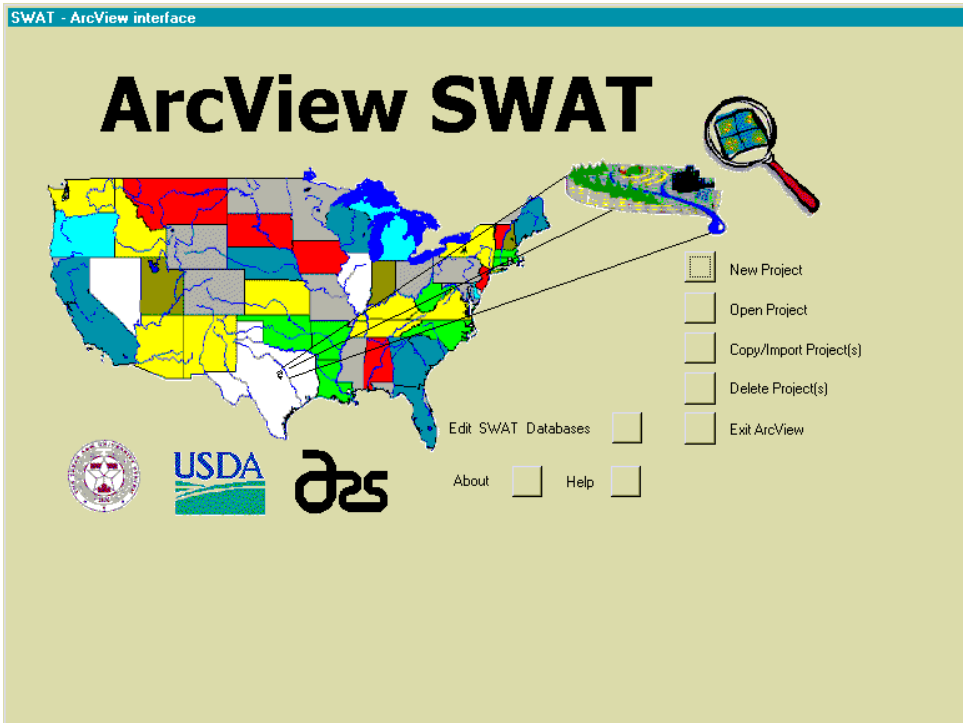


Figure 15.2

5. Click the box beside **New Project**.

6. A browser will be displayed requesting a name for the new project. Type ***lakefork*** in the text box labeled **File Name** (Figure 15.3).

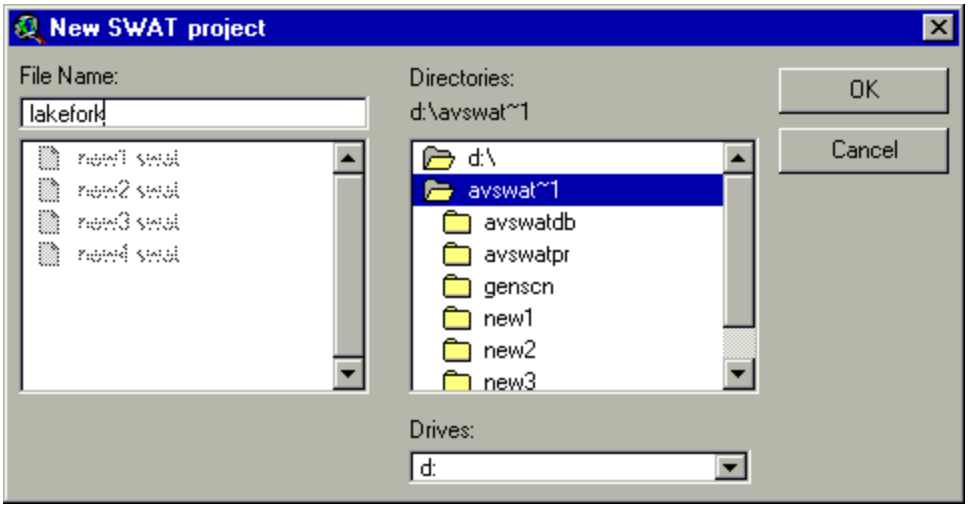


Figure 15.3

7. Click **OK**.

Note: Once **OK** is clicked, the interface creates a subdirectory called ***lakefork*** within the active directory

displayed in the directory tree on the dialog box. This directory is used to store maps and database tables created by the interface as well the input files for SWAT.

8. After the project name is specified, the interface brings up the Environmental Variables prompt box (Figure 15.4). This prompt box lists the directories in which the interface searches for the information needed to create the SWAT input files.

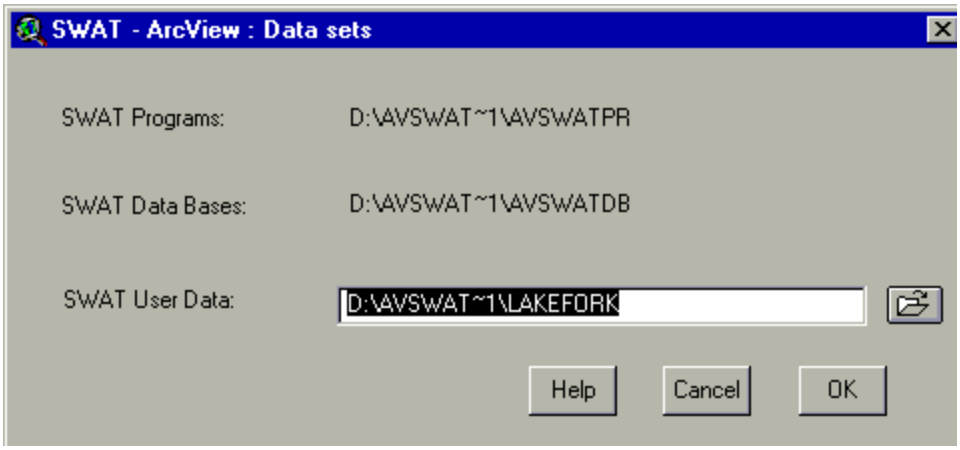


Figure 15.4

Programs used by the interface are stored in the directory listed next to **SWAT Programs**. This directory is defined when the interface is installed.

The directory listed next to **SWAT Data Bases** contains all the database (.dbf) tables used by the interface to set default input values and define the upper and lower limits for variable values. This directory also contains the soil and weather generator databases included with the interface. As with the previous directory, this directory is defined when the interface is installed.

The third directory is the **SWAT User Data** directory. When the interface brings up the Environmental Variables prompt box, the project directory created by the interface is listed in the text box (the interface *output* directory). This needs to be changed to the directory where the DBF tables containing the measured precipitation and temperature data are stored.

9. The maps and database tables required for the example project are stored in :/*Installation dir*/AvSwatDB/Example1. The name in the text

box may be changed by: 1) typing the directory pathway in the text box, or 2) searching for the correct directory with a browser.

To activate a browser:

a. Click the button to the right of the text box.

b. A directory browser will appear with the directory listed in the text box visible (Figure 15.5).

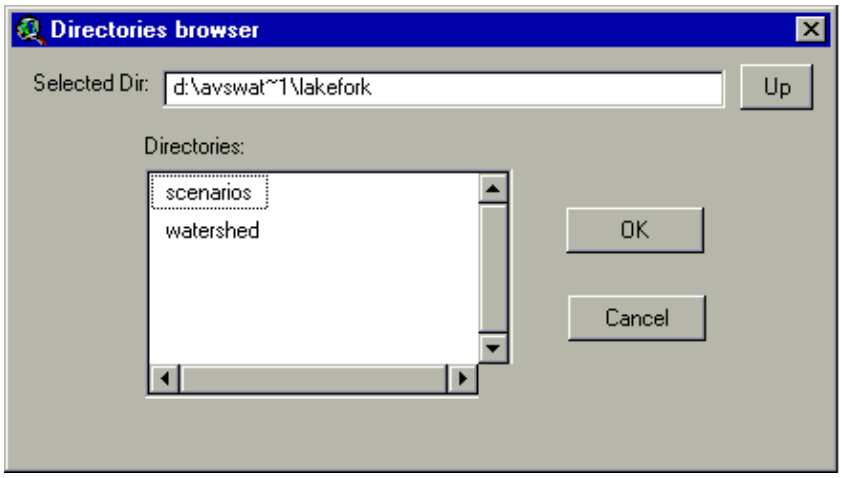


Figure 15.5

c. Click the button labeled **Up**. This will make the *installation dir* directory active. You will see at least three directories listed: *avswatdb, avswatpr,* and *lakefork*. Select *avswatdb* by clicking on the name in the list of directories. When the *avswatdb* directory is active, several subdirectories will be listed: *allus*, *example1*, *example2*, *helpdoc*, and *exinputs*. Select *example1* by clicking on the name in the list of directories. The Directory browser will now look like Figure 15.6.

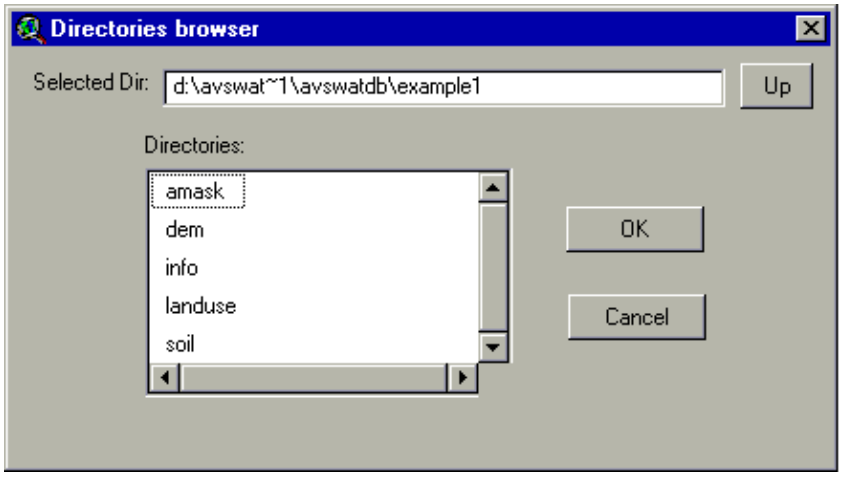


Figure 15.6

d. Once the proper directory is listed in the text box next to **Selected Dir**, click the button labeled **OK**.

10. The **SWAT User Data** directory in the prompt box will show the directory chosen with the browser (Figure 15.7).

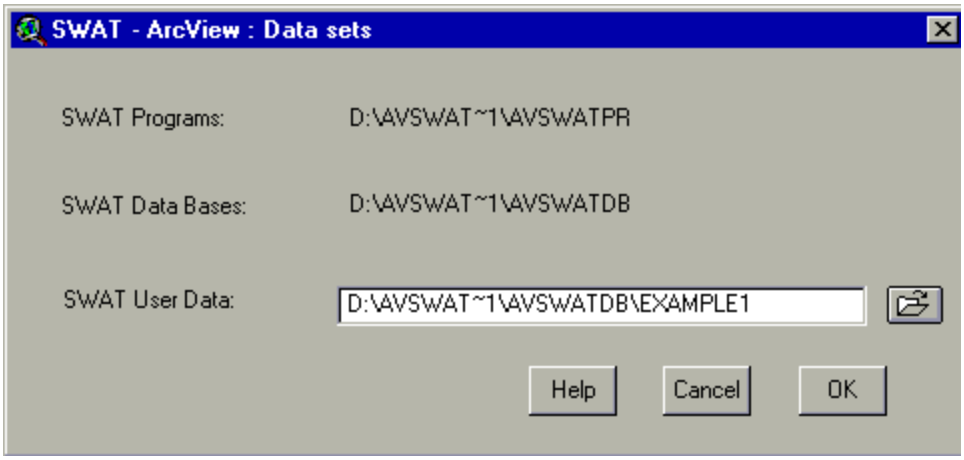


Figure 15.7

11. Click **OK** to confirm the choice.

> Note: To access the Environmental Variables prompt box at any time, select **Avswat Main Interface Dialog** from the **Avswat** menu to bring up the Main Interface screen, and then click the button next to **Environmental Variables**.

12. Once the User Data directory is defined, the interface will display the SWAT Project Window and automatically activate the Watershed Delineation dialog box (Figure 15.8).

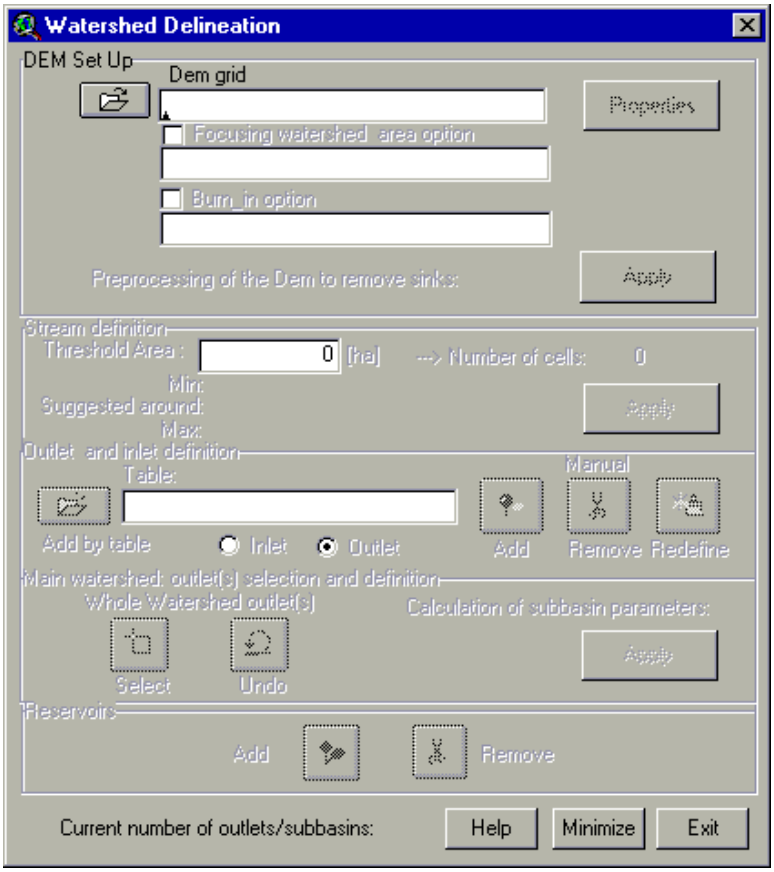


Figure 15.8

SECTION 15.1.1: PROCESSING THE ELEVATION MAP GRID

1. To load the example DEM, click beside the **DEM grid** text box.
2. A prompt box is opened (Figure 15.9)

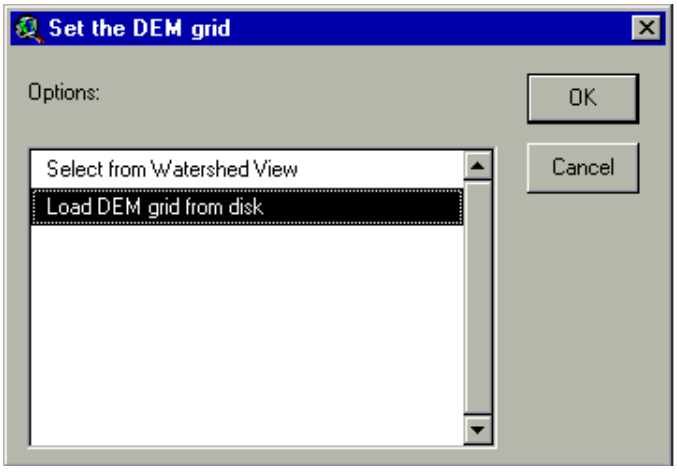


Figure 15.9

Highlight **Load DEM grid from disk** and click **OK**.

3. A grid dataset file browser will appear with the User Data directory active (Figure 15.10).

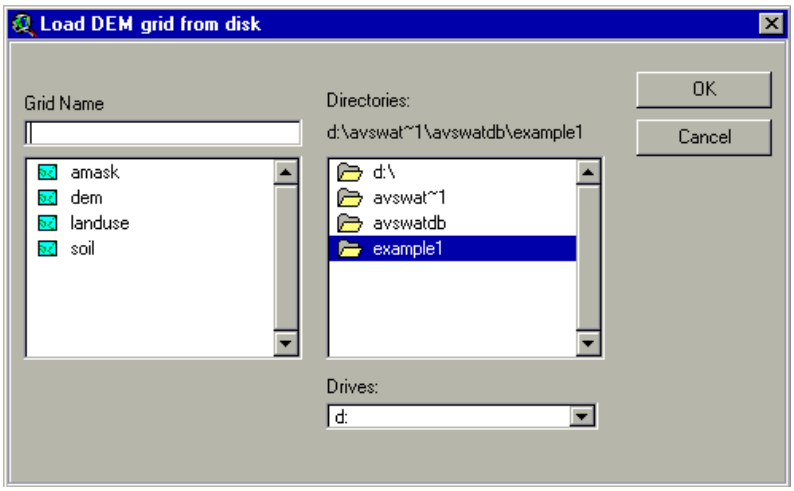


Figure 15.10

4. Click the name of the elevation map grid (*dem*). The name of the elevation map grid will then be displayed in the text box below **Grid Name** on the browser. Click **OK** to confirm the choice.

5. The name of the elevation map grid will be displayed in the **DEM grid** text box on the Watershed dialog box, and the elevation map will be displayed (Figure 15.11).

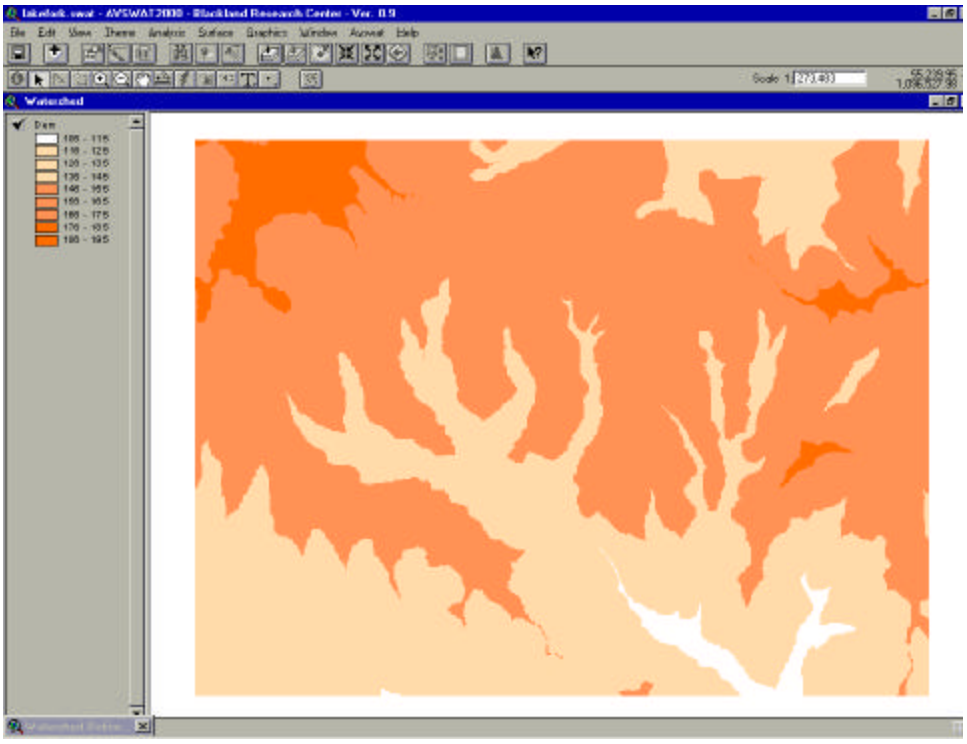


Figure 15.11

6. A prompt box will appear reminding the user to verify DEM properties (Figure 15.12).

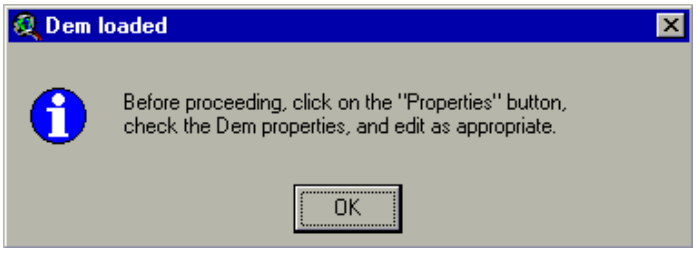


Figure 15.12

Click **OK**.

7. Click the properties button

next to the DEM grid text box. The interface will activate the map Properties prompt box (Figure 15.13).

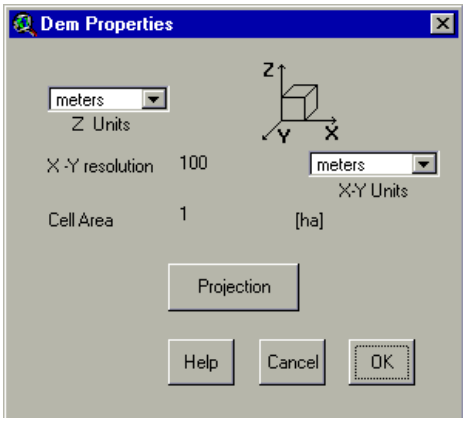


Figure 15.13

8. The units for the X-Y and Z resolutions should be set to meters. To define the projection, click the projection button

. A prompt box will appear listing projection details for the DEM (Figure 15.14).

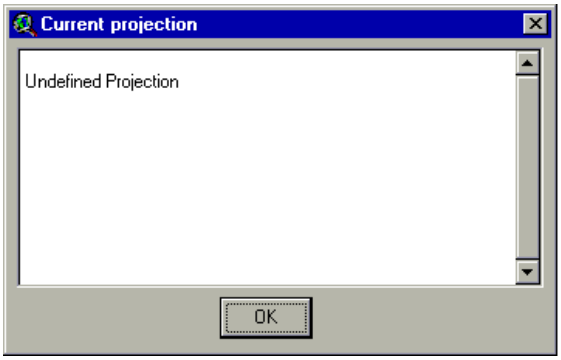


Figure 15.14

Click **OK**.

9. A prompt box will appear asking if the user wishes to modify the projection information. Click **Yes**.

10. The interface will activate the map projection prompt box (Figure 15.15).

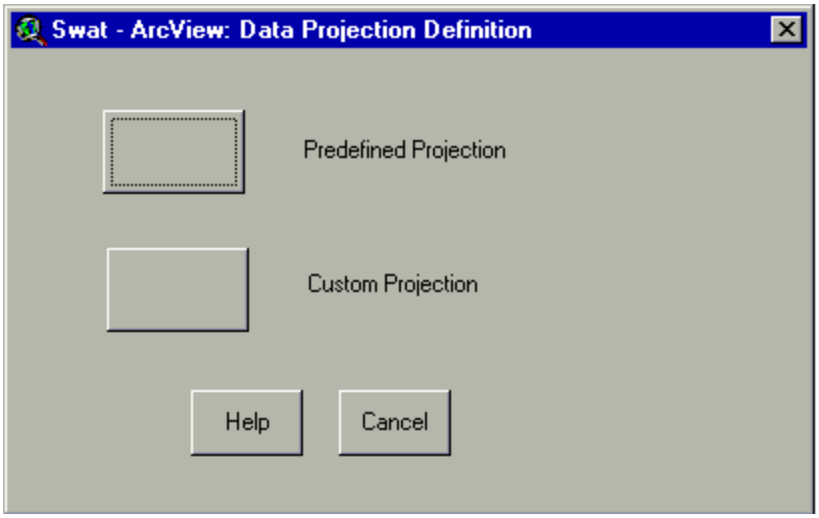


Figure 15.15

The maps in the example data set were created in the Albers Equal Area projection, which is a Predefined Projection in the interface. Click the button to the left of **Predefined Projection** on the map projection prompt box. A list of predefined projections will be displayed. Highlight *Albers Equal-Area (Conterminous U.S.)* by clicking on the name. Click **OK** to select this projection, and then click **OK** on the DEM Properties dialog box.

The map resolution units and projection are properties of the map that are set when the map is created. While the interface can utilize maps in any projection, all maps used for a project must all be in the same projection.

11. **Optional:** At this point the user may load a masking map grid and/or perform a burn-in of the stream network with a shape file containing the stream delineation for the watershed. A burn-in is useful in watersheds with very little relief (e.g. delta regions) or where the elevation map is not detailed enough to accurately predict the stream network. The example data set contains a masking map grid.

To load the masking map grid:

a. On the Watershed Delineation dialog box, check the box next to **Focusing watershed area option**. A check will appear in the box

and a prompt box will open Figure 15.16).

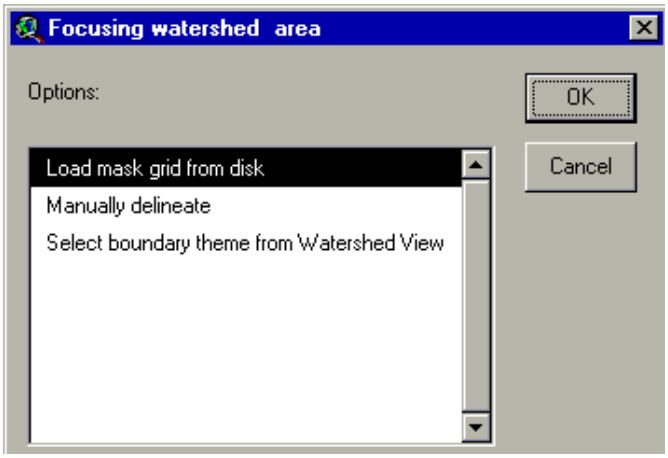


Figure 15.16

Highlight **Load mask grid from disk** and click **OK**.

b. A browser is displayed. Click the name of the masking map grid, *amask*, and then click **OK**.

c. The masking map grid will be displayed on the screen (Figure 15.17).

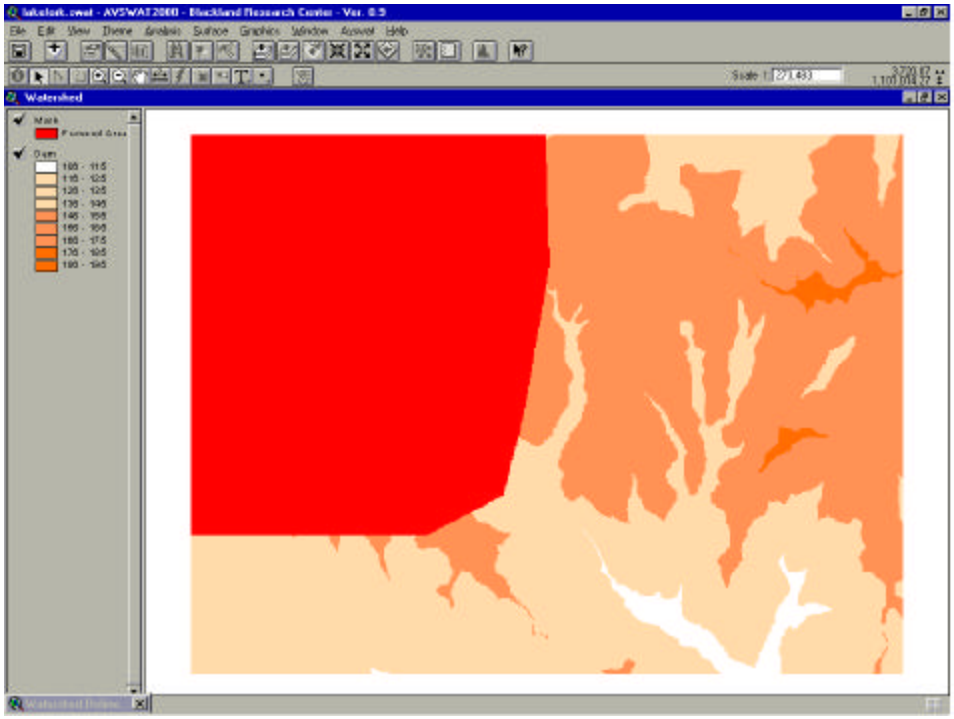
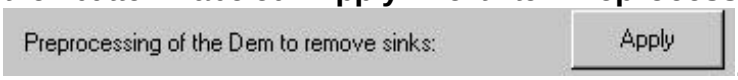


Figure 15.17

When a masking map grid is displayed, the stream network will be delineated only for the area of the DEM covered by the masking map grid.

12. Once the elevation map is displayed, the map must be preprocessed. The preprocessing feature 'smooths" the elevation grid by filling in areas of the map that drain to a point rather than drain to a channel. Preprocessing speeds up the amount of time it takes for the interface to define the channel network. To preprocess the elevation map, click the button labeled **Apply** next to **Preprocessing of the DEM**



.

While processing the DEM, a prompt box will appear (Figure 15.18).

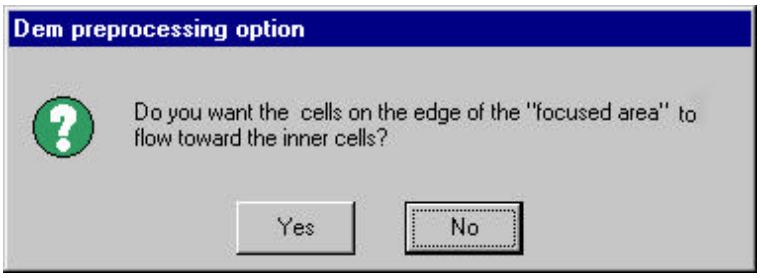


Figure 15.18

Click **No**.

13. A prompt box will appear after the map preprocessing is complete. Click **OK**.

14. Once the elevation map has been preprocessed, the threshold area used to define the origin of a stream needs to be specified. The smaller the number, the more detailed the stream network generated by the interface. Figure 15.19 shows the stream network generated with the threshold set to 100 ha, while Figure 15.20 shows the stream network generated with the threshold set to 1000 ha.

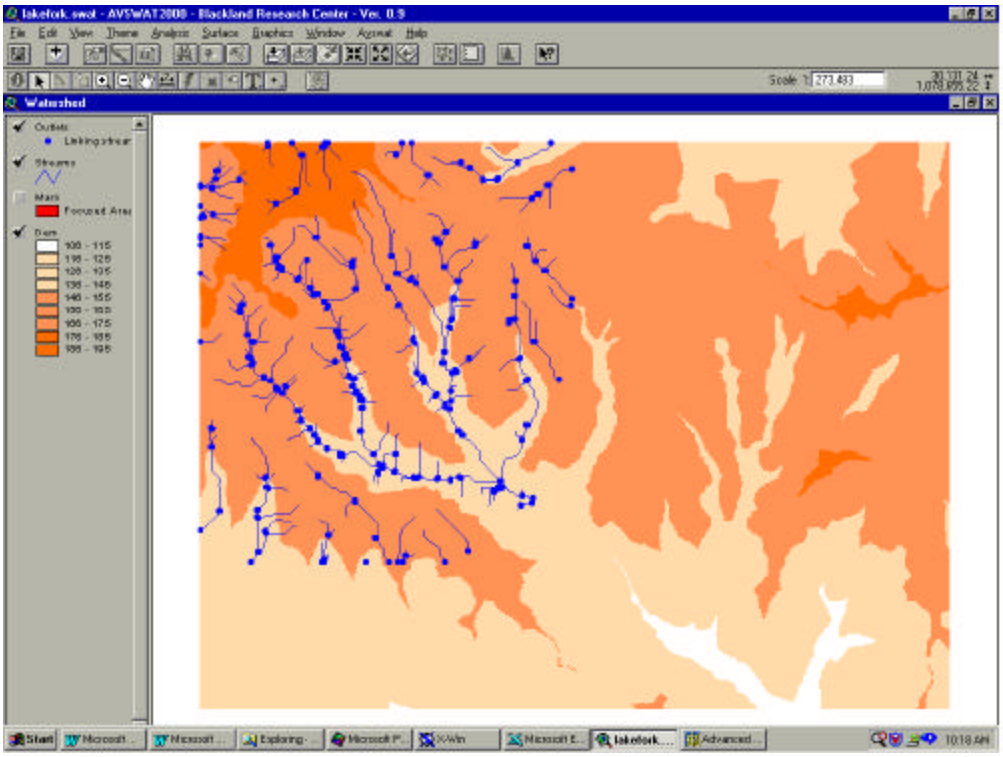


Figure 15.19

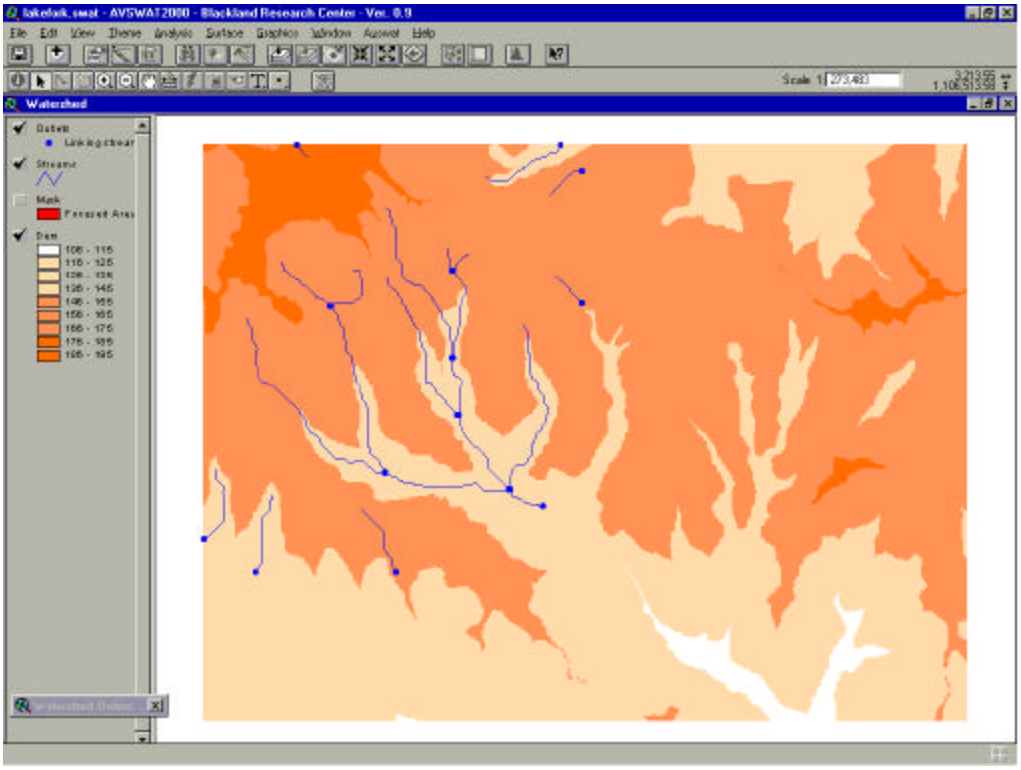


Figure 15.20

For the example project, set the threshold area to *1000*. Once the proper area is displayed in the text field next to **Threshold Area**, click **Apply**.

15. The stream network will be displayed upon completion of the calculations (Figure 15.20). Subbasin outlets defined by the junction of two streams are denoted on the network by blue dots.

> The user may modify the number of subbasin outlets manually or by importing an Avenue database (.dbf) table containing outlet location coordinates. Points added via the table or manually will be snapped to the closest point on the delineated stream channels.

16. A table of locations where nutrient data was collected has been included in the example data set. To load the table, first verify that the **Outlet** radio button is selected

. Then click next to the text field labeled **Table** on the Watershed Delineation dialog box.

17. A browser will be displayed (Figure 15.21).

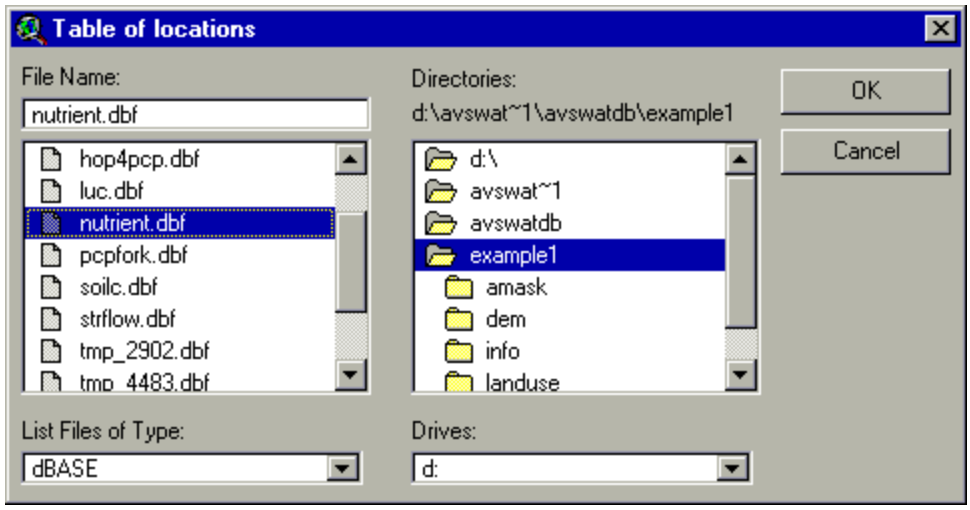


Figure 15.21

Select *nutrient.dbf* from the list of tables and click **OK**. The subbasin outlet locations loaded from the table are displayed as white dots (Figure 15.22).

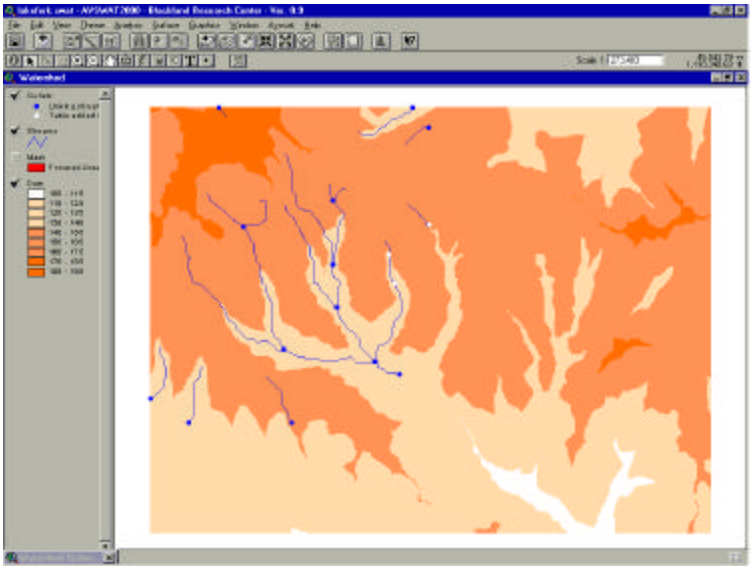


Figure 15.22

18. To manually add subbasin outlets, first verify that the **Outlet** radio button is selected

. Then click the button labeled **Add**

.

19. The dialog box will be minimized. Use the mouse to move around the map, and click with the left mouse button to place a subbasin outlet where the mouse is positioned. Subbasin outlets added manually will be displayed as red dots. Add four outlets so that the map looks similar to Figure 15.23.

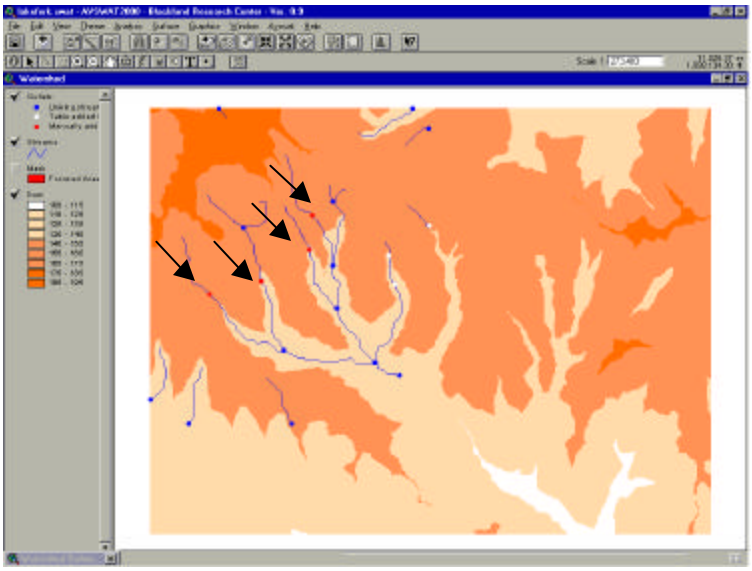


Figure 15.23

20. Once the display of subbasin outlets is satisfactory, the watershed outlet must be selected. Click the button above **Select**

. The dialog box will be minimized and a prompt box will appear (Figure 15.24).

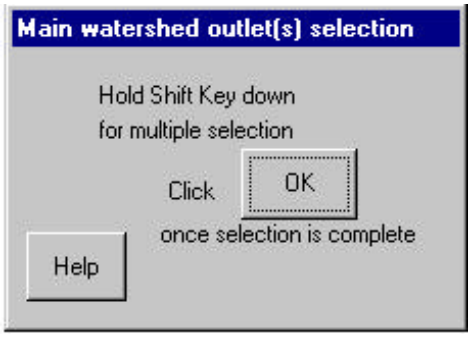


Figure 15.24

21. Select the subbasin outlet located on the lower right (Figure 15.25) to be the subbasin outlet by holding down the left mouse button and moving the mouse to form a box around the outlet dot. The outlet dot will turn yellow when it is selected.

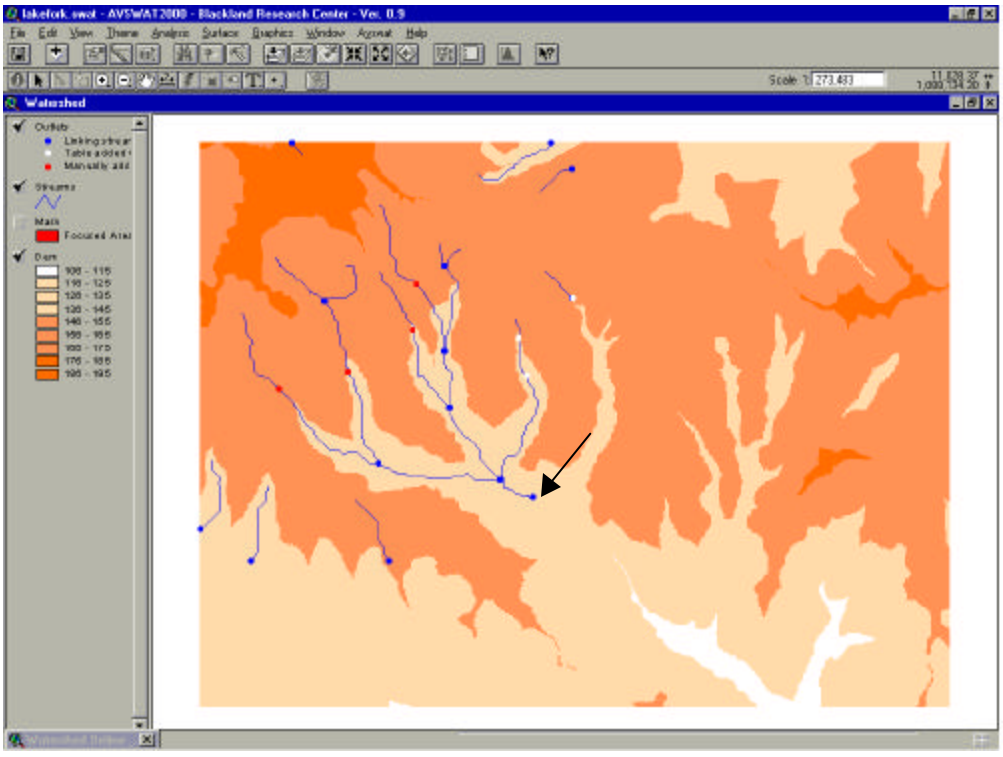


Figure 15.25

22. Click **OK** on the prompt box once the correct outlet is selected. Another prompt box will appear to verify the outlet choice. Click **Yes** to continue with the processing.

23. The subbasin delineation for the watershed will be displayed (Figure 15.26).

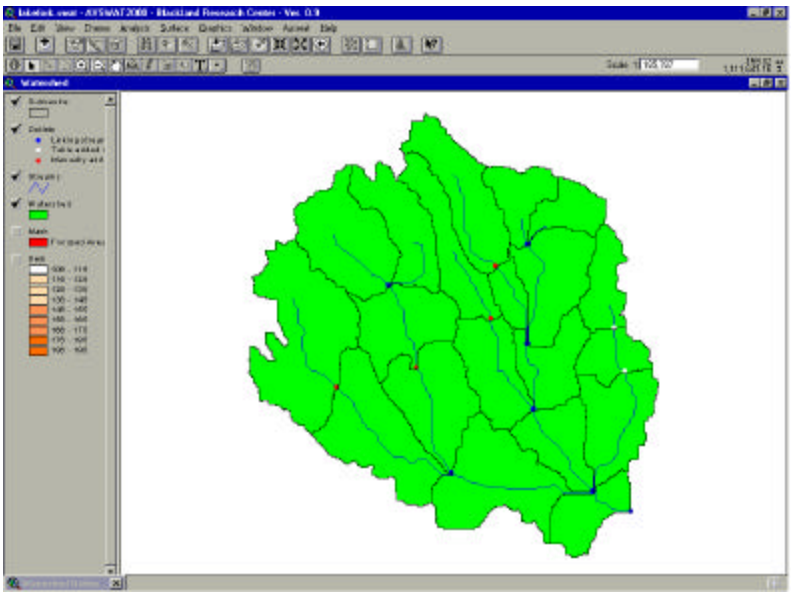
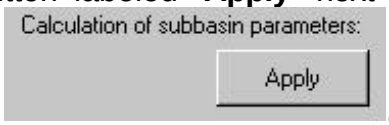


Figure 15.26

24. Click the button labeled **Apply** next to **Calculation of subbasin parameters** 

.

25. Once the calculation of subbasin parameters is complete, a prompt box will appear. Click **OK**.

SECTION 15.1.2: PROCESSING THE LAND USE/SOIL MAP GRIDS

1. Select **Land Use and Soil Definition** in the **Avswat** menu (Figure 15.27).

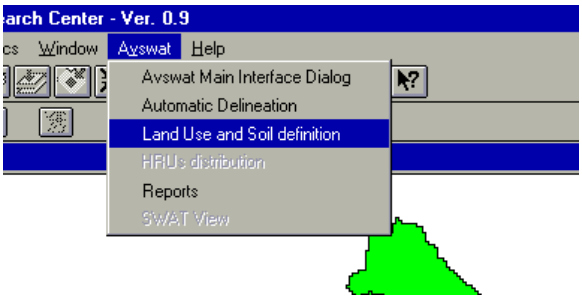


Figure 15.27

2. The Definition of Landuse and Soil themes dialog box will open (Figure 15.28).

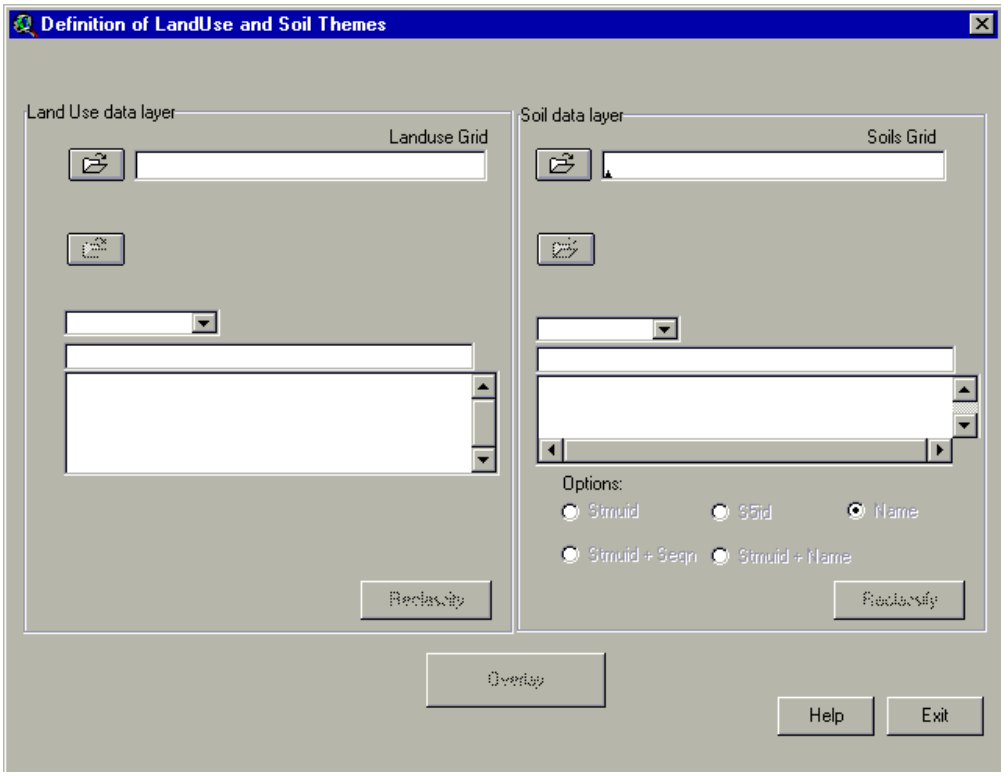


Figure 15.28

3. To load the example land use grid, click beside the **Landuse Grid** text field.

4. A prompt box will appear (Figure 15.29).

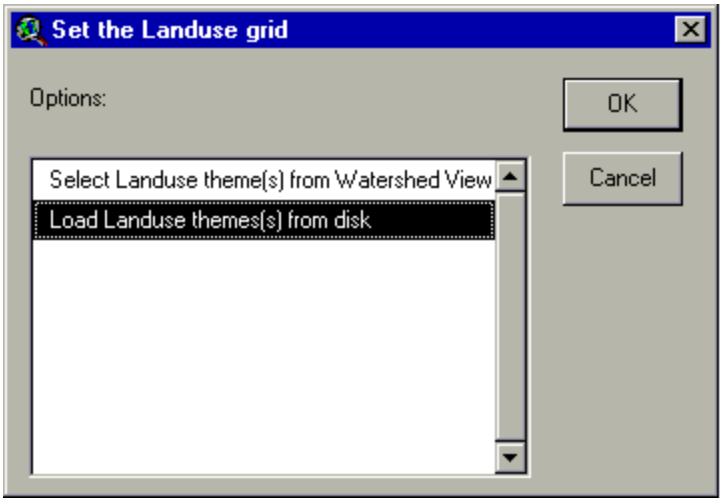


Figure 15.29

Highlight **Load Landuse theme(s) from disk** and click **OK**.

5. Another prompt box will appear for the user to define the map format. Select **Grid** and click **OK**.

6. A message box will appear reminding the user that the data must be projected. Click **Yes**.

7. A browser will appear with the User Data directory active. Click the name of the land use map grid (*landuse*). Click **OK** to confirm the choice.

8. The raw land use grid will be displayed and clipped to the watershed area (Figure 15.30).

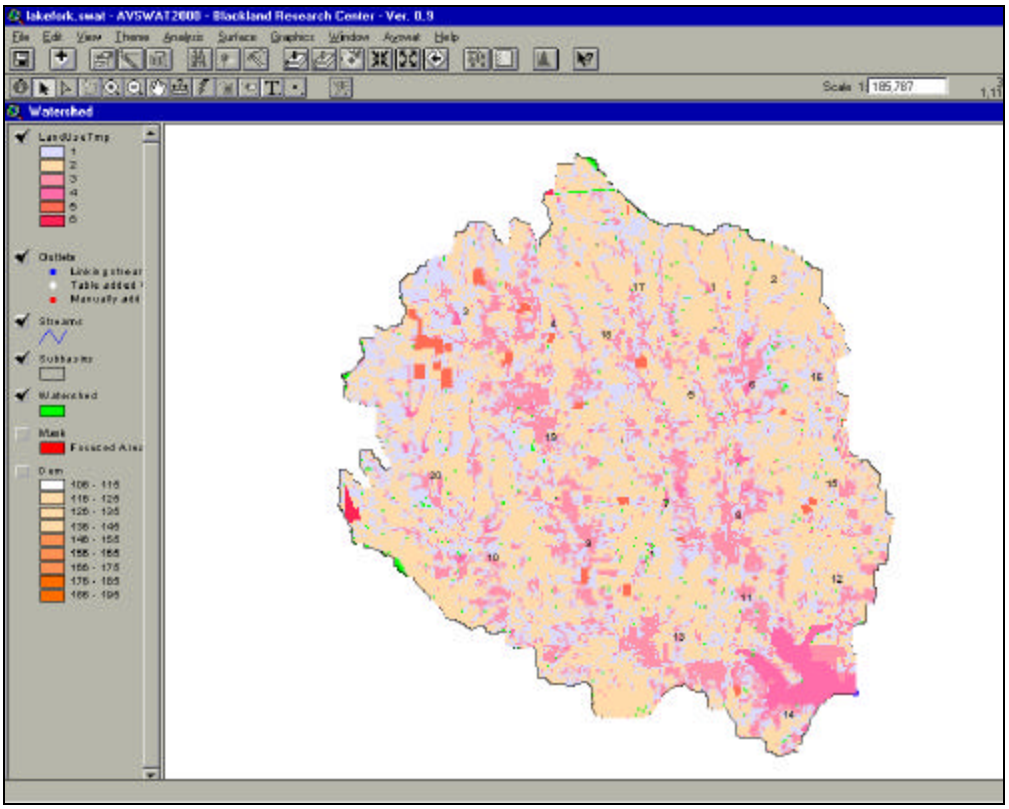


Figure 15.30

A message box will also appear reminding the user to load the look-up table for the map. Click **OK**.

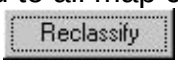
9. When the land use map grid is loaded, the interface does not know which SWAT land use code to assign to the different categories. Three options for loading this information are described in Section 6.

   The example data set includes a custom look up table to define the SWAT land use to be modeled for each category. Click beside **LookupTable Grid Values→ Land cover classes**.

10. A prompt box will be displayed for the user to select the type of table to be loaded. Highlight **User table** and click **OK**.

11. Another prompt box will appear for the user to define the format of the look up table. Select **.dbf file** and click **OK**.

12. A browser will be displayed. Click the name of the look up table (*luc.dbf*). Once the correct table is selected, click **OK**.

13. The SWAT land use categories will be displayed on the scrollable listing on the Land Use/Soil dialog box. Once a *LandUseSwat* code has been assigned to all map categories, the **Reclassify** button will be enabled. Click the 

button.

14. The category display for the map will show the SWAT land use codes (Figure 15.31).

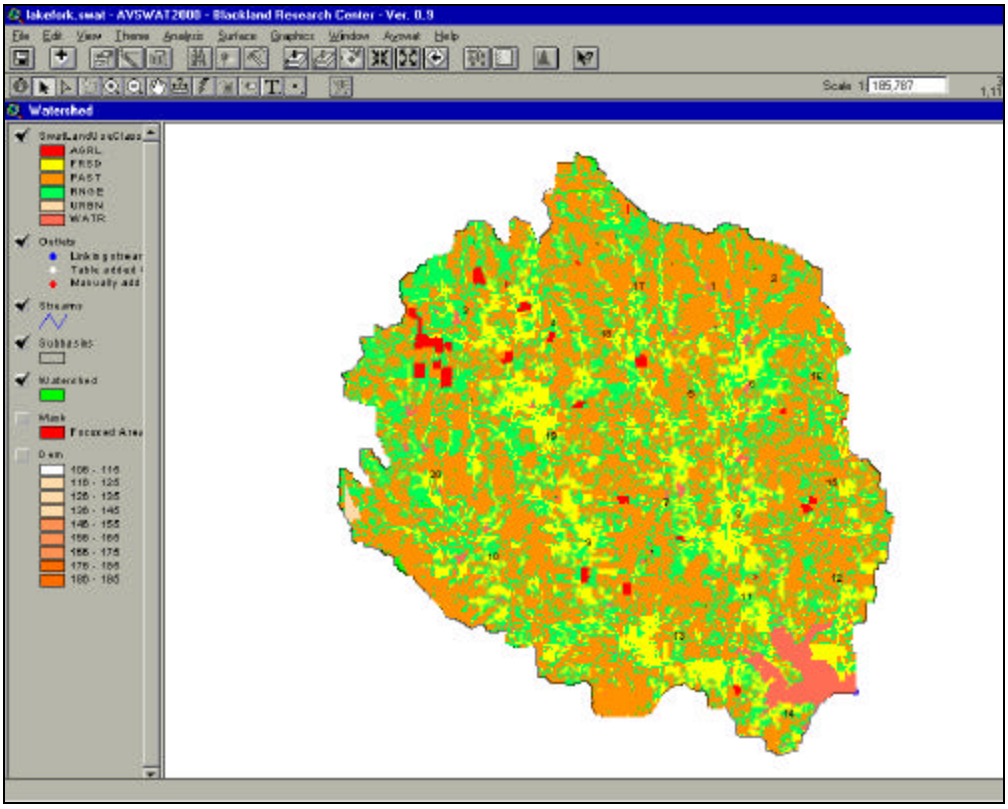


Figure 15.31

15. To load the example soil grid, click beside the **Soil Grid** text field.

16. A prompt box will appear (Figure 15.32).

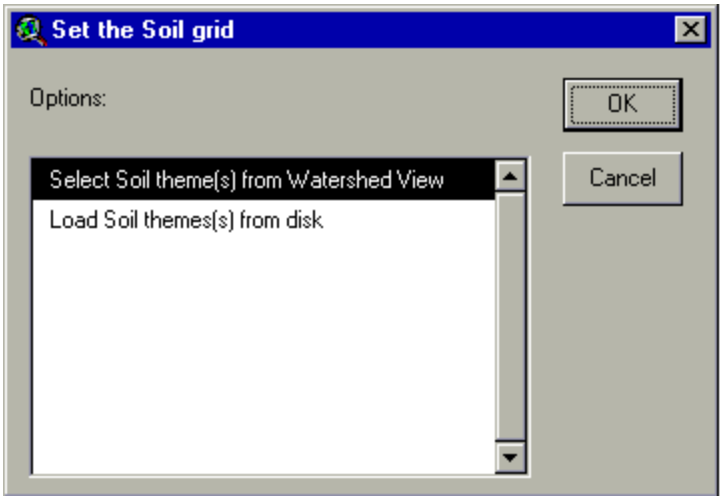


Figure 15.32

Highlight **Load Soil theme(s) from disk** and click **OK**.

17. Another prompt box will appear for the user to define the map format. Select **Grid** and click **OK**.

18. A message box will appear reminding the user that the data must be projected. Click **Yes**.

19. A browser will appear with the User Data directory active. Click the name of the soil map grid (*soil*). Click **OK** to confirm the choice.

20. The raw soil grid will be displayed and clipped to the watershed area (Figure 15.33).

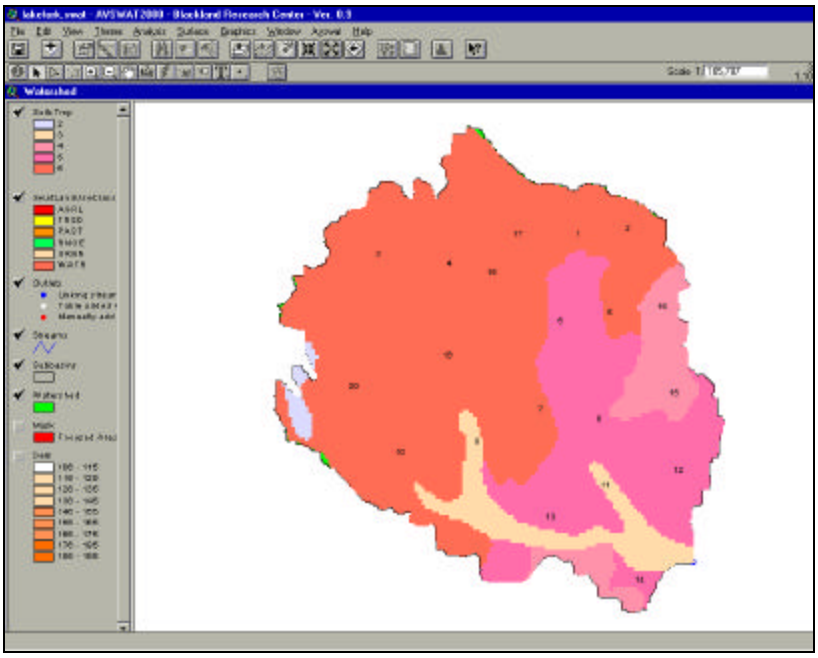
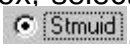


Figure 15.33

A message box will also appear reminding the user to load the look-up table for the map. Click **OK**.

21. Five options for linking the soil map grid to the soil database are described in Section 6. The example data set is set up to link via STATSGO polygon numbers. On the Land Use/Soil dialog box, select the **Stmuid** option for linking the soil grid to the soil database 

. Then click beside **Lookup Table Grid Values→Soils attributes**.

22. A prompt box will appear for the user to define the look up table type. Select **.dbf file** and click **OK**.

23. A browser will be displayed. Click the name of the look up table (*soilc.dbf*). Once the correct table is selected, click **OK**.

24. The soil linkage information will be displayed in the scrollable listing on the Land Use/Soil dialog box). Once an *Stmuid* code has been assigned to all map categories, the **Reclassify** button will be enabled. Click the

button.

25. The category display for the map will show the soil codes (Figure 15.34).

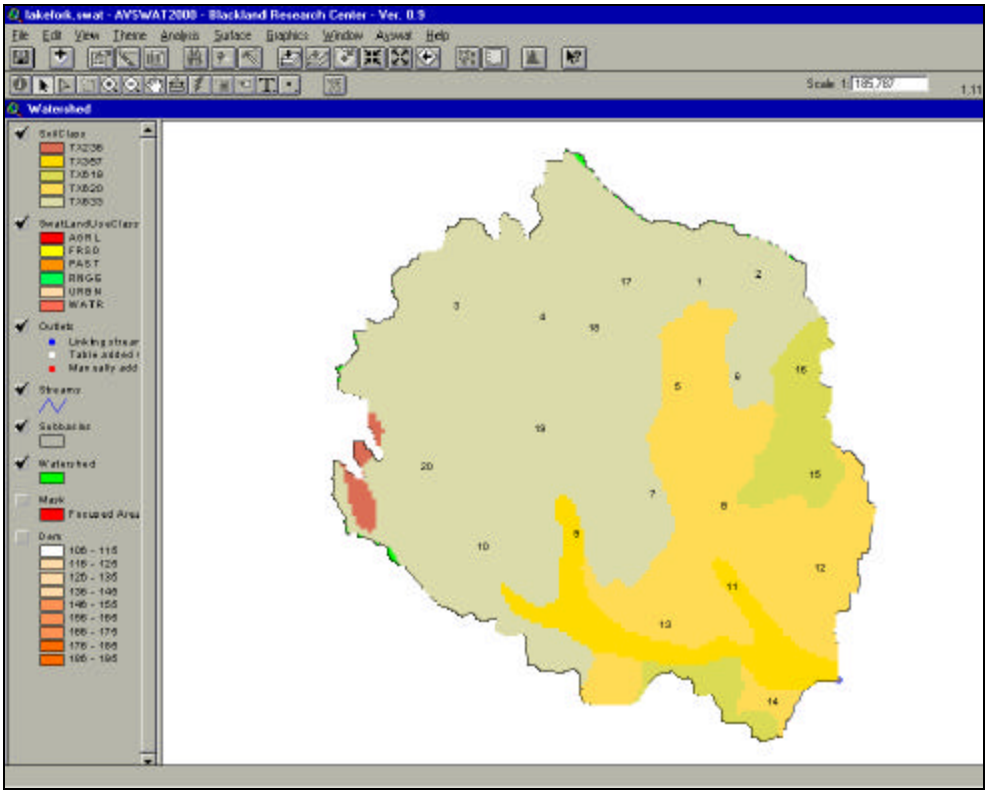


Figure 15.34

26. Once the land use and soil map grids have been loaded and reclassified, click the button labeled **Overlay** at the bottom of the Land Use/Soil dialog box.

27. When the overlay of the land use and soil map grids is complete, a prompt box will notify the user that the overlay process is complete. Click **OK**.

28. A report is generated during the overlay process. To access the report, select **Reports** under the **Avswat** menu. From the list of reports, select *SWAT model: LandUse and Soil Distribution* and click **OK**.

29. Close the report after viewing.

SECTION 15.1.3: LAND USE/SOIL DISTRIBUTION

1. Select **HRU distribution** from the **Avswat** menu (Figure 15.35).

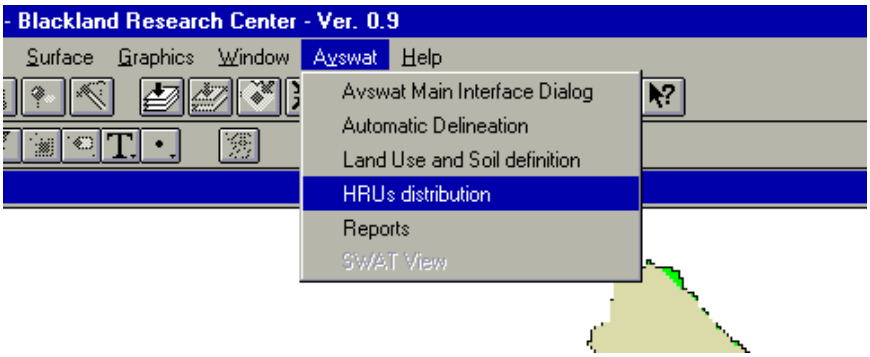


Figure 15.35

2. The Land Use/Soil Distribution dialog box will be displayed (Figure 15.36).

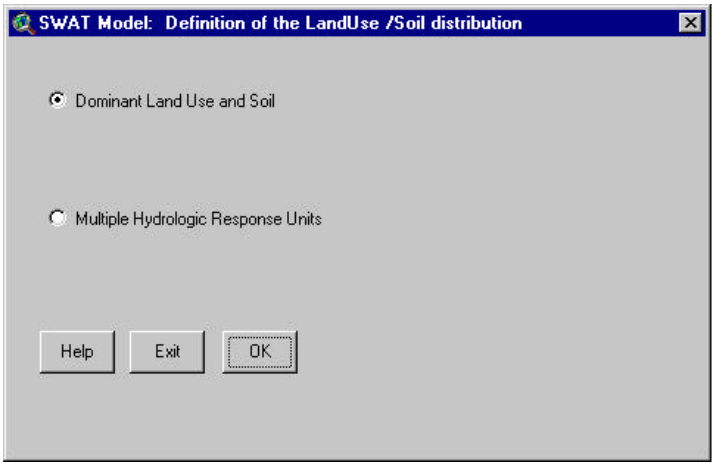


Figure 15.36

Select **Dominant Land Use and Soil** and click **OK**.

3. A message box will be displayed notifying the user when setup of HRUs is completed. Click **OK**.

4. A second message box is displayed notifying the user that the SWAT View is now active. Click **OK**. The interface will automatically switch to the SWAT View.

5. A report is generated during the HRU creation process. To access the report, select **Show List** under the **Reports** menu. From the list of reports, select *SWAT model: LandUse and Soil Distribution (after threshold application)* and click **OK**. The total number of HRUs created in the watershed is listed in the top section of the report in bold letters. The remainder of the report lists the land use and soil modeled in every subbasin and the percent area distribution of 1) subbasins within the watershed and 2) HRUs within the subbasins.

6. Close the report after viewing.

SECTION 15.1.4: WEATHER STATIONS

1. To load the example weather data, click **Weather Stations** under the **Input** menu (Figure 15.37).

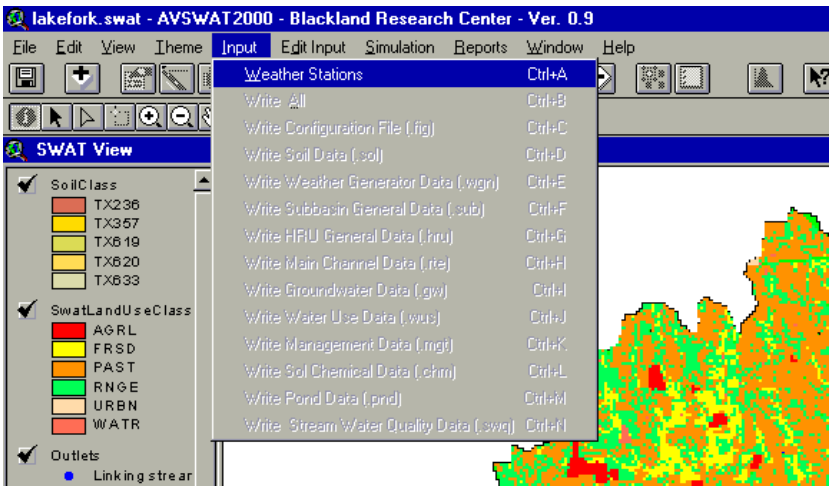


Figure 15.37

2. The Weather Data dialog box will be displayed (Figure 15.38).

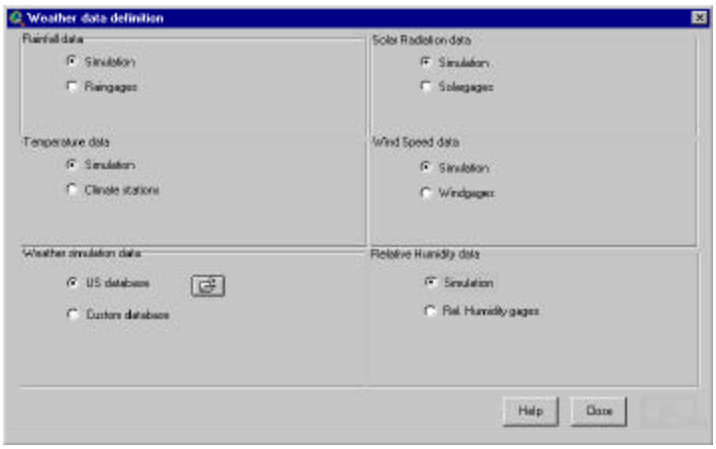


Figure 15.38

The example data set contains data files with measured precipitation and temperature for weather stations around the watershed.

3. To load the table containing the locations of the rain gage stations, click the radio button next to **Raingages** in the section of the dialog box labeled **Rainfall data**.

   A text box will appear at the bottom of the **Rainfall data** section. Click beside the text field.

   A browser will be displayed. Click the name of the rain gage location table (*pcpfork.dbf*) then click **OK**.

   The locations of the rain gages will be displayed as squares (Figure 15.39).

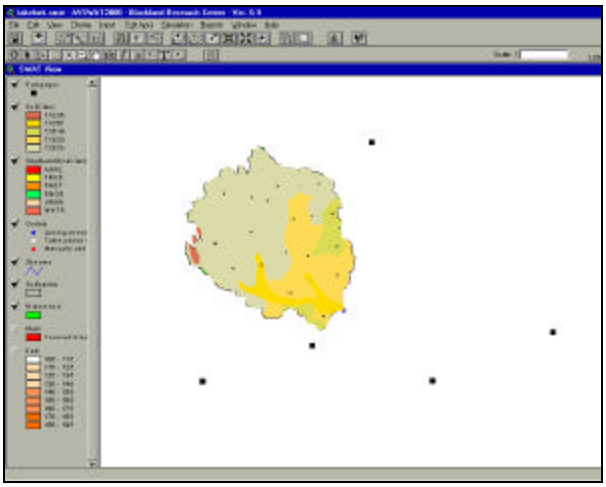


Figure 15.39

4. To load the table containing the locations of the temperature gage stations, click the radio button next to **Climate stations** in the section of the dialog box labeled **Temperature data**.

A text box will appear at the bottom of the **Temperature data** section. Click beside the text field.

A browser will appear. Click the name of the climate station location table (*tmpfork.dbf*) and then click **OK**.

The locations of the temperature gages will be displayed as triangles.

5. For a SWAT simulation using measured weather data, weather simulation information is needed to fill in missing data and to generate relative humidity, solar radiation, and wind speed. The example data set uses weather generator data loaded into the custom database. Click the radio button next to **Custom data base**.

   A text box will appear at the bottom of the **Weather simulation data** section. Click beside the text field.

   A browser will appear. Select the name of the weather generator stations location table from the list (*wgnstations.dbf*) and then click **OK**.

   The locations of the weather generator stations will be displayed as stars.

6. Once the weather generator data is loaded, a button labeled **OK** will appear at the bottom of the Weather Data dialog box. Click this button. The interface will assign the different weather station data sets to the subbasins in the watershed.

7. A prompt box will appear when processing of the weather data is complete. Click **OK.**

SECTION 15.1.5: CREATE ARCVIEW DATABASES & SWAT INPUT FILES

1. On the **Input** menu, click **Write All** (Figure 15.40).

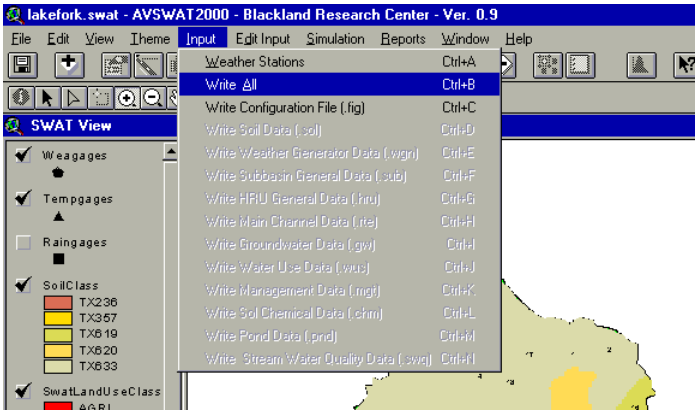


Figure 15.40

This creates the ArcView databases and SWAT input files containing default settings for SWAT input.

2. A message box will appear requesting the user to confirm the **Write All** command. Click **Yes**.

3. When the interface reaches the point where general subbasin data is compiled, a prompt box will appear asking the user if the default Manning's n value of 0.014 for overland flow should be changed. Click **No**.

4. When the interface reaches the point where main channel data is compiled, a prompt box will appear asking the user if the default Manning's n value of 0.014 for channel flow should be changed. Click **No**.

5. When the interface reaches the point where management data is compiled, a prompt box will appear asking if plant heat units should be estimated or set to a default value. Click **Yes** to estimate.

6. A message box will be displayed upon completion of the SWAT input database initialization. Click **OK**.

SECTION 15.1.6: RUN SWAT

1. On the **Simulation** menu, click **Run SWAT** (Figure 15.41).

Figure 15.41

2. A dialog box will be brought up (Figure 15.42).

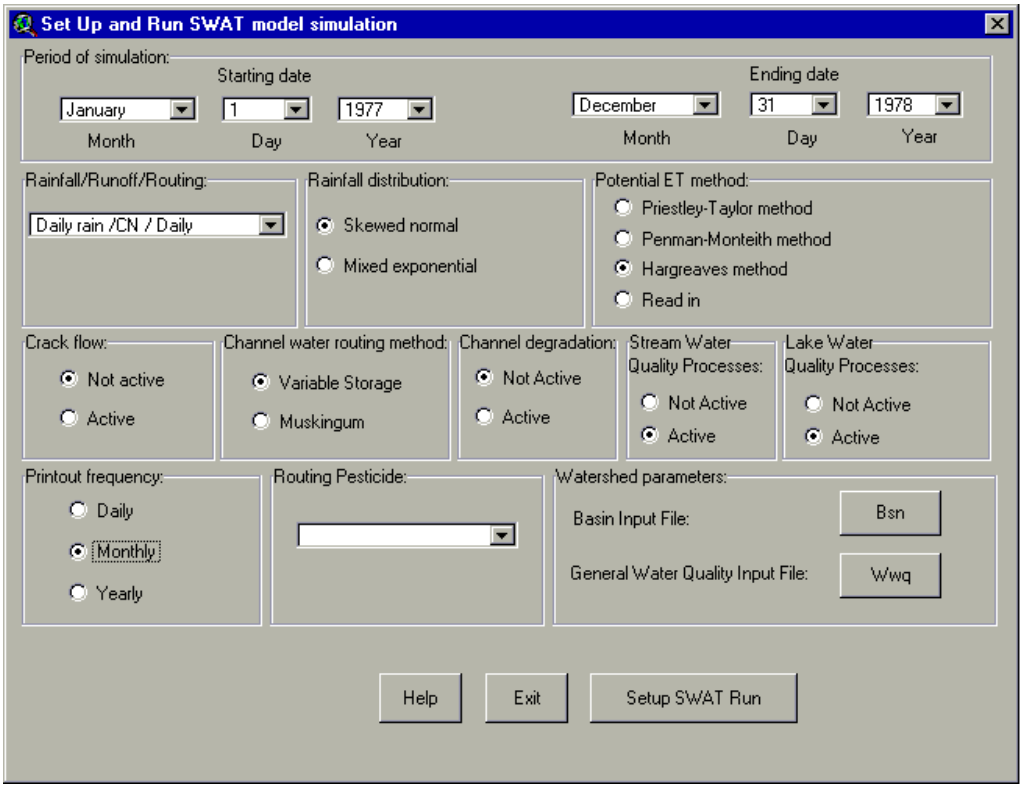


Figure 15.42

3. The initial and final day of simulation are set to the first and last days of measured weather data. Leave those values set to 1/1/1977 and 12/31/1978. Set the Potential ET method to **Hargreaves** and the Printout Frequency to **Monthly**. Leave all other settings as is.
4. Click the button labeled **Setup SWAT Run** to build the climate and watershed level input files.
5. A prompt box will appear asking if any input files need to be rewritten from modified .dbf files. Click **No**.
6. Click the **Run SWAT** button.
7. When the SWAT run is finished, a message box will be displayed noting that the simulation was successfully completed. Click **OK**.

***SECTION 15.1.7:* VIEW RESULTS**

1. A prompt box will be displayed asking if the user wishes to read the ASCII outputs. Click **Yes**.
2. The SWAT output data is loaded into dBASE tables and displayed (Figure 15.43).

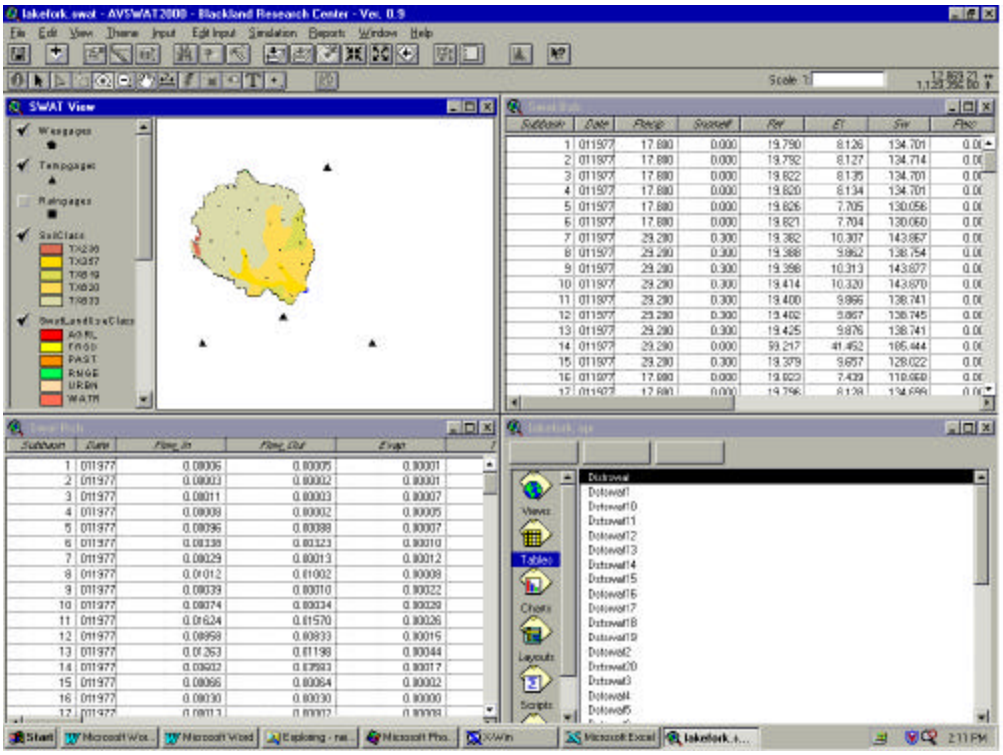


Figure 15.43

3. To graph and map results, on the **Reports** menu click **Map-Chart** (Figure 15.44).

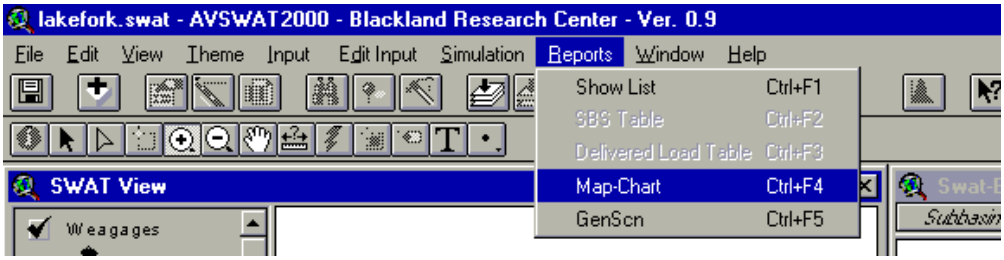


Figure 15.44

4. Select the **BSB** output file. Then Select **1** under **Subbasins**, **WYLD** under **Variables**, and **011977** through **121977** (to select multiple months hold down the `Shift` key while clicking the desired months). Once all the settings have been made, click **OK**.

5. A map of the water yield for all subbasins in 01/1997 will be plotted and the water yield for subbasin 1 will be graphed for the period specified (Figure 15.45).

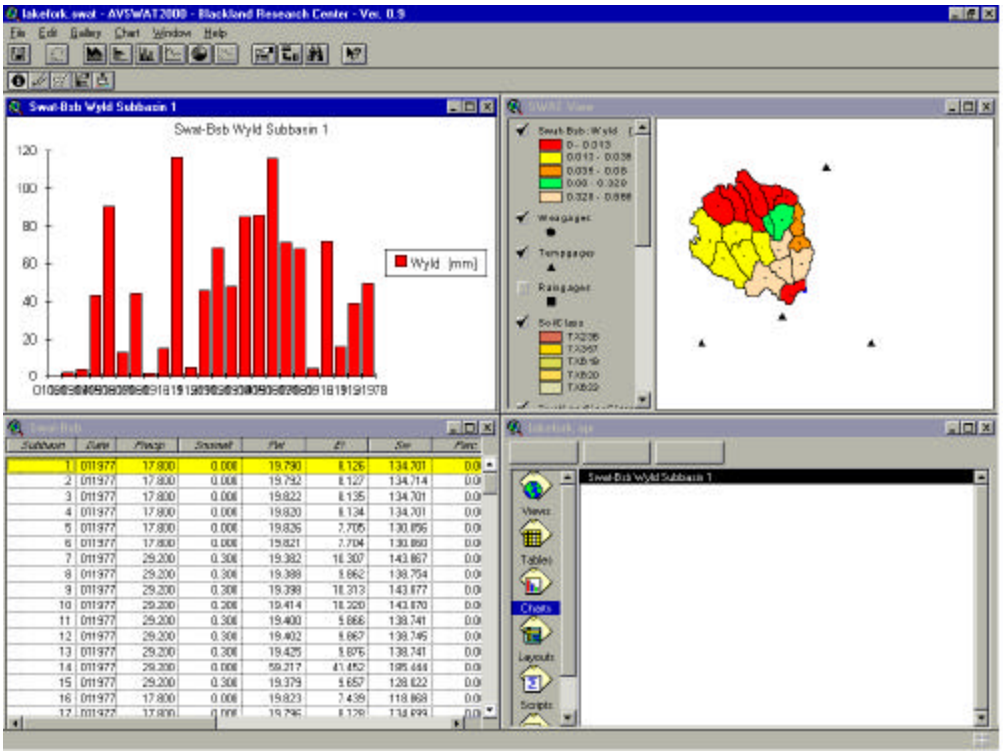


Figure 15.45

6. To make other plots, access the dialog box by clicking **Map-Chart** on the **Reports** menu. If the SWAT/ArcView menus are not listed, click the Output map, and they should appear.

7. If desired, input data may be edited from the **Edit Inputs** menu. After editing changes have been made, repeat the steps in Section 15.1.6 and 15.1.7 to generate and view the new output.

8. A Calibration Tool and Delivery Load Table Tool are available under the **Simulation** menu. Section 13 describes these tools.

9. To exit the SWAT/ArcView interface, click in the map display area with the right mouse button to make the pop-up menu appear. Select **Avswat Main Interface dialog** on the pop-up menu. Then click the button next to **Exit ArcView**. A prompt box will appear to confirm the exit selection. Click **Yes**.

APPENDIX 1:
WATERSHED DELINEATION OUTPUT DATA

The following sections describe the ArcView themes and tables generated during the watershed delineation process.

Subbasins Theme Data Fields

Field Name	Description
Id	ArcView internal field
GridCode	ArcView internal field
Subbasin	Subbasin number
Area	Subbasin area [hectares]
Len1	Stream reach (longest path within the subbasin) length [meters]
Slo1	Subbasin slope [%]
Sll	Field slope length [meters]
Csl	Stream reach (longest path within the subbasin) slope [%]
Wid1	Stream reach (longest path within the subbasin) width [meters]
Dep2	Stream reach (longest path within the subbasin) depth [meters]
Latitude	Latitude of the subbasin centroid
Elevation	Elevation of the subbasin centroid [meters]
Bname	String available for labeling the theme

Streams Theme Data Fields

Field Name	Description
Arcid	ArcView internal Field
From_node	ArcView internal Field
To_node	ArcView internal Field
Subbasin	Subbasin number
Subbasinr	Subbasin number receiving surface water from the subbasin
Numin	Number of inlet subbasins
Areac	Cumulated drainage area [hectares]
Len2	Stream reach length [meters]
Slo2	Stream reach slope [%]
Wid2	Stream reach width [meters]
Dep2	Stream reach depth [meters]
MinEl	Minimum elevation of the stream reach [meters]
MaxEl	Maximum elevation of the stream reach [meters]
Sname	String available for labeling the theme

Outlets Theme Data Fields

Field Name	Description
PointId	ArcView internal field
Grid_Code	ArcView internal field
Xpr	X coordinate in the current projection
Ypr	Y coordinate in the current projection
Lat	Latitude - decimal degrees
Long	Longitude - decimal degrees
Type	Outlet type
Id	Outlet ID

Reservoirs Theme

Field Name	Description
Subbasins	Subbasin number
Xpr	X coordinate in the current projection
Ypr	Y coordinate in the current projection
Lat	Latitude - decimal degrees
Long	Longitude - decimal degrees

Outlets Theme Type Field Values

Type	Meaning
L	Linking stream added outlet
O*	Table added outlet
T	Manually added outlet
P	Manually added point source
D*	Table added point source
I*	Table added draining watershed inlet
W	Manually added draining watershed inlet

* Acceptable values for importing outlet-inlet table.

APPENDIX 2: STATE FIPS CODES

Alabama	AL	01	Nebraska	NE	31
Arizona	AZ	04	Nevada	NV	32
Arkansas	AR	05	New Hampshire	NH	33
California	CA	06	New Jersey	NJ	34
Colorado	CO	08	New Mexico	NM	35
Connecticut	CT	09	New York	NY	36
Delaware	DE	10	North Carolina	NC	37
Florida	FL	12	North Dakota	ND	38
Georgia	GA	13	Ohio	OH	39
Idaho	ID	16	Oklahoma	OK	40
Illinois	IL	17	Oregon	OR	41
Indiana	IN	18	Pennsylvania	PA	42
Iowa	IA	19	Rhode Island	RI	44
Kansas	KS	20	South Carolina	SC	45
Kentucky	KY	21	South Dakota	SD	46
Louisiana	LA	22	Tennessee	TN	47
Maine	ME	23	Texas	TX	48
Maryland	MD	24	Utah	UT	49
Massachusetts	MA	25	Vermont	VT	50
Michigan	MI	26	Virginia	VA	51
Minnesota	MN	27	Washington	WA	53
Mississippi	MS	28	West Virginia	WV	54
Missouri	MO	29	Wisconsin	WI	55
Montana	MT	30	Wyoming	WY	56