Prepared in cooperation with the U.S. Agency for International Development, Office of Foreign Disaster Assistance (USAID/OFDA)

Users Manual for the Geospatial Stream Flow Model (GeoSFM)

By Guleid A. Artan, Kwabena Asante, Jodie Smith, Shahriar Pervez, Debbie Entenmann, James Verdin, and James Rowland

Open-File Report 2007–1440

U.S. Department of the Interior
U.S. Geological Survey

U.S. Department of the Interior
DIRK KEMPTHORNE, Secretary

U.S. Geological Survey
Mark D. Myers, Director

U.S. Geological Survey, Reston, Virginia 2008

For product and ordering information:
World Wide Web: http://www.usgs.gov/pubprod
Telephone: 1-888-ASK-USGS

For more information on the USGS—the Federal source for science about the Earth, its natural and living resources, natural hazards, and the environment:
World Wide Web: http://www.usgs.gov
Telephone: 1-888-ASK-USGS

Suggested citation:
Artan, G.A., Asante, K., Smith, J., Pervez, S., Entenmann, D., Verdin, J., and Rowland, J., 2008, Users Manual for the Geospatial Stream Flow Model (GeoSFM): U.S. Geological Survey Open-File Report 2007–1440, 146 p.

Contents

Figures

Tables

Users Manual for the Geospatial Stream Flow Model (GeoSFM)

By Guleid A. Artan[1], Kwabena Asante[1], Jodie Smith[1], Shahriar Pervez[1], Debbie Entenmann[1], James Verdin[2], and James Rowland[1]

[1] Science Applications International Corporation, Contractor to the U.S. Geological Survey (USGS) Earth Resources Observation and Science (EROS) Center, Sioux Falls, SD, work performed under USGS contract 03CRCN0001

[2] U.S. Geological Survey

Abstract

The monitoring of wide-area hydrologic events requires the manipulation of large amounts of geospatial and time series data into concise information products that characterize the location and magnitude of the event. To perform these manipulations, scientists at the U.S. Geological Survey Center for Earth Resources Observation and Science (EROS), with the cooperation of the U.S. Agency for International Development, Office of Foreign Disaster Assistance (USAID/OFDA), have implemented a hydrologic modeling system. The system includes a data assimilation component to generate data for a Geospatial Stream Flow Model (GeoSFM) that can be run operationally to identify and map wide-area streamflow anomalies. GeoSFM integrates a geographical information system (GIS) for geospatial preprocessing and postprocessing tasks and hydrologic modeling routines implemented as dynamically linked libraries (DLLs) for time series manipulations. Model results include maps that depicting the status of streamflow and soil water conditions. This Users Manual provides step-by-step instructions for running the model and for downloading and processing the input data required for initial model parameterization and daily operation.

Introduction to GeoSFM

The Geospatial Stream Flow Model (GeoSFM) was developed to establish a common visual environment for the monitoring of hydrologic conditions over wide areas (Artan and others, 2001). The monitoring activities include topographic analysis, data assimilation, time series processing, and presentation of the results. The GeoSFM is designed to use remotely sensed meteorological data in data sparse parts of the world (Artan and others, 2007). Many of the data sets involved in these processes are raster grids. The spatially distributed nature of the raster grids used in these processes required the adoption of a customizable geographic information system with excellent raster functionality. The ArcView GIS software was adopted for the implementation because it provided a visual, customizable development environment with excellent support of raster operations. An ArcView extension was developed (in Avenue) for the geospatial processing operations and for initializing time series analysis tasks. Routines for performing the hydrologic computations involved in mass balance and routing were developed in a mixed programming environment (C/C++ and Visual Fortran) and compiled as dynamically linked libraries (DLLs). These DLLs are called up directly from within ArcView scripts, eliminating the need to develop a separate interface for hydrologic routing operations. Descriptions of the algorithms used within these programs are presented in the GeoSFM Technical Manual (Asante and others, 2007a). This Users Manual describes the software components, data sets, processing steps, and input/output files associated with running the model. It also includes appendices which provide step-by-step instructions, with associated screen images, to guide users through model set up procedures.

Preprocessing Module

GeoSFM's preprocessing module consists of routines for performing terrain analysis, extracting subbasin characteristics, and estimating unit hydrograph responses. The terrain analysis routine delineates rivers, subbasins, and other elevation derivatives such as slope and drainage area. The subbasin characterization routines extract hydrologic parameters from elevation, soil, and land cover data into files for use in subsequent model operations. The response estimation routine computes the distribution of flow at the subbasin outlet following an instantaneous input event. It stores the result in a text file for use in flow simulations. This section of the Users Manual begins with instruction on installing GeoSFM, followed by instructions for completing the preprocessing tasks.

Model Installation

As shown in figure 1, GeoSFM consists of a series of nine directories containing programs, documents, and data sets. The PROGRAMS directory contains the software executables and DLLs used in GeoSFM. It also contains a batch file for installing and registering the software on the local computer. The DOCUMENTATION directory contains this Users Manual and the accompanying Technical Manual. The PRESENTATION directory contains illustrated presentations used for classroom instruction of new users. The DEMDATA directory contains elevation data from the USGS HYDRO1k data set (Verdin and Greenlee, 1996). The SOILDATA directory contains soil parameters derived from the FAO/UNESCO Digital Soil Map of the World (FAO, 1995). The LANDCOV directory contains a grid of vegetation data from the Global Land Cover Characteristics (GLCC) database (Loveland and others, 2000). The RAINDATA and EVAPDATA directories contain samples of the rainfall and evapotranspiration grids used in model computations. The SAMPLES directory contains input and output text files from a sample application of GeoSFM for the Limpopo Basin (Asante and others, 2007b), which is used for illustrating model functionality in this manual. The data files are distributed by CD-ROM or by download from the USGS EROS Web site.

Figure 1. Components of the Geospatial Stream Flow Model.

GeoSFM			
DEMDATA	**RAINDATA**	**EVAPDATA**	**LANDCOV**
elevation	rain_1999 rain_19991 • rain_1999365	evap_1999 evap_19991 • evap_1999111	usgslandcov
SOIL and L C Data	**PROGRAMS**	**PRESENTATIONS**	**DOCUMENTATION**
ks maxcover rcn soildepth texture whc veg	geosfm.avx v1.0 geosfm.dll geosfmcalib.exe geosfmgzip.exe geosfmpost.exe geosfmstats.dll geosfmtar.exe install.bat		GeoSFM Technical Manual GeoSFM Users Manual

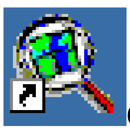
SAMPLES				
Grids	**Shapefiles**	**Text files**	**Text files**	**Project**
basins downstream elevations flowacc flowdir flowlen hilllength outlets slope streams strlinks traveltime velocity dem	basply1.shp rivlin1.shp limpbas.shp gauges2.shp	actualevap.txt balfiles.txt balparam.txt baseflow.txt basin.txt basinrunoffyield.txt cswater.txt damlink.txt damstatus.txt describe.txt evap.txt evapstations.txt excessflow.txt forecast1.txt forecast2.txt forecast3.txt gwloss.txt inflow.txt initial.txt interflow.txt	localflow.txt logfileflow.txt logfilesoil.txt mssbalance.txt maxtime.txt obsflow.txt order.txt rain.txt rainstations.txt rating.txt response.txt river.txt riverdepth.txt routfiles.txt routparam.txt soilwater.txt streamflow.txt testfile.txt times.txt	project.apr

To install GeoSFM, download or copy from CD all files described above to your C: drive. The GeoSFM software files listed in table 1 must be installed in the ArcView 3.x extension directory. The other files will be needed to complete the tutorials.

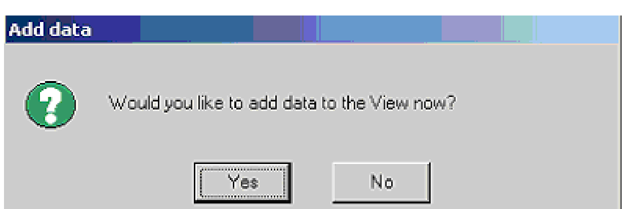
Table 1. Description of Software Programs in GeoSFM.

Files	Description
geosfm.avx	GeoSFM's ArcView 3.x Extension; provides the model interface
Geosfm.dll	GeoSFM's Dynamically Linked Libraries; contain hydrologic analysis routines
Geosfmcalib.exe	GeoSFM's Calibration module; executes the multi-objection parameter estimation algorithm.
Geosfmgzip.exe	Freeware for unzipping weather data files downloaded via FTP, renamed for easy identification
Geosfmtar.exe	Freeware for untarring weather data files downloaded via FTP, renamed for easy identification
Geosfmpost.exe	GeoSFM utility for postprocessing calibration module results
Geosfmstats.dll	GeoSFM Dynamically Linked Libraries for postprocessing simulation time series results.
Geosfmstats.exe	GeoSFM executable utility for postprocessing simulation time series results.
INSTALL.bat	GeoSFM installation utility for copying and registering program files to the local computer.

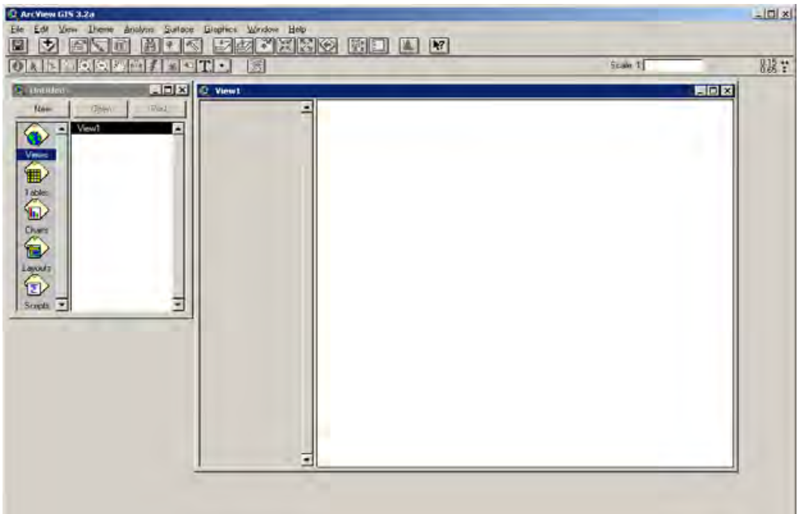
In c:\GeoSFM\Programs, double-click the INSTALL.bat file and installation is complete. This installation will copy all GeoSFM files and register the .dll files to your computer. You will need to create a new directory, c:\GeoSFM\workspace, for output files.


Open ArcView GIS by clicking the shortcut on your desktop or by selecting it from your Programs menu. When ArcView opens, the **Welcome to ArcView GIS** dialog box is displayed. Depending on the setup configuration, there are different ways to create a new project. If the dialog box below is displayed, click the **with a new View** radio button under **Create a new project** and click **OK**.

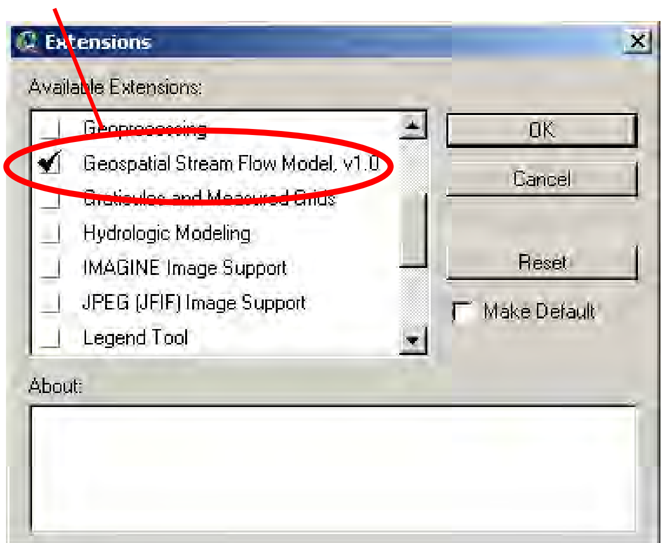
The **Add data** dialog box appears asking if you would like to add data to the View now. Click **No**.



If the dialog box is not displayed, as shown below, click the **Views** icon and then click the **New** button in the untitled Project window. This will open the **View1** window. Click and drag the bottom right corner to expand the view, and position the window next to the untitled Project window.

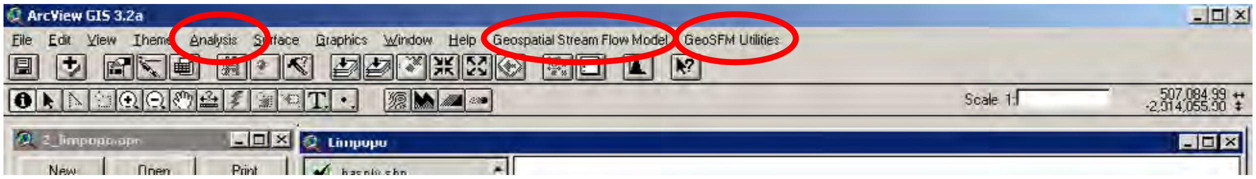


Next, from the **File** menu, select **Extensions** to load Geospatial Stream Flow Model and the Spatial Analyst.

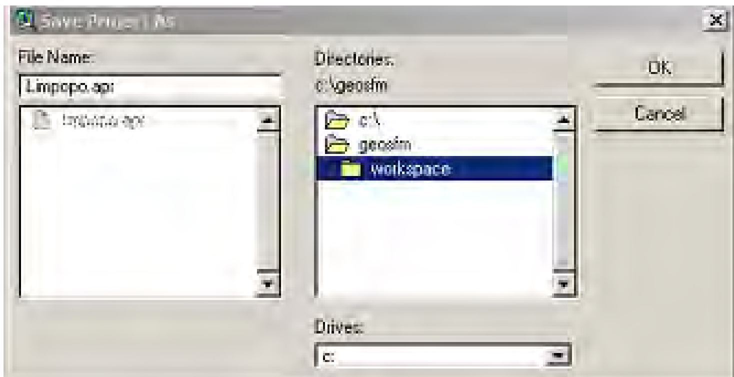


Check the boxes next to **Geospatial Stream Flow Model** and **Spatial Analyst** to load the extensions to the project and click **OK**.

The Menu and tool bar will update to reflect the additional functions of the Geospatial Stream Flow Model and the Spatial Analyst extensions.

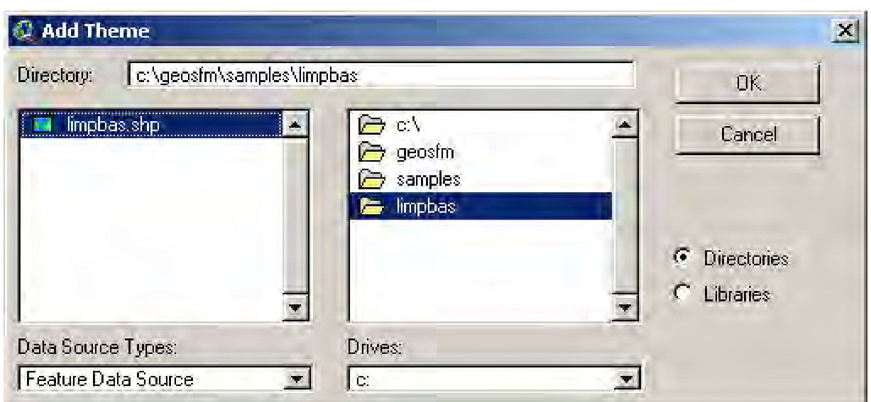


From the **File** menu, select **Save Project As**. Save your project to your workspace **c:\GeoSFM\workspace** with the file name **Limpopo.apr**. The extensions will then be preloaded next time you open the project.

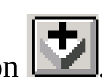


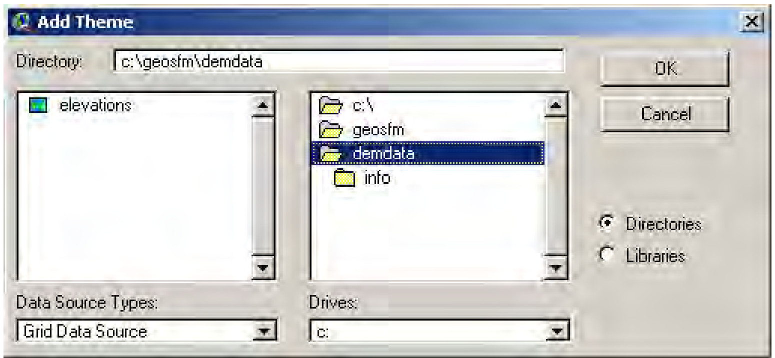
Terrain Analysis

Click the **Add Theme** button  to add the Limpopo Basin shapefile. Change the **Data Source Types** to **Feature Data Source**. Add the shapefile named **limpbas.shp** from the **c:\GeoSFM\samples\limpbas** directory. Click **OK**.

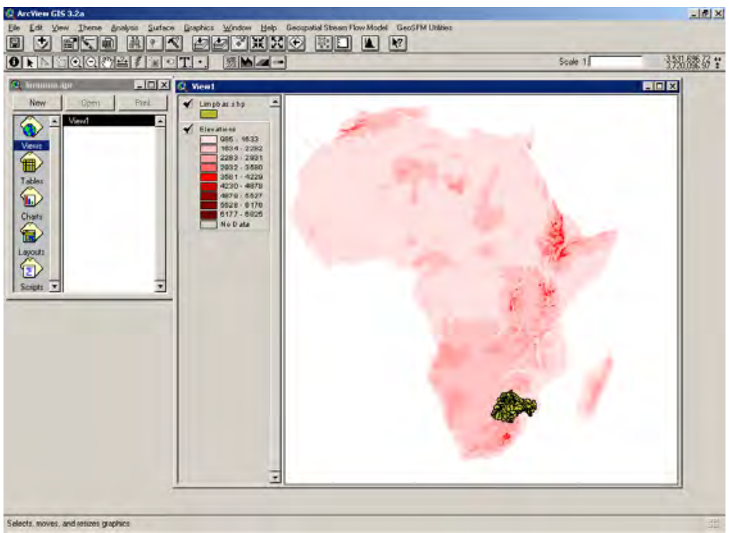


The Geospatial Stream Flow Model uses a digital elevation model for the delineation of hydrologic modeling units.

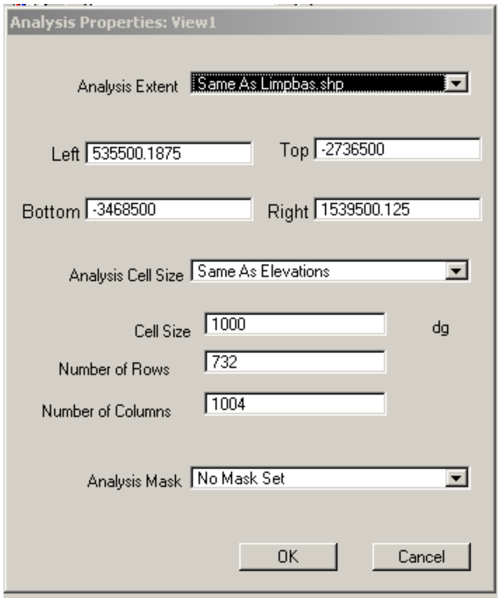
Add the **elevations** grid to the View using the **Add Theme** button . Change the **Data Source Types** to **Grid Data Source**. Click **elevations** from the **c:\GeoSFM\demdata** directory and click **OK** to add the DEM to the View.



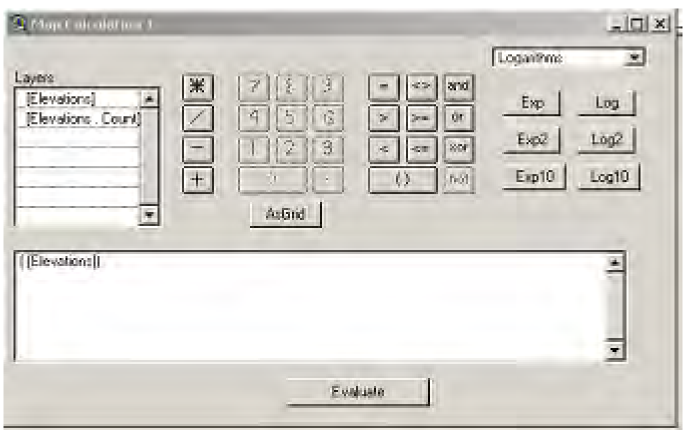
Click and drag the **Limpbas.shp** theme to the top of the table of contents and check the box so that it is visible over the **elevations** grid.



Next, set the analysis environment from the **Analysis** menu by selecting **Properties**. Change the **Analysis Extent** to **Same As Limpbas.shp** and the **Analysis Cell Size** to **Same As Elevations**. All other parameters will adjust themselves. Click **OK**.



Begin by clipping the DEM to the extent of the analysis area. In the **Analysis** menu, select the **Map Calculator.** Double-click **[Elevations]** from the **Layers** list and click **Evaluate**.



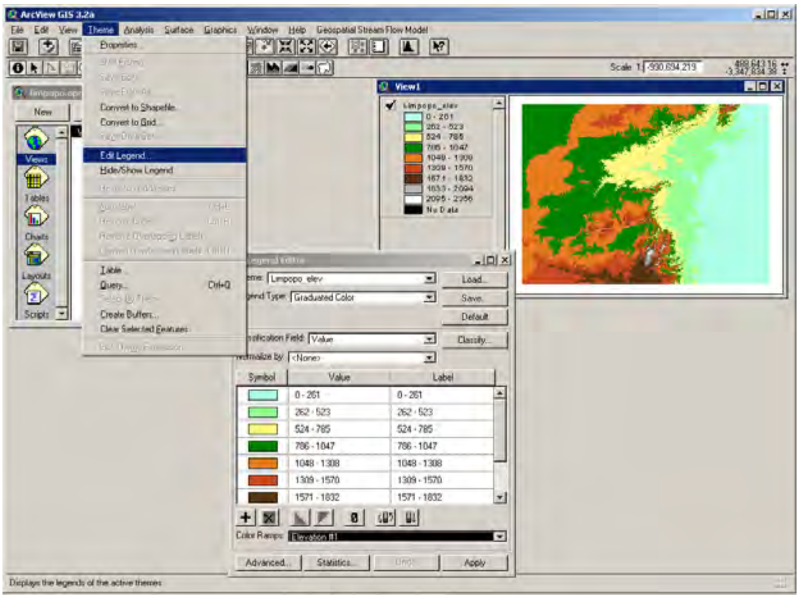
Select **Map Calculation 1** in the table of contents to display **Theme** in a raised box. From the **Theme** menu, select **Save Data Set**. In the **Save Data Set: Map Calculation 1** dialog box, navigate to the **c:\GeoSFM\workspace** directory, and in **Grid Name,** name your new grid extent **Limpopo_elev**. Click **OK**.

Click the **Add themes** button  to add the new permanent **Limpopo_elev** grid to the View. Change **Data Source Type** to **Grid Data Source** and click **Limpopo_elev** to add to the View.

Next, remove all **Themes** except for the new **Limpopo_elev** theme. Select the **Theme** to be removed by clicking **Theme**, which is now a raised box. In the **Edit** menu**,** select **Delete Themes** to remove the selected **Theme**. Continue until all themes are removed except for the **Limpop_elev** theme. Multiple themes can be selected by holding down the shift key while selecting the themes.

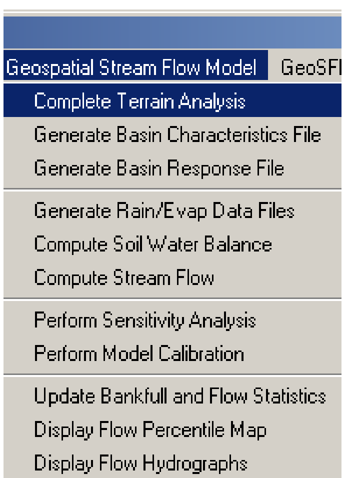
From the **View** menu**,** select **Zoom To Themes** to focus on the new extent area. You may want to apply a legend suitable for elevation to the theme. To do so, from the **Theme** menu select **Edit Legend**.

In the **Color Ramps** drop-down list at the bottom of the **Legend Editor,** select **Elevation #1** and click **Apply**.



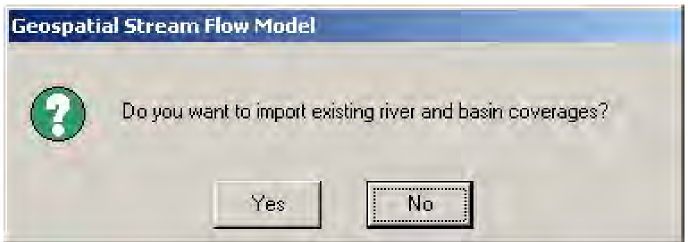
You are now ready to begin running the Geospatial Stream Flow Model!

In the **Geospatial Stream Flow Model,** select **Complete Terrain Analysis** from the drop-down list.

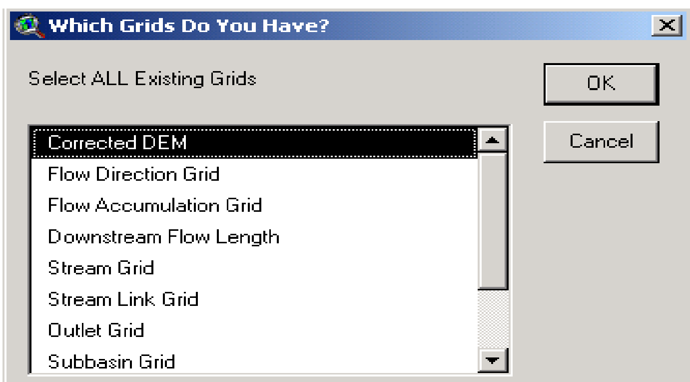


Confirm your working directory as **c:\GeoSFM\workspace** and click **OK**.

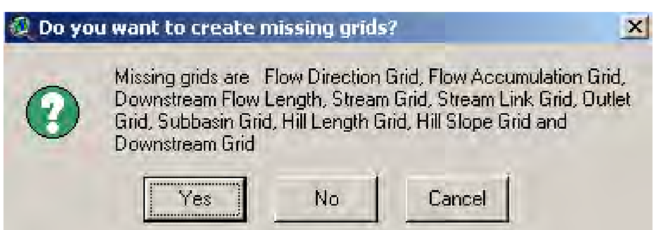
If you have existing river or basin coverages, you may add them. For this tutorial, select **No**.



Select the **Corrected DEM** as the only existing grid and click **OK**.



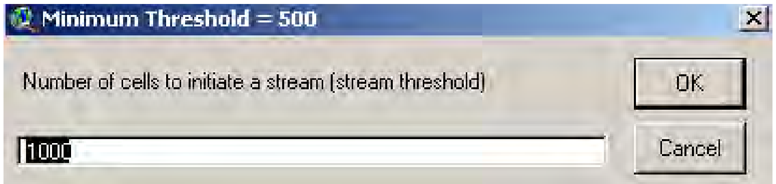
Select **Yes** to confirm that you want to create the missing grids.



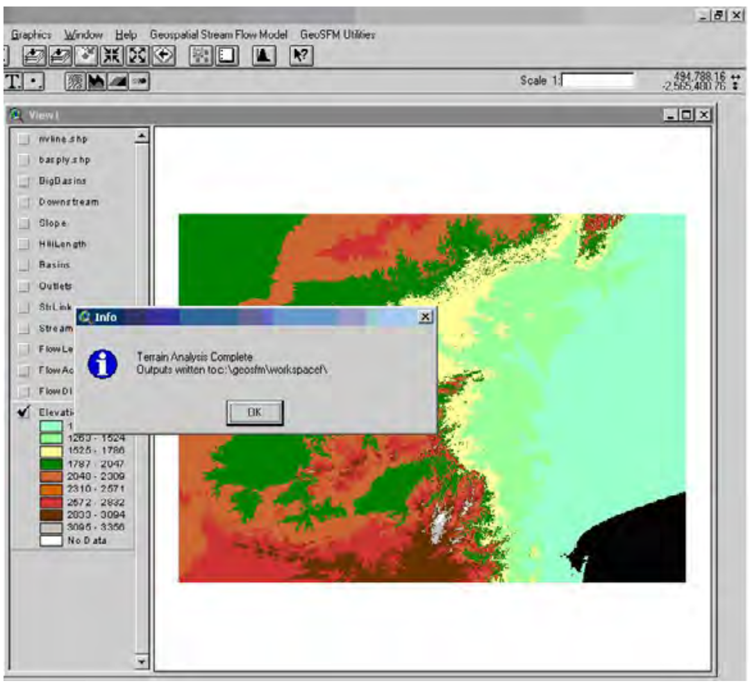
Confirm that the grid called **Limpopo_elev** is indeed the Corrected DEM.
Click **OK.**

The program should begin performing the terrain analysis. After a while, depending of DEM data resolution and modeling window spatial extend (time could vary from few seconds to hours), it will ask you to input the stream delineation threshold. This is the minimum number of cells that must be upstream of a given location before a river can be initiated.

Use the suggested default of **1000** and click **OK**. Using a different threshold will result in a model with a different number of streams and watersheds.

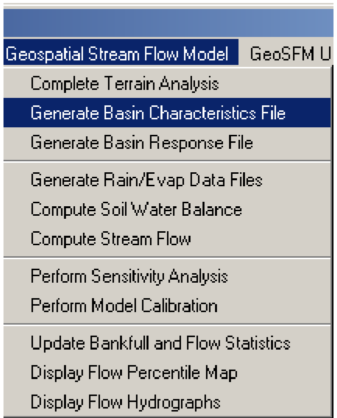


In a few minutes, you should get a message telling you the terrain analysis is complete. Click **OK**. (**Limpopo.elev** theme is replaced with **Elevations** theme during the processing.)



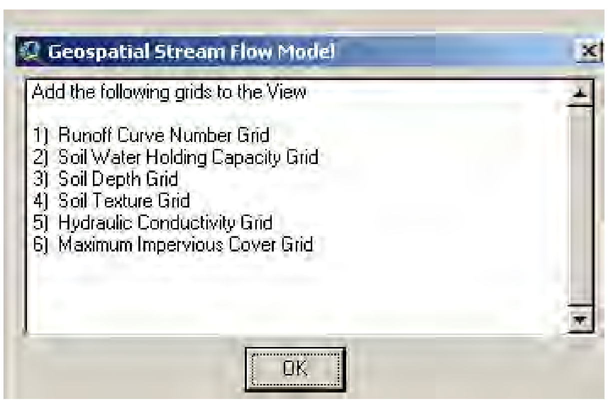
Basin Characterization

Next, you need to generate a file that summarizes basin characteristics. From the **Geospatial Stream Flow Model** menu, select **Generate Basin Characteristics File.**

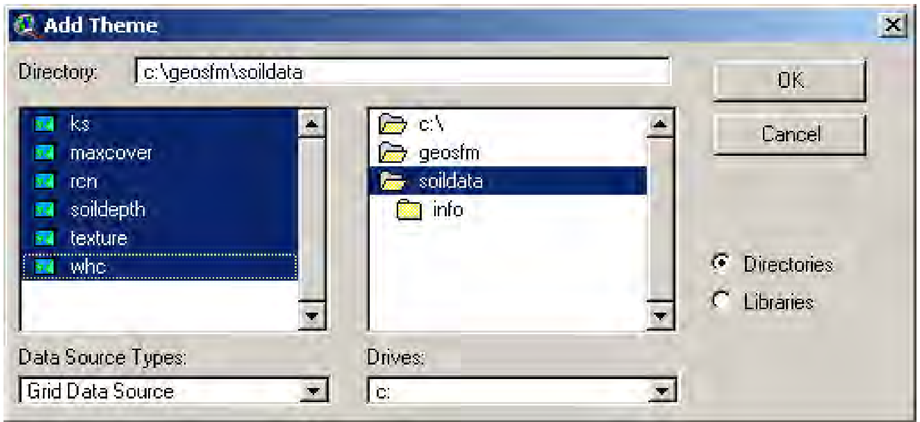


Confirm your working directory as **c:\GeoSFM\workspace.** Select **Yes** when presented with the question **Add Soils & LandCover Data to View?**

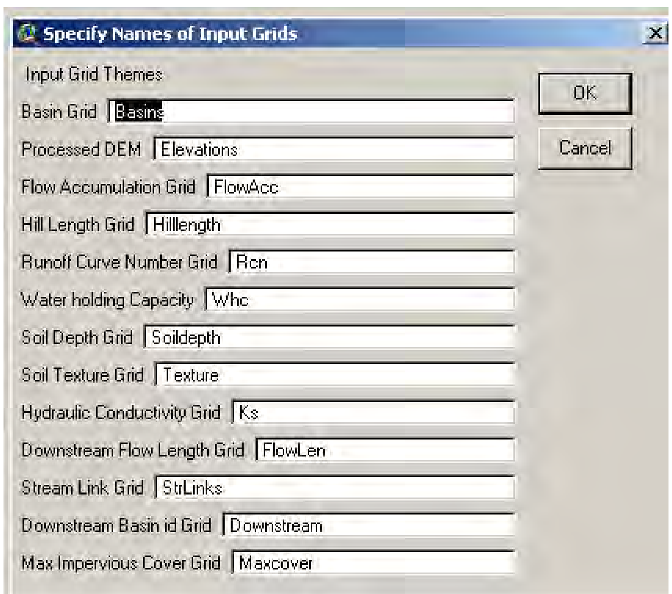
A list of the data sets you need will appear. Click **OK**.



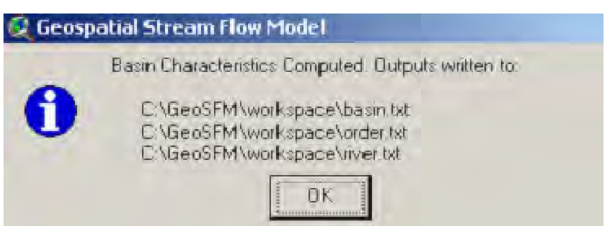
The required data sets are provided for you in the **c:\GeoSFM\soildata** directory. Change the **Data Source Types** to **Grid Data Source**. Hold the shift key down to select all the grids (ks, maxcover, rcn, soildepth, texture, and whc). Click **OK** to add them to the View.



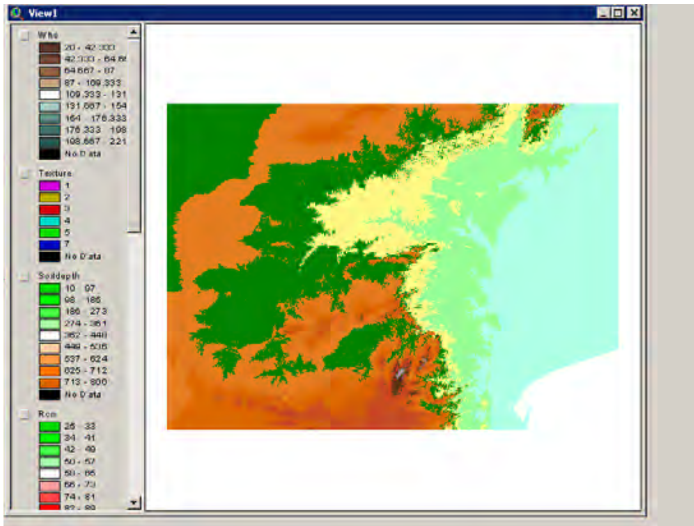
The program will present a new list of your input grids including all the input parameters.



The program will produce two files containing the characteristics of each subbasin and river. It will also produce a file containing the computational order, which is required for subsequent program operations. When the processing is complete, a dialog box will appear indicating the name and location of the output files. Click **OK**.

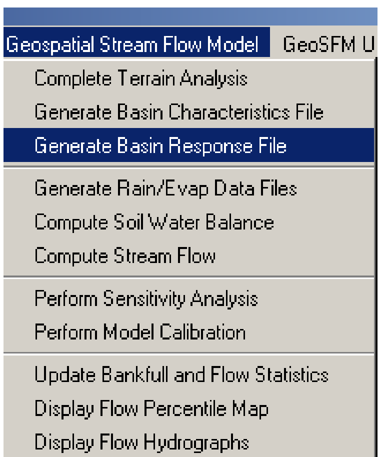


Below all the input grids are added to the table of contents.

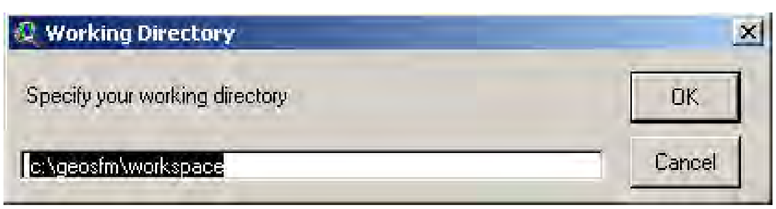


Unit Hydrograph Response

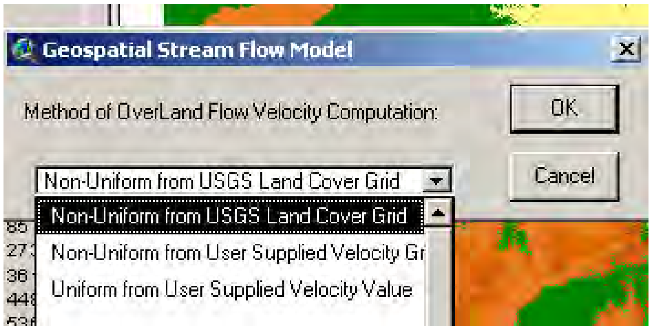
From the **Geospatial Stream Flow Model** menu, select **Generate Basin Response File**.



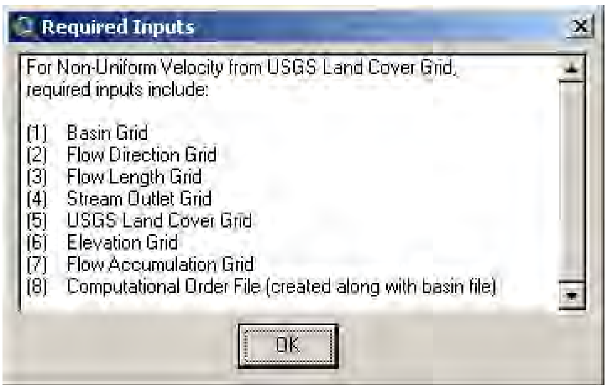
Confirm your working directory and click **OK**.



Select the **Non-Uniform from USGS Land Cover Grid** option for determining the overland flow velocity. Click **OK**.

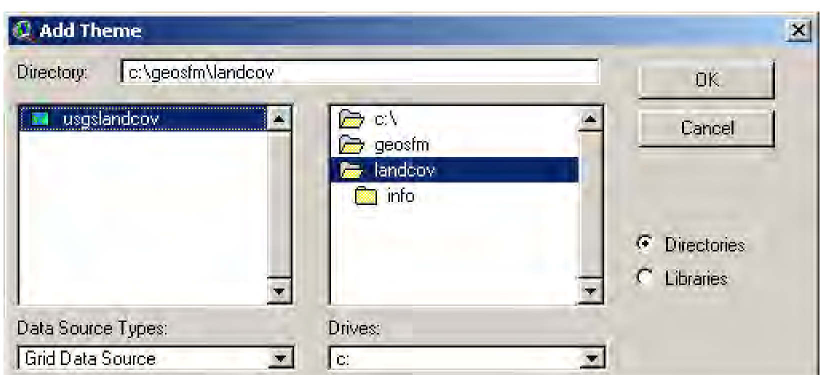


A list of the required inputs is displayed. Click **OK**.

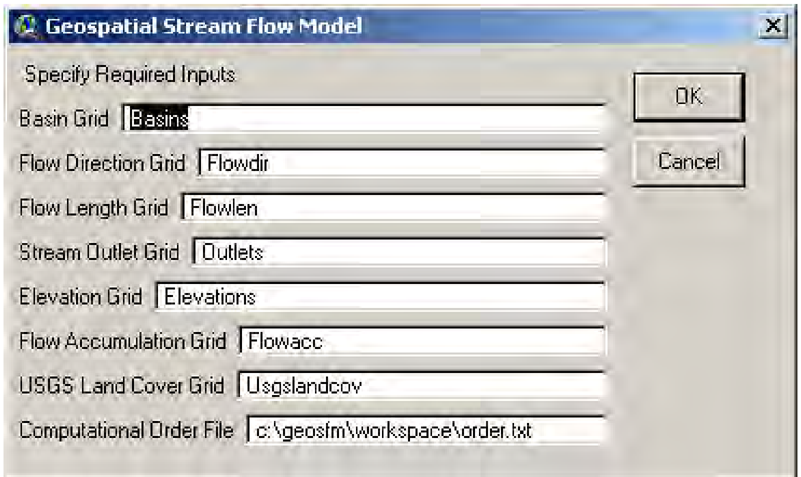


Click **Yes** when asked whether you want to add USGS Land Cover grid to the View.

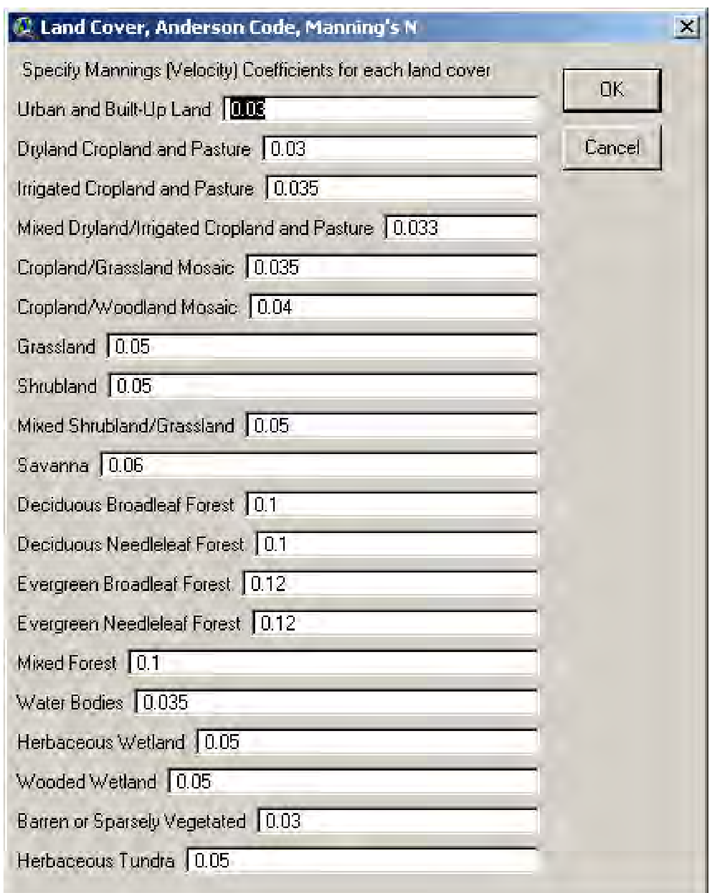
Change the **Data Source Types** to **Grid Data Source**. Select **usgslandcov** from **c:\GeoSFM\landcov** directory and click **OK** to add the land cover grid to the View.



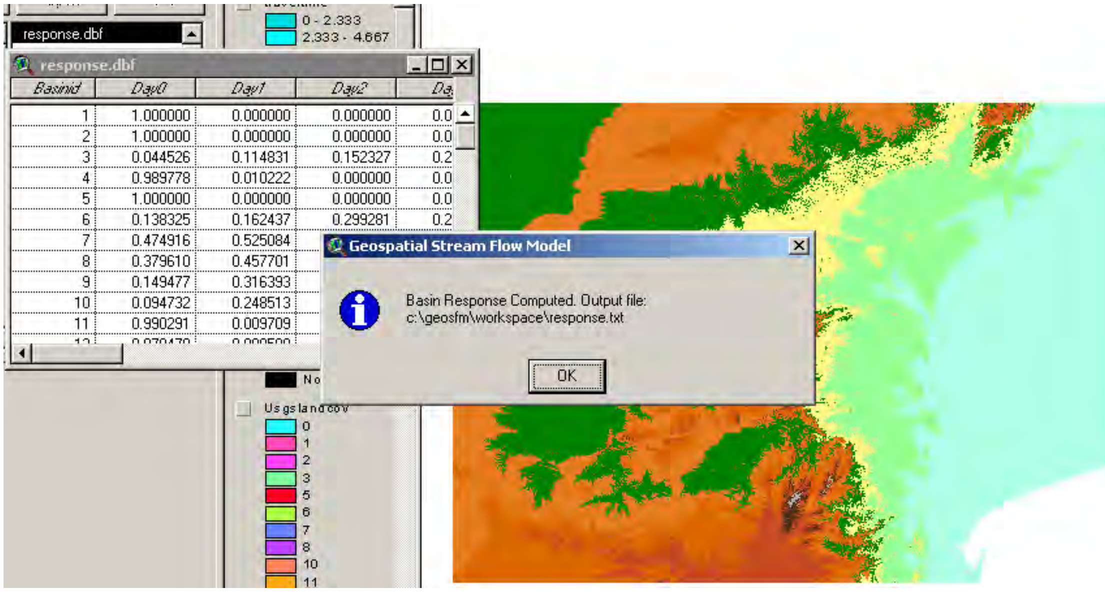
Confirm the names of the input grids and the **Computational Order File** with values displayed in the **Specify Required Inputs** dialog box. Click **OK**.



Manning's N for different land cover types, derived from (Maidment, 1993), should now be displayed. Use the default Manning's coefficient values for each of the land cover types. Click **OK**.



The program will compute a response file (similar to a unit hydrograph) for each subbasin. This may take a few minutes. After the computations are complete, a dialog box will appear and indicate to you the location of the output file.



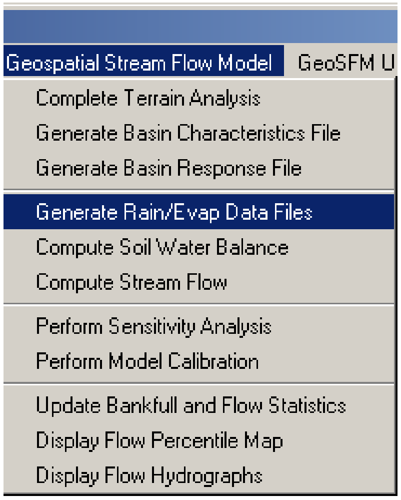
Traveltime and Velocity themes are added to the table of contents. Also, the response table is displayed. Click **OK** in the dialog box. You have now finished the terrain analysis and have generated the basin characteristics file and the response file. The basin and response files resulting from these computations are stored for use in subsequent model computations.

Hydrologic Analysis Module

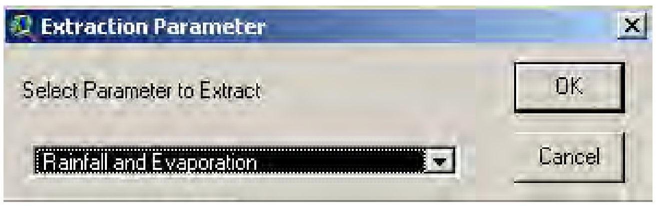
GeoSFM's hydrologic module contains several routines for moving water through the river basin. The module contains a data preparation routine for extracting time series data from geospatial grids into ASCII files by subbasins. The rainfall data presented here are derived from algorithms developed by Xie and Arkin (1997), while the evapotranspiration data are derived using the methods of Verdin and Klaver (2002). Two soil moisture accounting routines perform the vertical separation of incident rainfall into atmospheric releases, surface runoff, and subsurface flows. The module also contains three river flow transport routines for simulating the horizontal movement of water through the river network. The user is expected to prepare the input files, perform a soil moisture accounting, and then route the output with one river using one of the available routines. Instructions for completing the tasks are provided in the following sections, and detailed algorithm descriptions are provided in the accompanying Technical Manual.

Preparing Input Time Series Files

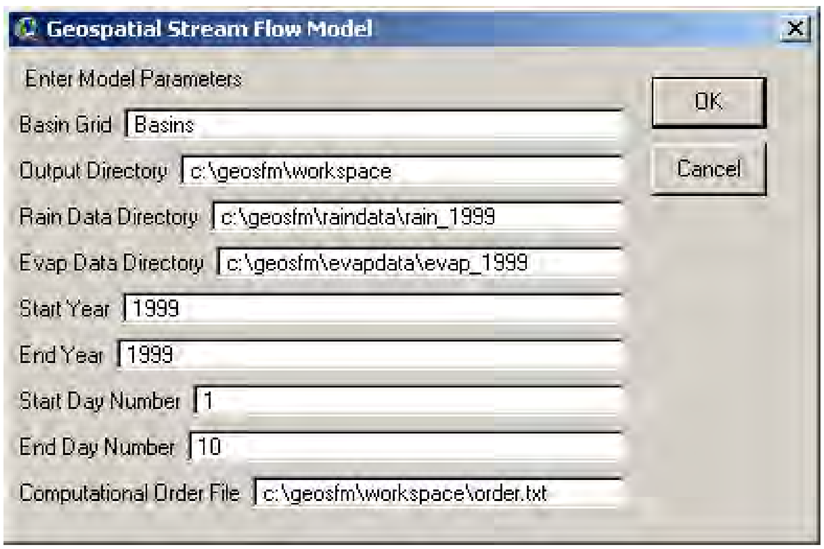
In this next step, a rainfall value will be estimated for each subbasin from the daily rainfall grids. To begin this computation, from the **Geospatial Stream Flow Model** menu, select **Generate Rain/Evap Data Files**.



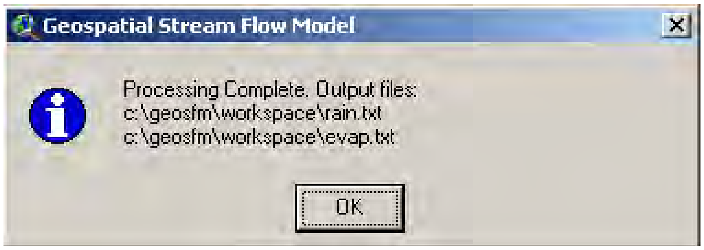
Select **Rainfall and Evaporation** when prompted to **Select Parameter to Extract**. Click **OK**.



Next, you will be prompted to **Enter Model Parameters**. Specify the location and dates to be processed as shown in the figure below. The **Rain Data Directory** and **Evap Data Directory** need to reflect the correct path; as seen below, these fields may need to be updated from the default path. The **End Day Number** field should be changed from the default of **240** to **10**, which will result in a shorter processing time. Click **OK**.



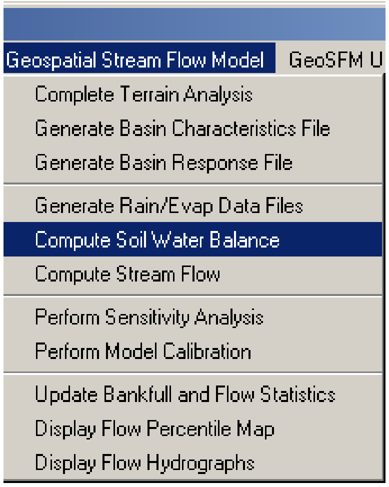
After a few moments, a message box will appear indicating that the processing is complete. The message also gives the location of the output files: **rain.txt** and **evap.txt**. Click **OK.**



The **rain.txt** file created from this process contains an average rainfall value in millimeters for each subbasin per day. The **evap.txt** file contains a potential evapotranspiration (PET) value in tenths of millimeters for each subbasin per day. The routing program will use these files for performing the soil water accounting.

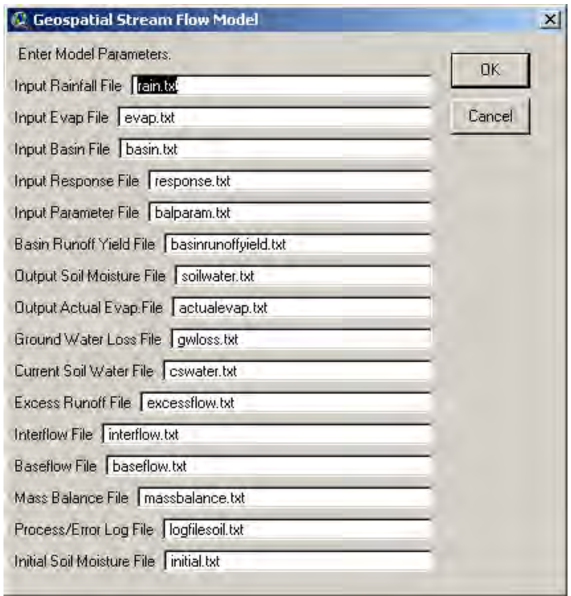
Soil Moisture Accounting

The next item in the **Geospatial Stream Flow Model** menu allows you to compute how much water is contributed to streamflow through the soil. From the **Geospatial Stream Flow Model** menu, select **Compute Soil Water Balance**.

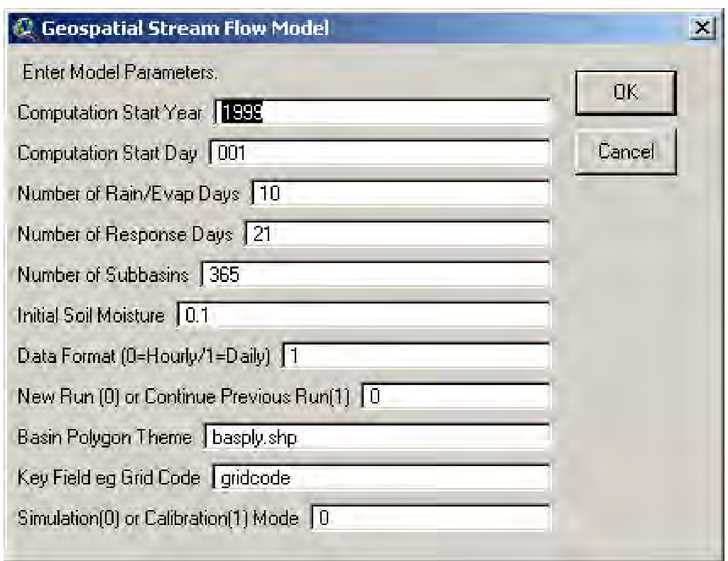


Confirm the working directory as **c:\geosfm\workspace**. Click **OK**.

You will then be prompted to verify the input and output files in the **Enter Model Parameters** dialog box; use the default files listed. Click **OK**.

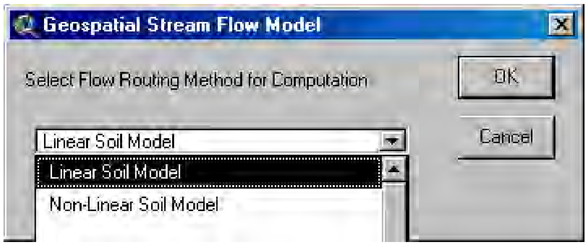


Next, **Enter Model Parameters** as shown below; use the default values. Click **OK**.

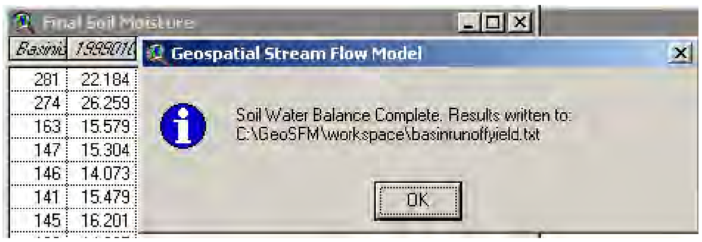


For short runs (30 days or less) getting the initial soil moisture content correct greatly influences the accuracy of your calculated flows. For this run, an initial soil wetness of 10 percent of storage capacity (**Initial Soil Moisture** value of 0.1) will be assumed.

Next, a prompt comes up to **Select Flow Routing Method for Computation**. The available choices are **Linear Soil Model** and **Non-Linear Soil Model**. For this tutorial, choose the **Linear Soil Model** option. Click **OK**.

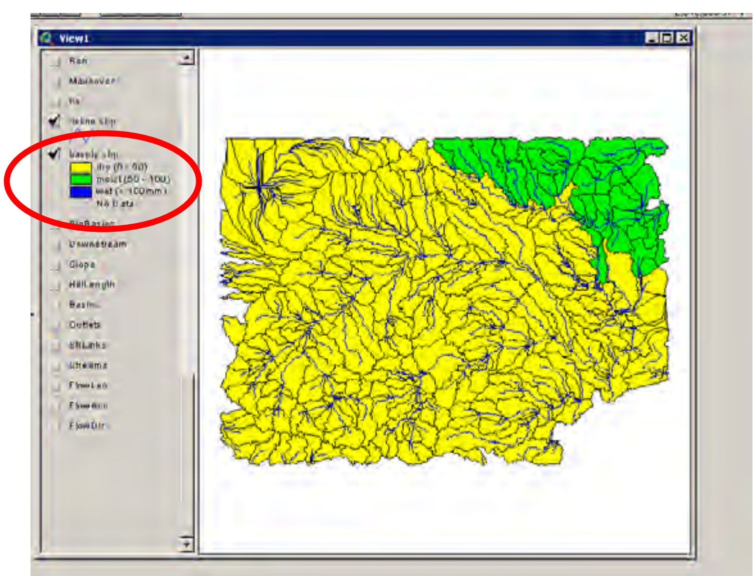


The model computes the soil water balance and indicates the location of the key output file containing the local contribution of each subbasin to downstream river flow. The **Final Soil Moisture** table is also displayed. Click **OK**.



The **basinrunoffyield.txt** file produced will be used by the flow routing program in subsequent operations. The basin polygon theme, **basply.shp**, is also color-coded to indicate the spatial distribution of soil moisture at the end of the simulation period.

Check the box next to this theme to turn it on, and make sure the theme is in a raised box. From the **Theme** menu, choose **Hide/Show Legend** to display the legend. There are three classes of soil moisture: dry, moist, and wet.

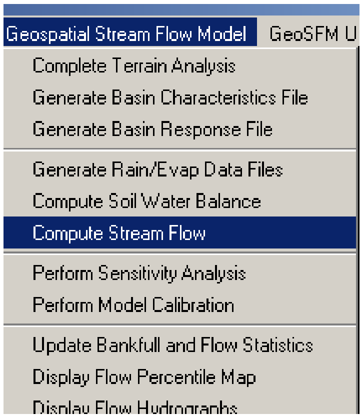


The image provides you with visual information of the distribution of soil moisture at the end of the simulation. The full time series of soil moisture values is written to the **soilwater.txt** file which is stored in the working directory. Another important file stored in the same directory is the

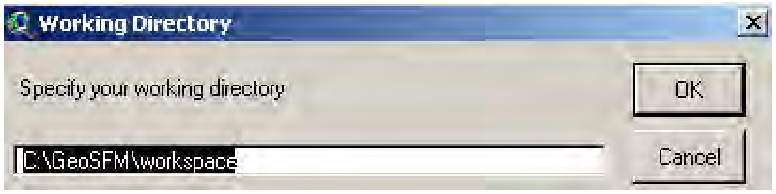
basinrunoffyield.txt file, which contains the runoff values to be passed to the river transport module.

Computing Streamflow

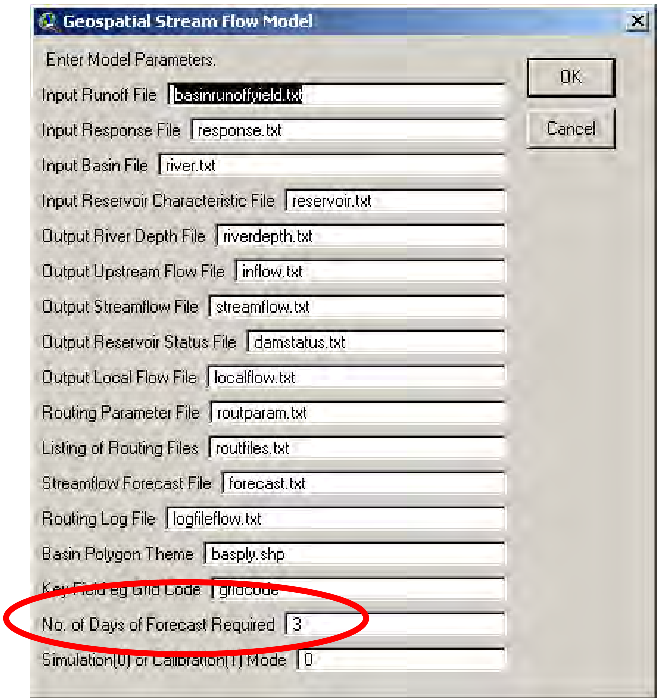
Finally, you get to the part engineers like; moving the water around. To begin, click the **Geospatial Stream Flow Model** menu and select **Compute Stream Flow**.



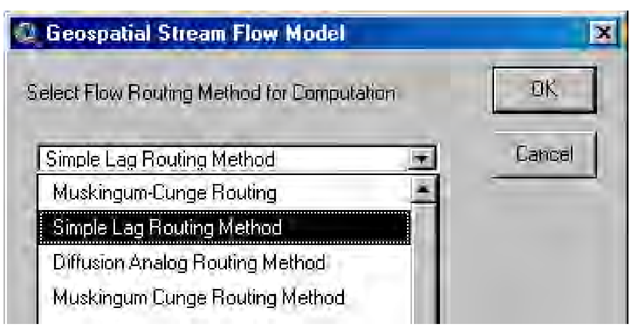
Specify your working directory. Click **OK**.



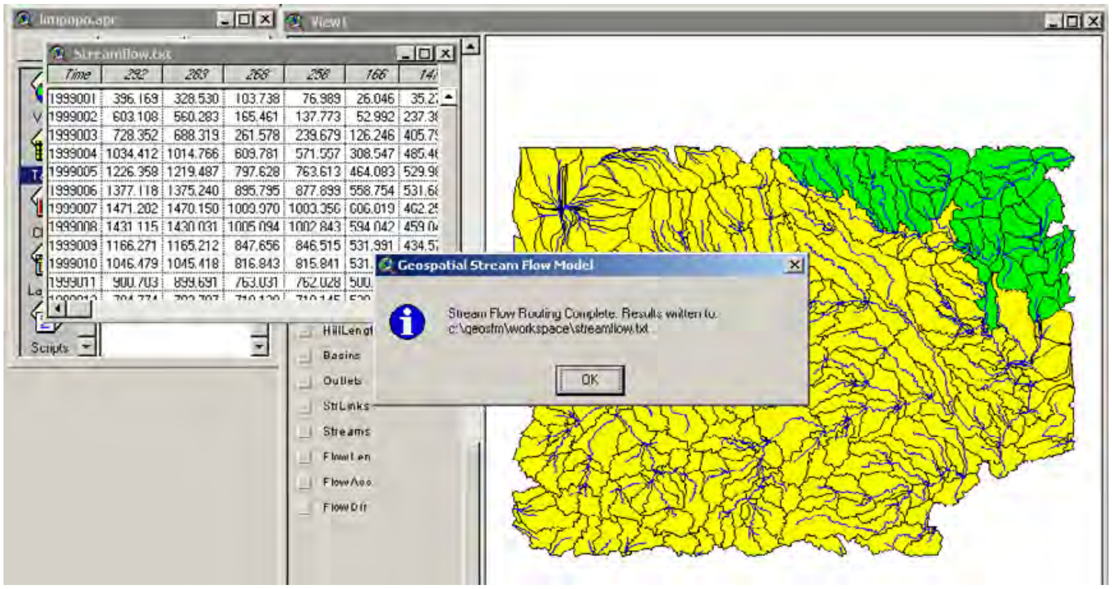
The next dialog box that appears allows you to verify or enter the simulation input files and output files. Pay particular attention to the **No. of Days of Forecast Required** field. In this example the forecast will be for a 3-day period. Click **OK**.



You will next be prompted to select the **Flow Routing Method for Computation**. The choices include: **Simple Lag Routing Method, Diffusion Analog Routing Method**, and **Muskingum Cunge Routing Method**. Select the **Simple Lag Routing Method** for this tutorial. Click **OK**.



The model will soon indicate that it has finished the computation and written the results to the **streamflow.txt** file in the working directory. Click **OK** in the **Stream Flow Routing Complete** dialog box.



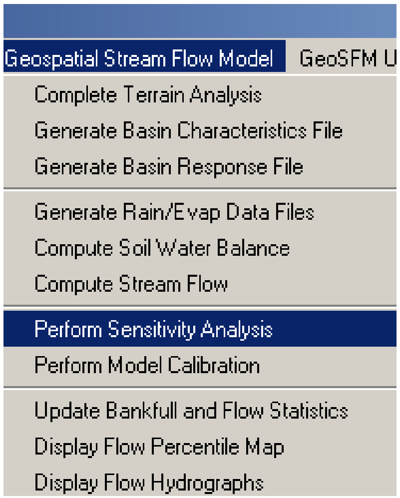
The **Streamflow.txt** table is displayed in ArcView to allow you to browse through the results of the analysis, and a simple ASCII version of the same table, the **streamflow.txt file**, is written to the working directory. The file contains a streamflow value in cubic meters per second for each stream during each simulation time step, in comma delimited format.

Parameter Calibration Module

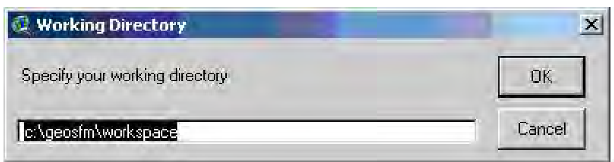
While the goal of GeoSFM is to allow for rapid assessment of streamflow conditions in basins with little ground-based information, its precision in representing flow processes can be greatly improved when observed flow information is available for adjusting initial parameter estimates. A parameter calibration module is incorporated into GeoSFM to facilitate such parameter adjustment within user-specified ranges. This module consists of one routine for analyzing the sensitivity of model results to changes in parameter estimates and another routine for testing different sets of selected parameters to determine which set results in the most accurate model results. User instructions for these two routines are provided in the following section. Detailed descriptions of the algorithms in the routines are provided in the accompanying Technical Manual.

Performing Sensitivity Analysis

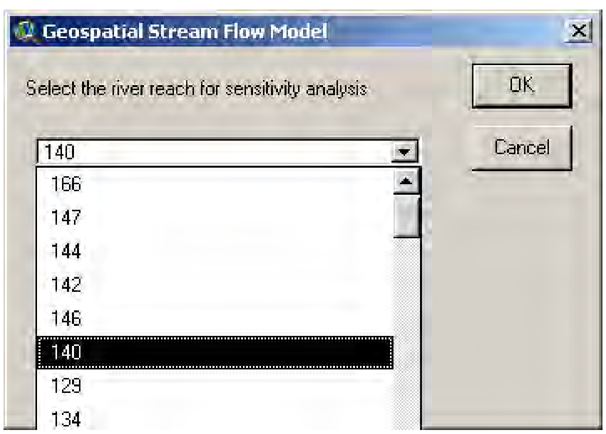
The sensitivity analysis evaluates the sensitivity of model outputs to changes in input parameters, and it is used to determine which parameters to modify during calibration. To initiate sensitivity analysis, select the menu item **Perform Sensitivity Analysis** from the **Geospatial Stream Flow Model** menu.



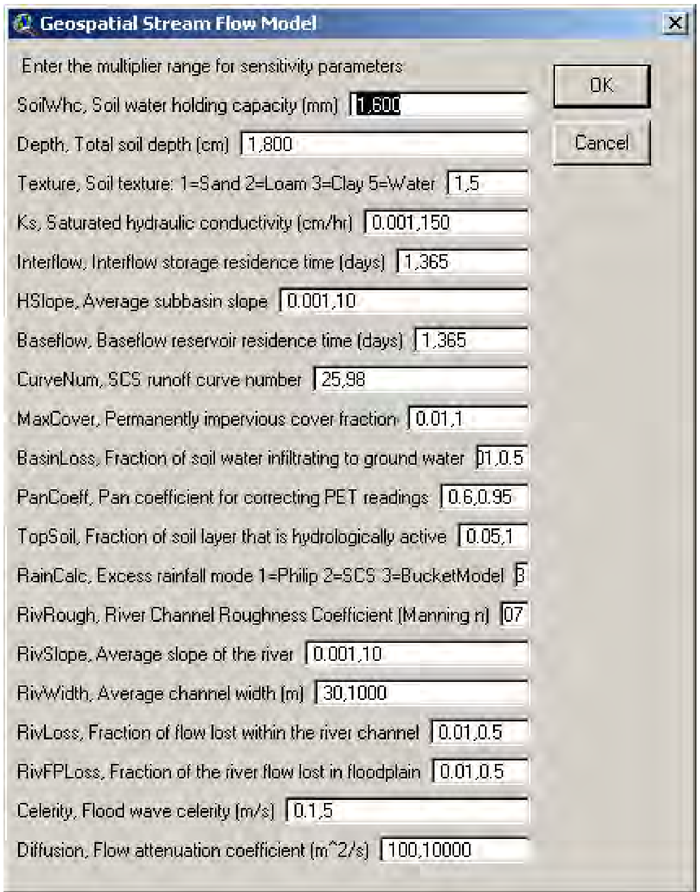
The **Working Directory** dialog box will ask you to **Specify your working directory**. Click **OK**.



The next dialog box displayed prompts you to **Select the river reach for sensitivity analysis**. In this tutorial, select basin 140 from the drop-down list and click **OK**.

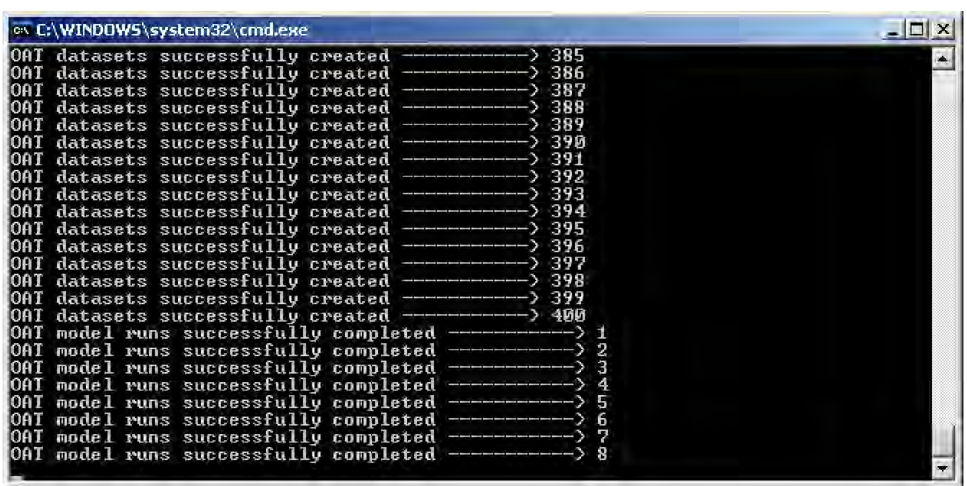


The next dialog box will list the absolute limits for the sensitivity parameters. Use the default values and click **OK**. This list shows the 20 different parameters that will be tested.

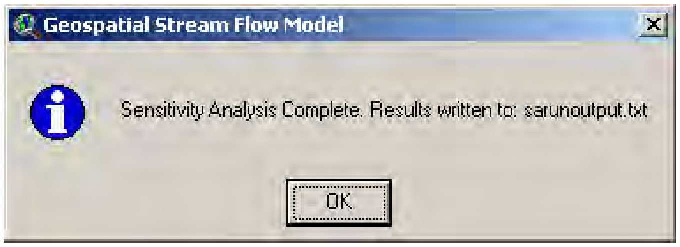


At this point, a window will open and display the number of data sets created and the number of model runs completed. This method is a one-at-a-time method where one model run will only have one parameter changed with all other parameters held constant. Twenty values are sampled at equal intervals for each of 20 parameters, resulting in a total of 400 model runs.

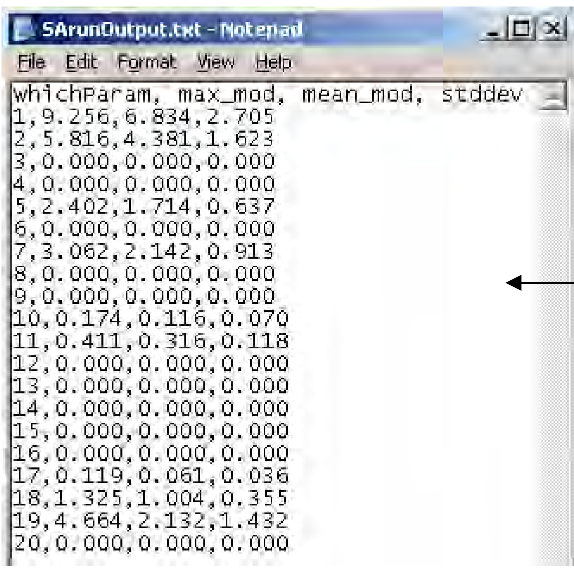
This will take a few minutes to complete.



When the process is finished, the number of model runs successfully completed will reach 400. A dialog box will display stating that the **Sensitivity Analysis is Complete** and the results have been written to **sarunoutput.txt**.



The output file **sarunoutput.txt** will give you the mean absolute difference of test results over the parameter range for each parameter. The greater the differences, the more sensitive the parameter. Sensitivity analysis is important when preparing to calibrate so that resources are not wasted on parameters that have little or no effect on model output.


SArunOutput.txt - Notepad

File Edit Format View Help

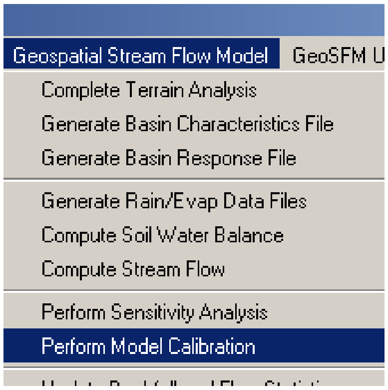
```
whichParam, max_mod, mean_mod, stddev
1,9.256,6.834,2.705
2,5.816,4.381,1.623
3,0.000,0.000,0.000
4,0.000,0.000,0.000
5,2.402,1.714,0.637
6,0.000,0.000,0.000
7,3.062,2.142,0.913
8,0.000,0.000,0.000
9,0.000,0.000,0.000
10,0.174,0.116,0.070
11,0.411,0.316,0.118
12,0.000,0.000,0.000
13,0.000,0.000,0.000
14,0.000,0.000,0.000
15,0.000,0.000,0.000
16,0.000,0.000,0.000
17,0.119,0.061,0.036
18,1.325,1.004,0.355
19,4.664,2.132,1.432
20,0.000,0.000,0.000
```

In this example, parameters 1, 2, 5, 7, 18, and 19 show the greatest differences as measured by the largest standard deviations. The parameters that are the most sensitive are SoilWhc, soil Depth, Interflow, BaseFlow, RivFPLoss, and Celerity.

The sensitivity analysis provides you with information on which parameters to focus on during the calibration described in the following section.

Performing Model Calibration

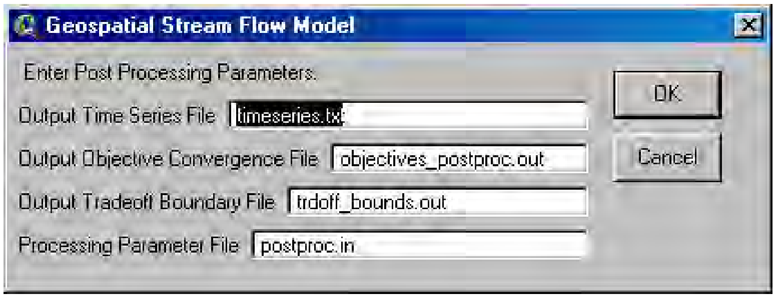
The next menu item is **Perform Model Calibration**; from the **Geospatial Stream Flow Model** menu, select **Perform Model Calibration**. The purpose of calibration is to adjust model parameters to closely match the real system.



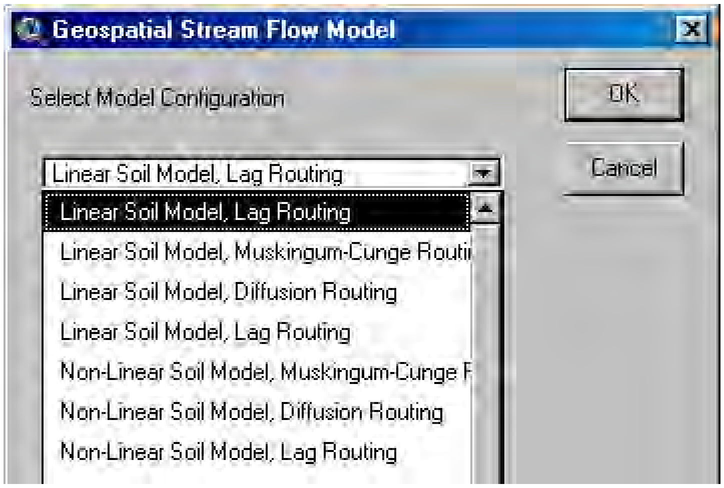
Confirm your working directory. Click **OK**.

The next box will display the model calibration parameters. Both input and output files will default in the display. Click **OK**.

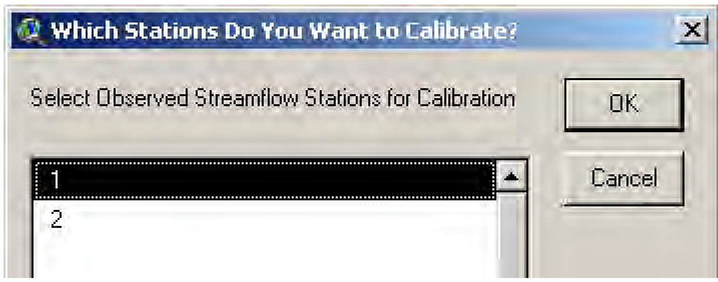
The following dialog box displays the input and output files needed for the postprocessing program. Click **OK**.



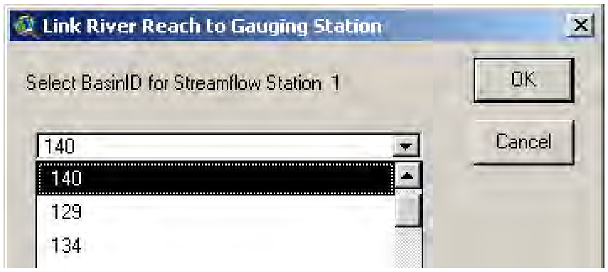
Select the model configuration from the following drop-down list. **Linear Soil Mode, Lag Routing** is the option selected for this tutorial. Click **OK**.



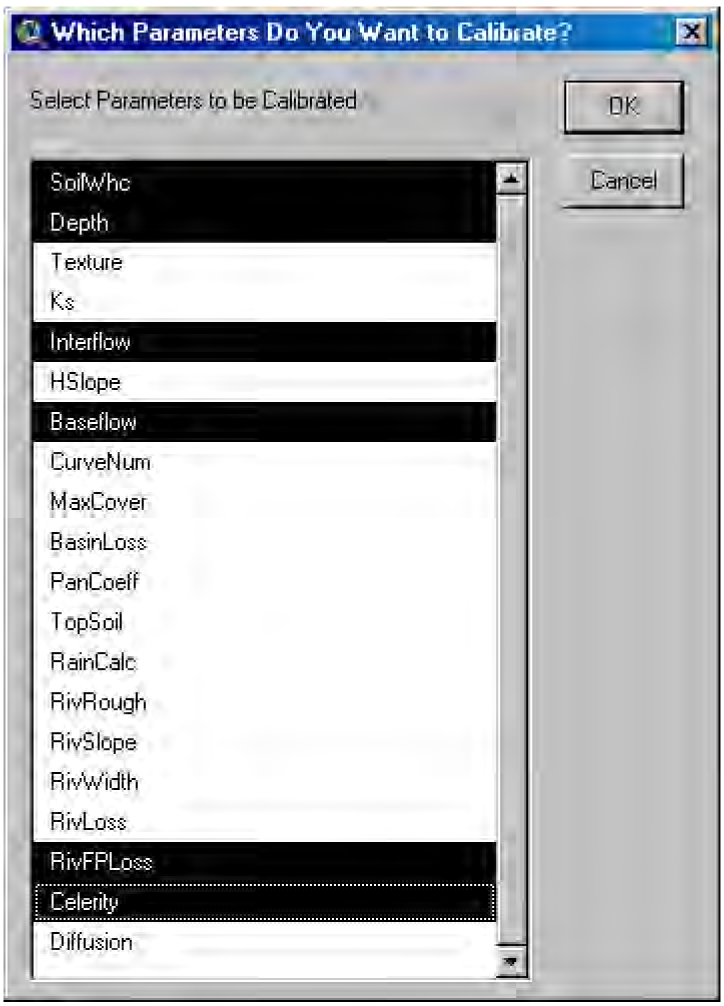
Copy the **observed_streamflow.txt** from the **geosfm\samples** directory and paste into your working directory—in this example **c:\GeoSFM\workspace**. In the next dialog box, select which observed streamflow stations will be used for calibration. In this example there are two stations; select **1** and click **OK**.



Next, select the basin ID for the streamflow station from the drop-down list. The streamflow station used for this tutorial is located in basin 140. Select **140** and click **OK**.

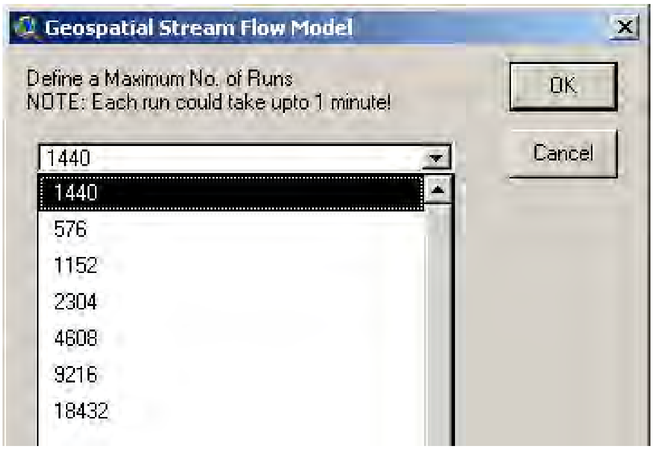


Next, the parameters to be calibrated are selected. While the results of the sensitivity analysis are important in this selection, your knowledge of the basin and your interpretation of the deficiencies of the current model are also important in deciding which parameters to calibrate. In this example, six parameters were chosen: SoilWhc, Depth, Interflow, BaseFlow, RivFPLoss, and Celerity. These six parameters were the most sensitive from the sensitivity analysis tutorial. Select **SoilWhc, Depth, Interflow, BaseFlow, RivFPLoss**, and **Celerity** from the parameter list by holding down the **Ctrl** key and clicking **OK**.

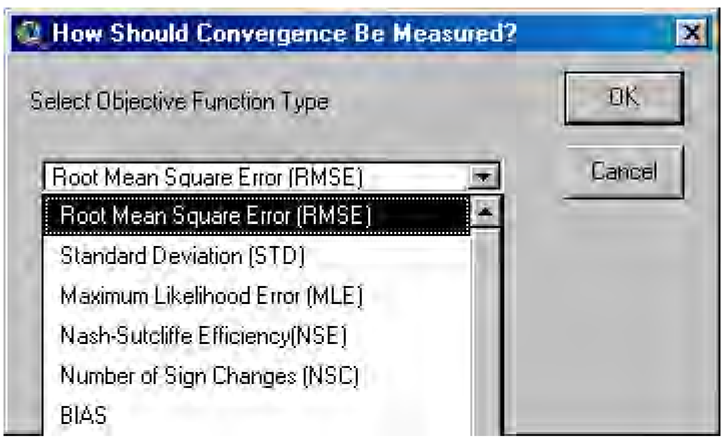


You are now prompted to **Define a Maximum No. of Runs**. The number of runs is dependent on the length of the streamflow record, the number of parameters being calibrated, and the complexity of the parameter dependent model response for the watershed being tested. In this tutorial, select **1440** as the default number of runs. There are other options in the drop-down list; you may select a smaller number for a shorter processing time. For optimum results, more model runs are better.

Click **OK**.

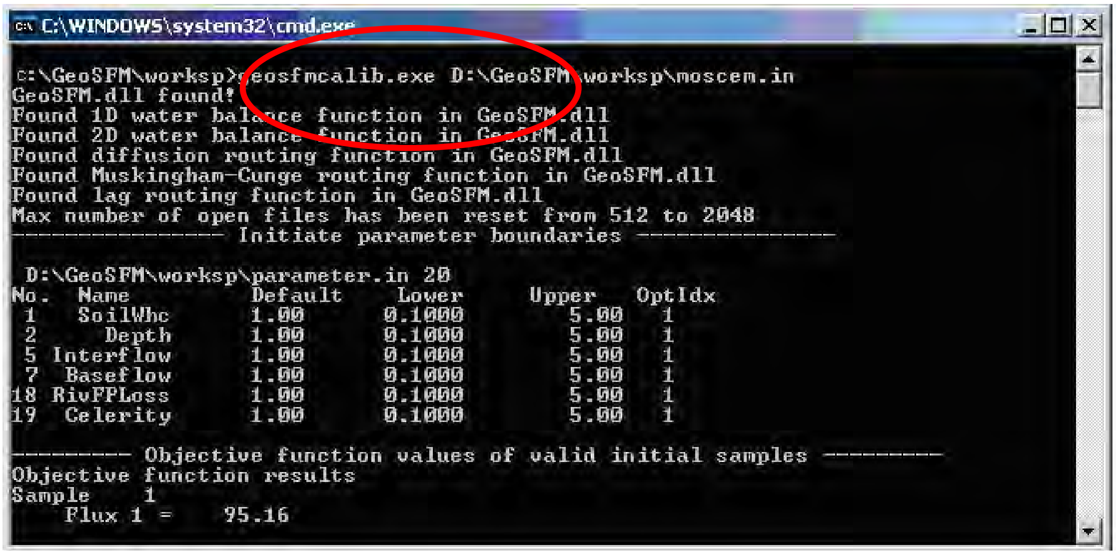


The **How Should Convergence Be Measured?** dialog box is displayed with a prompt. **Select Objective Function Type** and the **Root Mean Square Error (RMSE)**. Click **OK**.

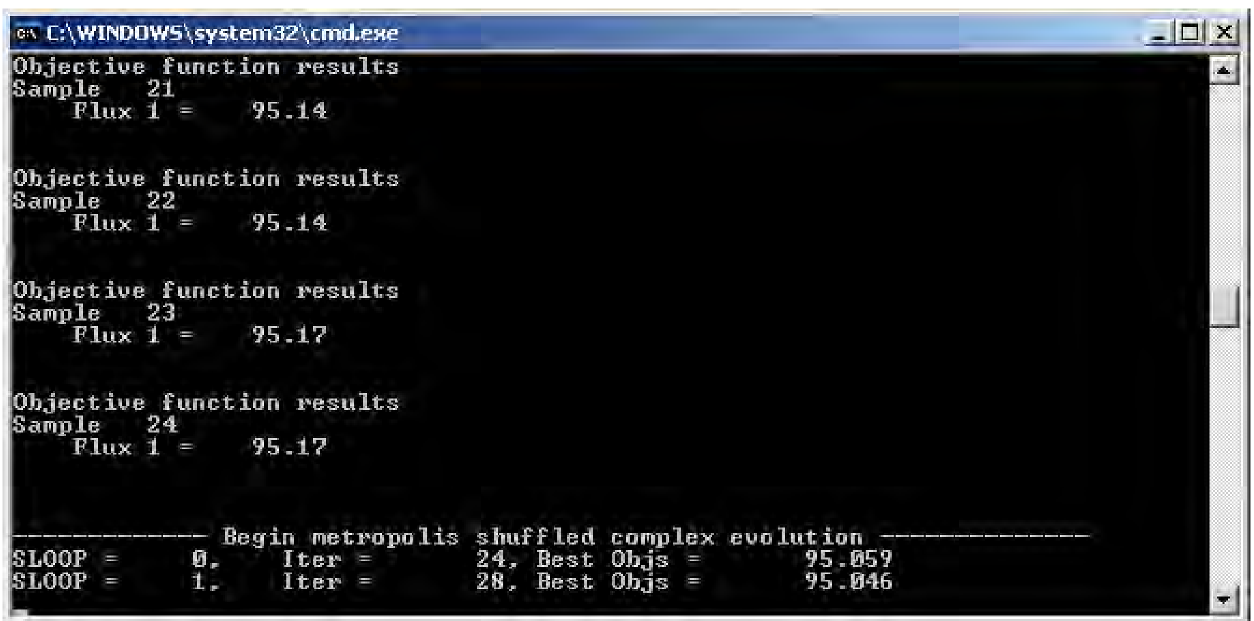


The following window will open and begin the calibration process. Notice the listed parameters. This process will take some time because it needs to run through 24 samples and 1,440 iterations.

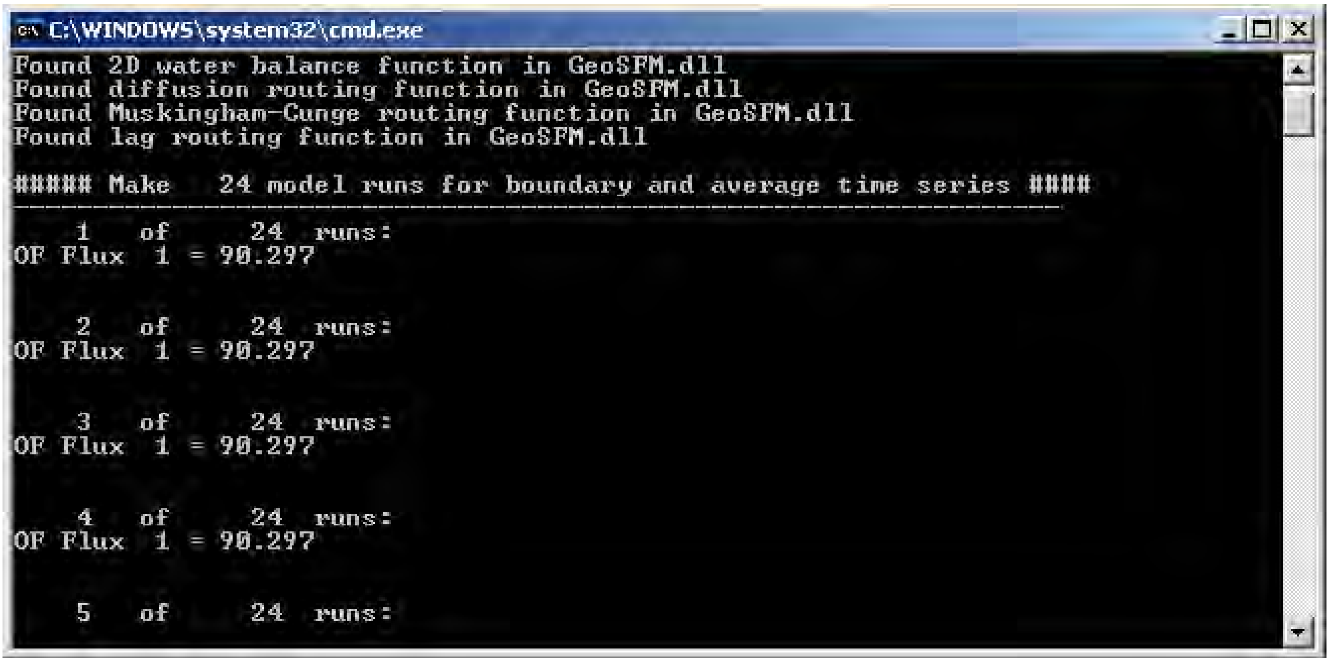
In this screen you see the calibration process is now running. The six parameters chosen are displayed along with the lower and upper values. This is the beginning of the objective function results for the samples. Each sample will display number of fluxes selected—in this example only one flux was selected.



When the sampling is completed, the next process is labeled "Begin metropolis shuffled complex evolution," which runs for 1,440 iterations.



The next window will open and start the postprocessing.



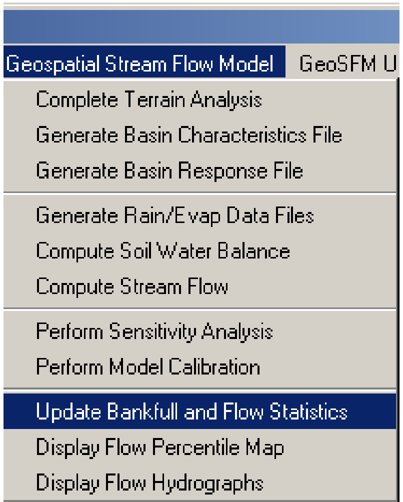
The postprocessing produces a time series file which lists the observed streamflow and calibrated streamflow output values. The **objectives_postproc.out** file lists the best calibration values in the time series file, and the **trdoff_bounds.out** displays the lower and upper boundaries with the average streamflow from the sample runs. New basin and river files (**basin.txt** and **river.txt**) containing the parameter set which resulted in the best fit between simulations and observations are also produced for use in subsequent model runs. You can also revert to the precalibration basin and river characteristics by restoring the information contained in the **basin_original.txt** and **river_original.txt** files.

Postprocessing Module

One major advantage of performing hydrologic modeling in a geospatial environment is that simulation results can be linked to river and basin feature representations, allowing the user to view and interact with the results in a visual environment. Three postprocessing routines are provided in GeoSFM to facilitate such interaction. The first routine computes characteristic flows for each river reach and stores the results in the associated river shapefile. The second routine displays flow results in color-coded maps by linking flow time series with the associated basin shapefile. The third routine allows hydrographs to be plotted with a simple click any river reach. Instructions for running these routines are described in the following sections.

Updating Bankfull and Flow Statistics

From the **Geospatial Stream Flow Model** menu, select **Update Bankfull and Flow Statistics** to compute characteristic flows for each river reach and store the results in an associated river shapefile.



Confirm the working directory as **c:\geosfm\workspace**. Click **OK**.

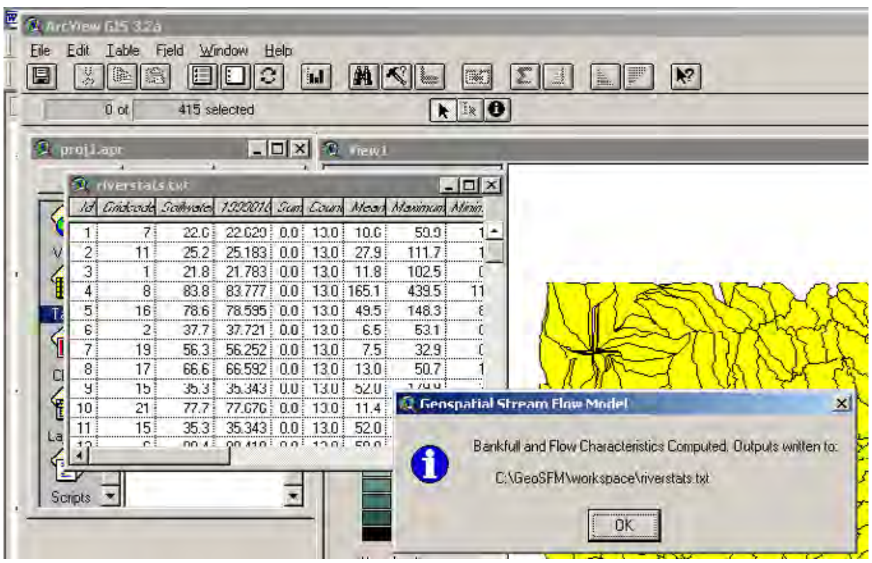
The **Basin Theme** dialog box displays asking you to **Select basin coverage/grid theme** from the drop-down list. Select **basply.shp** (basin polygon shapefile). Click **OK**.

Next, the **GeoSFM Utilities** dialog box opens with a prompt to **Enter Model Parameters**. The input file defaults to **streamflow.txt** and the two output files default to **monthlyflow.txt** and **annualflow.txt**. Click **OK**.

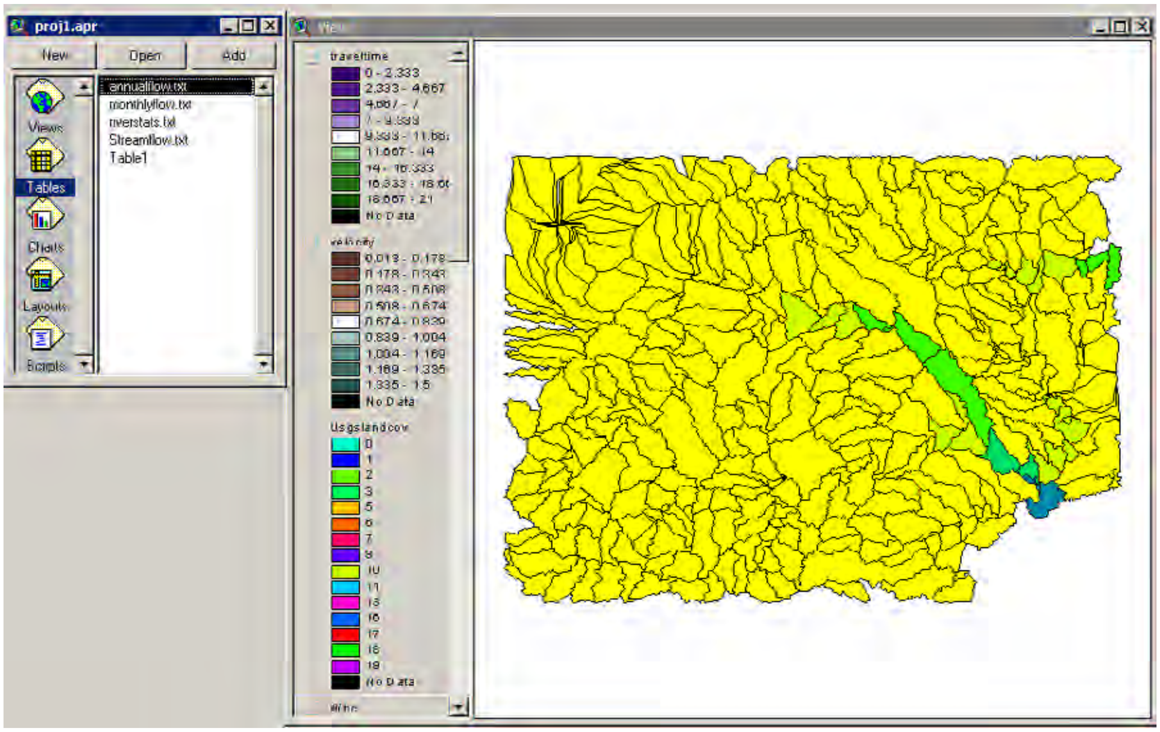
If less than 9 months of data are supplied, the following **Info** dialog box will open. Click **OK**.

The model will soon indicate that it has finished processing and has written the results to the **riverstats.txt** file in the working directory. The **riverstats.txt** file contains the statistical estimate of how much water is needed to fill a river channel (carrying capacity.)

Click **OK** in the **Bankfull and Flow Characteristics Computed** dialog box. Notice the **riverstats.txt** table is also displayed.

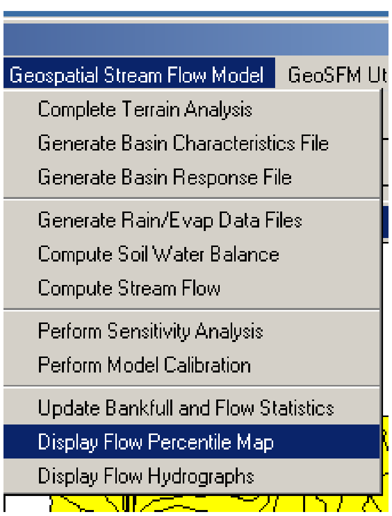


The View is updated and the **annualflow.txt, monthlyflow.txt, riverstats.txt**, and **streamflow.txt** files are added to ArcView as tables.

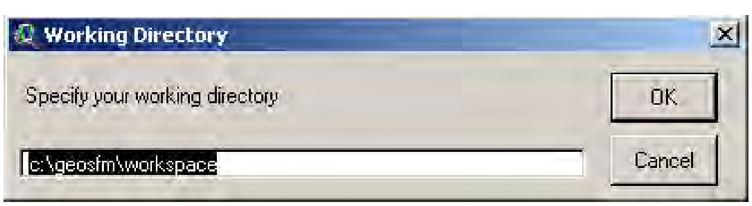


Display Flow Percentile Map

From the **Geospatial Stream Flow Model** menu, select the next item in the list: **Display Flow Percentile Map**.



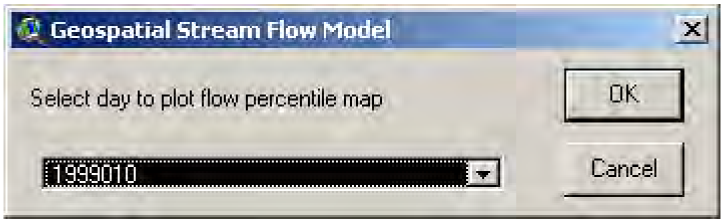
Confirm the working directory as **c:\geosfm\workspace**. Click **OK**.



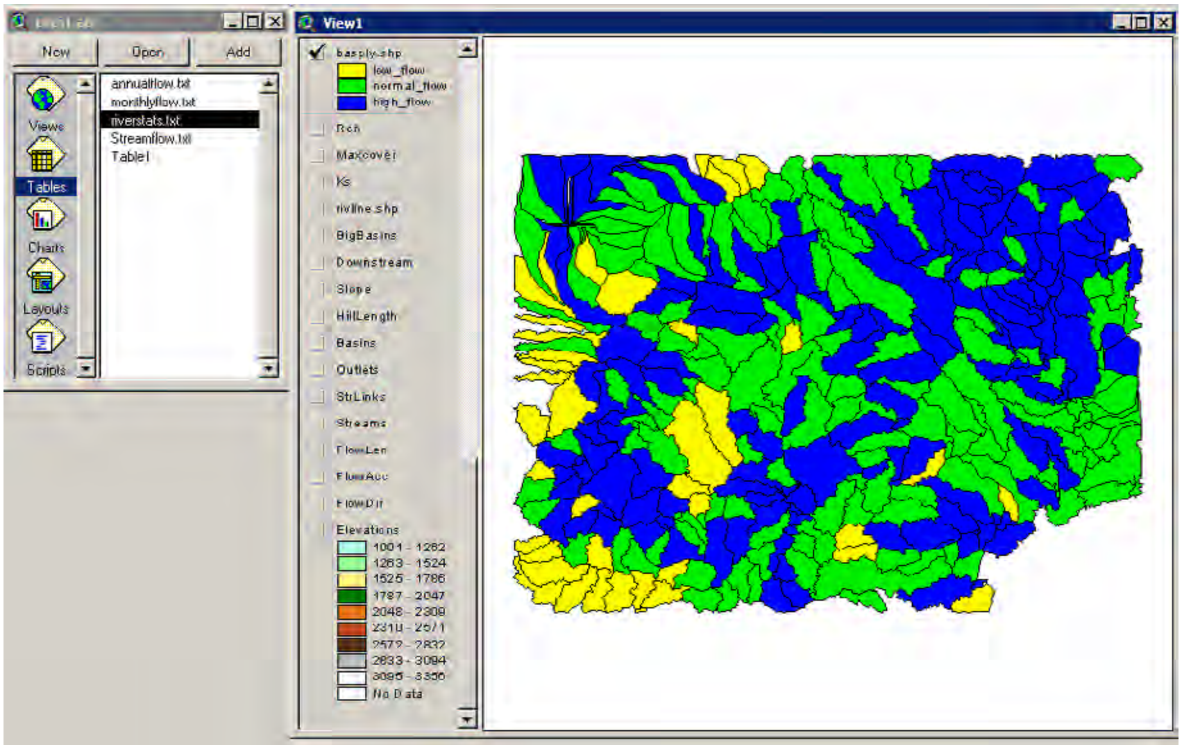
Select **Streamflow.txt** when prompted to **Select Input Data File** in the **Input Flow Time Series** dialog box. Click **OK**.

Select **basply.shp** when prompted to **Select basin coverage/grid theme** in the **Basin Theme** dialog box. Click **OK**.

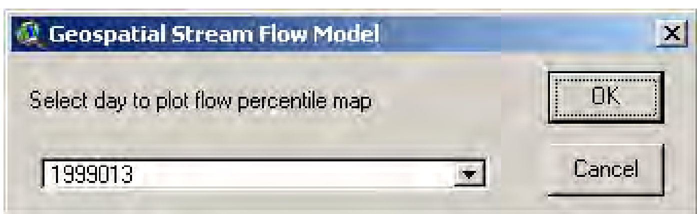
Next, at the prompt **Select day to plot flow percentile map**, choose 1999010 from the drop-down list. This example represents day 10 in the year 1999. Click **OK**.



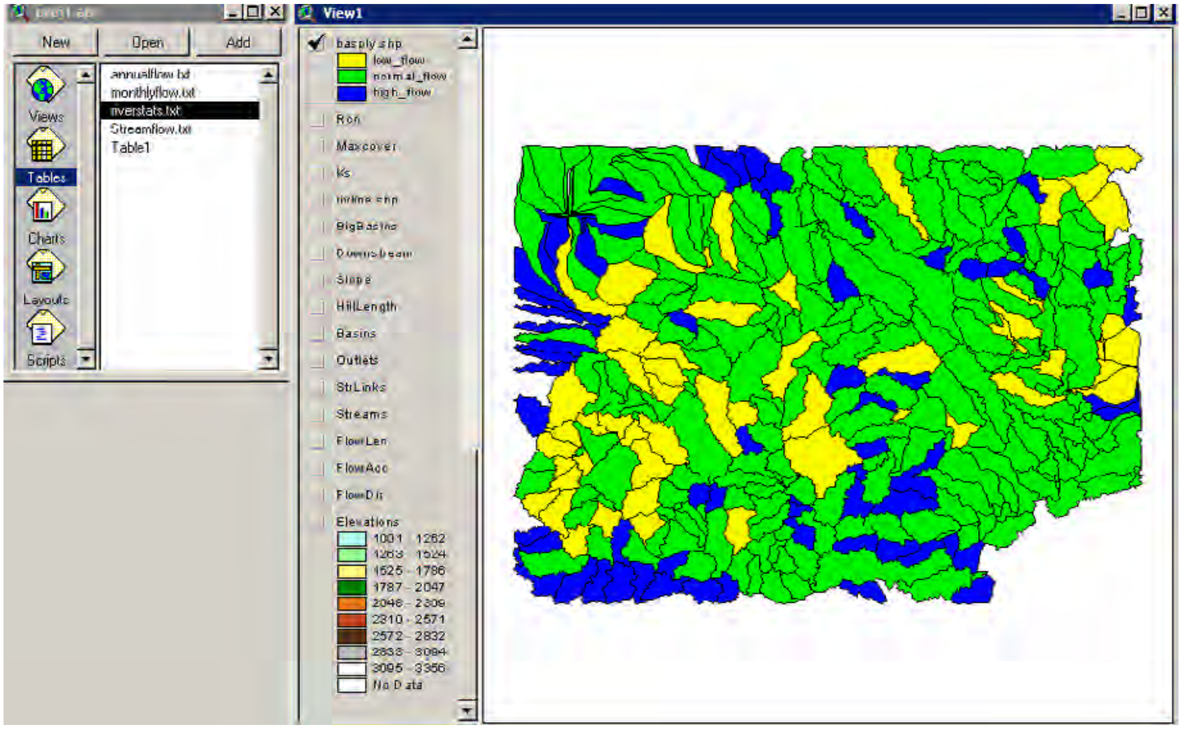
After a few seconds, a map will display. The basins are now classified into three categories: low flow, normal flow, and high flow.



For the next example, day 13 was selected. Click **OK**.



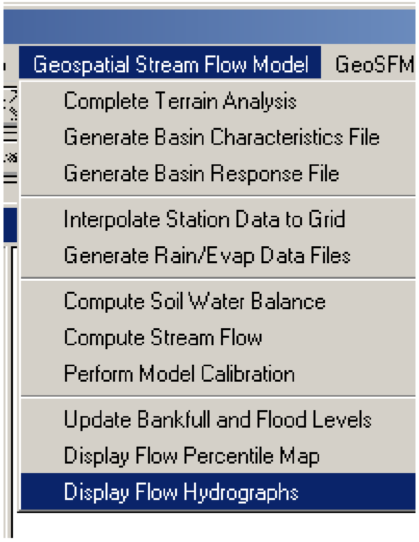
Results for day 13 are seen below.



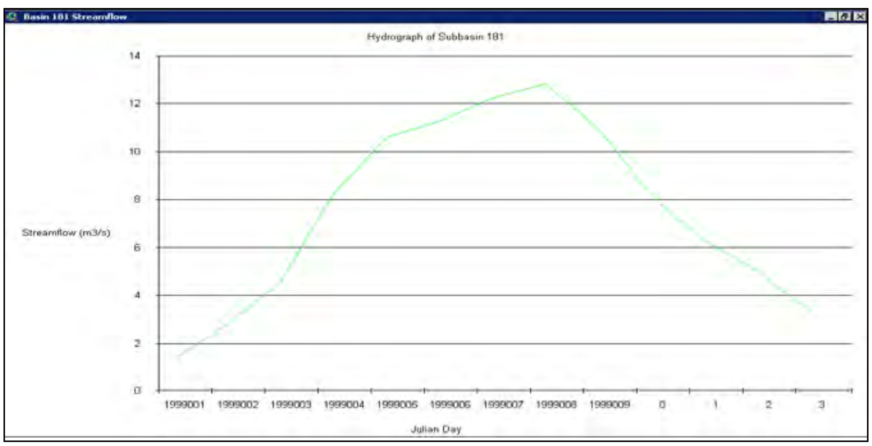
Notice the decreasing flow trend in many headwater subbasins between day 10 and day 13. Such images provide a visual representation of simulated flows, which allows you to quickly identify areas of concern within the study area.

Displaying Hydrographs

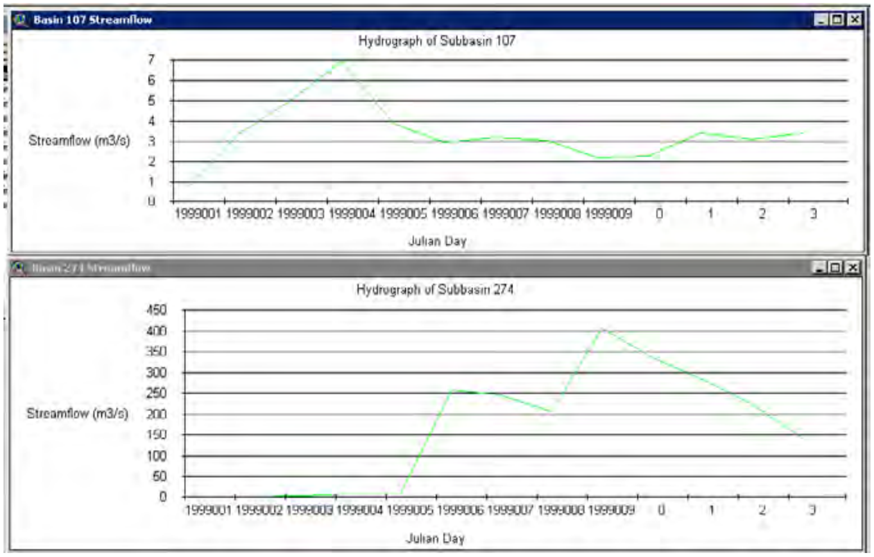
GeoSFM provides a tool for plotting and displaying hydrographs. You may also open the resulting file in a spreadsheet to plot out the hydrographs; Microsoft Excel spreadsheet software provides some additional plotting options, and it does a good job displaying a long time series. In the View, make sure that you have **basply.shp** selected and displayed in a raised box. To plot the hydrographs in ArcView, go to the **Geospatial Stream Flow Model** menu and select **Display Flow Hydrographs** to activate the plotting tool.



After you select **Display Flow Hydrographs,** your cursor changes to a "+". Now click in one of the subbasins for which you would like to create a hydrograph. The subbasin is now highlighted to show that it has been selected, and a window opens with the hydrograph displayed. An ArcView hydrograph is produced showing the daily variation of flow at the outlet of the selected subbasin.



Click several different subbasins to see how different hydrographs compare. See if you can observe the downstream propagation of flow by comparing hydrographs from the headwater subbasins to those farther downstream.



You can analyze questions such as

- Does the 3-day forecast indicate increasing or decreasing streamflows?
- Are the basins with high soil moisture the same basins with the most streamflow?

ArcView's graphing capabilities are limited in the number of data points that can be plotted and in the ability to manipulate graph elements to enhance the display. You may want to import individual simulation output files into other programs, such as Microsoft Excel spreadsheet software, for presentation quality graphs.

GeoSFM Utilities

A number of utilities are included in GeoSFM for its use in operational settings.

- RAINDATA tools for downloading and processing data produced at USGS EROS for daily GeoSFM runs
- GISDATA tools for performing routine tasks associated with comparing and integrating satellite-derived data in raster format with vector station data
- DEMDATA tools for removing spurious pits which may exist in elevation data sets other than the HYDRO1k
- TIMESERIES tools for processing output files from hydrologic analysis to determine representative flows at daily, monthly, and annual time scales

Instructions for using these utilities are given in the following sections.

Rain Data Tool for Downloading Data

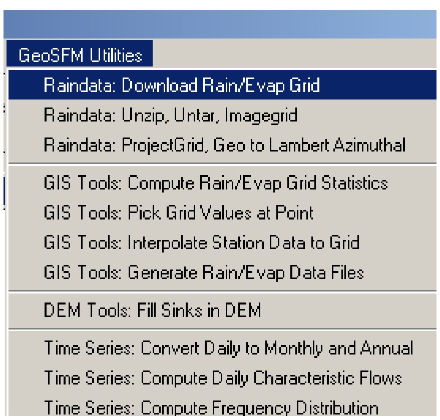
GeoSFM contains several menu items designed to assist the operational user in retrieving and processing daily rainfall and evapotranspiration data. The first **Raindata** option is **Download Rain/Evap Grid**, which uses file transfer protocol (FTP) and a live Internet connection to download data from the USGS EROS anonymous FTP server. A listing of the data sets available for download from the USGS EROS FTP site for 16 days after initial collection is provided in table 2.

Table 2. Operational geospatial time series data available for download from the USGS EROS FTP site.

Region	Data Source	Naming Convention	Comments
		Rainfall	
Global	NASA TRMM	TRMM<YY><MM><DD>g	
Africa	NOAA RFE	Rain_<YYYY><JJJ>g	Integrates in situ gauge data
Asia	NOAA RFE	Rain_<YYYY><JJJ>g	For limited project sites only
		Potential Evapotranspiration	
Global	USGS GPET[1]	Evap_<YYYY><JJJ>g	

[1]Global Potential Evapotranspiration data set from USGS EROS

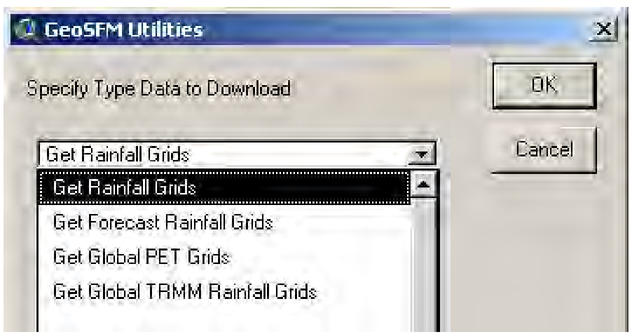
From the main toolbar, go to **GeoSFM Utilities** and select **Raindata: Download Rain/Evap Grid** from the drop-down list.



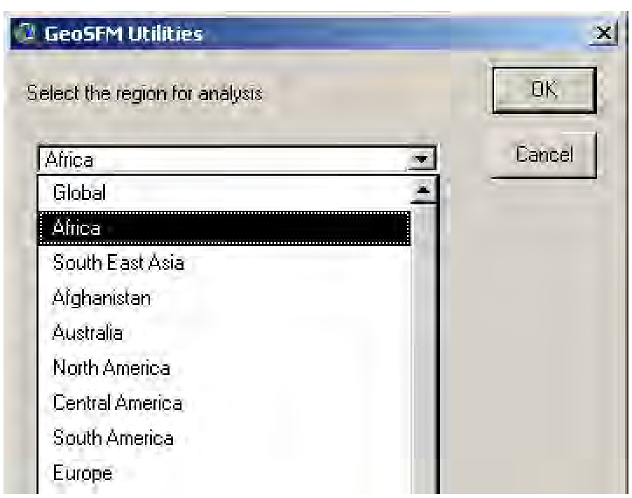
Specify your working directory. Click **OK**.

At the prompt, use the default entries. Click **OK**.

In the **Specify Type Data to Download** dialog box, select **Get Rainfall Grids** from the drop-down list. Click **OK**. (**Get Global PET Grids** would be selected for PET data.)

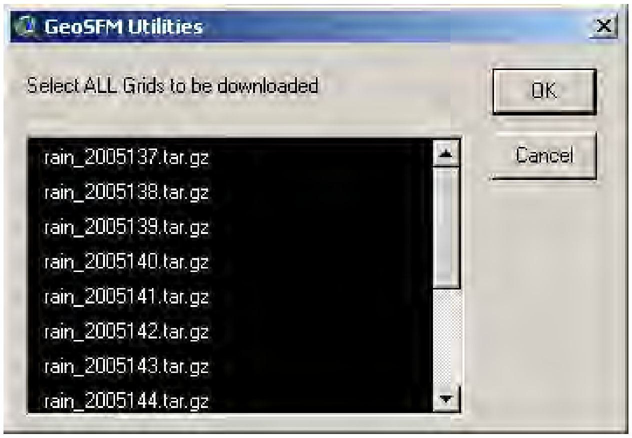


In the **Select the region for analysis** dialog box, select **Africa** from the drop-down list. Click **OK**. The latest available data will be displayed for the region selected.



Wait for the FTP transfer to complete (Internet connection must be open). The program writes a local file with the selected regional data sets. The available rain/evap data will be displayed for downloading. Generally, 16 days of data are available for downloading from the FTP site.

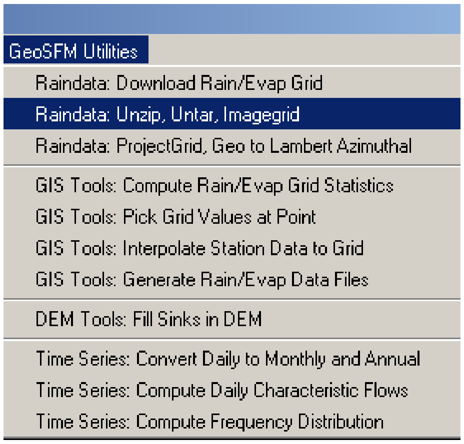
Select ALL Grids to be downloaded by clicking and highlighting each selection that you want to download. For this tutorial, 10 days of data were used. Click **OK**. The naming convention for rain data is rain_YYYYJJJ.tar.gz ("Y"–year, "J"–Julian day). The naming convention for evapotranspiration data is YYMMDD.tar.gz ("Y"–year, "M"–month, "D"–day).



The **GeoSFM Utilities** box appears with the message **Data Transfer Complete** and the number of files downloaded. All downloaded files will now be in your working directory. Click **OK**. This process can be repeated for downloading PET data.

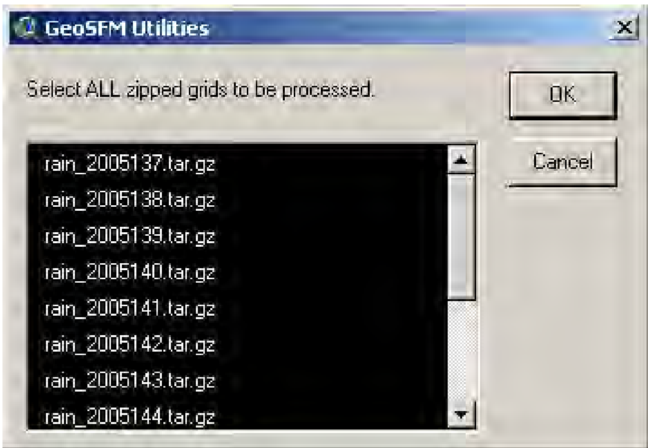
Rain Data Tool for Image to Grid Conversion

The second **Raindata** option is a tool for extracting and converting downloaded time series images into ArcInfo grid format. From the main toolbar, go to the **GeoSFM Utilities** tool and select **Raindata: Unzip, Untar, Imagegrid** from the drop-down list.

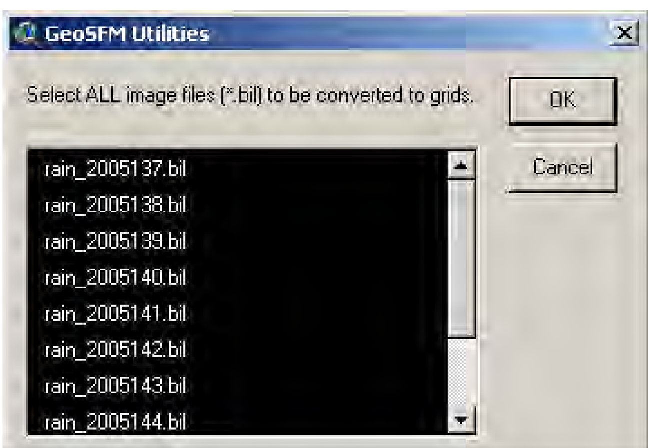


Specify your working directory. Click **OK**.

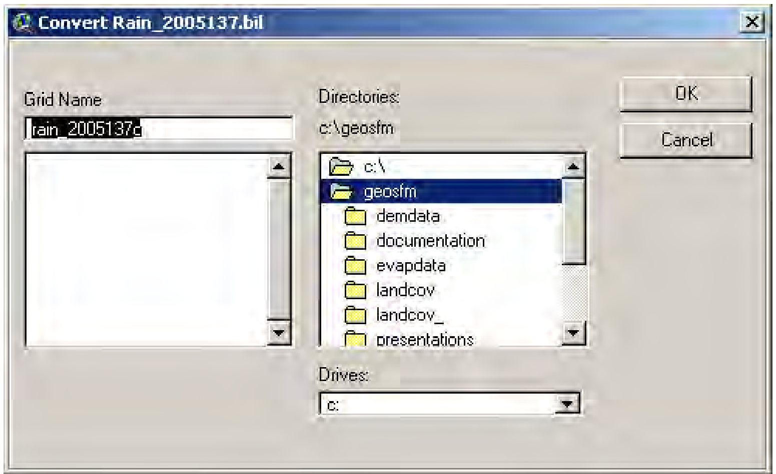
Select ALL zipped Grids to be processed by clicking and highlighting each selection that you want to process. Click **OK**.



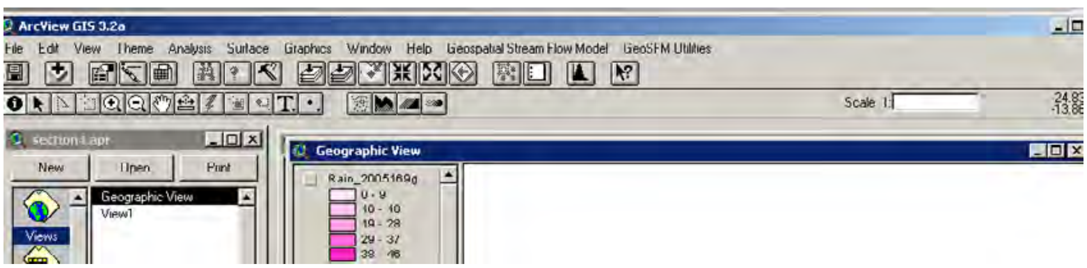
Next, **Select ALL image files (*.bil) to be converted to grids** by clicking and highlighting each item. Click **OK**.



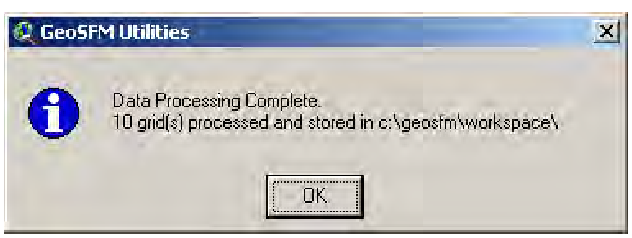
A dialog box will appear for each file; confirm your directory. Click **OK** for each dialog box—one for each rain file to download.



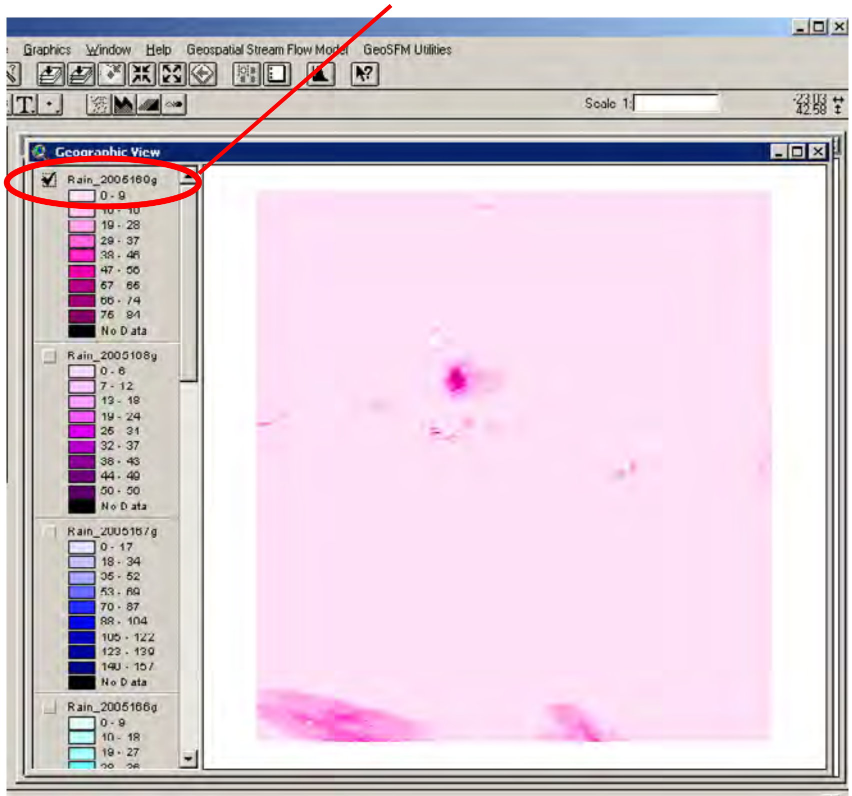
At the same time, the rain grids will be added to the table of contents in a new View labeled **Geographic View**.



When all processing is complete, the **GeoSFM Utilities** dialog box will appear, stating **Data Processing Complete**, along with the number of grids processed and directory information. This process can be repeated for PET data. The grid naming convention for raindata is Rain_YYYYJJJg (for example, Rain_1999001g), and for evapotranspiration data Evap_YYYYJJJg (for example, Evap_1999001g).

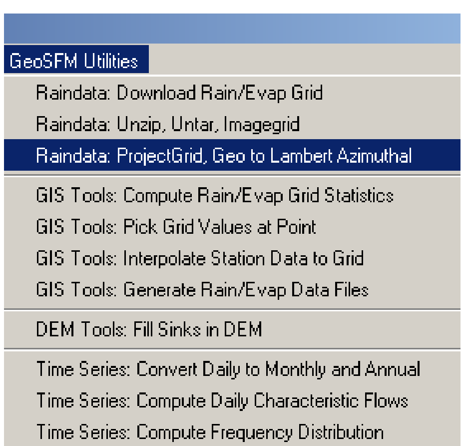


All **Rain_YYYYJJJg** themes are added to the table of contents as seen below. (A *g* at the end of a grid name indicates the geographic projection.)



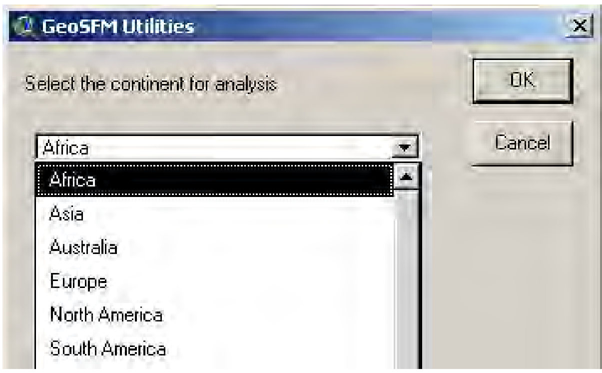
Rain Data Tool for Projecting Grids

From the main toolbar, go to the **GeoSFM Utilities** tool and select **Raindata: ProjectGrid, Geo to Lambert Azimuthal** from the drop-down list to project the grids from geographic to the Lambert Azimuthal projection.

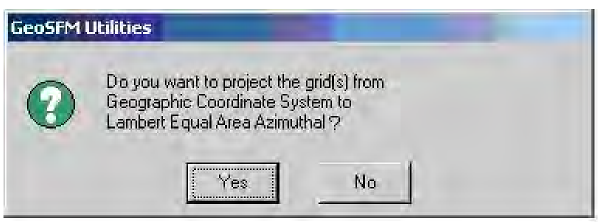


Specify your working directory. Click **OK**.

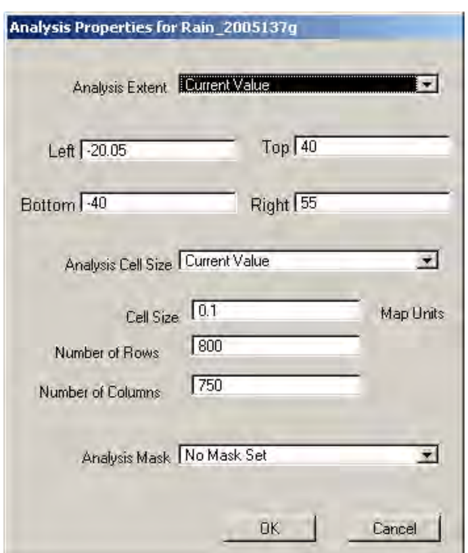
In the **Select the continent for analysis** dialog box, select **Africa** from the drop-down list. Click **OK**.



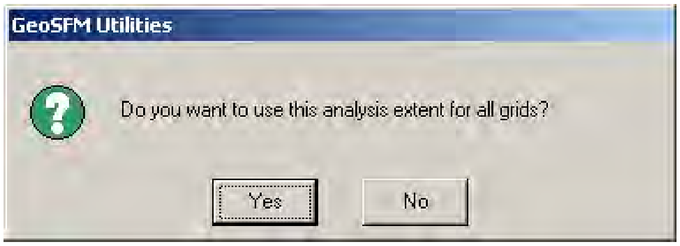
Select **Yes** in the **Do you want to project the grid(s) from Geographic Coordinate System to Lambert Equal Area Azimuthal** dialog box.



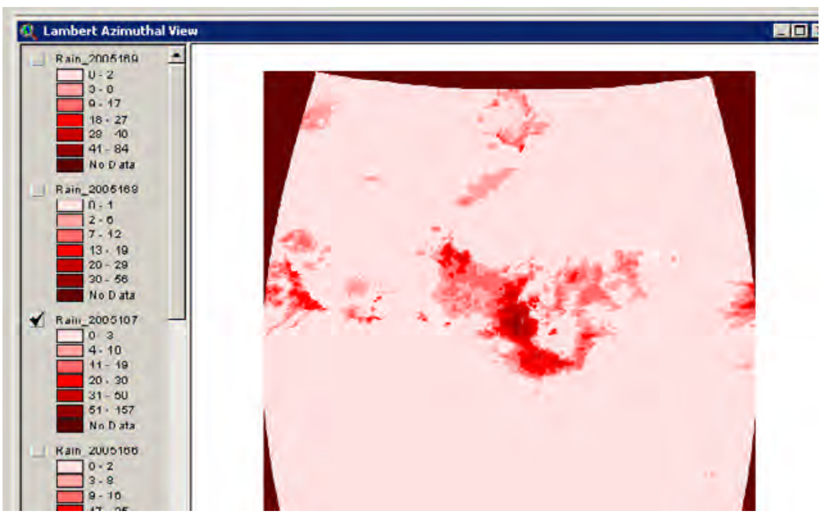
Next, the **Analysis Properties for Rain_YYYYJJJg** is displayed; accept the default values and click **OK**.



Select **Yes** when prompted **Do you want to use this analysis extent for all grids?**



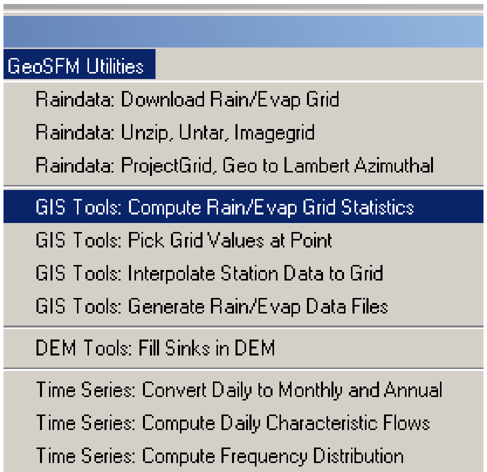
After a few seconds, the **Lambert Azimuthal View** window opens with the **Rain_YYYYJJJ** themes projected in **Lambert Azimuthal Equal Area** projection. (Notice the *g* at the end of the grid name is no longer displayed.)



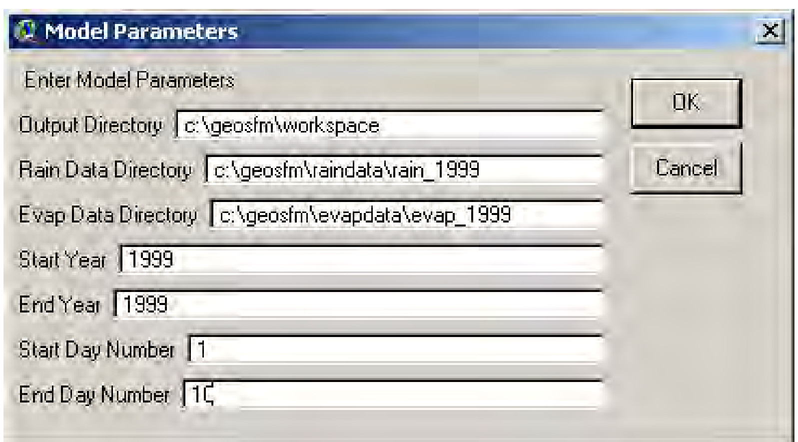
Now the **Rain_YYYYJJJ** grids can be added to the View when prompted during the **Generate Rainfall and Evaporation Basin Files**.

GIS Tool for Computing Grid Statistics

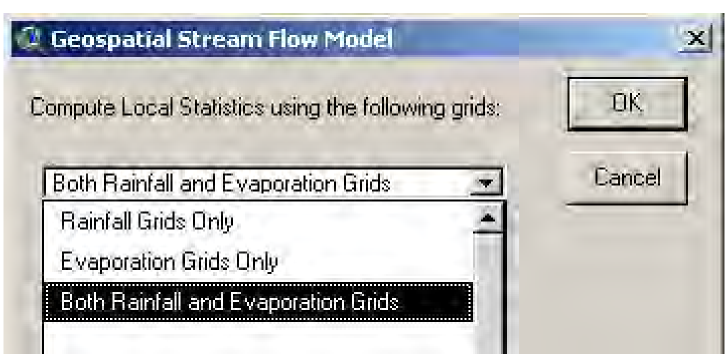
GeoSFM contains a number of tools designed to facilitate some commonly performed operations. From the **GeoSFM Utilities** menu, select **GIS Tools: Compute Rain/Evap Grid Statistics**.



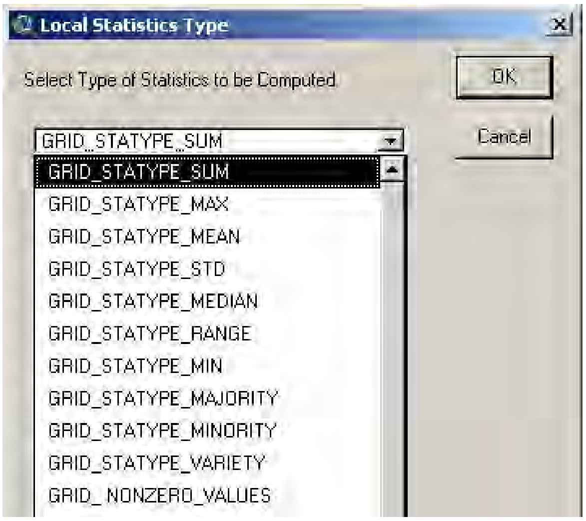
The **Model Parameters** dialog box is displayed; confirm the **Output Directory** and type in the correct path for the **Rain Data Directory** and **Evap Data Directory**. Confirm the **Start Year** and **End Year** fields. Enter the **Start Day Number** and the **End Day Number** field for the time frame you want to compute. In this example, the **End Day Number** was changed to 10 from the default value for faster processing time. Click **OK**.



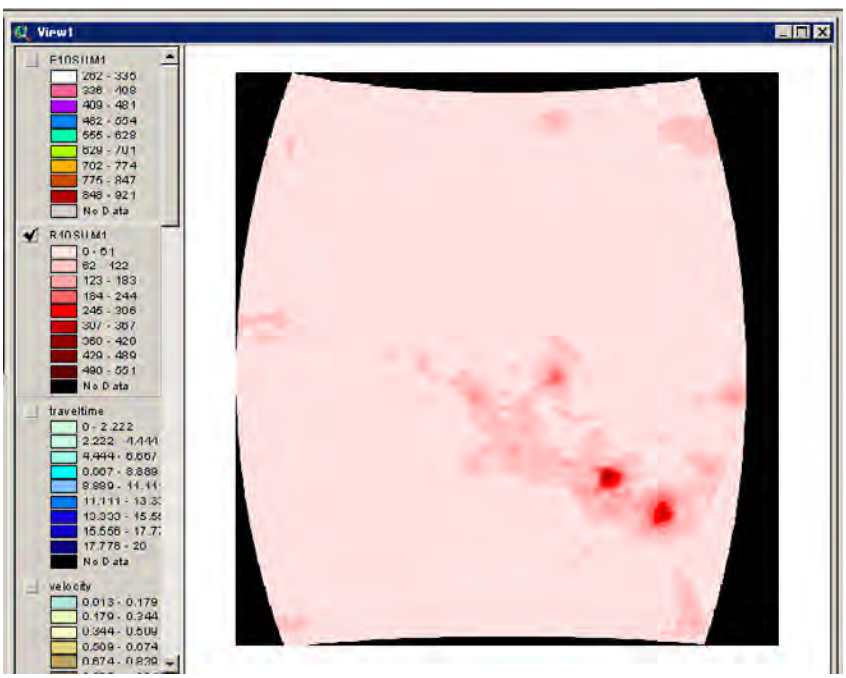
At the prompt, select **Both Rainfall and Evaporation Grids** to compute local statistics. Click **OK**.



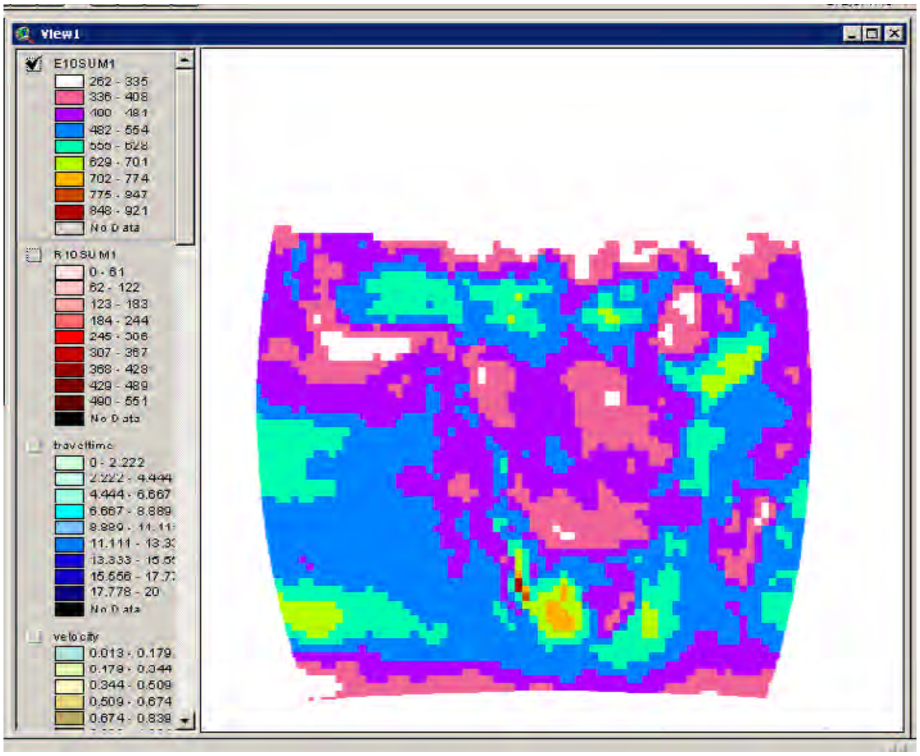
Next, the **Local Statistics Type** dialog box appears with a list of various statistics that can be computed. Select the statistic to be computed from the drop-down list. For this tutorial, select **GRID_STATYPE_SUM**, which will display the sum rain/evap for the 10-day date range defined in the **Model Parameters** dialog box. Click **OK**.



After a few seconds for processing, the R10SUM1 Theme (sum of the rain data for 10 days) is displayed.



After a few more seconds if it's correct, the E10SUM1 Theme (sum of the evap data for 10 days) is displayed.



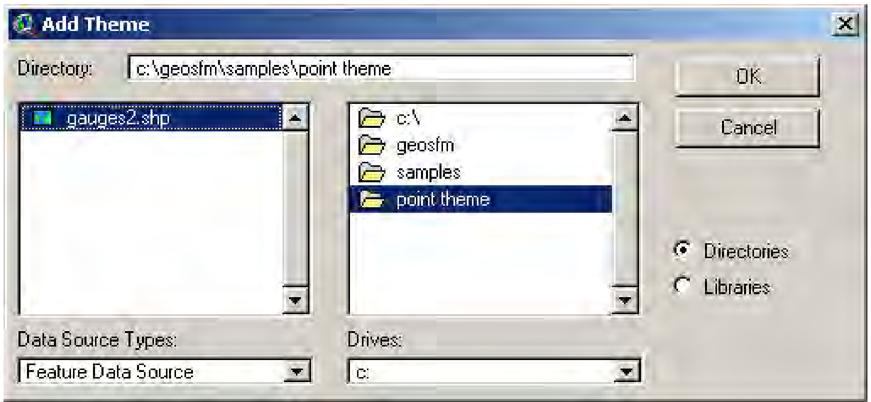
Run through the process again and select a different type of statistic to compute; compare the differences in the displayed data.

GIS Tool for Point Sampling of Grids

The second menu item is **GIS Tools: Pick Grid Values at Point**. Use this menu option to extract rainfall and evaporation grid data at points into a shapefile/coverage. To begin this process, a point theme is required. Instructions for creating such a theme are provided in Appendix 5 of this document. For completing this tutorial, a point theme is provided in the **Geosfm\samples\point theme** directory.

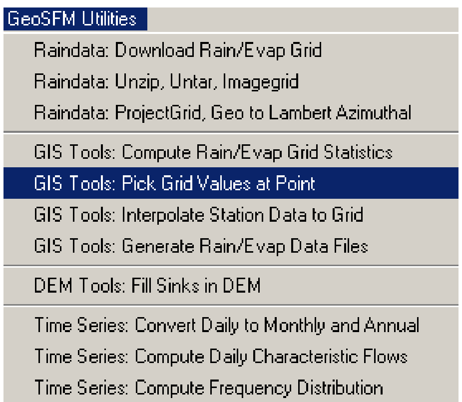
First, to add a **Theme**, click the **Add Theme** button . From the **Add Theme** dialog box, navigate to the **c:\geosfm\samples\point theme** directory.

In the **Data Source Types** box, make sure **Feature Data Source** is selected. Highlight **gauges2.shp** and click **OK**.



This shapefile contains randomly defined points just for use in this tutorial; these points do not reflect actual rain gauge station sites. This is to help you get an understanding of the functionality available to you through the GeoSFM Utilities tool.

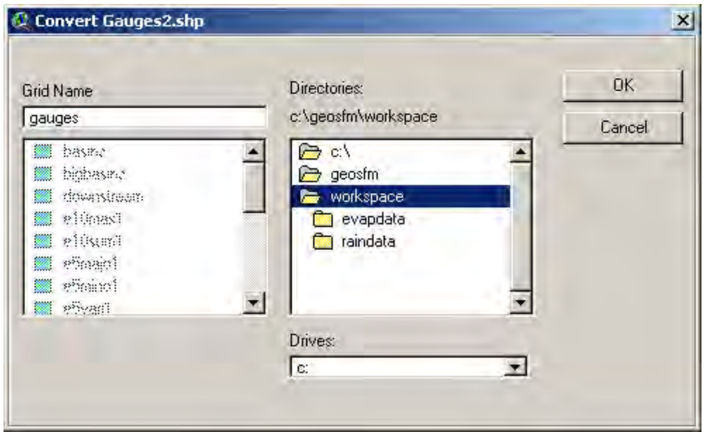
From the **GeoSFM Utilities** menu, select **GIS Tools: Pick Grid Values at Point.**



Confirm your working directory. Click **OK**.

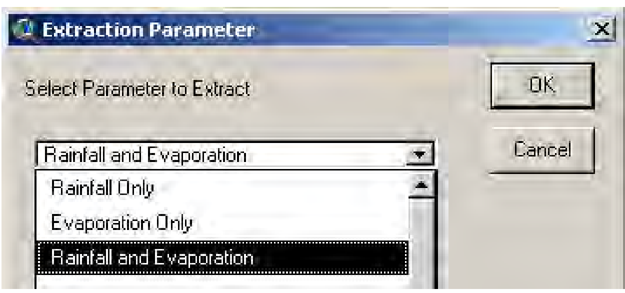
Next, at the prompt to **Select the Point Theme to be used**, select **gauges2.shp** from the drop-down list. Click **OK**.

The **Convert gauges2.shp** dialog box opens. Convert the shapefile to a grid. Navigate to the **c:\geosfm\workspace** directory. Name your new grid in the **Grid Name** field. Click **OK**.

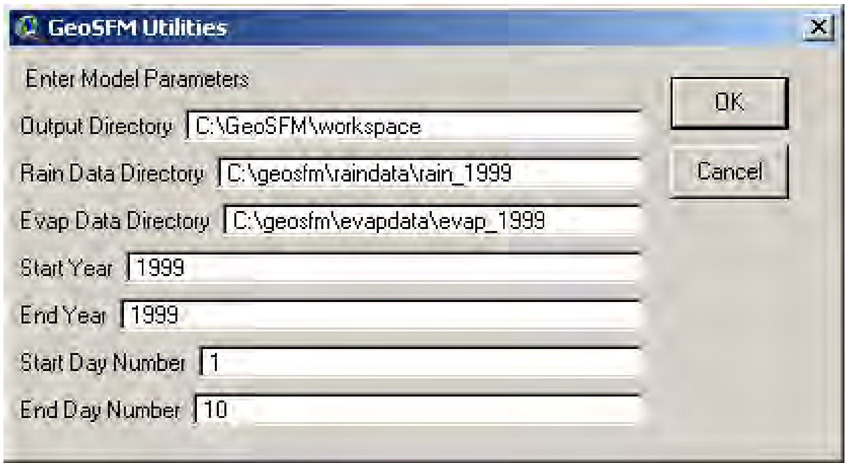


In the **Conversion Field: gauges2.shp** dialog box at the prompt to **Pick field with output header ID values**, select **Stations_i**. Click **OK**.

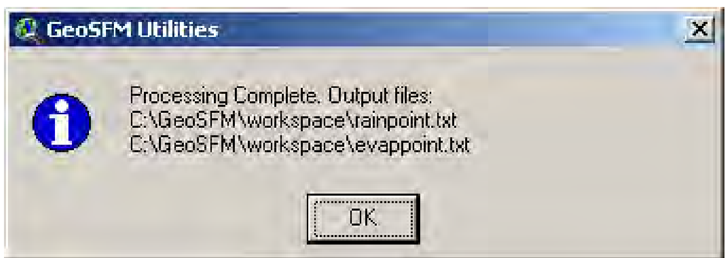
Next, at the prompt to **Select Parameter to Extract**, for this tutorial select **Rainfall and Evaporation**. Click **OK**.



At the **Enter Model Parameters** prompt, confirm the **Output Directory** and type in the correct paths for the **Rain Data Directory** and **Evap Data Directory**. Confirm the **Start Year** and **End Year** fields. Enter the **Start Day Number** and the **End Day Number** for the time frame you want. In this example, the **End Day Number** was changed to 10 from the default value for faster processing time. Click **OK**.



It will take a few seconds for processing. When the processing is complete, a dialog box will display showing the path of the two new output files. Click **OK**.

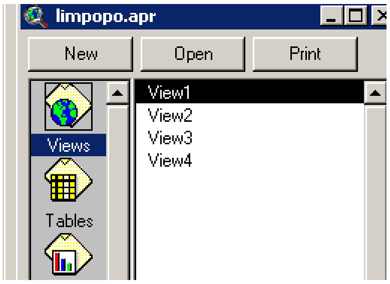


These two files contain the rain and evaporation data for the user-defined points (gauges) for the first 10 days in 1999.

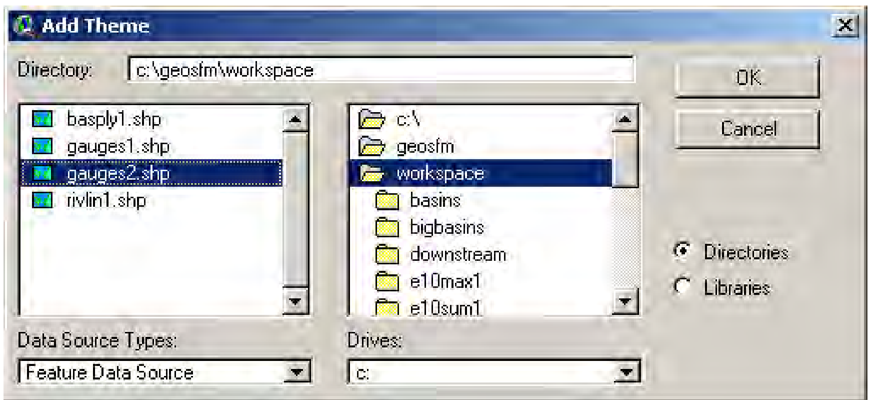
GIS Tool for Interpolating Point Data

The third **GIS Tools** menu item listed in the **GeoSFM Utilities** is **Interpolate Station Data to Grid**. This function is used when there are areas over which spatially distributed precipitation data is not available. Data interpolation routines convert station readings into a continuous surface. Both rainfall and evaporation tables are required. These tables need to contain station IDs along with daily amounts of rainfall/evaporation for each station. Instructions for creating these tables are included in the appendices of this Users Manual. For the purposes of this tutorial, tables are provided in the **c:\geosfm\samples\stations** directory.

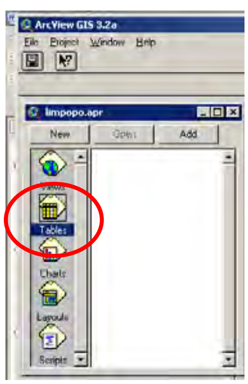
Create two new folders in your working directory labeled **Raindata** and **Evapdata**; this will help in managing the data files that will be created from this process. Start by opening a new View by clicking **View** and the **New** button from the project window.



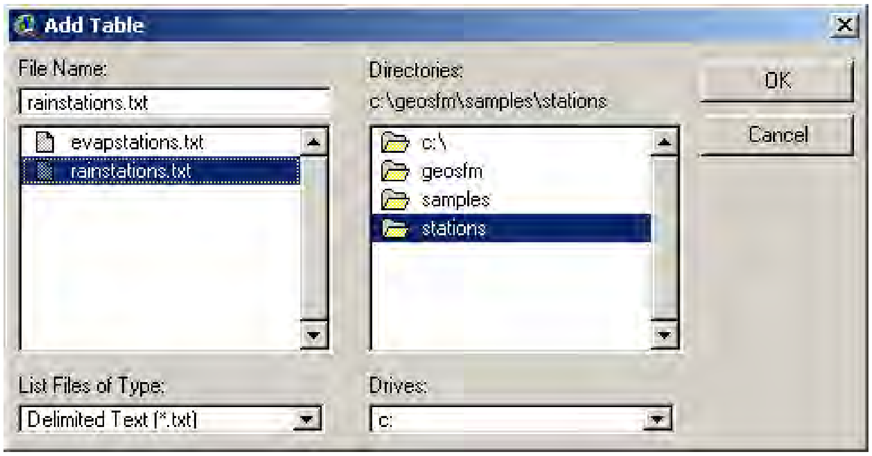
Add the **gauges2.shp** file to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Feature Data Source**. Click the **gauges2.shp** file from your working directory and click **OK** to add to the View.



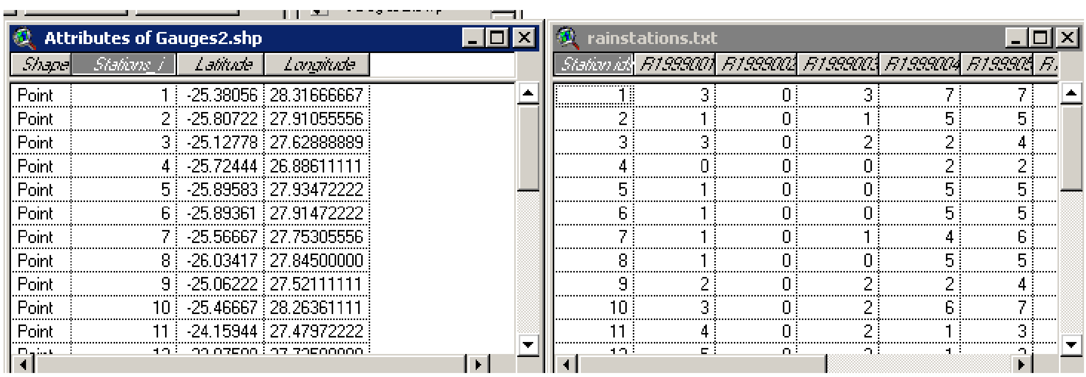
Next, add the rain station table to ArcView by selecting **Tables** and clicking the **Add** button seen below.



Navigate to your working directory. In the box to **List Files of Type**, select **Delimited Text (*.txt)** and select your .txt file (in this example, **rainstations.txt**). Click **OK**.

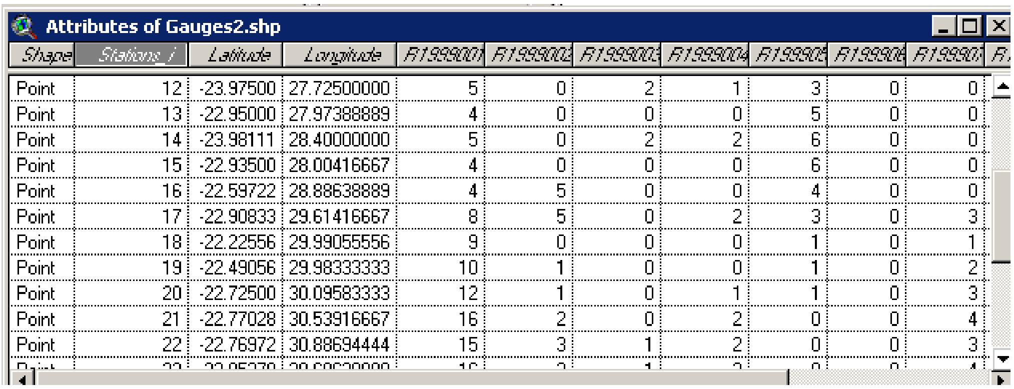


Next, join the new **rainstations.txt** file to the attribute table of the **gauges2.shp** file. Arrange both tables as seen below. Highlight the common column headers in both tables—**Stations_i** in **Attributes of Gauges2.shp** and **Station ids** in **rainstations.txt**. Make sure the destination table is active (adding fields to **Attributes of Gauges2.shp**).



Click the **Join** icon. The new columns are added to the **Attributes of Gauges2.shp** table.

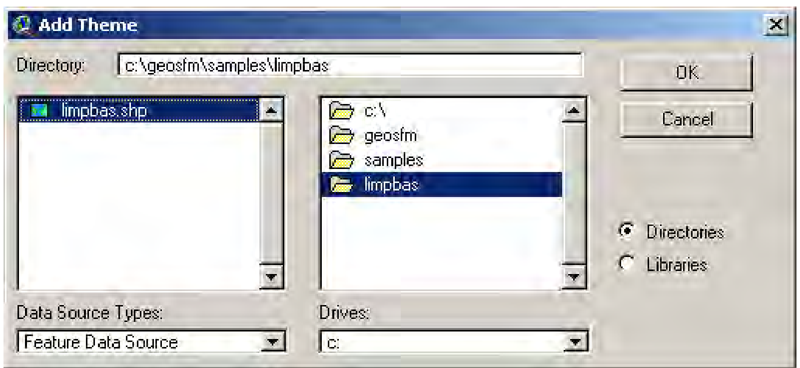
The new rain data fields are also added to the attribute table.



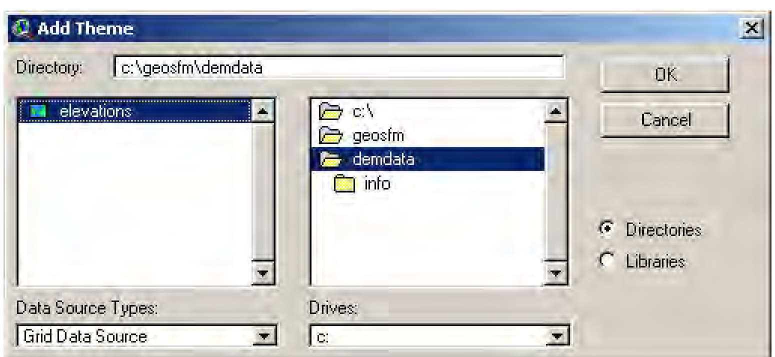
Shape	Stations_i	Latitude	Longitude	R1999001	R1999002	R1999003	R1999004	R19990	R19990	R19990	R
Point	12	-23.97500	27.72500000	5	0	2	1	3	0	0	
Point	13	-22.95000	27.97388889	4	0	0	0	5	0	0	
Point	14	-23.98111	28.40000000	5	0	2	2	6	0	0	
Point	15	-22.93500	28.00416667	4	0	0	0	6	0	0	
Point	16	-22.59722	28.88638889	4	5	0	0	4	0	0	
Point	17	-22.90833	29.61416667	8	5	0	2	3	0	3	
Point	18	-22.22556	29.99055556	9	0	0	0	1	0	1	
Point	19	-22.49056	29.98333333	10	1	0	0	1	0	2	
Point	20	-22.72500	30.09583333	12	1	0	1	1	0	3	
Point	21	-22.77028	30.53916667	16	2	0	2	0	0	4	
Point	22	-22.76972	30.88694444	15	3	1	2	0	0	3	

Repeat this process for the evaporation data. You can choose to process rainfall and evaporation data separately or at the same time. The attribute table will need all the necessary data for the selection made during processing.

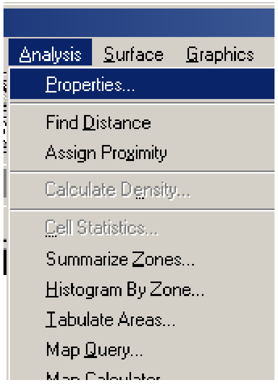
Click the **Add Theme** button  to add the Limpopo Basin shapefile. Change the **Data Source Types** to **Feature Data Source**. Add the shapefile named **limpbas.shp** from the **c:\GeoSFM\samples\limpbas** directory. Click **OK** to add shapefile to the View.



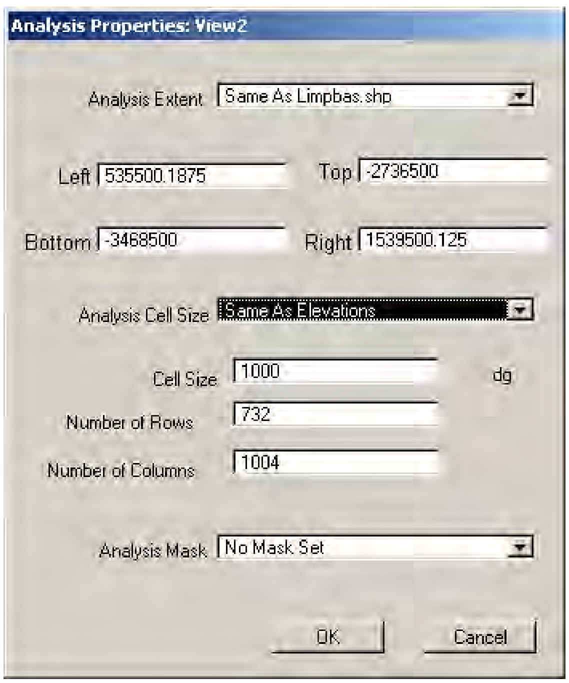
Next, add the **elevations** grid to the View using the **Add Theme** button . Change the **Data Source Types** to **Grid Data Source**. Click **elevations** from the **c:\GeoSFM\demdata** directory and click **OK** to add the DEM to the View.



Next, set the analysis environment from the **Analysis** menu by selecting **Properties**.

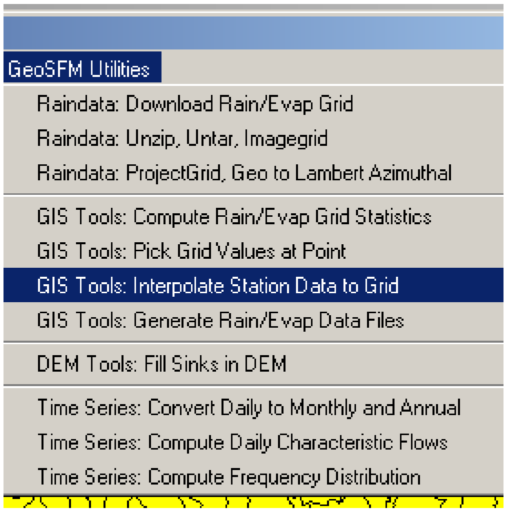


Change the **Analysis Extent** to **Same As Limpbas.shp** and the **Analysis Cell Size** to **Same As Elevations**. All other parameters will adjust themselves. Click **OK**.

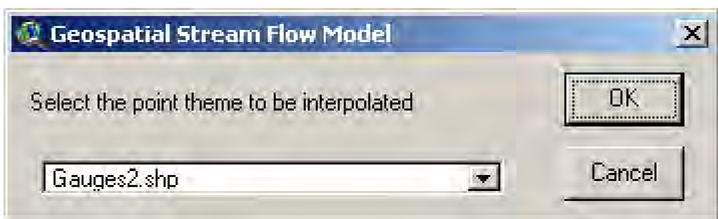


Now you are ready to start the process.

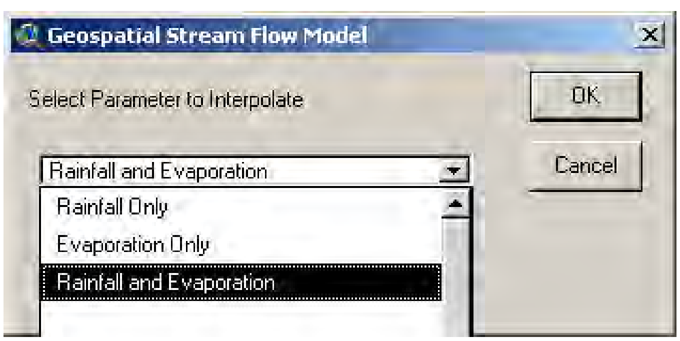
From the **GeoSFM Utilities** menu, select **GIS Tools: Interpolate Station Data to Grid**.



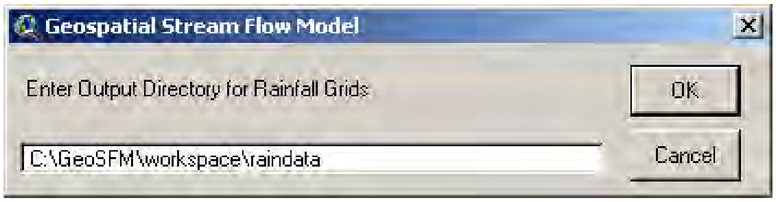
Select the point theme to be interpolated from the drop-down list. In this example, select **Gauges2.shp**. Click **OK**.



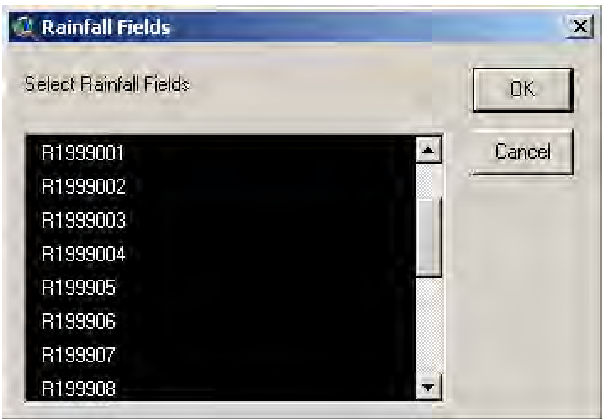
Select Parameter to Interpolate—in this example, select **Rainfall and Evaporation**. Click **OK**. If the evapotranspiration data are already provided as grids, they do not need to be interpolated in this tutorial. If the evapotranspiration data were in a text type formatted file, the data will need to be interpolated.



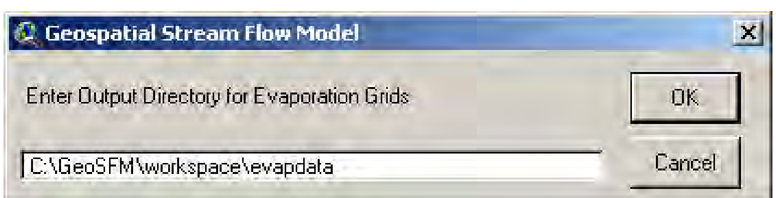
Enter the output directory for rainfall grids in a new raindata folder created in **c:\GeoSFM\workspace\raindata**. Click **OK**.



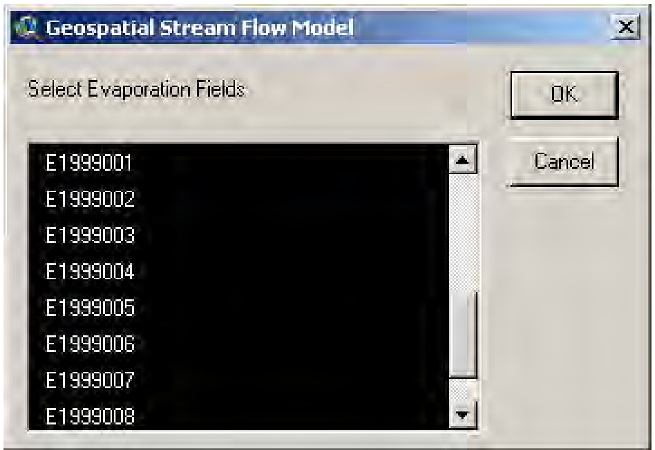
At the **Select Rainfall Fields** prompt, select the fields that contain the rainfall values. Select the 10 days from the list, **R1999001** through **R1999010**. You may select any number of days for which data are available. In this tutorial, select all 10 listed. Click **OK**.



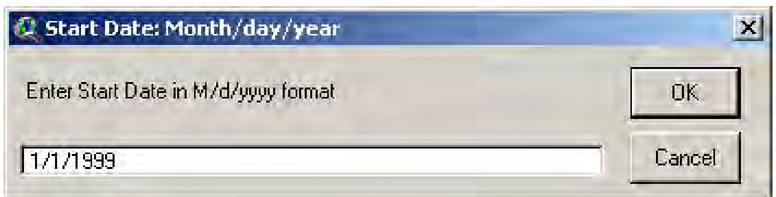
Enter the Output Directory for Evaporation Grids in a new evapdata folder created in **c:\GeoSFM\workspace\evapdata**. Click **OK**.



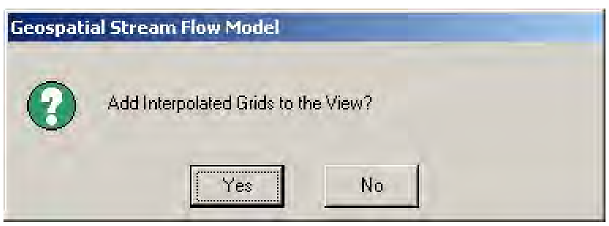
At the **Select Evaporation Fields** prompt, select the fields that contain the evaporation values. Select the 10 days from the list, **E1999001** through **E1999010**. Again, you may select any number of days for which data are available. For this tutorial, select all 10 listed. Click **OK**.



In the **Start Date** dialog box, change the default start date to **1/1/1999** to match the date of the first rainfall/evap fields selected above. The first field selected was **1999001** and represents **1/1/1999**. All the grids created will be based on the Julian day. Click **OK**.

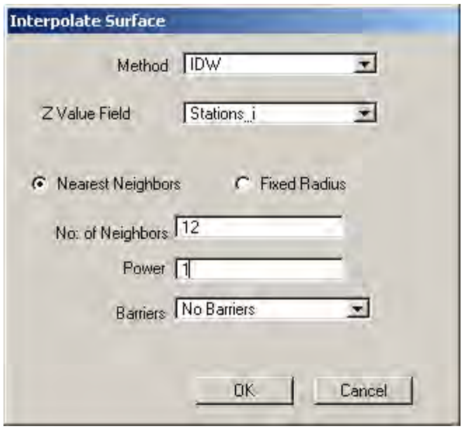


When asked if you would like to **Add Interpolated Grids to the View**, click **Yes**.

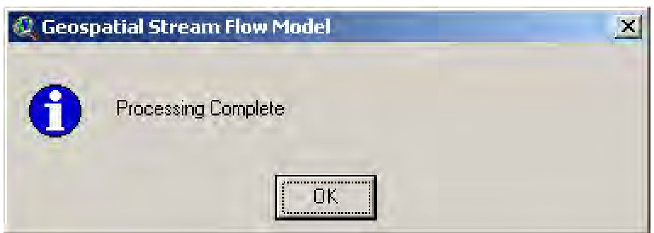


Next, the **Interpolate Surface** dialog box is displayed.

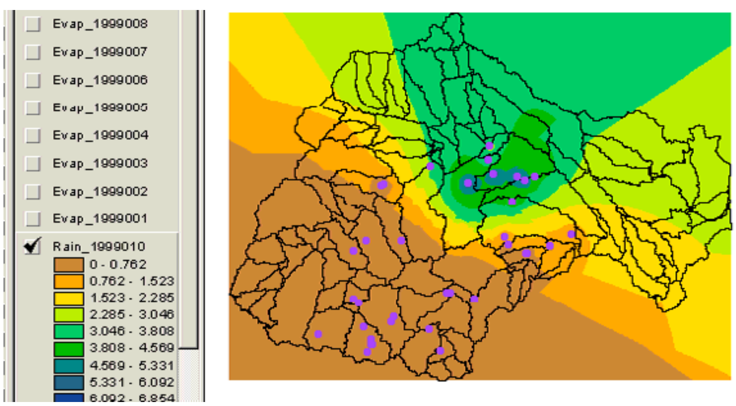
Populate:
Method: options are **IDW** and **Spline**, select **IDW** (Inverse Distance Weighting)
Z Value Field: select the default, **Stations_i**
Nearest Neighbors or **Fixed Radius:** select **Nearest Neighbors**
No: of Neighbors: select the default, **12**
Power: change the **Power** field to **1** to create a smoother surface (default is 2)
Barriers: select the default, **No Barriers**
Once parameters are selected, click **OK**.



It will take a few seconds to process the rainfall/evap grids. A process completion window appears when it is complete. Click **OK**.



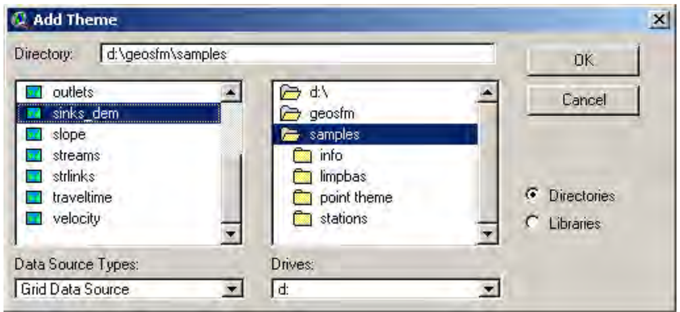
From the **Theme** menu, you may want to select **Edit Legend** and apply a precipitation color ramp like the one below.



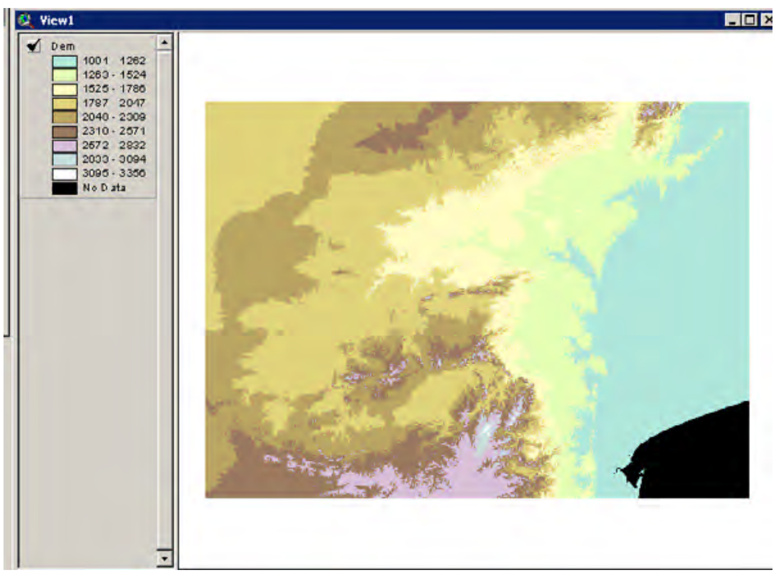
DEM Tool for Sink Filling

In this tutorial, GeoSFM's elevation grid filling tool is presented. The tool is used for filling spurious sinks, while maintaining sinks that are natural occurrences in the landscape. This time-consuming process yields a DEM that will properly transport water across its surface. This process is best done on a smaller basin region than for an entire continent. If the elevation data are from the HYDRO1k DEM data set developed at USGS EROS, then this process is not needed. HYDRO1k is a hydrologically corrected DEM, which implies that it is devoid of spurious pits that interrupt hydraulic connectivity over the land surface.

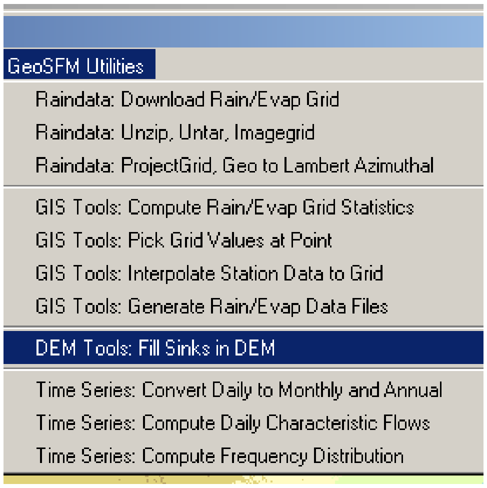
To begin the process, add the **Sinks_dem** grid to the View using the **Add Theme** button . Change the **Data Source Types** to **Grid Data Source**. Click **Sinks_dem** from the **c:\GeoSFM\samples** directory and click **OK** to add the DEM to the View.



The DEM is added to the View, as shown below.



The only menu item under **DEM Tools** is **Fill Sinks in DEM**. From the **GeoSFM Utilities** menu, select **DEM Tools: Fill Sinks in DEM**.

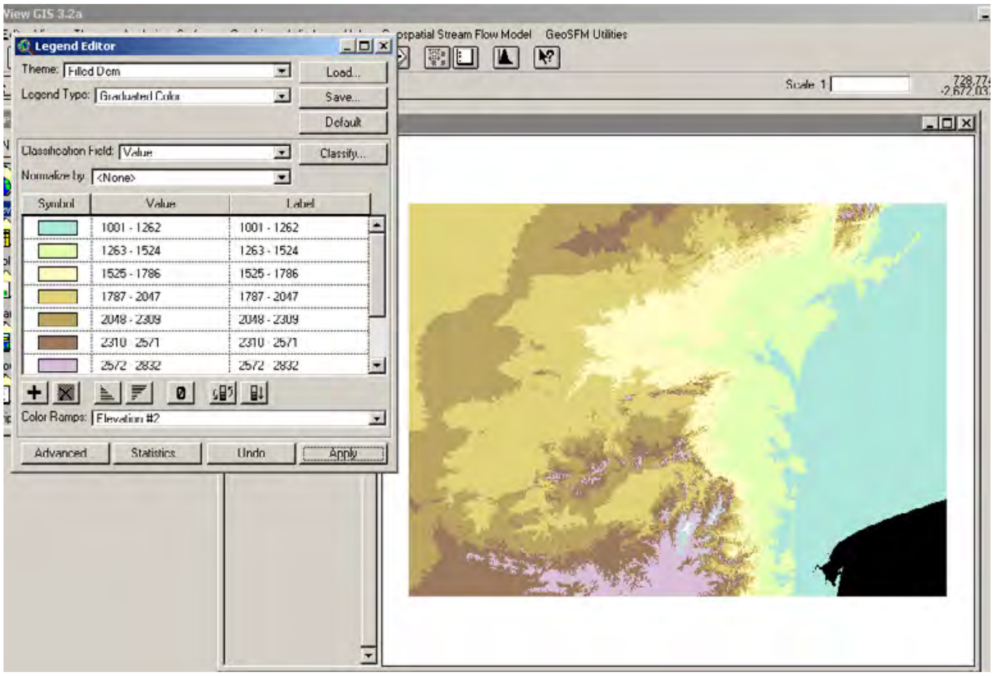


Confirm your working directory. Click **OK**.

Next, the **Identify Input DEM** dialog box is displayed. At the **Select the Grid to be Filled** prompt, select **Sinks_dem** from the drop-down list. Click **OK**.

After a few moments for processing, the new **fill1** grid is added to the table of contents. You may want to apply a legend appropriate to the elevation. To do so, from the **Theme** menu, select **Edit Legend**.

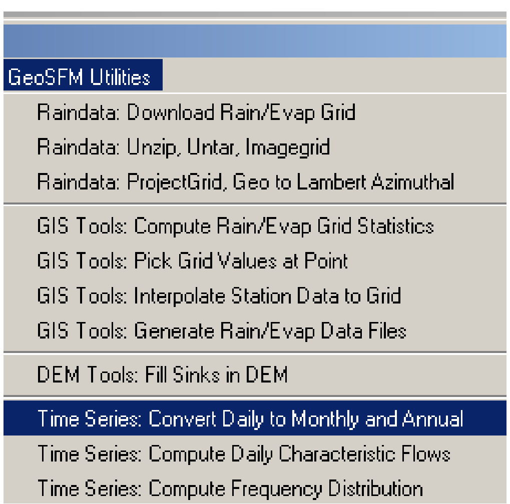
In the **Color Ramps** drop-down list, select one of the **Elevation** choices and click **Apply**.



The **Sinks_dem** file in the samples folder is removed and a new **fill1** grid is added to your workspace directory. The resulting elevation grid is ready for use in terrain analysis.

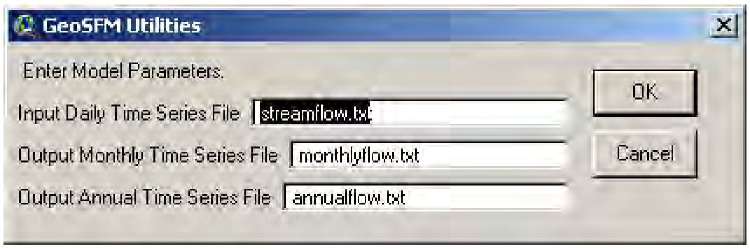
Time Series Tool for Changing Temporal Resolution

The **GeoSFM Utilities** contains three time series tools for performing tasks commonly required when examining simulation results. The first of the three functions is **Convert Daily to Monthly and Annual**. From the **GeoSFM Utilities** menu, select **Time Series: Convert Daily to Monthly and Annual**.

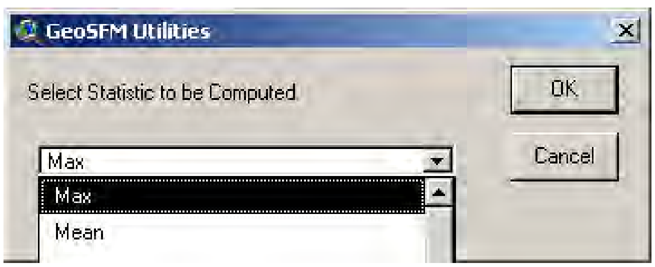


Confirm your working directory. Click **OK**.

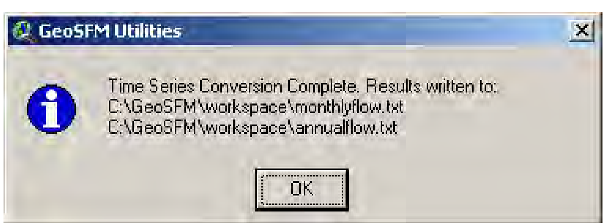
From the **GeoSFM Utilities** dialog box, **Enter Model Parameters**. The input file defaults to **streamflow.txt**, and the output files are **monthlyflow.txt** and **annualflow.txt**. Click **OK**.



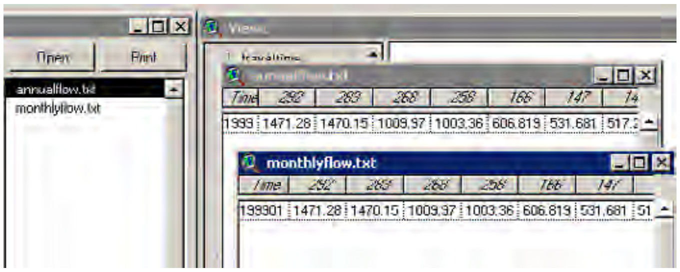
Next, at the **Select Statistic to be Computed** prompt, select **Max**. Click **OK**.



After a few moments of processing, the **Time Series Conversion Complete** dialog box will display as seen below. The program will produce two files containing the monthly and annual streamflow amounts for each subbasin. The message contains the name and location of the output files. Click **OK**.

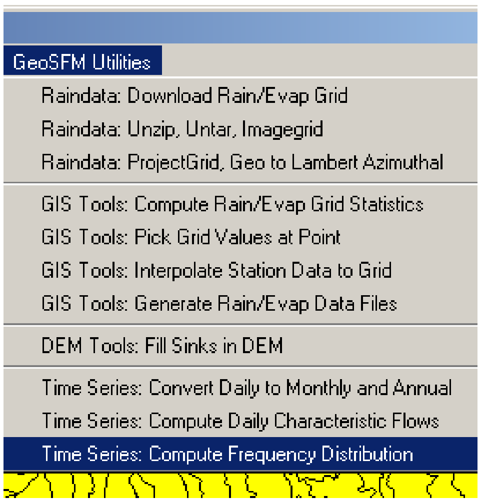


Two new tables are shown below; the streamflow data was for a few days in January 1999.



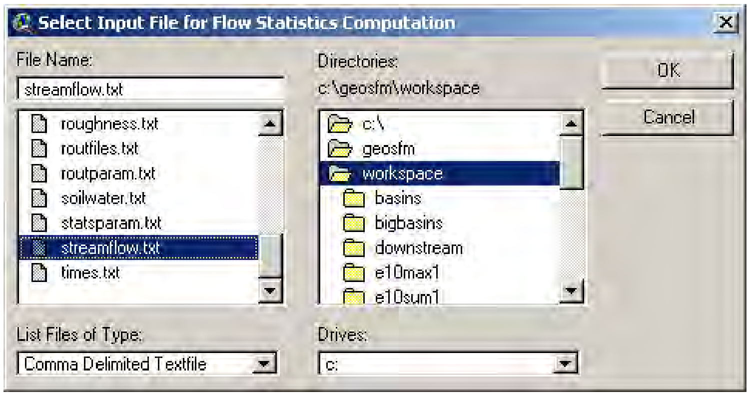
Time Series Tool for Frequency Analysis

The last tool in this tutorial is **Compute Frequency Distribution**. From the **GeoSFM Utilities** menu, select **Time Series: Compute Frequency Distribution**.



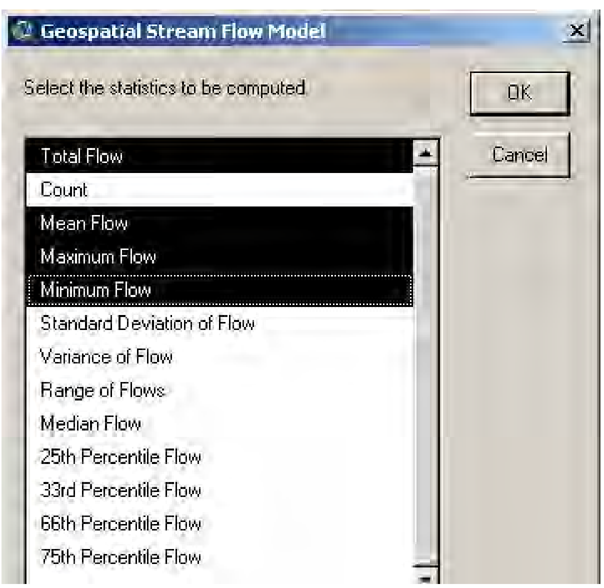
Confirm your working directory. Click **OK**.

Next, the **Select Input File for Flow Statistics Computation** dialog box is displayed. Navigate to your working directory and under **List Files of Type**, select **Comma Delimited Textfile** from the drop-down list. Select **streamflow.txt** as seen below. Click **OK**.

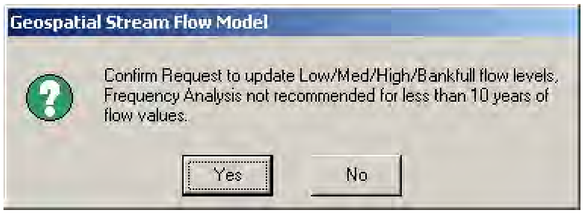


When the **Basin Theme** dialog box is displayed, select your basin coverage/grid theme. In this tutorial, the basin coverage selected is **basply.shp**. Click **OK**.

Next, the **Select the statistics to be computed** dialog box opens. In this example, **Total Flow, Mean Flow, Maximum Flow,** and **Minimum Flow** have been selected. Notice that more than one statistical type can be selected at a time from the list of options provided. Click **OK**.

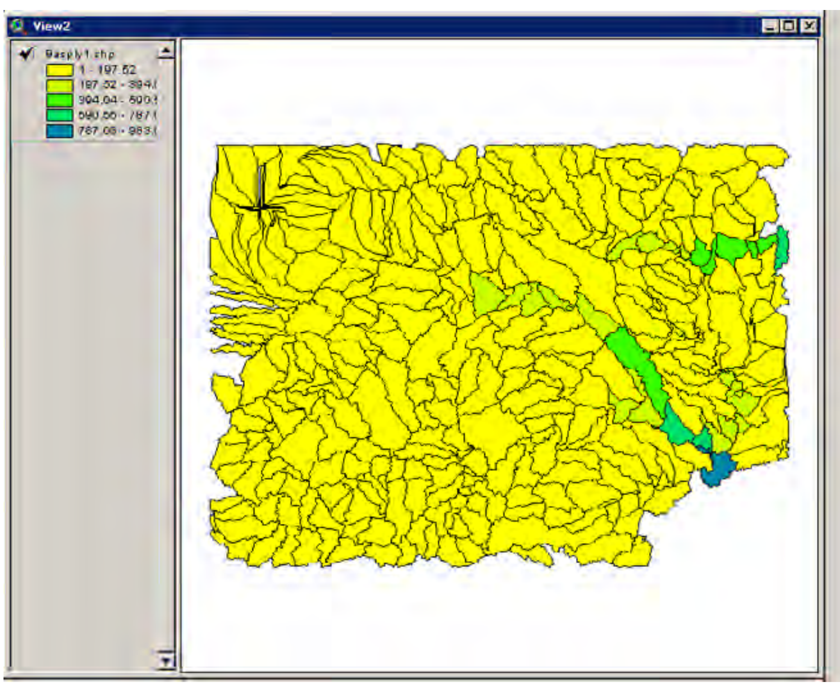


Confirm Request to update Low/Med/High Bankfull flow levels is displayed. Select **Yes**. This analysis is not recommended for less than 10 years of data. Because you have only been working with 10 days of data, the results of the analysis will not be meaningful. The objective is to familiarize you with the functionality available in the **GeoSFM Utilities** tool.

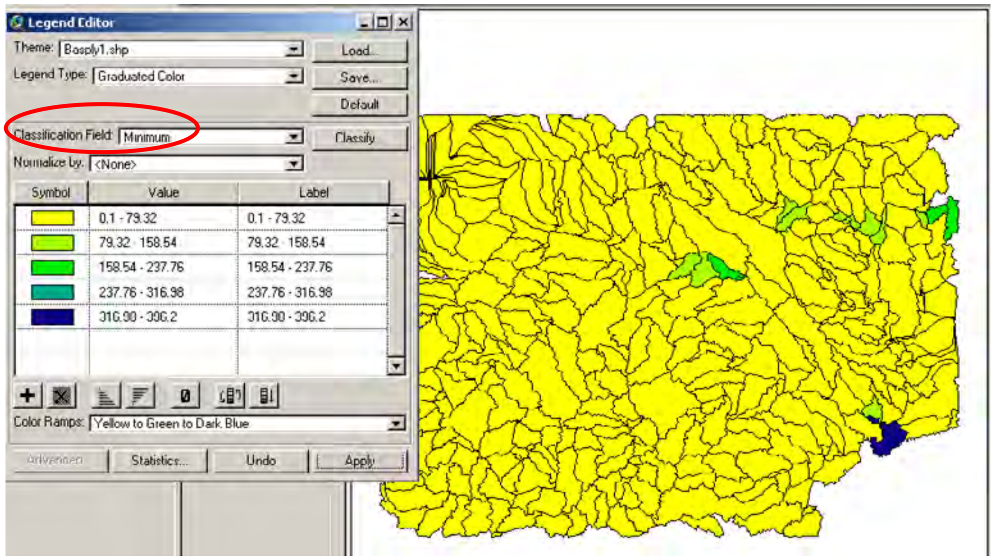


The next dialog box displayed is the **Select field to display**; in this example, **Mean** has been selected. Click **OK**. Again, notice the list of options available for displaying the results. The new calculations will be displayed in the **basply.shp**, overwriting what was displayed.

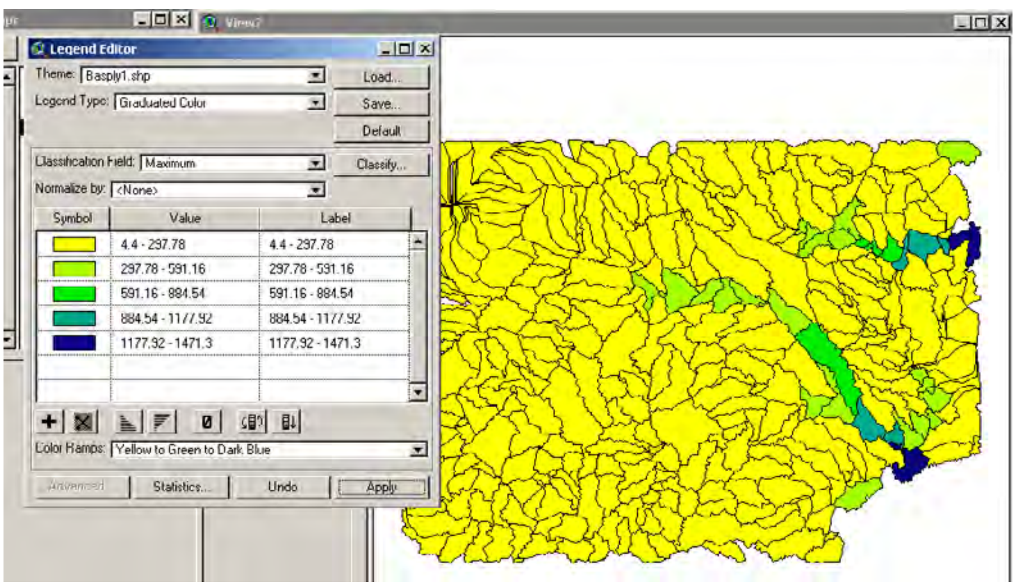
Below is the updated **basply.shp** with the **Mean** value displayed.



You can change the value displayed. From the Theme menu, select Legend Editor. Then change the Classification Field and click the Apply button. The Color Ramp selected is Yellow to Green to Dark Blue. Below, you can see the results with the Minimum classification field selected.



Below is the result when the **Maximum** classification field is selected.



Notice the differences among the three different classifications. The maximum flow classification is useful for quickly identifying subbasins through which the highest flows passed during the analysis period.

Conclusions

This manual provides step-by-step instructions for running the GeoSFM software and accompanying data sets for performing model setup, preprocessing, flow simulation, calibration, and postprocessing tasks. It also provides instructions for running utilities for downloading daily data and processing of required time series grids and performing some common processing tasks for elevation and time series grids and files. Additionally, five appendices are included with detailed instructions for downloading and processing elevation, soil, and land cover data sets used in parameterizing GeoSFM. The accompanying GeoSFM Technical Manual details the functions underlying the menus described in this manual. Together, these two manuals should serve to document the GeoSFM system for potential users seeking to assess hydrologic conditions in data sparse settings (Artan and others, 2007).

References Cited

Artan, G., Gadain, H., Smith, J., Asante, K., Bandaragoda, C.J., Verdin, J., 2007, Adequacy of satellite derived rainfall data for stream flow modeling, Natural Hazards, vol. 43, no. 2, p. 167-185.

Artan, G., Verdin, J., and Asante, K., 2001, A wide-area flood risk monitoring model, Fifth International Workshop on the Applications of Remote Sensing in Hydrology, Montpellier, France, October 2–5, 2001.

Asante, K.O., Dezanove, R.M., Artan, G.A., Lietzow, R., and Verdin, J., 2007, Developing a flood monitoring system from remotely sensed data for the Limpopo Basin, Transaction on Geoscience and Remote Sensing, vol. 45, no. 6, p. 1907–1914.

Food and Agriculture Organization of the United Nations (FAO), 1995, Digital Soil Map of the World (DSMW), CD-ROM V.3.5, FAO, Rome, Italy.

Loveland, T.R., Reed, B.C., Brown, J.F., Ohlen, D.O., Zhu, J., Yang, L., and Merchant, J.W., 2000, Development of a Global Land Cover Characteristics Database and IGBP DISCover from 1-km AVHRR Data, Int. J. Remote Sensing, vol. 21, no. 6, p. 1303–1330.

Maidment, D., ed., 1993, Handbook of Hydrology, McGraw Hill, New York, NY.

Verdin, J.P., and Klaver, R.W., 2002, Grid cell based crop water accounting for the Famine Early Warning System, Hydrol. Processes, vol. 16, p. 1617–1630.

Verdin, K.L., and Greenlee, S.K., 1996, Development of continental scale digital elevation models and extraction of hydrographic features, Proceedings of the Third International Workshop on Integrating GIS and Environmental Modeling at Santa Fe, New Mexico, National Center for Geographic Information and Analysis, Santa Barbara, California.

Webb, R.S., Rosenzweig, C.E., and Levine, E.R., 1993, Specifying land surface characteristics in general circulation models: soil profile data set and derived water-holding capacities, Global Biogeochemical Cycles, vol. 7, p. 97-108.

Xie, P., and P.A. Arkin, 1997, A 17-year monthly analysis based on gauge observations, satellite estimates, and numerical model outputs, Bulletin of the American Meteorological Society, 78(11), 2539–2558.

Zobler, L., 1986, A World Soil File for Global Climate Modelling. NASA Technical Memorandum 87802. NASA Goddard Institute for Space Studies, New York, New York, U.S.A.

Glossary

EROS	Earth Resources Observation and Science Center
NOAA	National Oceanic and Atmospheric Administration of the United States
RFE	Satellite-derived Rainfall Estimate from NOAA Climate Prediction Center
TRMM	Tropical Rainfall Measuring Mission
USGS	U.S. Geological Survey

Appendix 1: Processing Elevation Data

GeoSFM can use any digital elevation model for the delineation of hydrologic modeling units. The use of the HYDRO1k data set is described in this document because it is readily available and easily distributed. The descriptions encompass the processes to download data and initial processing to remove **nodata** cells and prepare the data set for use in GeoSFM. As with other tutorials in this document, the Limpopo Basin in southern Africa is used as the test basin.

These data are available from the U.S. Geological Survey Earth Resources Observation Science (EROS) Center at *http://edcdaac.usgs.gov/gtopo30/hydro/africa.html*. This Web site brings you to the Land Processes Distributed Active Archive Center. From this page, scroll down to the list of downloads. (Data for other regions are found at the same Web site at the following address: *http://edcdaac.usgs.gov/gtopo30/hydro/*). The Web site allows you to download the entire African data set (including elevation, compound topographic index, aspect, flow direction, flow accumulation, slope, drainage basins, and streams) or any of its individual components.

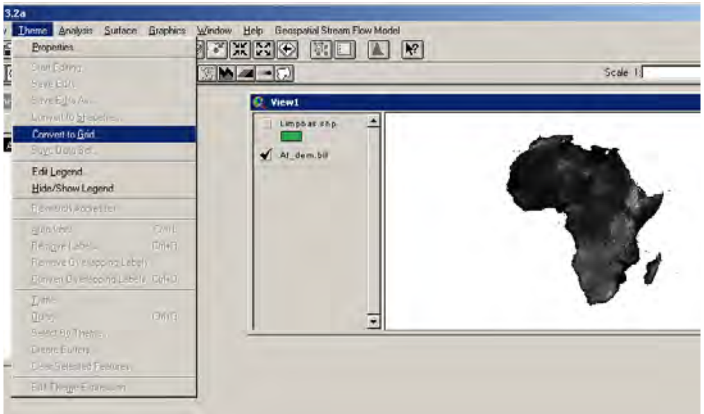
Download the elevation data set by clicking **Download HYDRO1k DEM for Africa as a tar file (34.0MB)**. Save the file to the workspace, **c:\GeoSFM\workspace**. Use WinZip to select and extract all files to the workspace. Extract the **af_dem.bil** from the **af_dem.bil.gz** file. If you do not already have WinZip installed, download a free trial version from *http://www.winzip.com/winzip/download.htm.*

Extracted files:
af_dem.gz
af_dem.bil
af_dem.blw
af_dem.hdr
af_dem.stx
readme.af

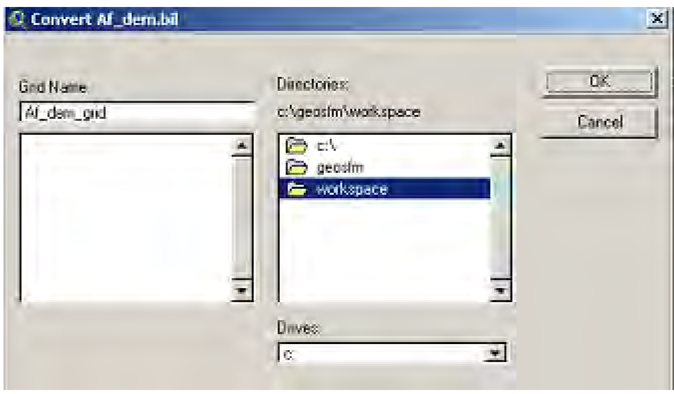
Next, you need to add the elevation data to the ArcView project, convert it into the required ArcInfo grid format, and make some corrections. Add the Digital Elevation Model to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Image Data Source**. Click the **af_dem.bil** file from the **c:\GeoSFM\workspace** directory and click **OK** to add the DEM to the View.

Next, the DEM will need to be converted to a grid. Select the theme **Af_dem.bil** so that it appears in a raised box. Checking the box next to the theme name will make it visible.

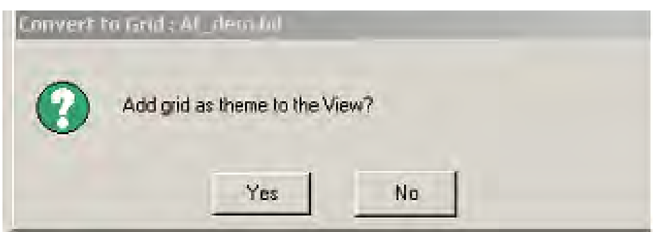
From the **Theme** menu, select **Convert to Grid**.



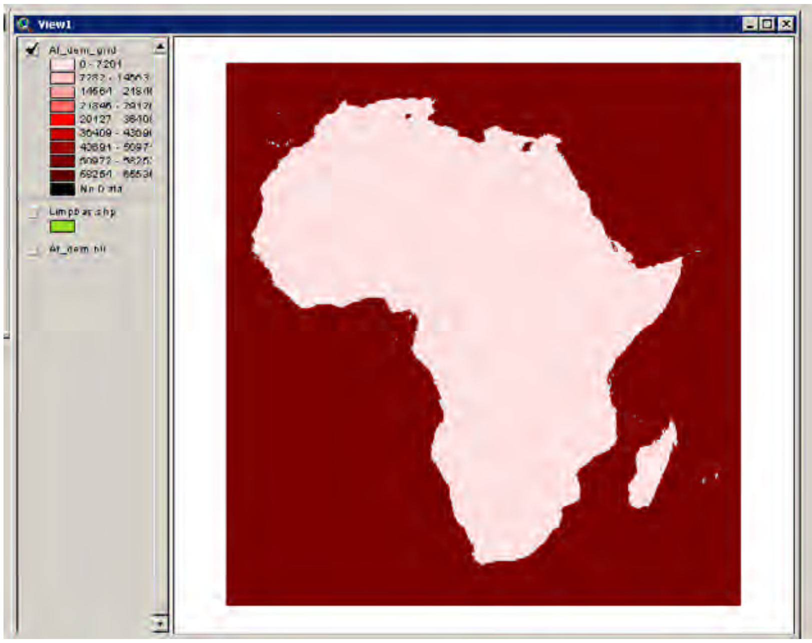
The **Convert Af_dem.bil** dialog box appears. Navigate to the **c:\GeoSFM\workspace** directory and name your new grid **Af_dem_grid in Grid Name**. Click **OK**.



A dialog box appears asking if you would like to **Add grid as theme to the View**. Click **Yes**.



The new **Af_dem_grid** theme is shown below.

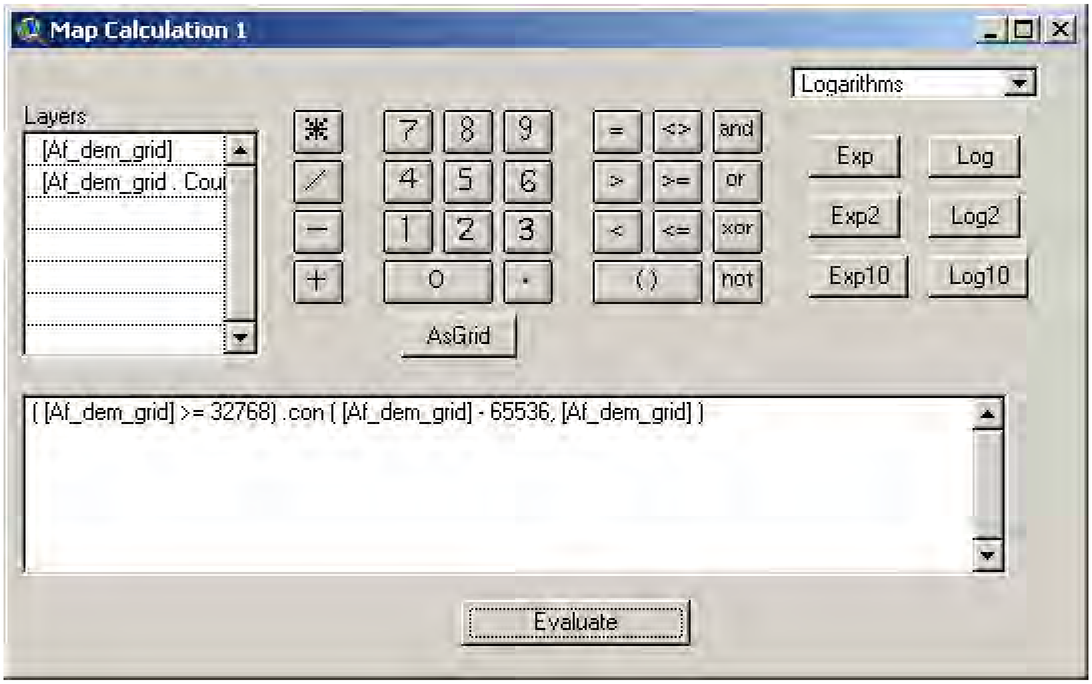


After adding the grid to the View, some map calculations must be performed to correct abnormally high values that are introduced into the DEM to avoid having to store negative values in the *.bil format. From the **Analysis** menu, choose **Map Calculator**.

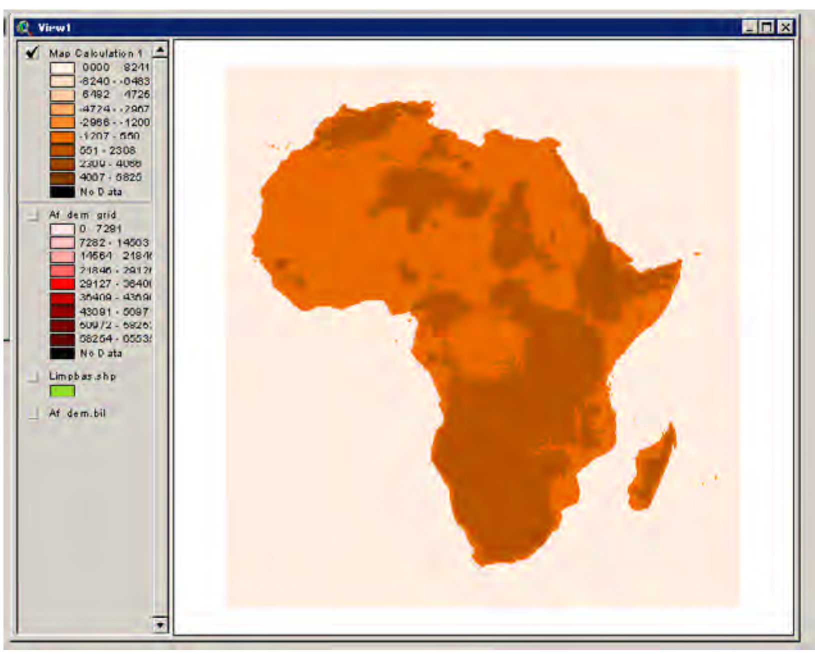
The first correction subtracts 65536 from all the cells with elevation higher than 32768, which restores the original negative values. The result of this computation is that all NODATA cells are assigned values of –9999 while all other cells will have their true elevation (negative or positive).

In **Map Calculator**, type the following expression exactly as shown:

([Af_dem_grid] >= 32768).con([Af_dem_grid] – 65536, [Af_dem_grid])

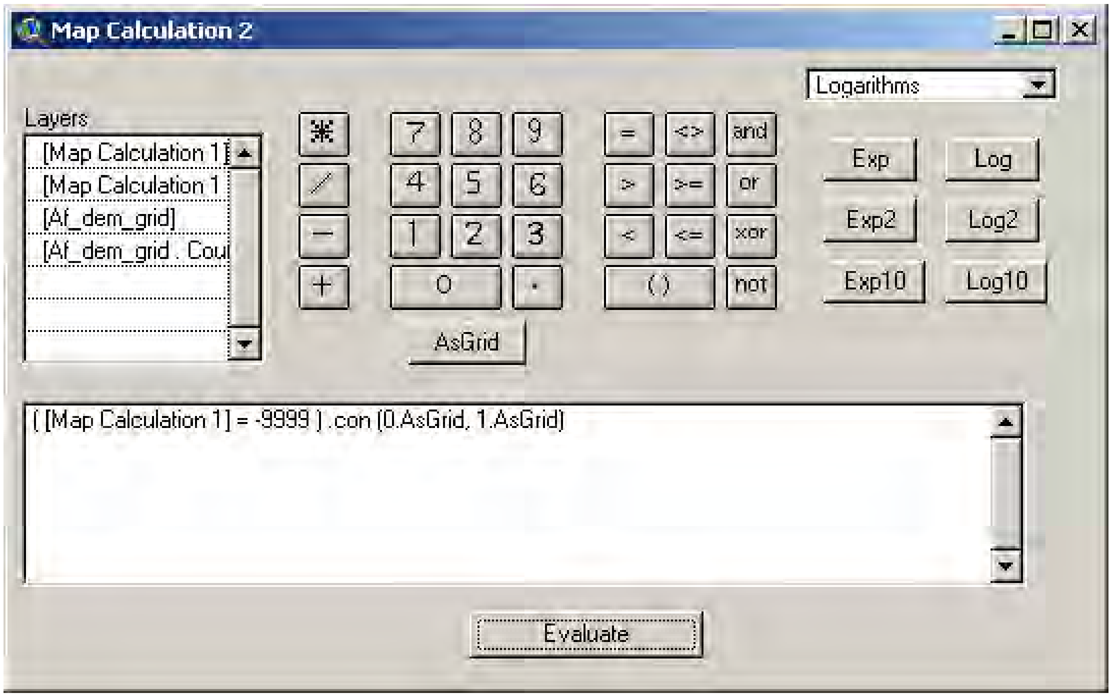


Click the **Evaluate** button. This produces a temporary grid called **Map Calculation 1**, seen below.

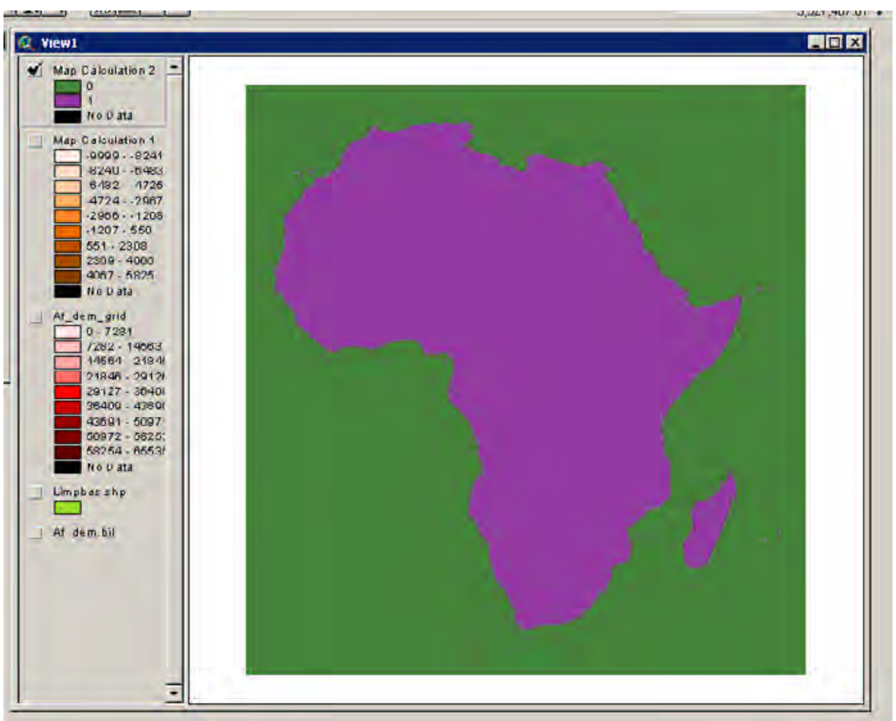


Choose **Map Calculator** from the **Analysis** menu again to do the second calculation. The second calculation assigns values of zero to NODATA cells and values of one to all other cells so that the NODATA cells can be eliminated in the final step. Type in the following expression exactly as shown:

([Map Calculation 1] = -9999).con (0.AsGrid, 1.AsGrid)

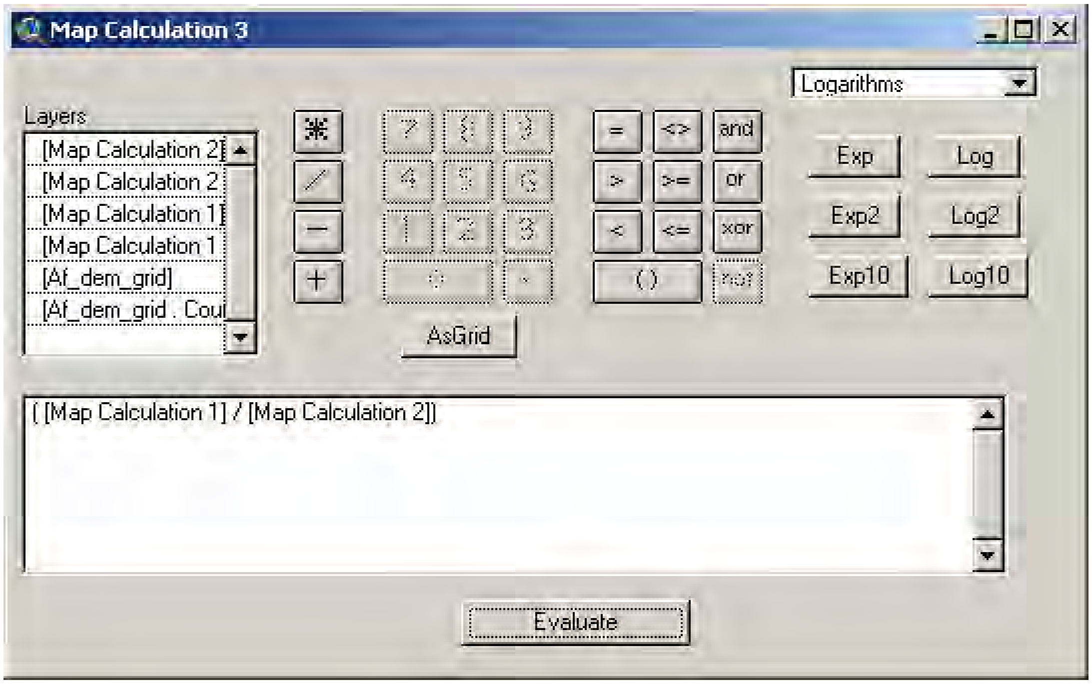


Click the **Evaluate** button. This produces a temporary grid called **Map Calculation 2**, seen below.

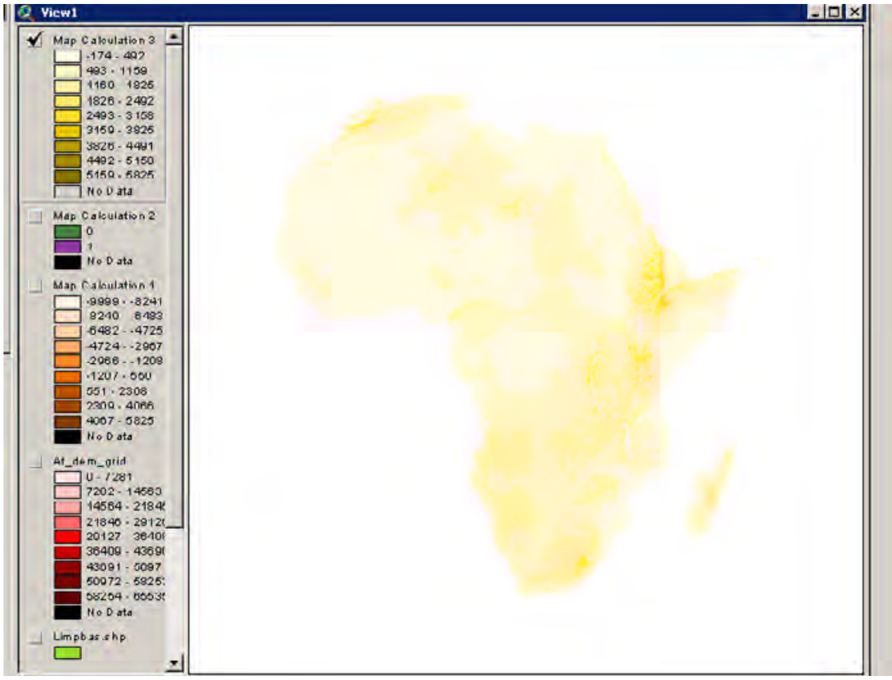


The final computation divides the result of the first correction with the result of the second correction. Zero divides values of –9999 while 1 divides all other values. Division by zero results in a NODATA cell being created while division by 1 leaves the original value unaltered. Choose **Map Calculator** from the **Analysis** menu again and type in the following expression exactly as shown:

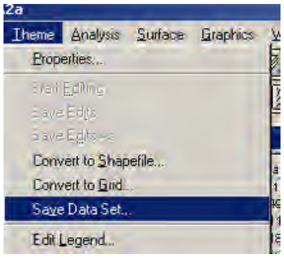
([Map Calculation 1] / [Map Calculation 2])



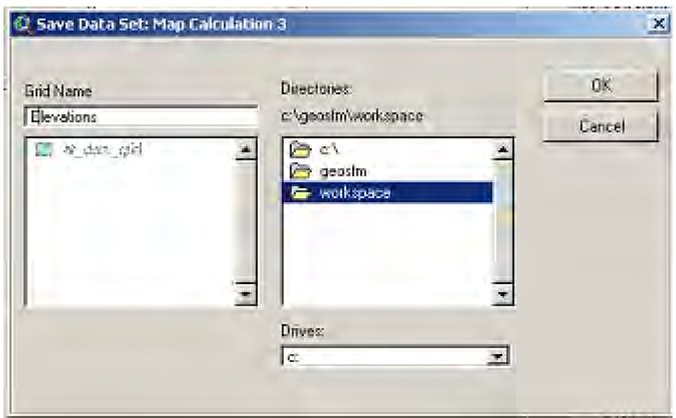
Click the **Evaluate** button. This produces a temporary grid called **Map Calculation 3**, seen below.



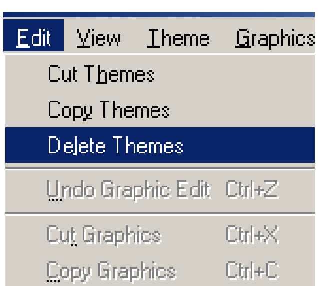
Because this final elevations grid will be used for the rest of the tutorial, it needs to be made permanent. Select the **Map Calculation 3 Theme** so it appears in a raised box. From the **Theme** menu**,** select **Save Data Set**.



In the **Save Data Set** Dialog Box, navigate to the **c:\GeoSFM\workspace** directory and name the new grid **Elevations** in the **Grid Name** field. Click **OK**.



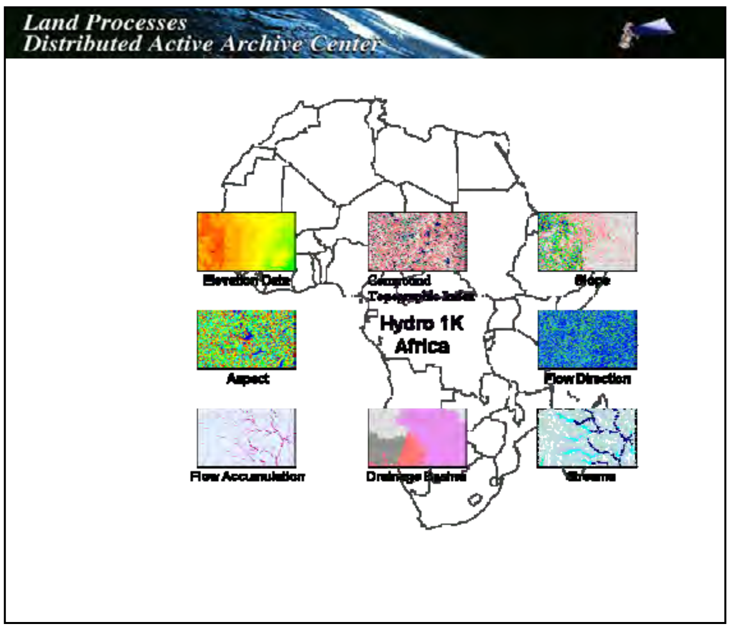
You can now remove all themes by selecting the theme so it appears in a raised box, and then from the **Edit** menu, select **Delete Themes**.



Now the permanent **Elevations** grid can be added to the View using the **Add Theme** button . The resulting elevation grid is ready for use in terrain analysis.

Appendix 2: Defining Analysis Extent

GeoSFM uses a shapefile to define the analysis extent for use in the modeling process. HYDRO1k Basins data sets are derived using the vector stream networks along with the flow direction data set. Each polygon in the basin data set has been tagged with a Pfafstetter code uniquely identifying each subbasin. Additional attributes defining the characteristics of each subbasin have also been developed. These data are available in an ArcInfo export format and shapefile format. The process for downloading data documented below captures data needed to complete the terrain analysis tutorial. This tutorial was developed for the Limpopo Basin in southern Africa. The elevation data are available from the U.S. Geological Survey Earth Resources Observation Science (EROS) Center at *http://edcdaac.usgs.gov/gtopo30/hydro/af_basins.asp.* Data for other regions are also found at the Web site of the Land Processes Distributed Active Archive Center at *http:/edcdaac.usgs.gov/gtopo30/hydro*.



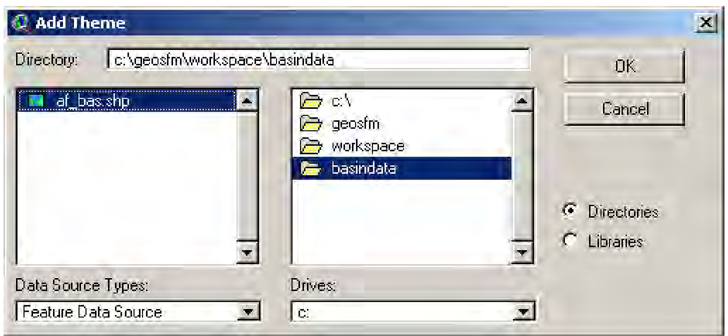
The African HYDRO1k data set can be downloaded in its entirety, or individual layers can be downloaded by selecting the data layer, see above. Detailed descriptions of the techniques used in the development of the HYDRO1k data set can be found in the Readme file. The page contains options for downloading the entire HYDRO1k data set for Africa as a tar file (215.5 MB) or downloading individual components in either ArcInfo export format or shapefile formats.

Click **Download HYDRO1k Basins data set for Africa as a tar file in shapefile format (7.4 MB)**. Save the file to your workspace, **c:\GeoSFM\workspace**. Use WinZip to select and extract all files to your workspace. Extract all files from the **af_base.tar.gz** file. If you do not already have WinZip installed, you can download a free trial version from *http://www.winzip.com/winzip/download.htm.*

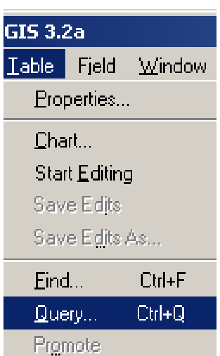
Extracted files:
af.readme
af_bas.dbf

af_bas.prj
af_bas.shp
af_bas.shp.xml
af_bas.shx

Next, you need to add the basin data to the ArcView project and select the basins for the analysis area, which is the Limpopo Basin in this tutorial. Add the basin data to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Feature Data Source**. Click the **af_bas.shp** file from the **c:\GeoSFM\workspace** directory and click **OK** to add to the View.

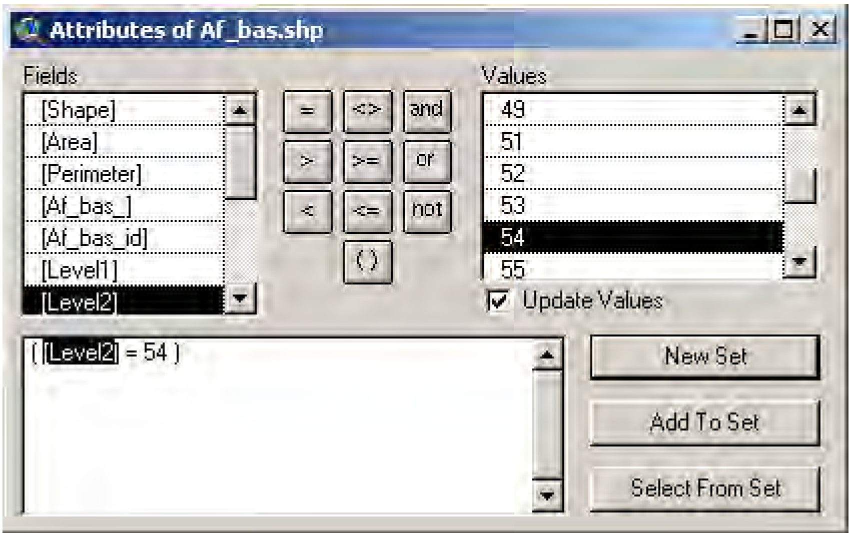


After adding the shapefile, select the basin area in the attribute table. Open the attribute table by clicking the **Table** icon. From the **Table** menu select **Query**.



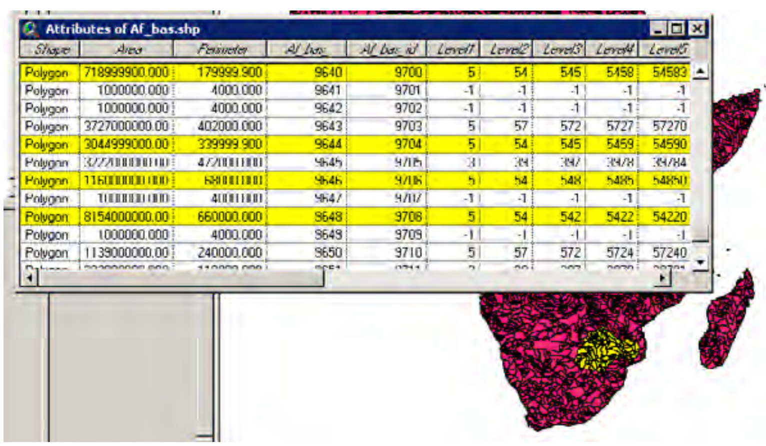
The **Attributes of Af_bas.shp** query box opens.

Double-click **Level 2** under **Fields**. Make sure the **Update Values** box is checked so all **Level 2** values will display under **Values**. Click the "=" button and then the value you need for this basin. Select **54** for this tutorial. Click **New Set**.

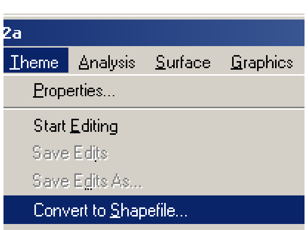


([Level2] = 54)

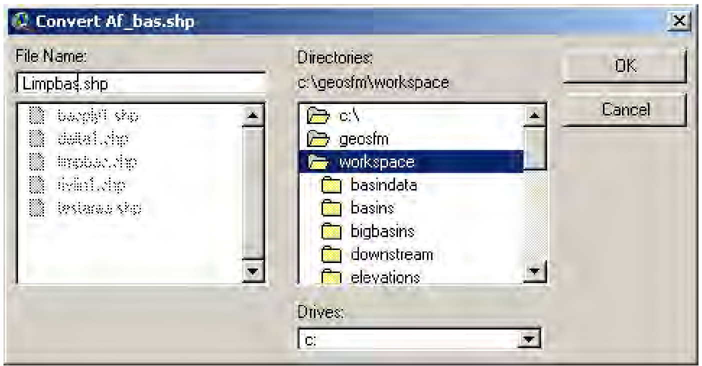
All attributes with a value of **54** in the **Level2** column will have a yellow highlight. The African shapefile will also show the subbasins selected highlighted in yellow.



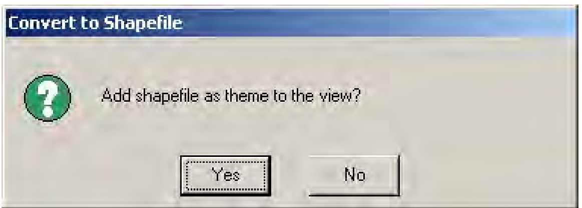
Now you can convert the selections to a shapefile. From the **Theme** menu, select **Convert to shapefile**.



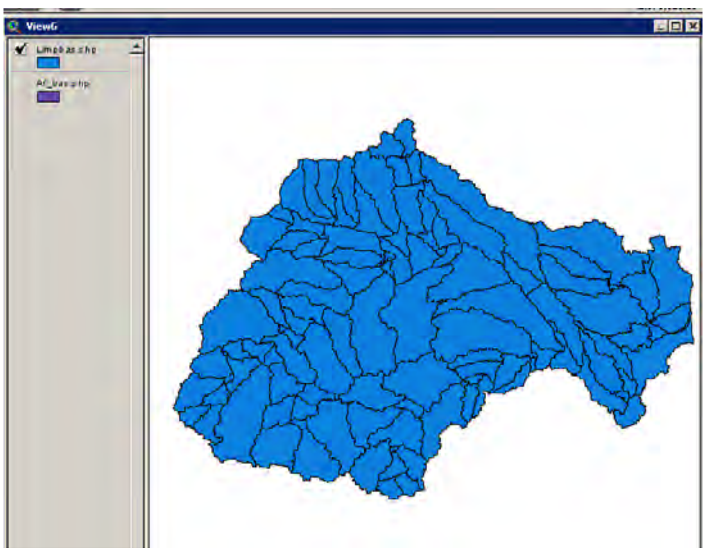
The **Convert Af_bas.shp** dialog box opens. Navigate to the appropriate directory and under **File Name** type in the name of the new shapefile. Click **OK**.



The **Add shapefile as theme to the view** dialog box will display. Click **Yes**.



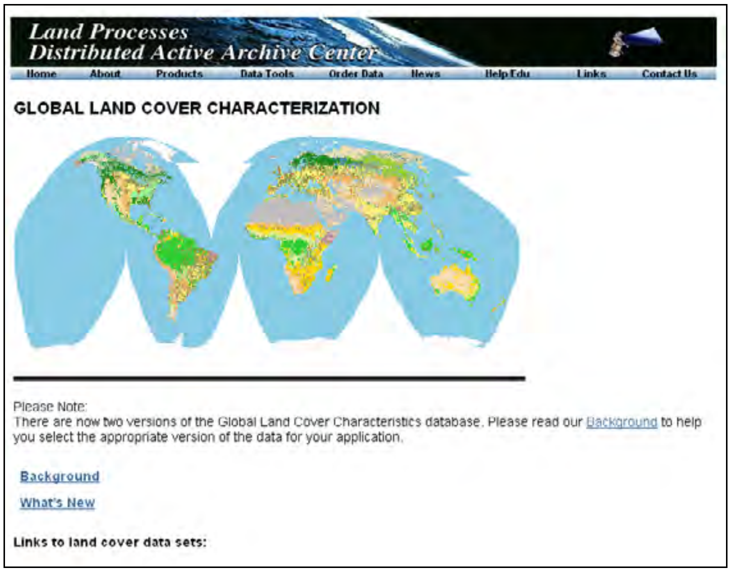
Below, the new shapefile is displayed.



The shapefile can be added to the View when prompted during the terrain analysis process.

Appendix 3: Processing Land Cover Data

Download the data from the Internet at *http://edcdaac.usgs.gov/glcc/af_int.html*.



This Web site brings you to the Land Processes Distributed Active Archive Center—Global Land Cover Characterization. From this page, scroll down to the **Links to land cover data sets**. Two different versions are available; select **Version 2**.

Select the appropriate area for download from the table shown below.

GLCC – Version 2		
Global	North America	Eurasia
Africa	South America	Australia Pacific

Select **Africa** because the study area is the Limpopo Basin in southern Africa. Create a new folder in the GeoSFM folder called **glcc** for all of the land cover data. Selecting Africa will display the Africa Land Cover Characteristics Data Base Version 2.0 page. From the Africa Land Cover page, select **Data in Lambert Azimuthal Equal Area Projection**. This transfers you to a page with links to data and documentation in either compressed or uncompressed format. Scroll down the page to the **image files** and click **Compressed/3.3 Mb**. Save the file to **c:\GeoSFM\glcc**. Use WinZip to select and extract all files to your workspace. From the **afusgs2_0l.img.gz** file, extract the **afusgs2_0l.img**. If you do not already have WinZip installed, you can download a free trial version from *http://www.winzip.com/winzip/download.htm.* You can also elect to download the uncompressed files directly if you have adequate bandwidth to support such a download.

The downloaded land cover data is provided in a generic image array format (flat, headerless, binary raster data) to maximize the flexibility for importing the data into a variety of software systems. The following procedure is needed before importing these raster image files into ArcView. First, rename the image file to **afusgs2_0l.bsq** (.bsq –image file– band sequential). Next, create a header file with the following fields:

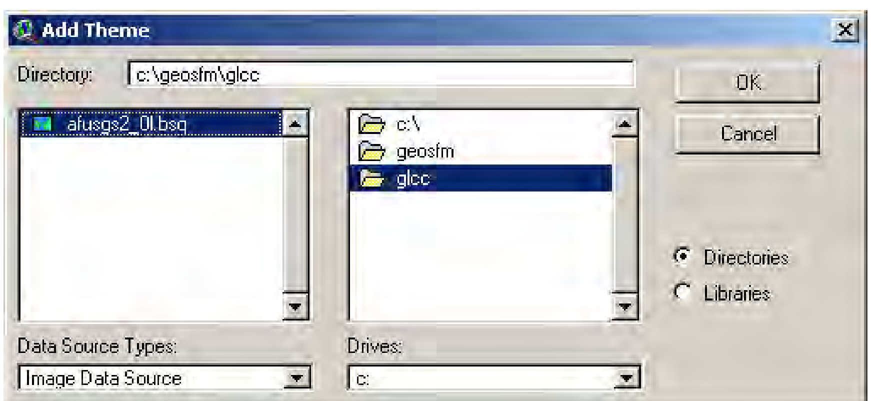
nrows—the number of rows in the image
ncols—the number of columns in the image
nbands—the number of spectral bands in the image; default is 1
nbits—the number of bits per pixel per band; default is 8
layout—the organization of the bands in the image file; acceptable values are bil, bip, and bsq
skpbytes—the number of bytes of data in the image file to skip to reach start of image data (bypass header information); default is 0
ulxmap—the x-axis map coordinate of the center of the upper-left pixel
ulymap—the y-axis map coordinate of the center of the upper-left pixel
xdim—the x dimension of a pixel in map units
ydim—the y dimension of a pixel in map units

The image parameters needed to populate the new header file can be found in the documentation for each of the continental databases. For example, the projection parameters for the Limpopo study area can be found by clicking **Africa Documentation** and scrolling down to **section 2.2 Lambert Azimuthal Equal Area Projection Parameters**. The data dimensions of the Africa land cover characteristics data set are 9,276 lines (rows) and 8,350 samples (columns), resulting in a data set size of about 77 megabytes for 8-bit (byte) images. The map projection parameters are as follows:

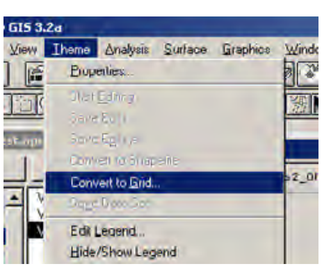
Projection Type: Lambert Azimuthal Equal Area
Units of Measure: meters
Pixel Size: 1,000 meters
Radius of sphere: 6,370,997 m
Longitude of origin: 20 00 00 E
Latitude of origin: 5 00 00 N
False easting: 0.0
False northing: 0.0
XY corner coordinates (center of pixel) in projection units (meters):
Lower left: (–4458000, –4795000)
Upper left: (–4458000, 4480000)
Upper right: (3891000, 4480000)
Lower right: (3891000, –4795000)

Prepare an ASCII header file in any text editor, such as WordPad, as shown below.

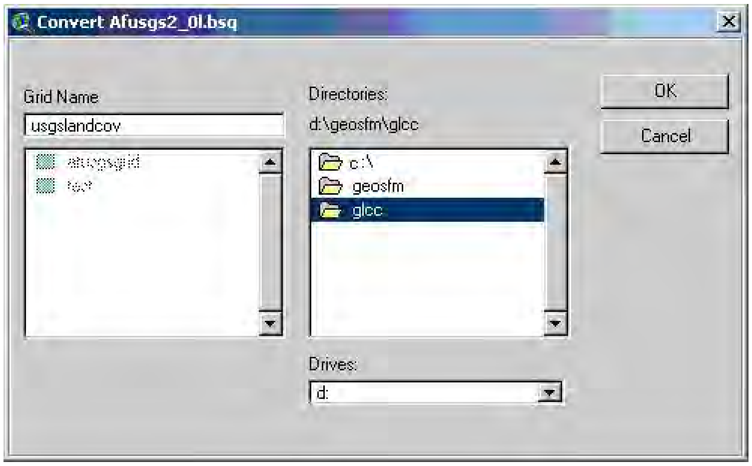
Save the document from the file under the name **afusgs2_0l.hdr** in the directory **c:\GeoSFM\glcc**. The land cover data must be added to the ArcView project and converted into a grid for use in GeoSFM. Add the **afusgs2_0l.bsq** file to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Image Data Source**. Click the **afusgs2_0l.bsq** file from the **c:\GeoSFM\glcc** directory and click **OK** to add the land cover data to the View.



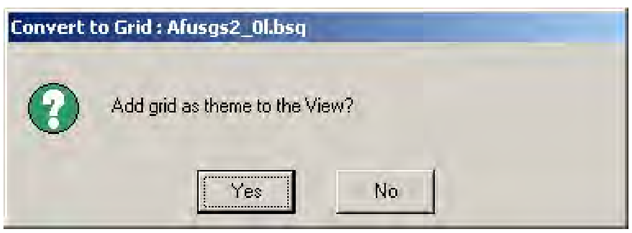
The land cover data will not display until it has been converted to a grid. To convert the land cover data to a grid, select the theme **afusgs2_0l.bsq** so that it appears in a raised box. From the **Theme** menu select **Convert to Grid**.



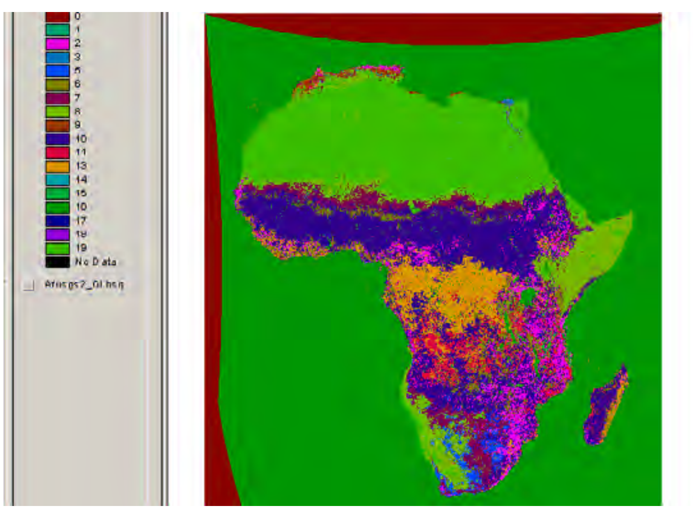
The **Convert Afusgs2_0l.bsq** dialog box appears. Navigate to the **c:\GeoSFM\glcc** directory and type in **usgslandcov** in **Grid Name**. Click **OK**.



A dialog box appears asking if you would like to **Add grid as theme to the View**. Click **Yes**.



A new **usgslandcov** theme is shown below.



Next, you need to add the **Lu_code** and **description** fields to the attribute table.

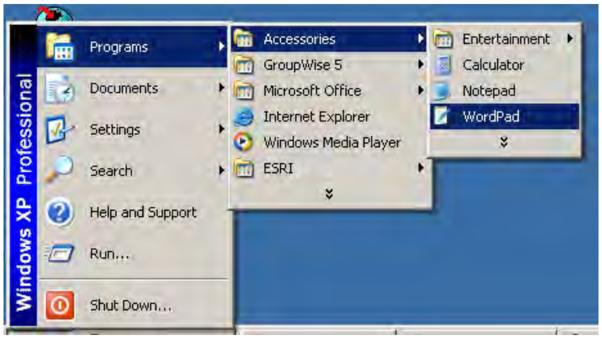
Africa Land Cover Characteristics Data Base Version 2.0

- Africa Readme
- Africa Land Cover Database Table
- Data in Interrupted Goode Homolosine Projection
- Data in Lambert Azimuthal Equal Area Projection

Value	Code	Description
1	100	Urban and Built-Up Land
2	211	Dryland Cropland and Pasture
3	212	Irrigated Cropland and Pasture
4	213	Mixed Dryland/Irrigated Cropland and Pasture
5	280	Cropland/Grassland Mosaic
6	290	Cropland/Woodland Mosaic
7	311	Grassland
8	321	Shrubland
9	330	Mixed Shrubland/Grassland
10	332	Savanna
11	411	Deciduous Broadleaf Forest
12	412	Deciduous Needleleaf Forest
13	421	Evergreen Broadleaf Forest
14	422	Evergreen Needleleaf Forest
15	430	Mixed Forest
16	500	Water Bodies
17	620	Herbaceous Wetland
18	610	Wooded Wetland
19	770	Barren or Sparsely Vegetated
20	820	Herbaceous Tundra
21	810	Wooded Tundra
22	850	Mixed Tundra
23	830	Bare Ground Tundra
24	900	Snow or Ice

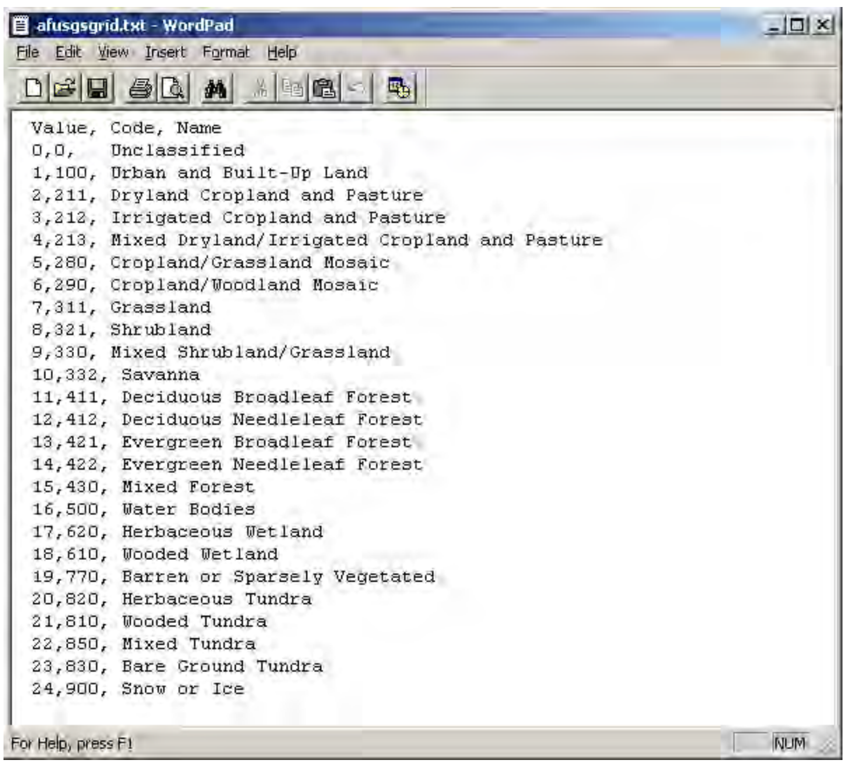
Value 330 not recognized in model need to add

Now that you have the data you need, you can prepare an ASCII file. Open WordPad (Start/Programs/Accessories/WordPad). Cut and paste the data into WordPad.



Commas need to be added to separate column values, and each line needs to be on a separate row. The header and the first line of data also need to be added.

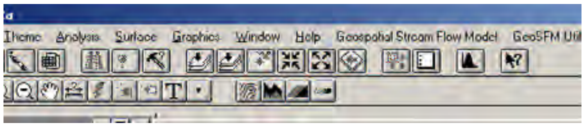
Add header names and values:
Value, Code, Name
0, 0, Unclassified—notice the commas separating field values
An example of the WordPad document can be seen below.




```
Value, Code, Name
0,0,   Unclassified
1,100, Urban and Built-Up Land
2,211, Dryland Cropland and Pasture
3,212, Irrigated Cropland and Pasture
4,213, Mixed Dryland/Irrigated Cropland and Pasture
5,280, Cropland/Grassland Mosaic
6,290, Cropland/Woodland Mosaic
7,311, Grassland
8,321, Shrubland
9,330, Mixed Shrubland/Grassland
10,332, Savanna
11,411, Deciduous Broadleaf Forest
12,412, Deciduous Needleleaf Forest
13,421, Evergreen Broadleaf Forest
14,422, Evergreen Needleleaf Forest
15,430, Mixed Forest
16,500, Water Bodies
17,620, Herbaceous Wetland
18,610, Wooded Wetland
19,770, Barren or Sparsely Vegetated
20,820, Herbaceous Tundra
21,810, Wooded Tundra
22,850, Mixed Tundra
23,830, Bare Ground Tundra
24,900, Snow or Ice
```

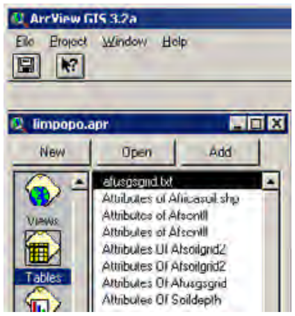
To save the document, from the **File** menu, select **Save As**.
File Name—**usgslandcov.txt**
Save as Type—**Text Document**
Directory—**c:\GeoSFM\glcc**

Click new grid theme so it appears in a raised box. Click table icon to open attribute table. The table should contain **Value** and **Count** fields.

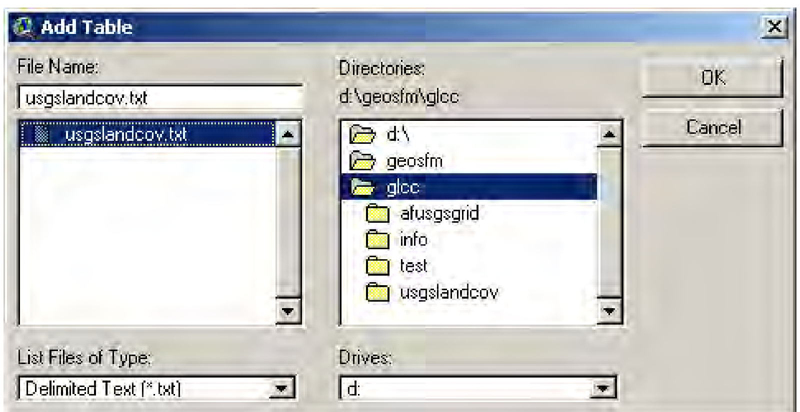


Next, you need to add the new **usgslandcov.txt** file that was created.

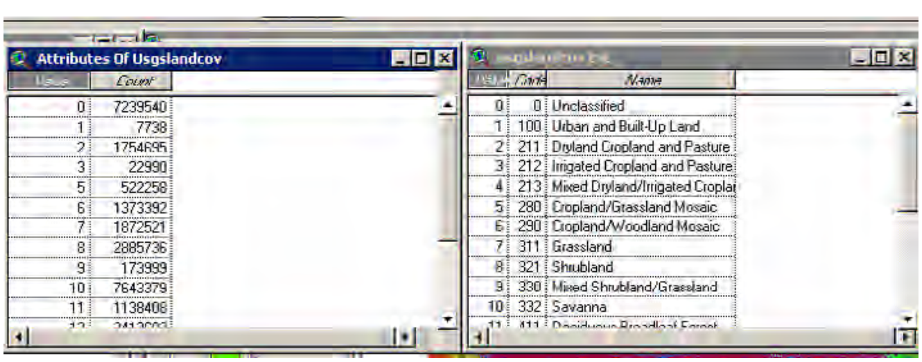
Click **Tables, Add** in the project window.



The **Add Table** dialog box will open. In **c:\geosfm\glcc**, select **usgslandcov.txt**. **List of Files of Type: Delimited Text (.txt)** should be displayed. Click **OK**.



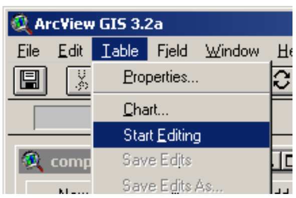
Arrange both tables as seen below. Highlight the common column headers in both tables—**Value** in **Attributes of usgslandcov** and **Value** in **usgslandcov.txt**. Make the destination table active (adding fields to **Attributes of usgslandcov**).



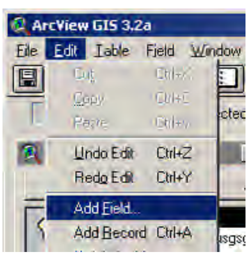
Click the **Join** icon. The new columns are added to the **Attributes of usgslandcov** table.

Once the new fields are added to the attribute table, you need to make the added data permanent.

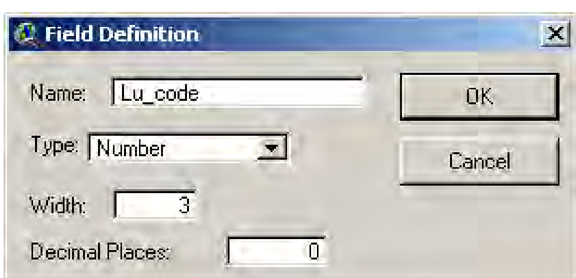
From the **Table** menu, select **Start Editing**.



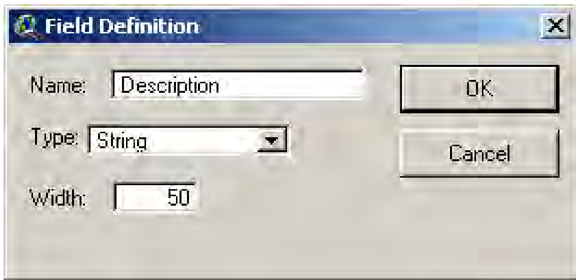
Then from the **Edit** menu, select **Add field**.



Two fields will be added.
Field One:
Name: **Lu_Code**
Type: **Number**
Width: **3**
Decimal Places: **0**



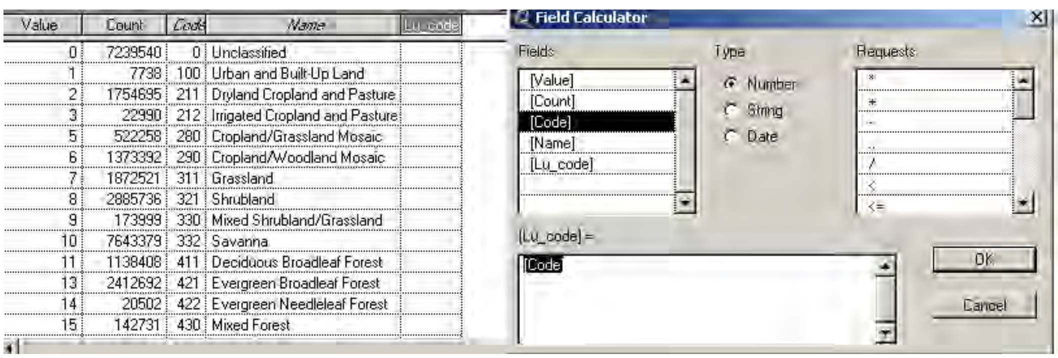
After **Field Definition** is populated, click **OK**.
Field Two:
Name: **Description**
Type: **String**
Width: **50**



After **Field Definition** is populated, click **OK**.

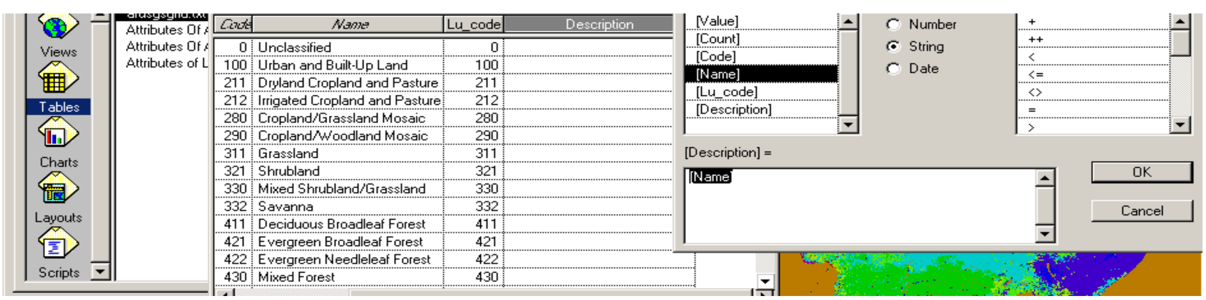
Next, populate the column values for the new fields by using the **Field Calculator** icon on the tool bar. Click the **Field Calculator** icon.

The **Field Calculator** dialog box opens—double-click **[Code]** under **Fields.**
The calculation is expressed as **[Lu_code] = [Code]**, as seen below. Click **OK**.



The new **Lu_code field** is populated with the same values as the **Code field**.

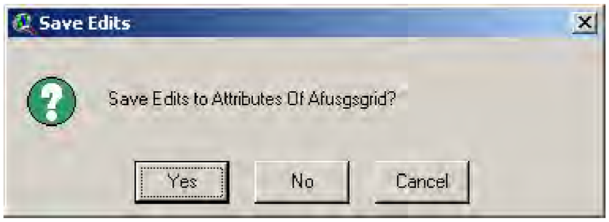
Click the **Field Calculator** again to populate the **Description** column field.



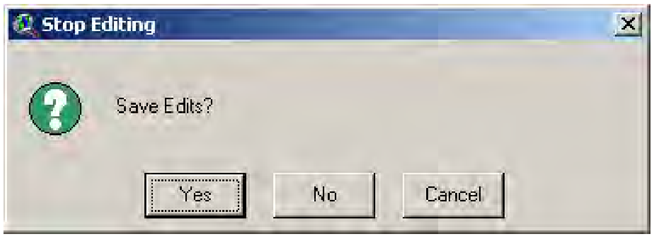
The **Field Calculator** dialog box opens—double-click **[Name]** under **Fields**.
The calculation is expressed as **[Description] = [Name]**. Click **OK**.
The new **Description** fields are populated with the same values as the **Name** fields.

Next, remove the join leaving **Value, Count, Lu_code**, and **Description** fields. From the **Table** menu, select **Remove All Joins**.

From the **Table** menu, select **Save Edits**. Click **Yes**.



From the **Table** menu, select **Stop Editing**. Click **Yes**.



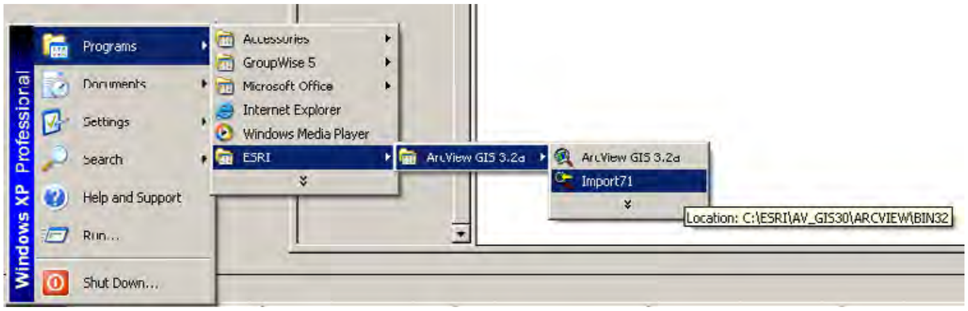
The USGS Land Cover grid can be added to the View when prompted during the **Method of OverLand Flow Velocity Computation** process in the unit hydrograph computation process.

Appendix 4: Processing Soil Characteristics

Create a new folder for **DSMW** data to be stored in the GeoSFM folder. Download DSMW data from the CD into **c:\geoSFM\dsmw**. The downloaded files are:

1STREAD
ERDAS
FAOSOIL
IDRISI
RASTEXPN
VECTOR
READ1ST.txt

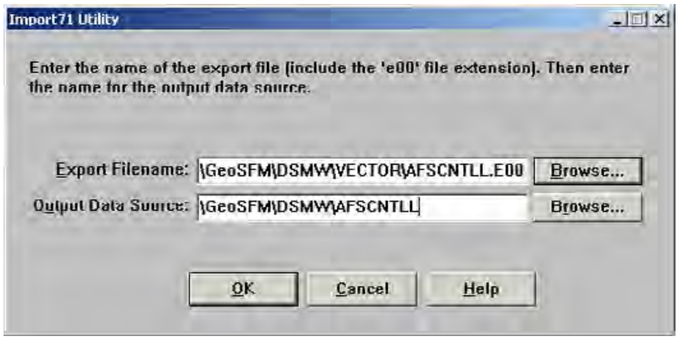
Now you can begin to import the data into an ArcView project for use in GeoSFM. First, import the vector files by clicking **Start/Programs/ESRI/ArcView 3.2a/Import71**, as seen below.



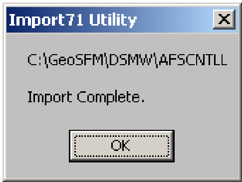
The vector files are found in the Vector folder where they are listed by 10 different regions. The files for the study area in this tutorial are the 10 files for Africa, **AFSNTLL.e00 – *.e09**. By selecting **.e00**, all files are added. **Browse** to populate **Export Filename**:

c:\geoSFM\dsmw\Vector\AFSCNTLL.E00.

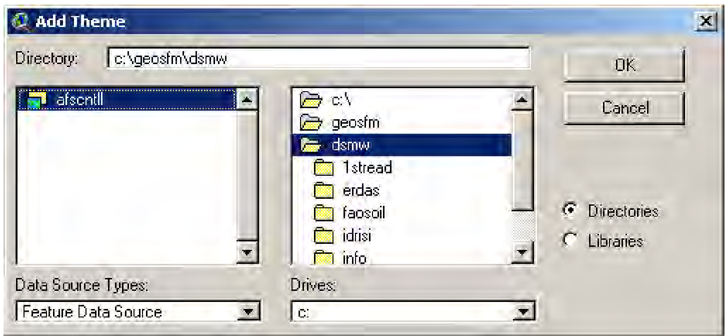
Browse to populate the **Output Data Source** and click **OK**.



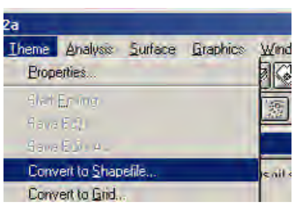
The **Import71 Utility** dialog box displays, stating **Import Complete**. Click **OK**.



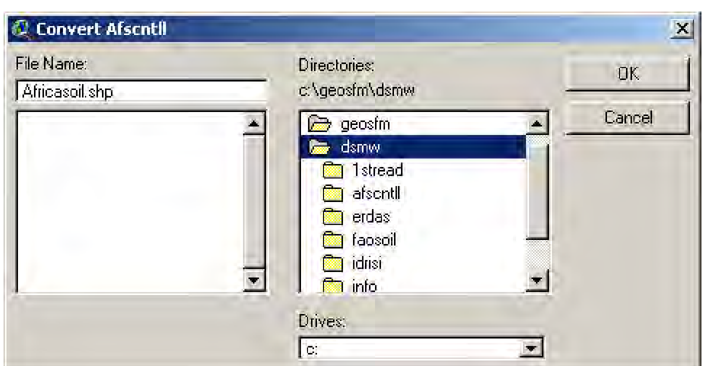
Add the **AFSCNTLL** file to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Feature Data Source**. Click the **afscntll** file from the **c:\GeoSFM\dsmw** directory and click **OK** to add to the View.



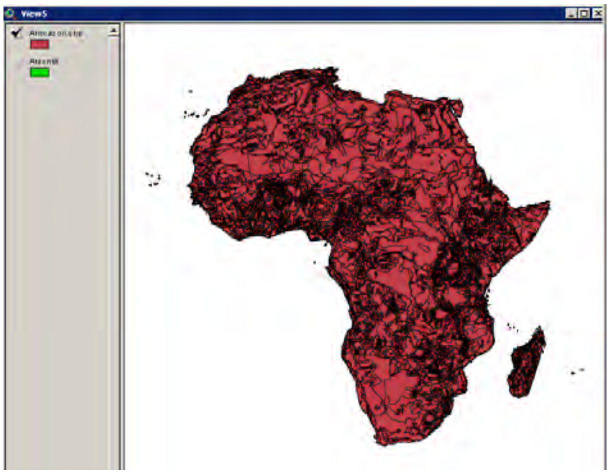
Next, convert **Afscntll** to a shapefile. Select the theme **Afscntll** so that it appears in a raised box. From the **Theme** menu, select **Convert to Shapefile**.



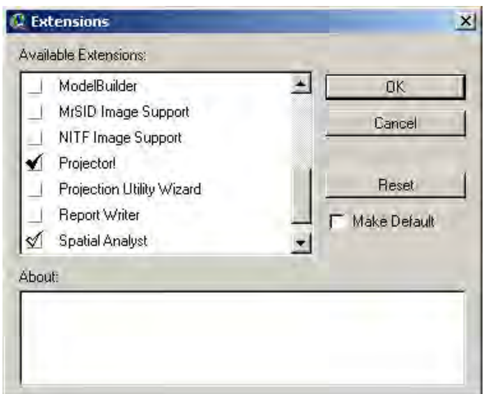
The **Convert Afscntll** dialog box appears. Navigate to the **c:\GeoSFM\dwms** directory and name your new shapefile in **File Name**. Click **OK**.



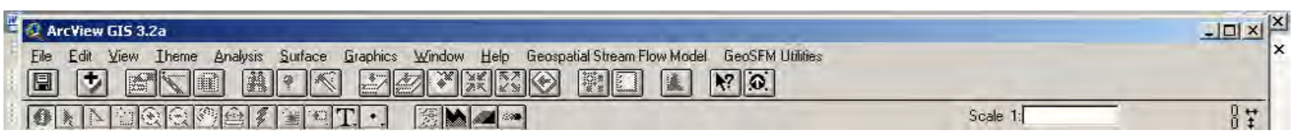
The new shapefile is added below.



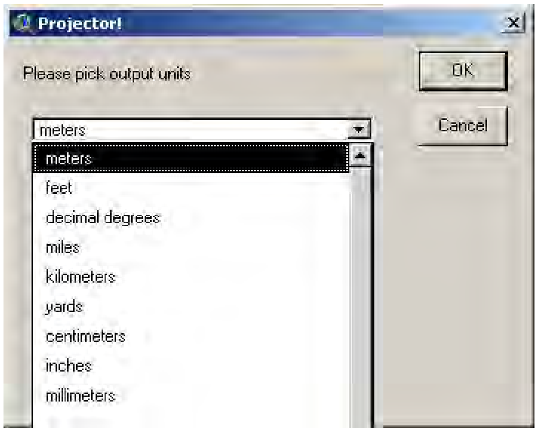
Next, you need to project the shapefile to Lambert Equal-Area Azumuthal projection.
Start by activating the **Projector!** Extension. From the **File** menu, select **Extensions**. Scroll down to **Projector!** and click the box to the left to select it. Click **OK**.



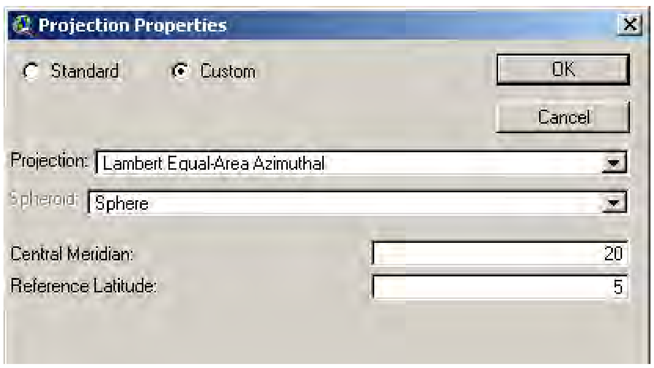
The tool bar is updated to add the **change projection** button.



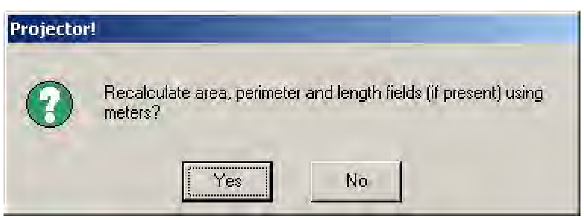
Click the **change projection** button to start the projection process. The first dialog box is displayed. Select meters from the drop-down list for the output units. Click **OK**.



Next, the **Projection Properties** box is displayed. Select the **Custom** radio button. In the **Projection** drop-down list, select **Lambert Equal-Area Azimuthal**. Change the **Central Meridian** to **20** and the **Reference Latitude** to **5** for the continent of Africa.



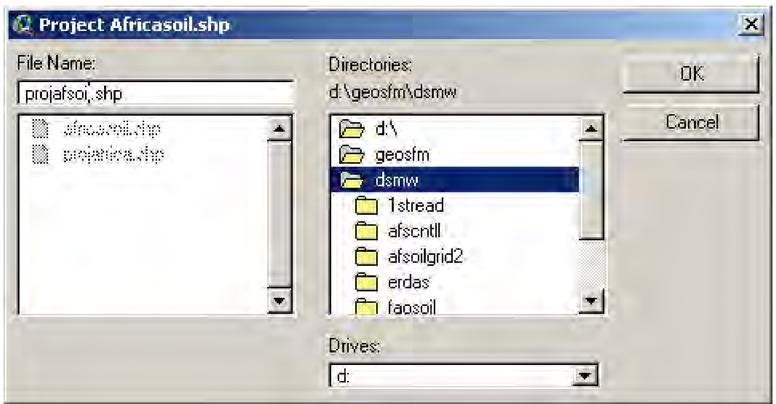
Click **Yes** when prompted, **Recalculate area, perimeter and length fields (if present) using meters?**



Click **Yes** when asked to **Add projected shapefiles as themes to a view**.

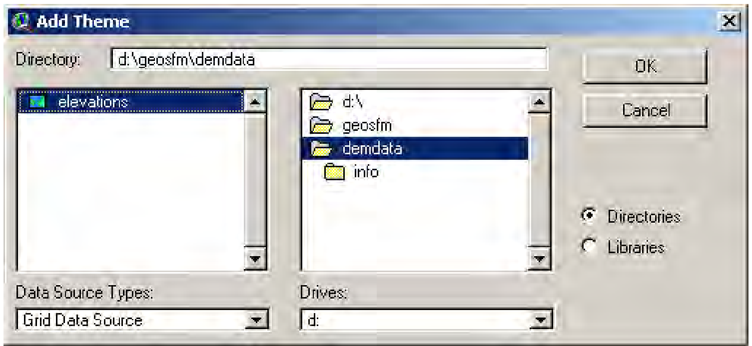
Next, select the view where you want the theme added. Select **View1** and click **OK**.

Navigate to your directory, and under **File Name**, name your new projected shapefile. Click **OK**.



Now you can delete the original **africasoil.shp**. From the **Edit** menu, select **Delete Theme**. (Make sure your theme is in a raised box.)

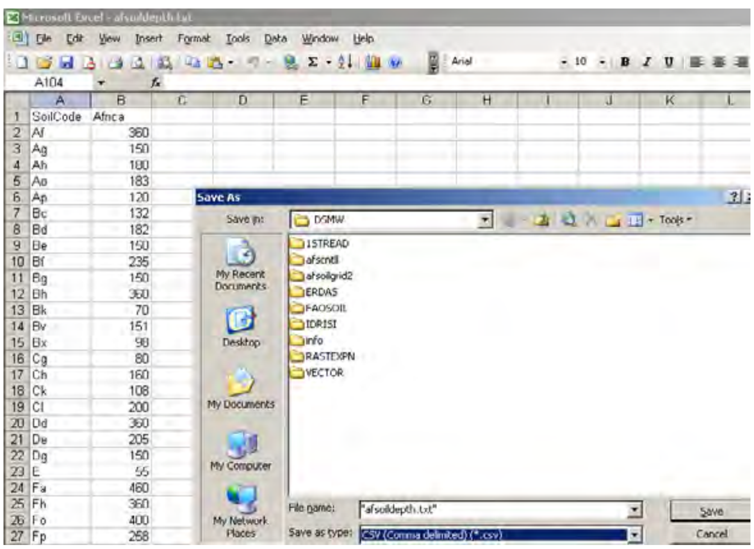
Before you begin creating the soil depth grid, add the **elevations** theme to your view. Use this theme to define the output grid extent and the cell size. Add the **elevations** file to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Click the **elevations** file from the **c:\GeoSFM\demdata** directory and click **OK** to add to the View.



Now you can begin to create six different soil characteristic grids; these grids will be needed for generating the **Basin Characteristics File**.

Soil Depth

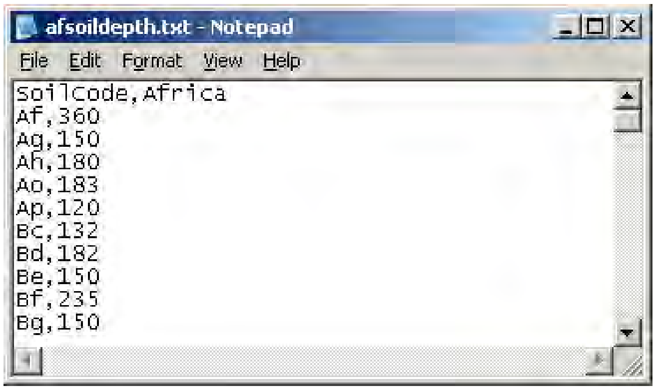
Soil depth will be the first grid created; it is a hydrological active soil layer depth grid. The source for this data will be a data set found in NASA Technical Memorandum 4286 (Webb and others, 1993). It contains a soil thickness data set with 106 soil types for six continents. The data in the **Soil Code** and **Africa** columns are required for completing this tutorial. Create a text document with two columns of data, as shown below.



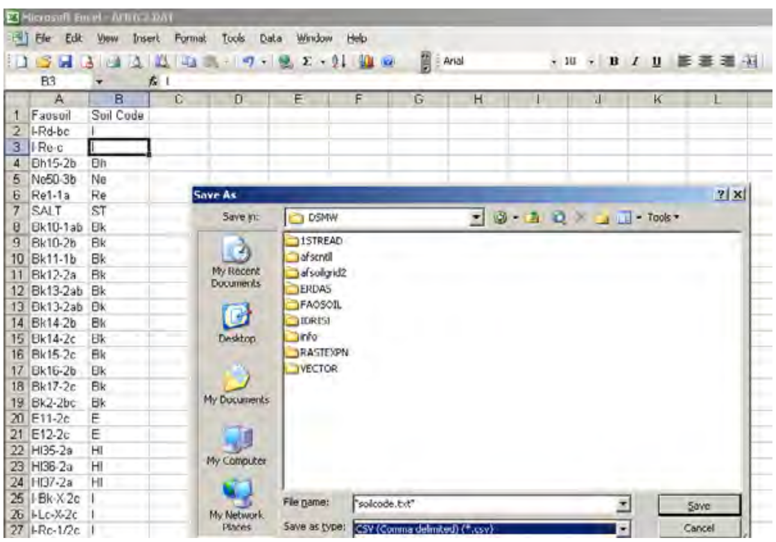
Populate:
File name:
"afsoildepth.txt"

Save as type:
CSV (Comma delimited) (*.csv)

The table is displayed in Notepad.



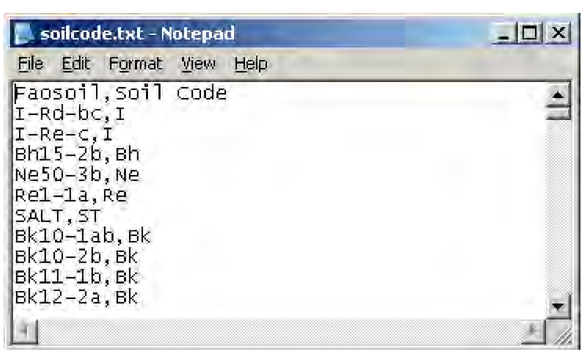
The soil depths in this file are associated with soil types (depicted by soil codes). To map the spatial distribution of soil depth, the soil codes must be associated with specific soil polygons so that the depths can be transferred to the polygons. The soil polygons for Africa are stored in two files, **AFRIC1.DAT** and **AFRIC2.DAT**, which are found in the FAOSOIL\DATA folder on the DSMW CD. The data in the **Faosoil** column (4th column) and the **Soil Code** column (8th column) of these two files must be extracted into a new text document.



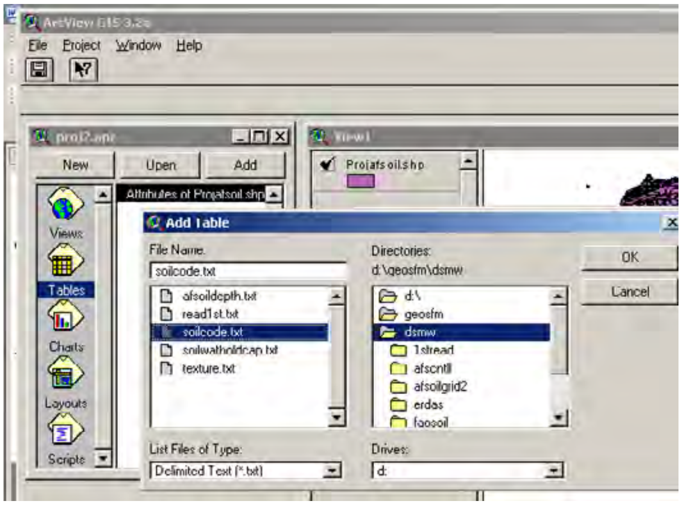
Populate:
File name:
"soilcode.txt"

Save as type:
CSV (Comma delimited) (*.csv)

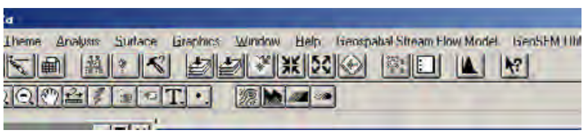
The table is displayed in Notepad.



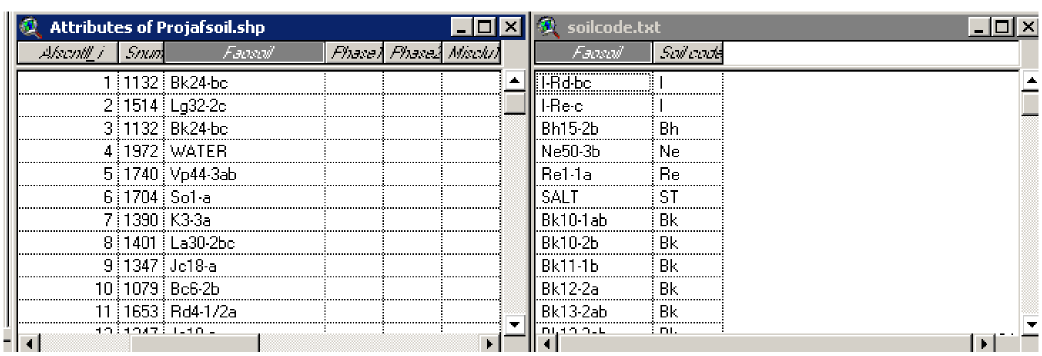
Now add the tables to a View by selecting **Tables** and clicking the **Add** button in the **Project** window. First, select the **soilcode.txt** as seen below. Click **OK**.



Open the attribute table of **Projafsoil.shp** by selecting the **Projafsoil** theme and clicking the **Table** icon on the tool bar.



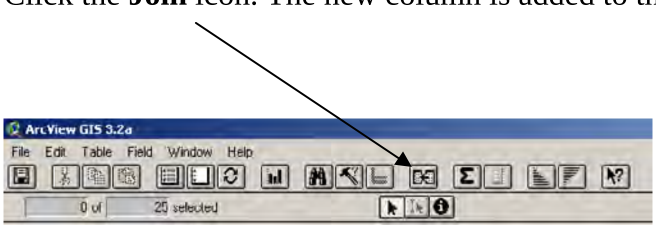
Position the tables as seen below. Highlight the common column headers in both tables—**Faosoil** in **Attributes of Projafsoil** and **Faosoil** in **soilcode.txt**. Make the destination table active (adding fields to **Attributes of Projafsoil.shp**).



Afscntl_i	Snum	Faosoil	Phase1	Phase2	Misclu
1	1132	Bk24-bc			
2	1514	Lg32-2c			
3	1132	Bk24-bc			
4	1972	WATER			
5	1740	Vp44-3ab			
6	1704	So1-a			
7	1390	K3-3a			
8	1401	La30-2bc			
9	1347	Jc18-a			
10	1079	Bc6-2b			
11	1653	Rd4-1/2a			

Faosoil	Soil code
I-Rd-bc	I
I-Re-c	I
Bh15-2b	Bh
Ne50-3b	Ne
Re1-1a	Re
SALT	ST
Bk10-1ab	Bk
Bk10-2b	Bk
Bk11-1b	Bk
Bk12-2a	Bk
Bk13-2ab	Bk

Click the **Join** icon. The new column is added to the **Attributes of Projafsoil** table.



Again, select **Tables** and click the **Add** button in the **Project** window to add the second table to the View. This time, select **afsoildepth.txt**. Click **OK**.

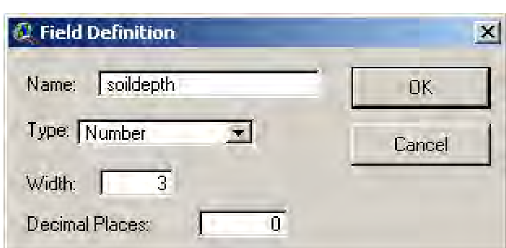
Position the tables side by side. Highlight the common column headers in both tables—**Soil code** in **Attributes of Projafsoil** and **Soilcode** in **afsoildepth.txt**. Make the destination table active (adding fields to **Attributes of Projafsoil.shp**).

Click the **Join** icon. The new column is added to the **Attributes of Projafsoil** table.

Now add a new field to the **Attributes of Projafsoil.shp**. From the **Table** menu, select **Start Editing**.

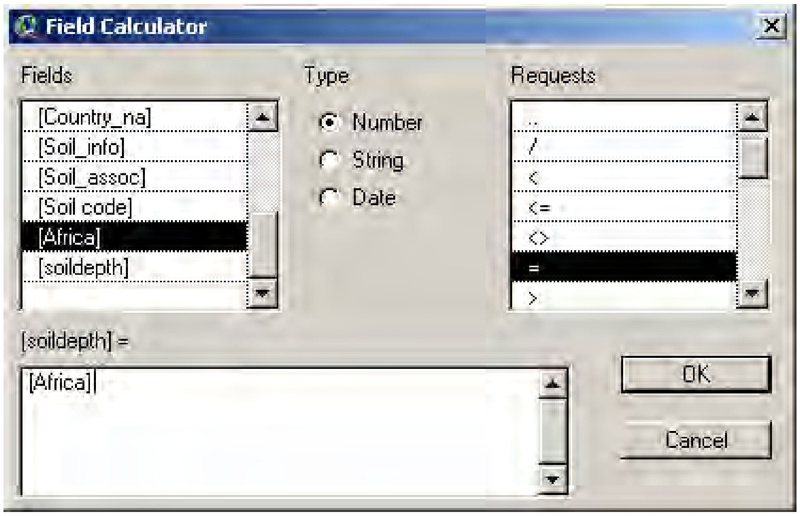
Then from the **Edit** menu, select **Add Field**.

The **Field Definition** dialog box opens.
Populate:
Name: with the name of your new field
Type: number
Width: defaults to 16; changed to 3
Decimal Places: 0



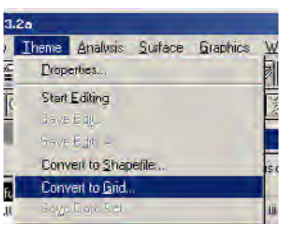
Select the calculator icon on the tool bar.

The **Field Calculator** opens. Enter the following calculation:
[soildepth] = [Africa]
Click **OK**.

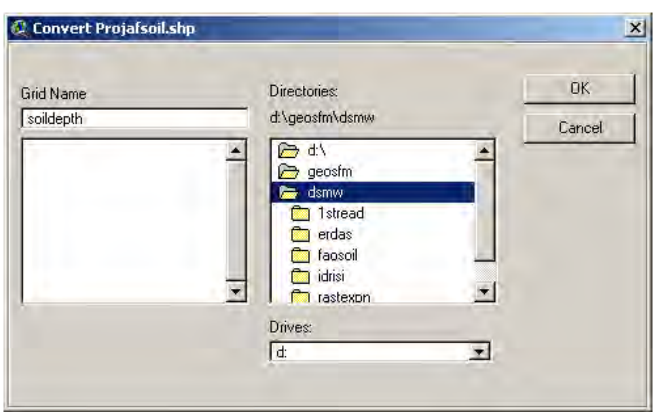


The new **soildepth** field will be populated with the values from the **Africa** field.

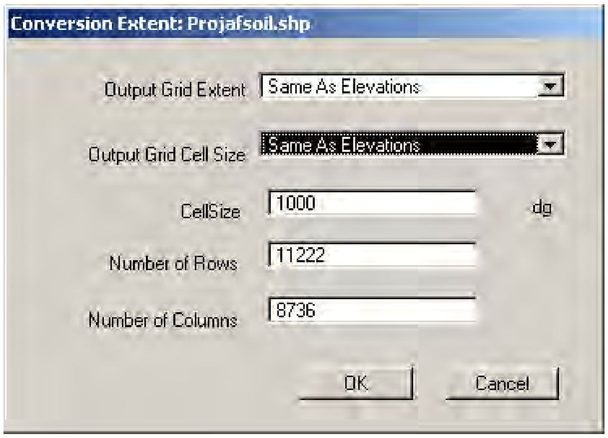
In the **Table** menu, select **Save Edits** to save the changes to the attribute table. Then in the **Tables** menu, select **Stop Editing**. Create a new grid with the data from the **soildepth** field. From the **Theme** menu, select **Covert to Grid**.



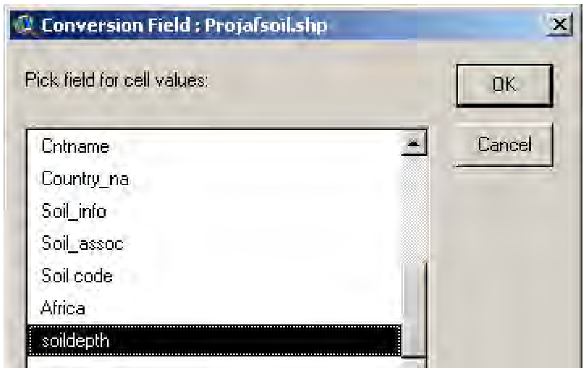
The **Convert Projafsoil.shp** dialog box opens. Select the **c:\geosfm\dsmw** directory and name your grid in **Grid Name**. Click **OK**.



The **Conversion Extent** dialog box opens. In **Output Grid Extent**, select **Same as Elevations** from the drop-down list. In **Output Grid Cell Size**, select **Same as Elevations**. The other values will default. Click **OK**.

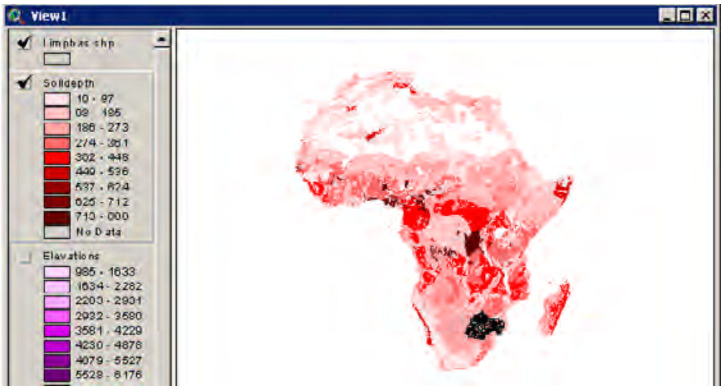


Next, the **Conversion Field** dialog box opens. In **Pick field for cell values**, select **soildepth**. Click **OK**.



When asked to **Add grid as theme to the view**, click **Yes**.

The new Africa soil depth grid is displayed.



Texture

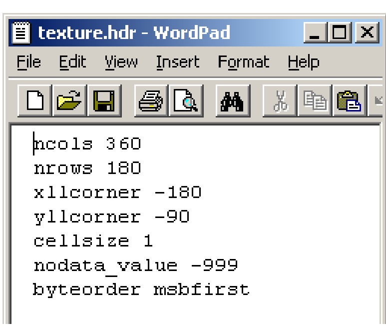
The second required data layer is the soil texture grid. GeoSFM uses three different texture classes (coarse, medium, and fine) for defining basin soil characteristics. The soil texture data set used in the model is from Zobler (1986). It is contained in a file named **FAO_soil.textur.1nnegl.bin**, which is stored at the following FTP site: ftp://daac.gsfc.nasa.gov/data/inter_disc/hydrology/soil/. The following procedure is needed before importing these binary files into ArcView. First, the file is renamed from **FAO_soil.textur.1nnegl.bin** to **texture.bil**. Next, a header file is created with the following parameters:

ncols: the number of columns in the data set
nrows: the number of rows in the data set
xllcorner: the x-coordinate of the center or lower-left corner of the lower-left cell
yllcorner: the y-coordinate of the center or lower-left corner of the lower-left cell
cell size: cell size of the data set

nodata_value: value in the file assigned to cells whose value is unknown. This keyword and value is optional. The nodata_value defaults to –9999.

byteorder: the byte order of the binary cell values. You can choose between two keywords, msbfirst or lsbfirst. Msbfirst is used for cell values written with the most significant bit first. Lsbfirst is used for cell values written with the least significant bit first.

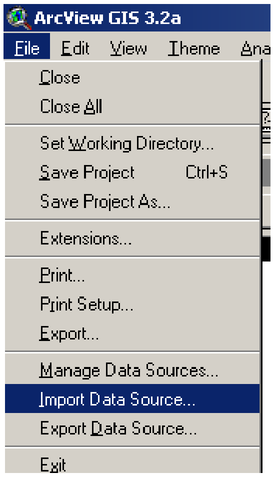
The parameters needed to populate the new header file can be found in the **Readme.fao_soil** documentation. The parameters of the header file are saved in an ASCII file using any simple text editor, such as WordPad, as shown in the example below.



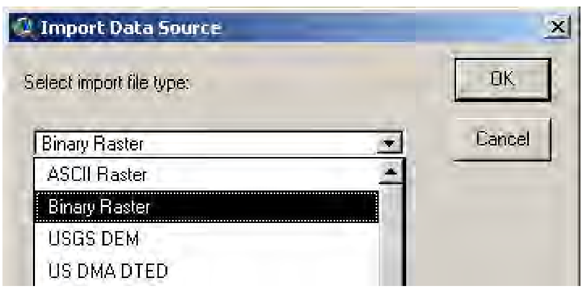
Save the document. From the **File** menu, select **Save As**.

File Name: **texture.hdr**
Save as Type: **Text Document**
Directory: **c:\GeoSFM\Zobler**

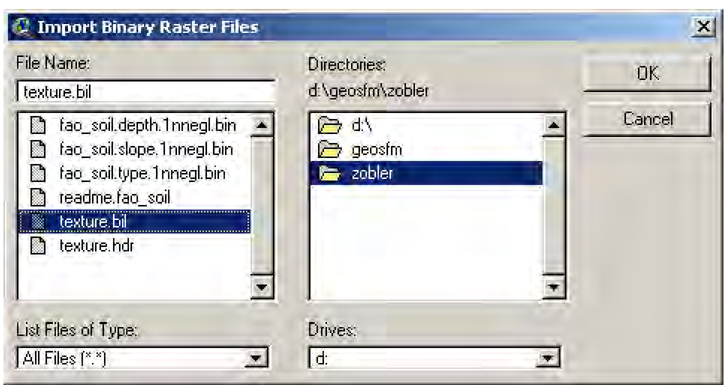
Next, import the **texture.bil** file. From the **File** menu, select **Import Data Source**.



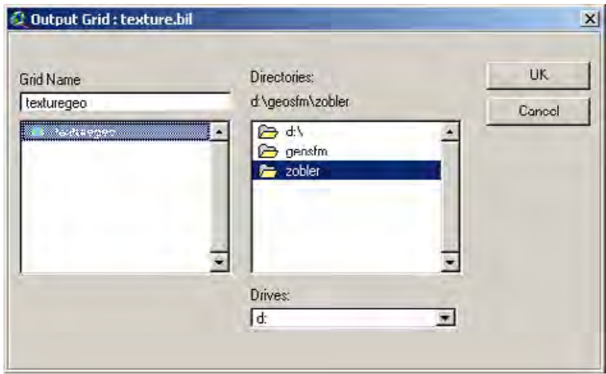
The **Import Data Source** dialog box is displayed; from the drop-down list, select **Binary Raster**. Click **OK**.



Navigate to your directory and change the **List Files of Type** to **All Files (*.*)**. Now you can select the **texture.bil** to populate the **File Name**. Click **OK**.

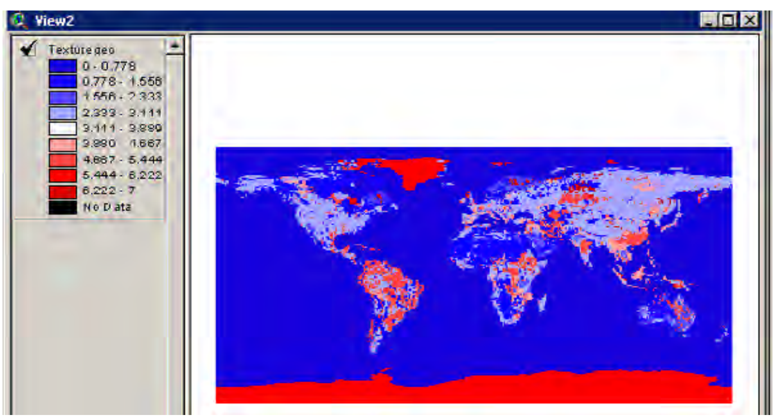


The **Output Grid** dialog box opens. Navigate to your directory and name your new grid under **Grid Name**. You are now converting the .bil file to a grid. Click **OK**.

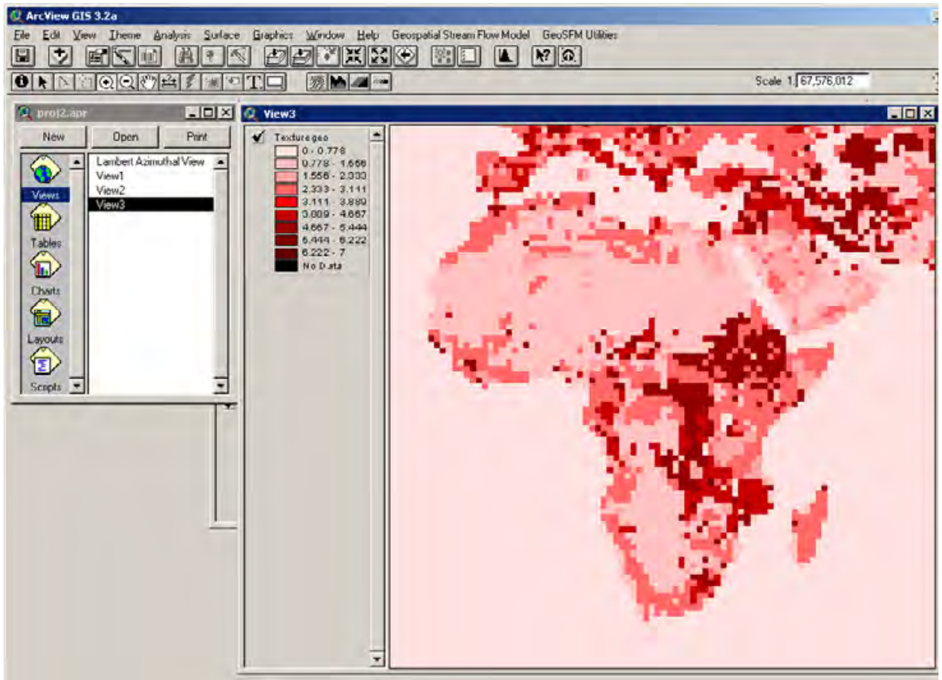


Select **Yes** when prompted to **Add grid as theme to the View**.

The new **Texturegeo** grid is displayed.

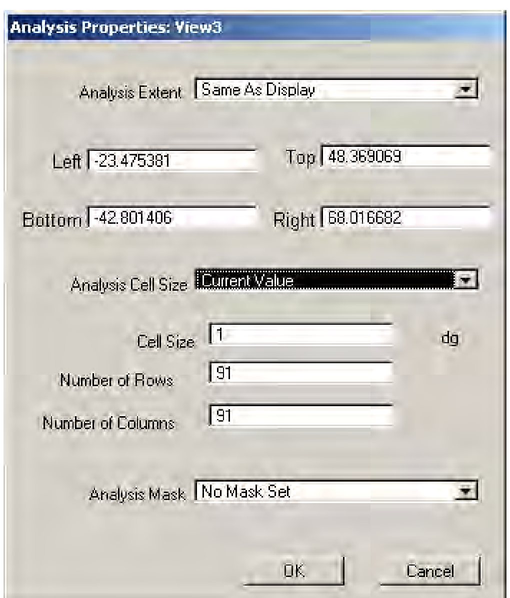


Zoom into the continent of Africa by clicking the **zoom** icon and putting a box around Africa. To accomplish this, click one corner of the area you want to capture, then hold the mouse button and drag to the opposite corner, boxing in Africa. See display below.

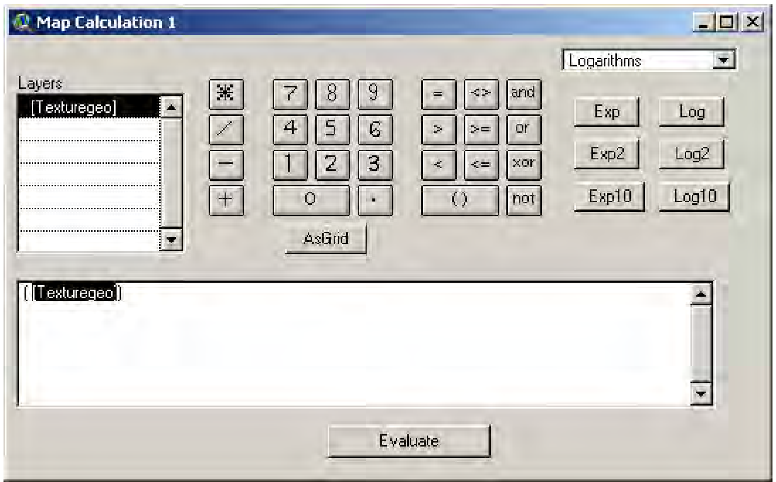


Now set the analysis extent to the display so you can clip the African continent (what is captured in our display area) from the global data set.

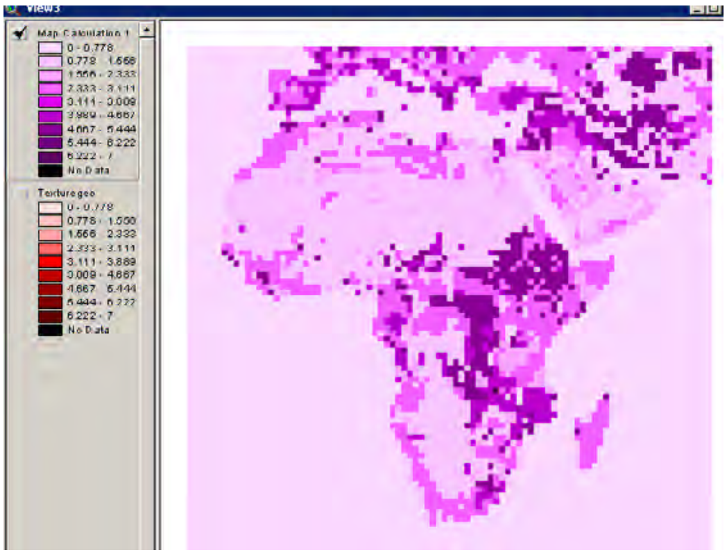
From the **Analysis** menu, select **Properties**. Select **Same As Display** from the drop-down list to define the **Analysis Extent**. Use the default values for all other fields. Click **OK**.



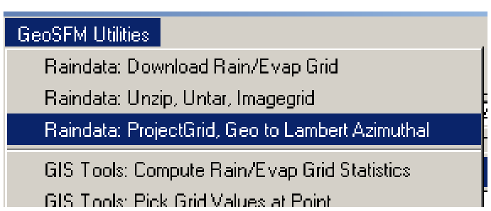
The **Analysis extent** is now defined; using the **Map Calculator**, you can clip the extent area from the global data area. To accomplish this, click the **Analysis** menu and select **Map Calculator**. From here, select the **Texturegeo** layer. Click **Evaluate**.



The **Map Calculation 1** theme is displayed below. You now have a grid of Africa.



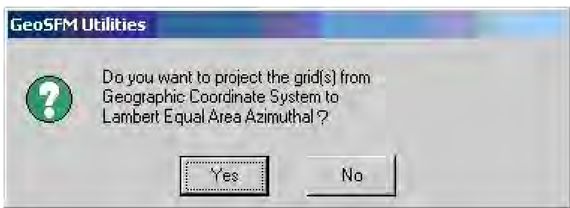
Next, the data set must be transformed from the geographic projection to the Lambert Equal-Area Azimuthal projection to match the other data sets used in the GeoSFM. From the **GeoSFM Utilities** menu, select **Raindata: ProjectGrid, Geo to Lambert Azimuthal**. Normally, this function is used for projecting rain and PET data, but you can project any grid except for large global data sets.



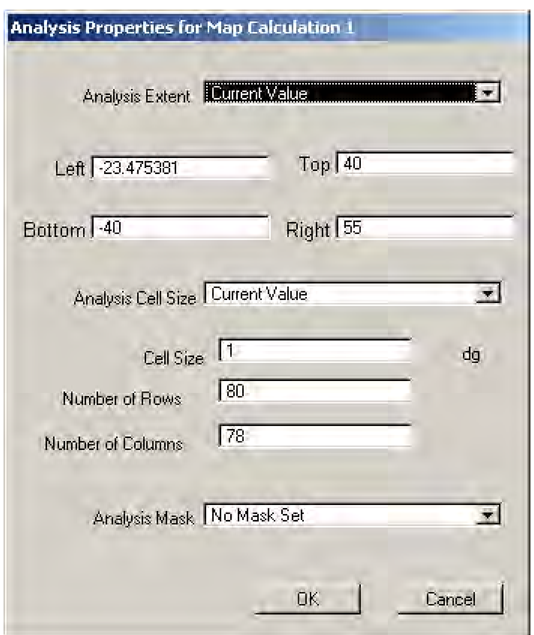
In the first window, **Specify your working directory** is displayed. Click **OK**.

Next, **Select the continent for analysis** is displayed; from the drop-down list, select **Africa**. Click **OK**.

Click **Yes** when prompted to project the grid to Lambert Equal-Area Azimuthal.

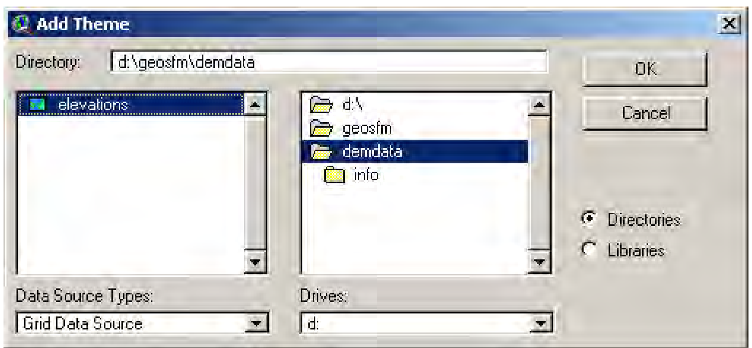


Use default values for **Analysis Properties**. Click **OK**.

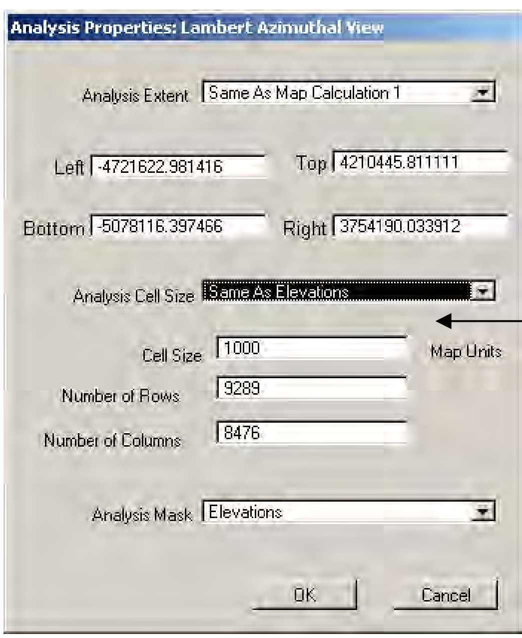


The last dialog box informs you that the data set will need to be saved. Click **OK**.

Before you save the data set, change the cell size to match the cell size of the other data you are using. First, add the **elevations** grid. Click the **Add theme** icon. Navigate to the **demdata** folder. Change **Data Source Types** to **Grid Data Source** and select **elevations**. Click **OK**.



After the elevations theme is added, click the **Analysis** menu and select **Properties**.

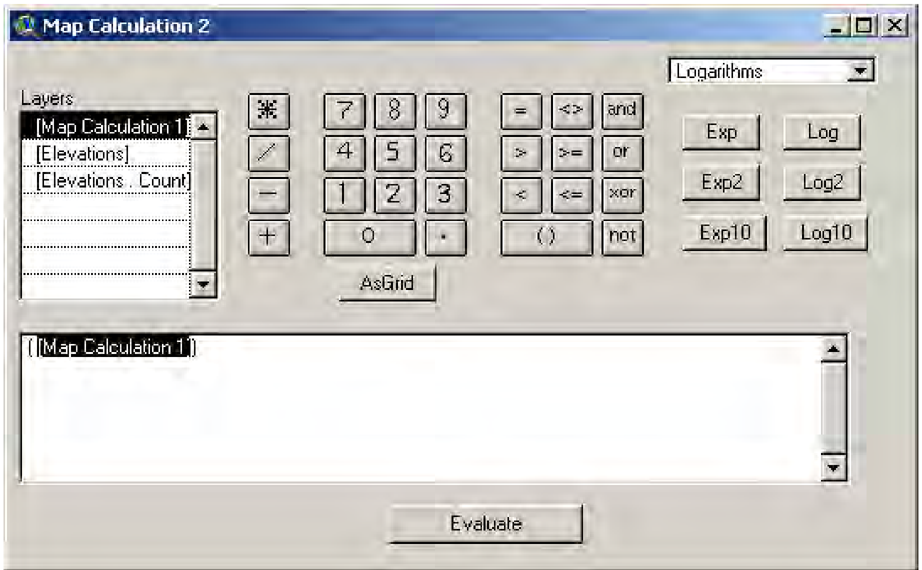


Populate:
Analysis Extent – Same As Map Calculation 1

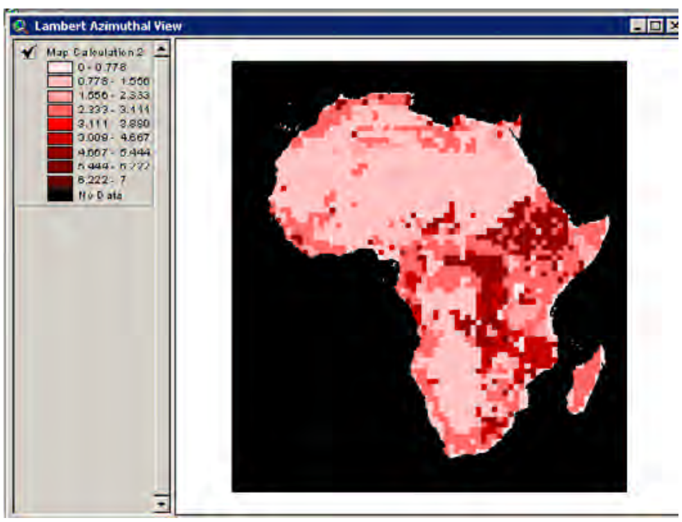
Analysis Cell Size – **Same As Elevations**

Analysis Mask – **Elevations**

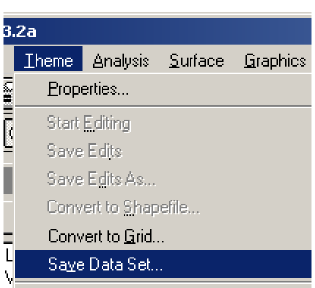
Click **OK** to define the cell size for resampling the **Map Calculation 1**. From the **Analysis** menu, select **Map Calculator**. Double-click **Map Calculation 1** under layers to populate the **Evaluate** window. Click the **Evaluate** button.



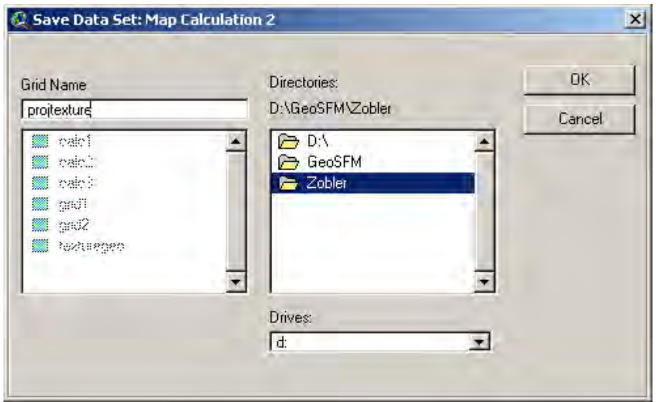
Map Calculation 2 is now projected with a cell size of 1,000.



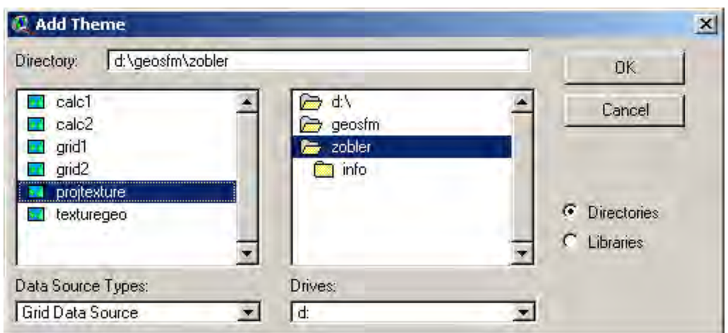
Save the data set. From the **Theme** menu, select **Save Data Set**.



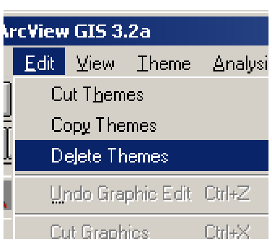
Navigate to your working directory and name your new grid. Click **OK**.



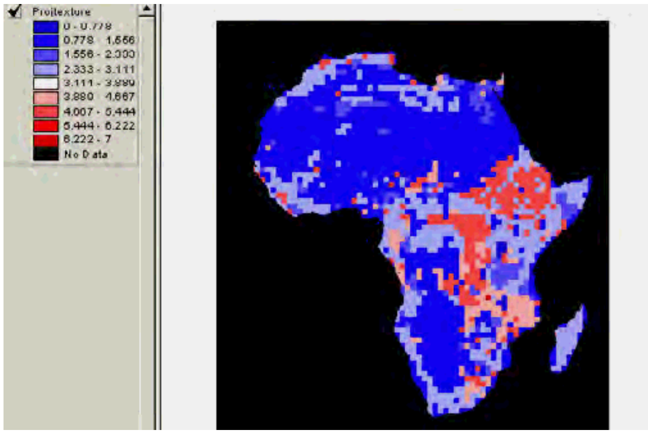
Add the **projtexture** file to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Navigate to your working directory and select **projtexture**. Click **OK** to add to the View.



To delete **Map Calculation 2**, select the theme in your table of contents. Then from the **Edit** menu, select **Delete Themes**.



A new projected texture grid is displayed below.

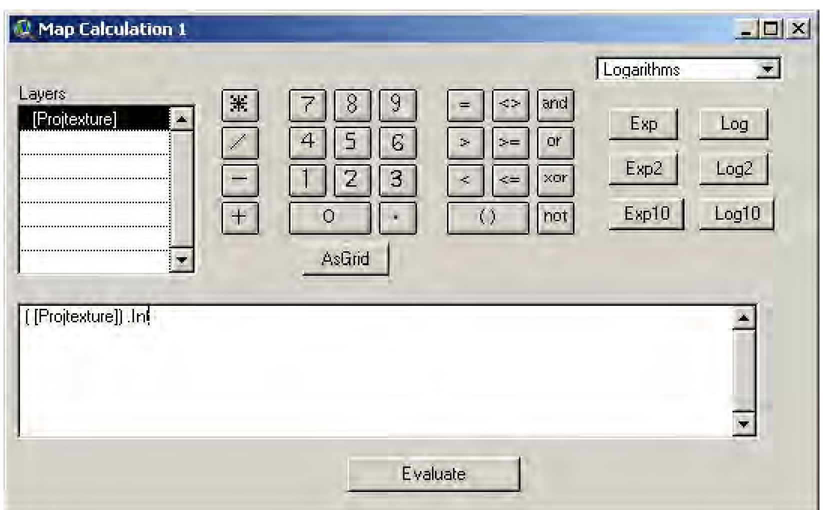


Next, you need to convert the seven different texture classes to GeoSFM’s three classes.

To begin this process, you need to work with integers. From the **Analysis** menu, select the **Map Calculator**. Double-click **projtexture** to populate the evaluation window and type in the following expression exactly as shown:

([Projtexture]) .Int

Click **Evaluate**.

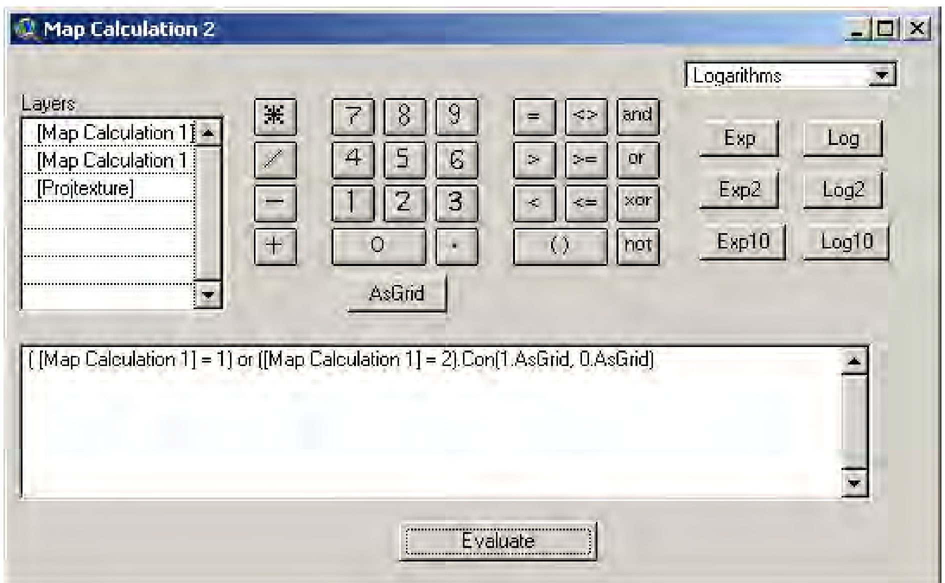


Now you are working with values 0–7. —**Map Calculation 1**.

Next, define GeoSFM class 1 as consisting of classes 1 and 2 from the **Map Calculation 1** theme. From the **Analysis** menu, select **Map Calculator** and type in the following expression exactly as shown:

([Map Calculation 1] = 1) or ([Map Calculation 1] = 2).Con(1.AsGrid, 0.AsGrid)

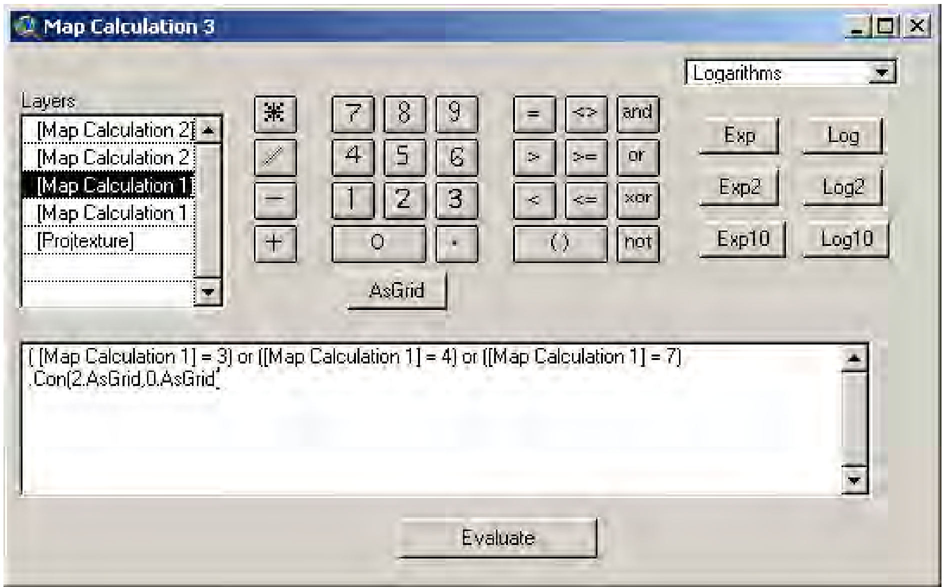
Click the **Evaluate** button**.**



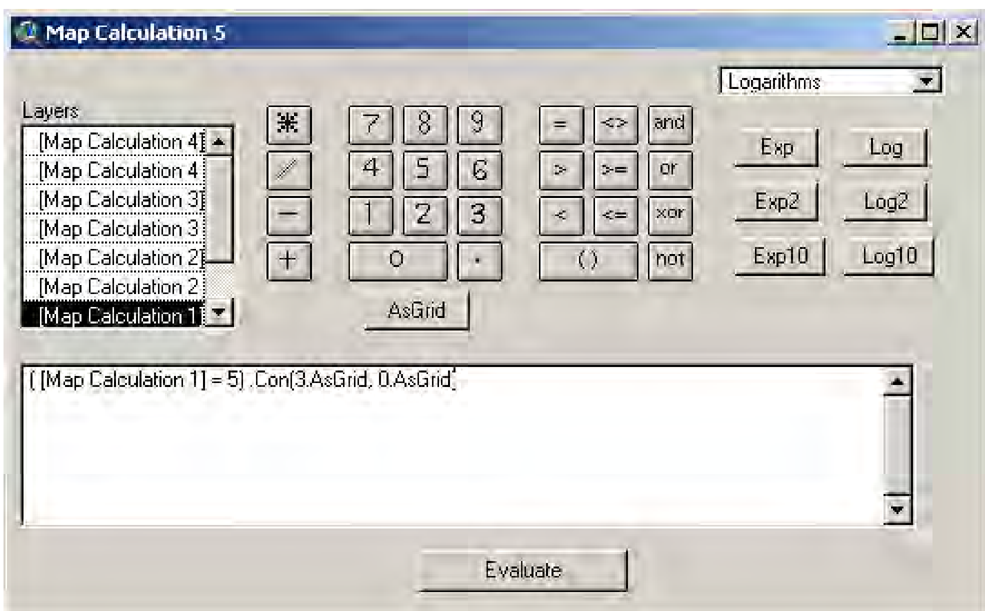
Next, define GeoSFM class 2, which is classes 3, 4, and 7 from the **Map Calculation 1** theme. From the **Analysis** menu, select **Map Calculator** and type in the following expression exactly as shown:

([Map Calculation 1] = 3) or ([Map Calculation 1] = 4) or ([Map Calculation 1] = 7).Con(2.AsGrid, 0.AsGrid)

Click the **Evaluate** button**.**

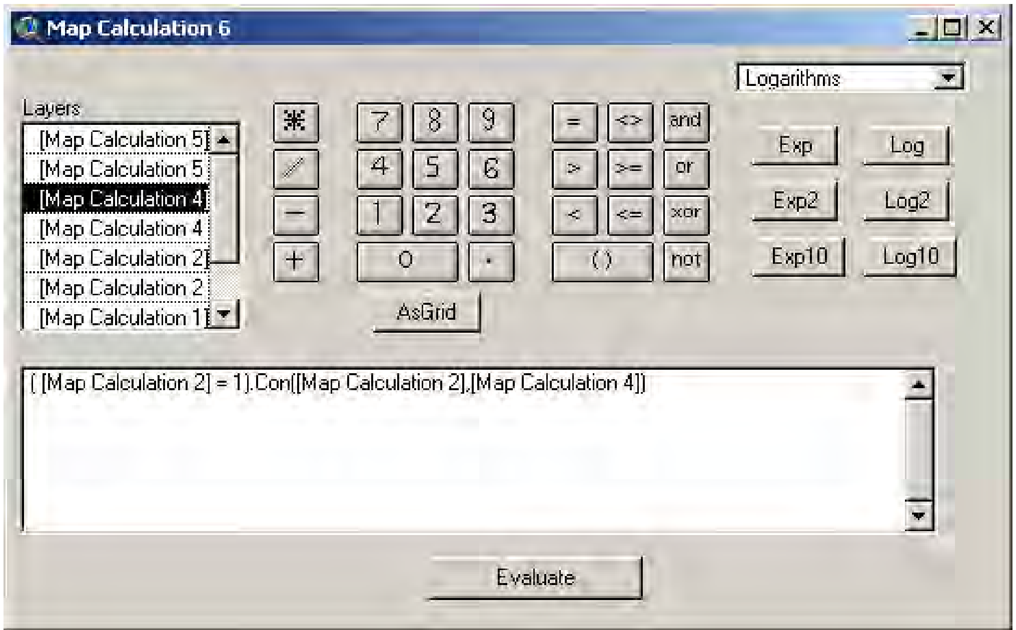


Similarly, define GeoSFM class 3 using class 5 from the **Map Calculation 1** theme by selecting **Map Calculator** from the **Analysis** menu and typing in the following expression and clicking the **Evaluate** button: **([Map Calculation 1] = 5).Con(3.AsGrid, 0.AsGrid)**



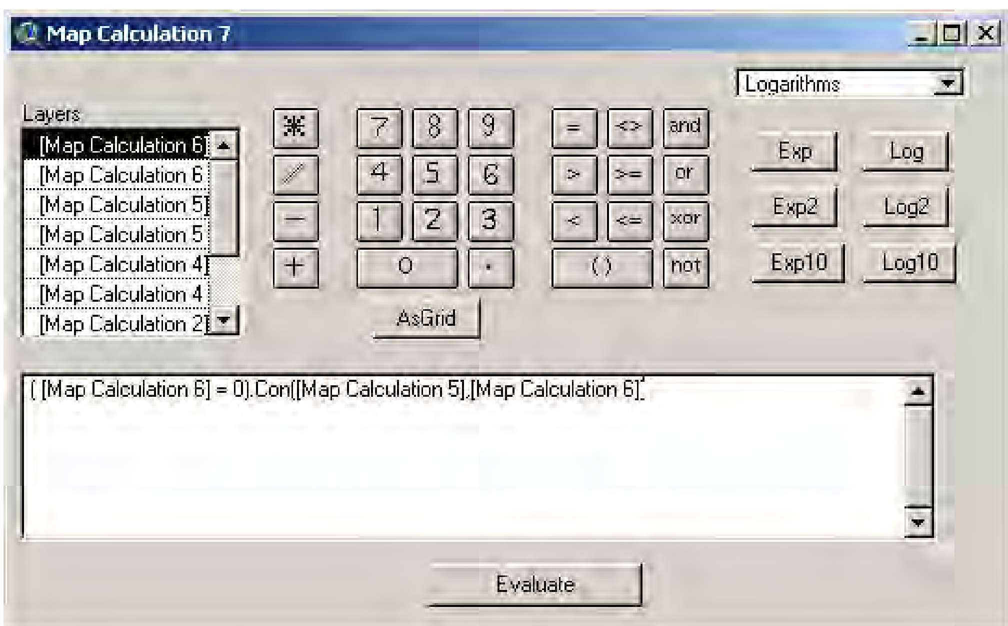
Next, you need to combine the results from Map Calculations 2 and 4 into a single grid with two classes. From the **Analysis** menu, select **Map Calculator** and type in the expression exactly as shown below and click the **Evaluate** button.

([Map Calculation 2] = 1).Con([Map Calculation 2],[Map Calculation 4])

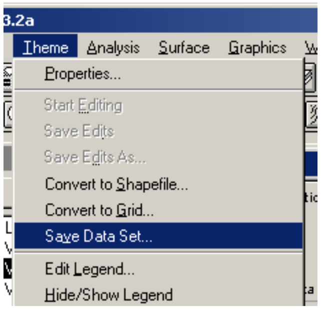


As a final calculation, allow the third class to be combined with the previously combined grid to create a new grid with all three classes. From the **Analysis** menu, select **Map Calculator** and type in the following expression exactly as shown and click the **Evaluate** button**.**

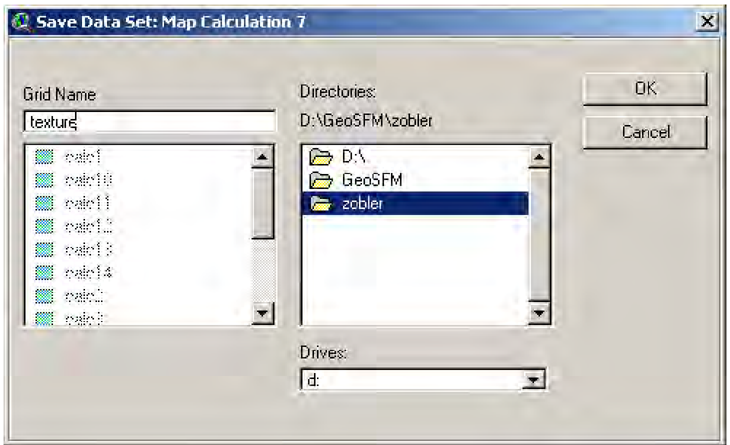
([Map Calculation 6] = 0).Con([Map Calculation 5],[Map Calculation 6])



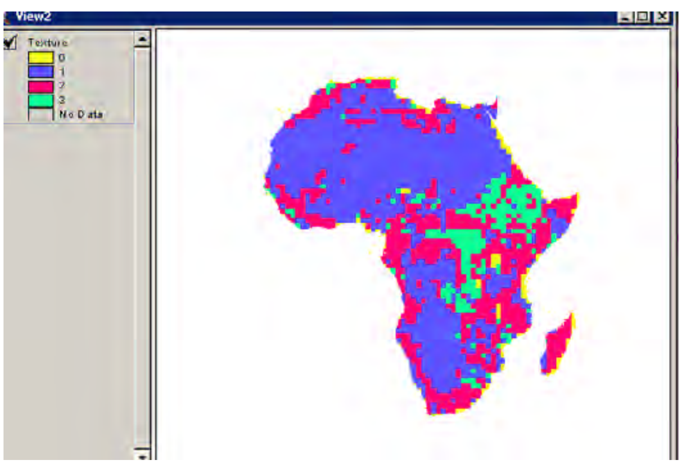
Map Calculation 7 is the final grid with the corrected texture classes. To save this data set, click **Map Calculation 7** to activate the theme in the table of contents. From the **Theme** menu, select **Save Data Set**.



Navigate to your working directory. Type in the grid name, **texture**, and click **OK**.

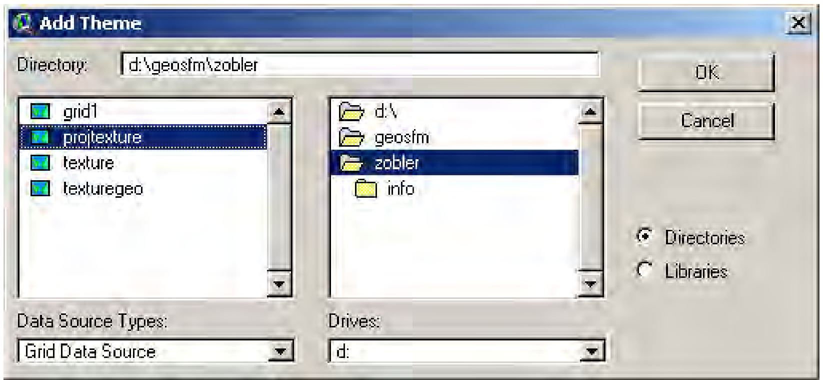


Final **Texture** grid is displayed below.

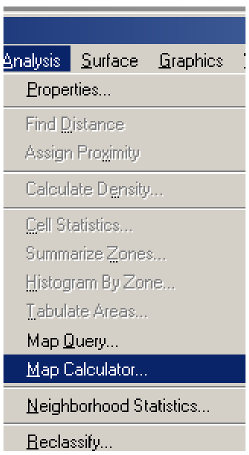


Hydraulic Conductivity

The third soil characteristic grid required is the hydraulic conductivity, Ks. The values are assigned based on the seven Zobler texture classes. First, add the **projtexture** grid used in defining the texture classes. Add the **projtexture** file to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Navigate to your working directory and select **projtexture**. Click **OK** to add to the View.

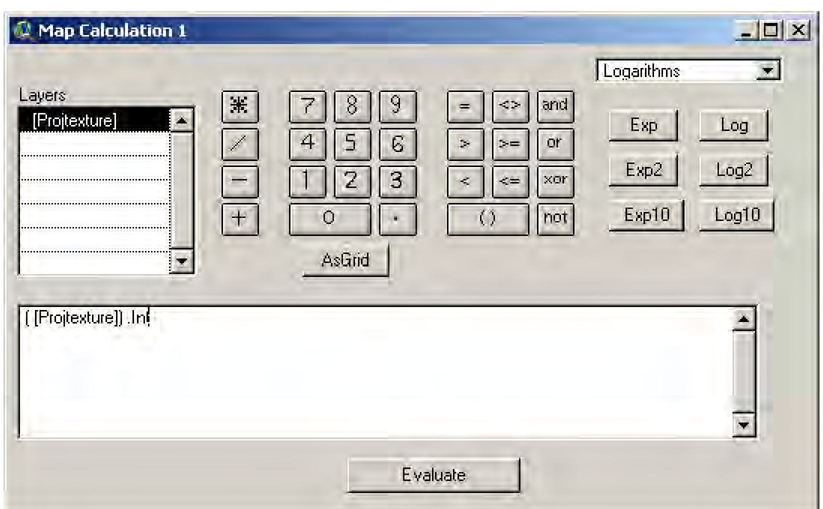


From the **Analysis** menu, select the **Map Calculator**.

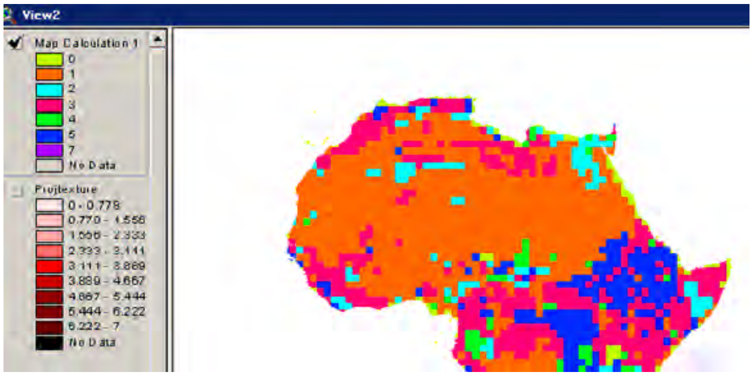


This opens the **Map Calculator** window. Double-click **projtexture** to populate the evaluation window and type in the following expression exactly as shown and click **Evaluate**.

([Projtexture]) .Int

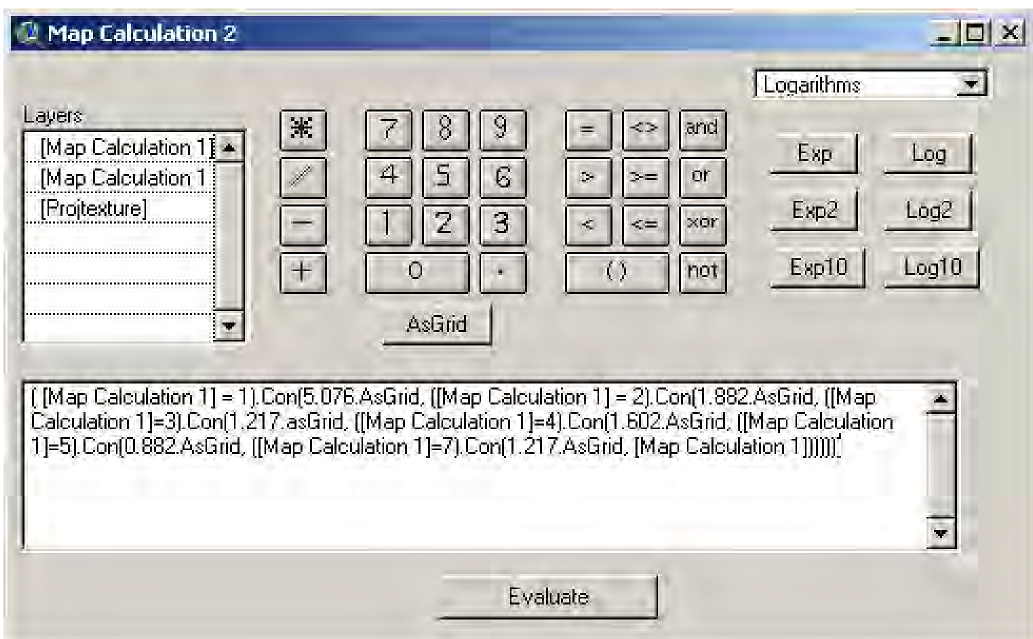


This produces the **Map Calculation 1** grid with values 0–7.



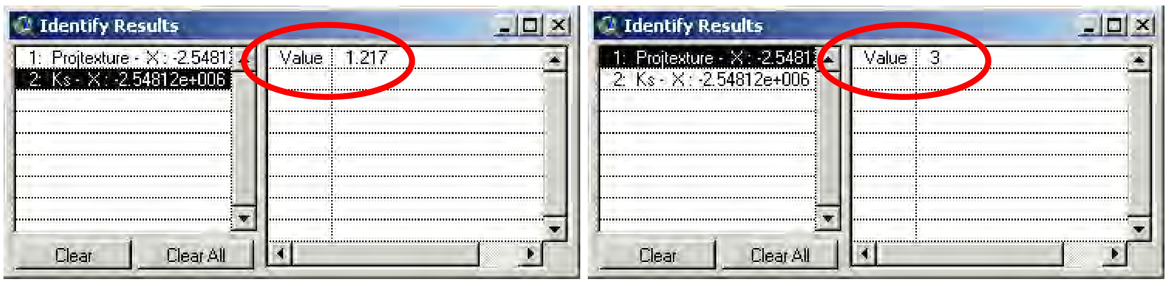
Ks values (in cm/hr) are assigned based on the seven texture classes in the Zobler (1986) data set as follows: 5.076 for coarse, 1.882 for medium/coarse, 1.217 for medium, 1.602 for fine/medium, 0.882 for fine, 0.0 for ice, and 1.217 for organic soils. From the **Analysis** menu, select the **Map Calculator**. Type in the following expression exactly as shown:

([Map Calculation 1] = 1).Con(5.076.AsGrid, ([Map Calculation 1] = 2).Con(1.882.AsGrid, ([Map Calculation 1]=3).Con(1.217.AsGrid, ([Map Calculation 1]=4).Con(1.602.AsGrid, ([Map Calculation 1]=5).Con(0.882.AsGrid, ([Map Calculation 1]=7).Con(1.217.AsGrid, [Map Calculation 1]))))))

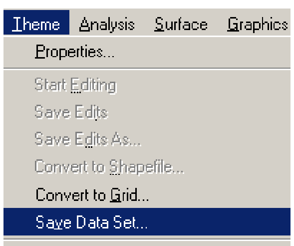


Once the new grid is created, you can check your data values by selecting both your new grid (Ks or **Map Calculation 2**) and the projtexture or (**Map Calculation 1**) grid. To select more than one **Theme** at a time, hold down the shift key while making your selections. Click the **Identify** icon and then click a cell in your grid.

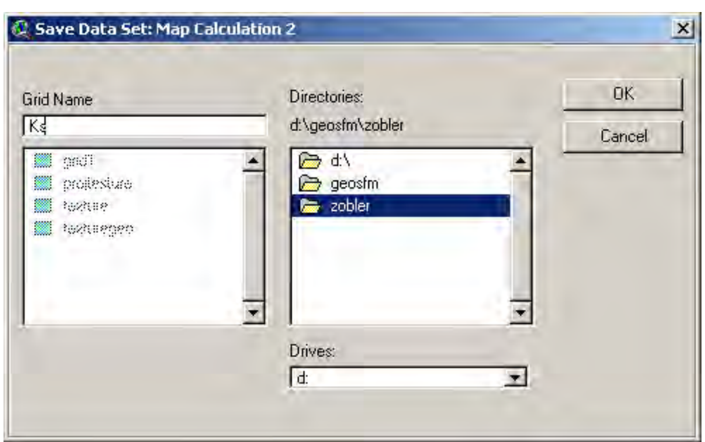
The **Identify Results** window will open showing the values. If the Ks grid is highlighted in the table, the Ks value will display—**1.217** as shown below. If the **projtexture** or **Map Calculation 1** is highlighted, then the value is **3**. This corresponds with our original table, where the texture value 3 has a Ks value of 1.217.



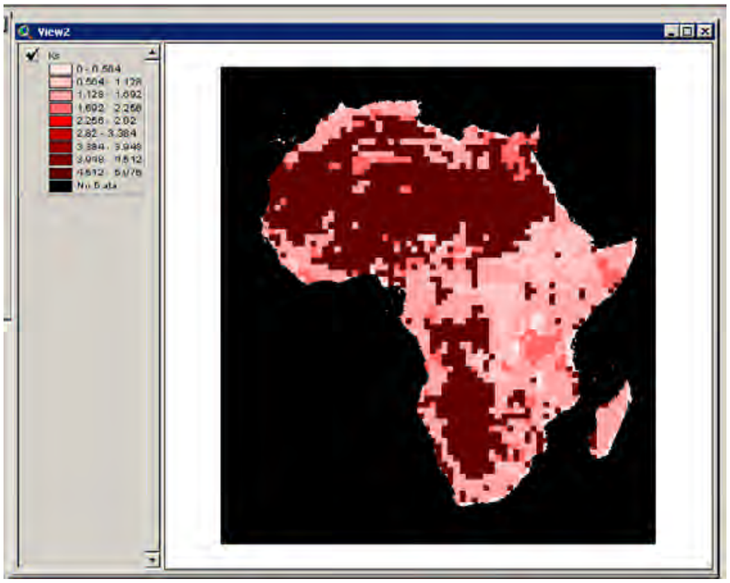
The **Map Calculation 2** grid is the final Ks grid, so we will save this data set. From the **Theme** menu, select **Save Data Set**.



Navigate to your working directory and name your new grid **Ks**. Click **OK**.

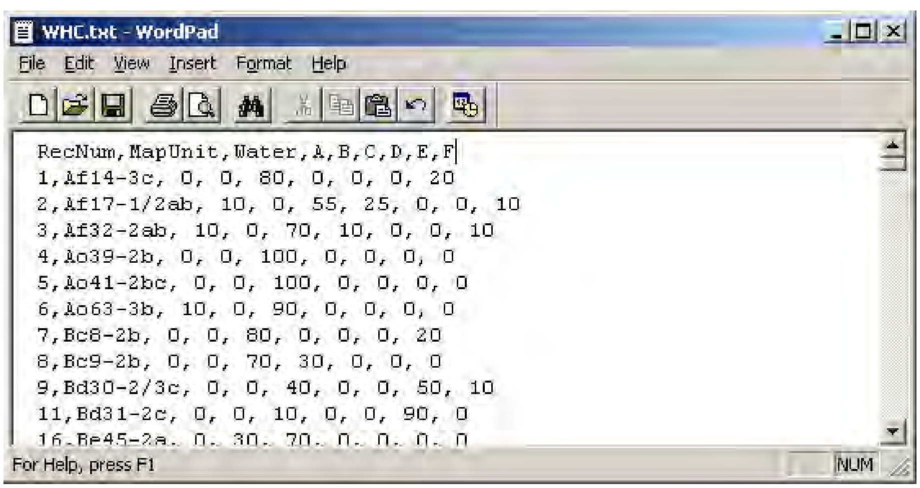


Add the **Ks** grid to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Navigate to your working directory and select **Ks**. Click **OK** to add to the View. The other **Themes** can be deleted by selecting the themes and then from the **Edit** menu selecting **Delete Themes**. Below is the final **Ks** grid.

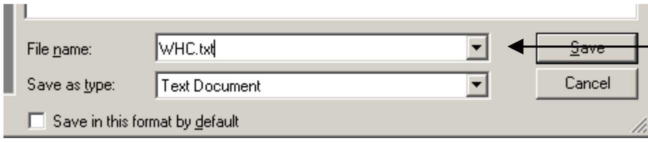


WHC – Soil Water Holding Capacity

The next required soil characteristic grid is the soil water holding capacity grid. The source for this data will be the **SMAXl.ASC** file found on the FAO Digital Soil Map of the World CD in the FAOSOIL\DATA\folder. The data in this file must be joined to the **projafrica.shp** file. Begin by opening the **SMAX1.ASC** file in WordPad. Add a line at the top for header information. The file contains a record number (**RecNum**), the mapping unit name (**MapUnit**), and the soil classes (**Water, A, B, C, D, E,** and **F**) as shown below.

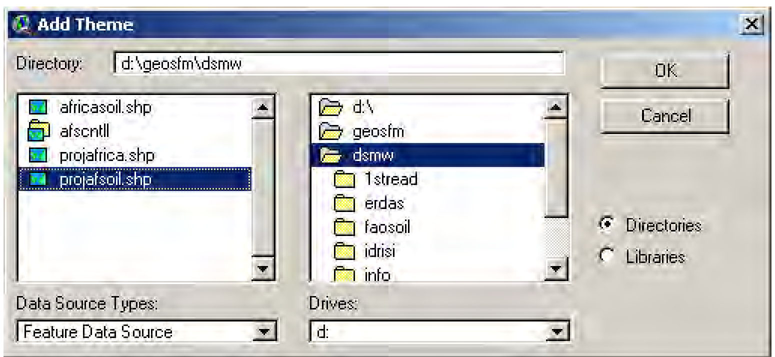


From the **File** menu, select **Save As.** Navigate to your directory**.**



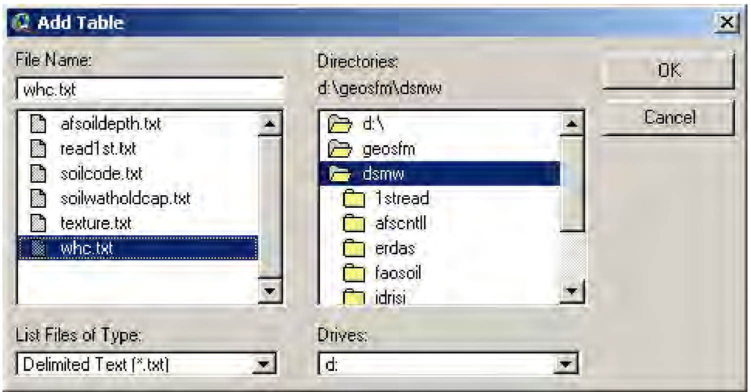
Populate:
File name:
WHC.txt
Save as type:
Text Document

Next, add the **projafsoil.shp** file to the View. Add the shapefile to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Feature Data Source**. Navigate to your working directory and select **projafsoil.shp**. Click **OK** to add to the View**.**

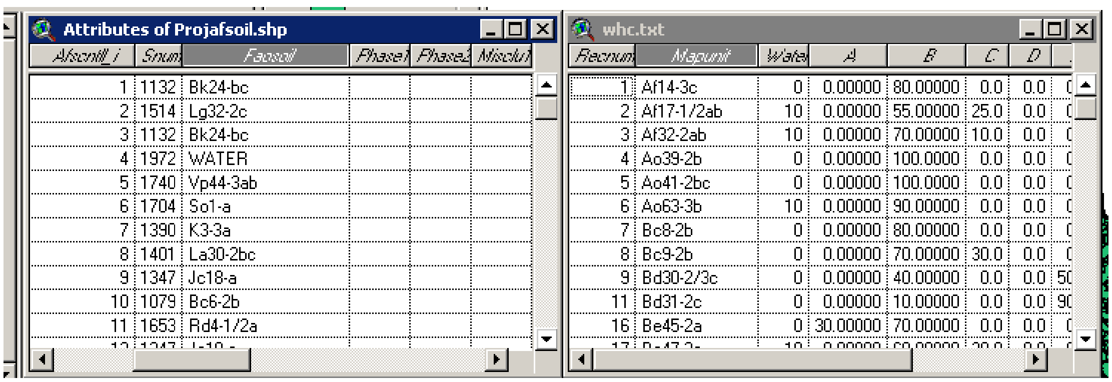


In **View,** click **projafsoil.shp** so it appears in a raised box. Click the **Open** theme **table** icon.

Next, you need to add the new **whc.txt** to the View. From the project window, select **Tables** and click the **Add** button. Navigate to your working directory and select **whc.txt**. **List Files of Type** which will need to be **Delimited Text (*.txt).** Click **OK.**



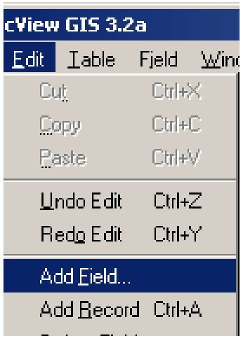
Arrange both tables as seen below. Highlight the common column headers in both tables—**Faosoil** in **Attibutes of projafsoil.shp** and **Mapunit** in **whc.txt** file. Make the destination table active (adding fields to **Attributes of Projafsoil.shp**).



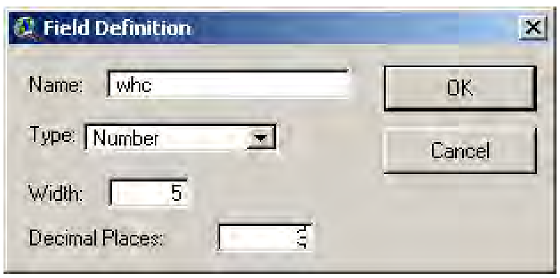
Click the **Join** icon; the new columns are added to the **Attributes of Projafsoil.shp** table.

Now add a new field to the attribute table. From the **Table** menu, select **Start Editing**.

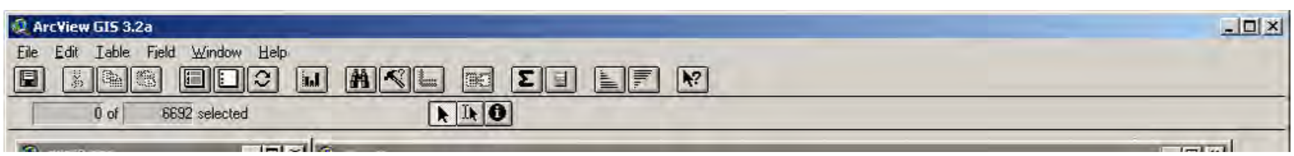
Then from the **Edit** menu, select **Add Field**.



The **Field Definition** box opens.
Populate:
Name: whc
Type: Number
Width: 5
Decimal Places: 3



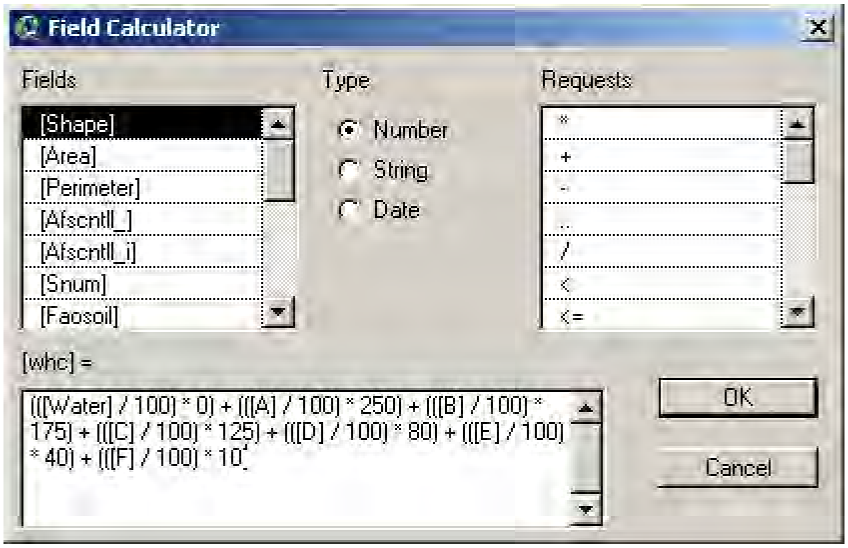
Click **OK**.
Select the **calculator** icon on the tool bar.



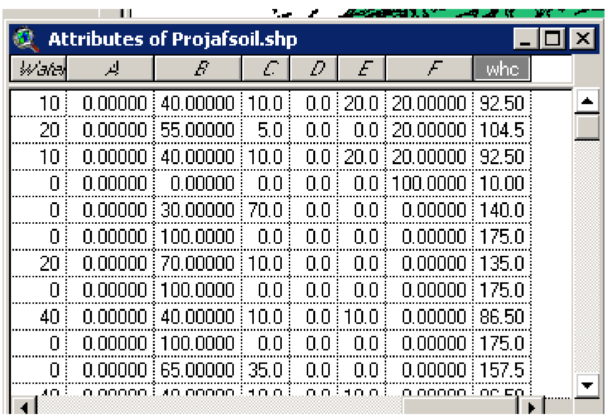
This will display the **Field Calculator**; enter the following expression:

(([Water] / 100) * 0) + (([A] / 100) * 250) + (([B] / 100) * 175) + (([C] / 100) * 125) + (([D] / 100) * 80) + (([E] / 100) * 40) + (([F] / 100) * 10)

Click **OK**.

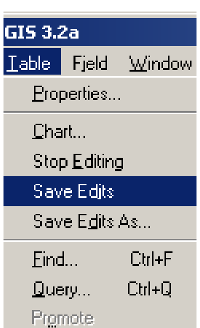


The water, A, B, C, D, E, and F values correspond with the soil class values listed in the attribute table. The values that are multiplied are the median soil moisture and the total available water. After a few moments for processing, the **whc** values are populated in the new column.



Water	A	B	C	D	E	F	whc
10	0.00000	40.00000	10.0	0.0	20.0	20.00000	92.50
20	0.00000	55.00000	5.0	0.0	0.0	20.00000	104.5
10	0.00000	40.00000	10.0	0.0	20.0	20.00000	92.50
0	0.00000	0.00000	0.0	0.0	0.0	100.0000	10.00
0	0.00000	30.00000	70.0	0.0	0.0	0.00000	140.0
0	0.00000	100.0000	0.0	0.0	0.0	0.00000	175.0
20	0.00000	70.00000	10.0	0.0	0.0	0.00000	135.0
0	0.00000	100.0000	0.0	0.0	0.0	0.00000	175.0
40	0.00000	40.00000	10.0	0.0	10.0	0.00000	86.50
0	0.00000	100.0000	0.0	0.0	0.0	0.00000	175.0
0	0.00000	65.00000	35.0	0.0	0.0	0.00000	157.5

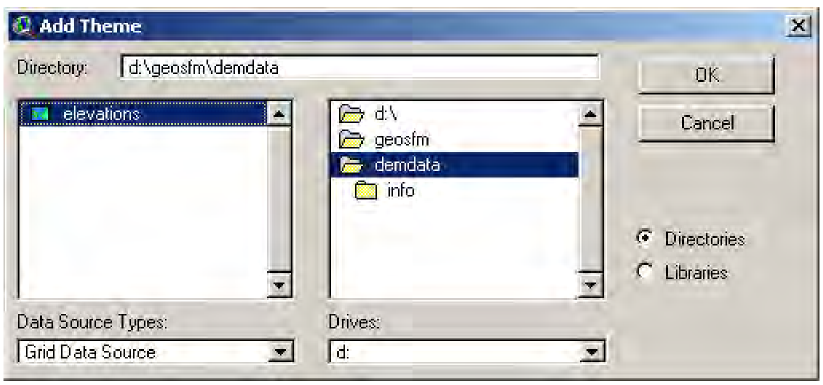
Next, you will save the edits; from the **Table** menu select **Save Edits**.



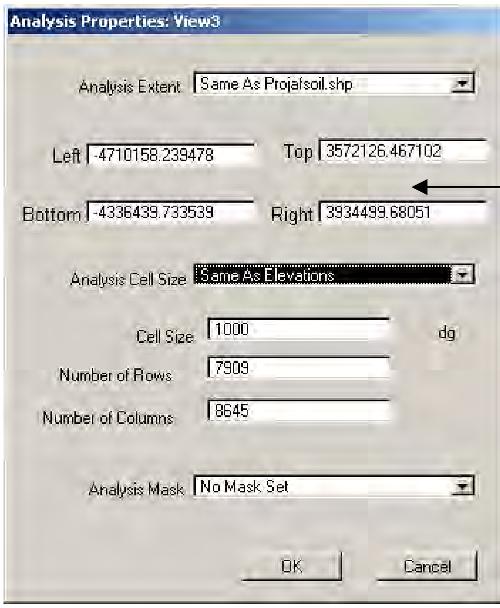
The **Save Edits** dialog box is displayed; click the **Yes** button.

From the **Table** menu, select **Stop Editing**. This will end the editing session.

Set the **Analysis extent** before you create the new **whc** grid. Add the **elevations** grid to the View. Add the grid to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Navigate to your working directory and select **elevations**. Click **OK** to add to the View**.**

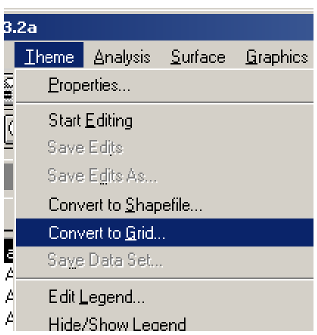


Now, from the **Analysis** menu, select **Properties**. The **Properties** dialog box is displayed.



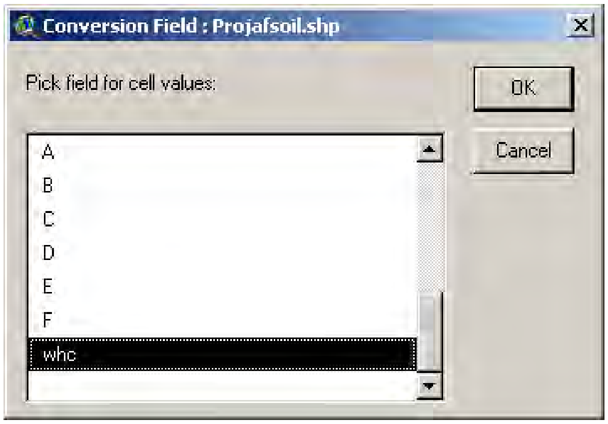
Populate:
Analysis Extent:
Same As Projafsoil.shp
Analysis Cell Size:
Same As Elevations

Click **OK**.
From the **Theme** menu, select **Convert to Grid**.



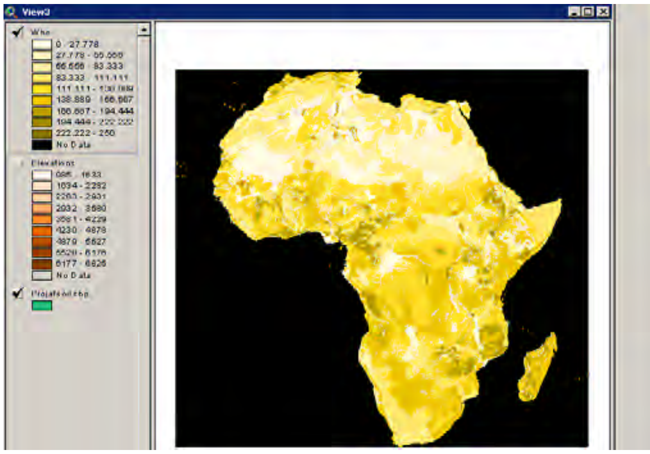
Navigate to your working directory and type in the **Grid Name—whc**. Click **OK**.

When prompted for **Conversion Field**, select **whc** from the drop-down list. Click **OK**.



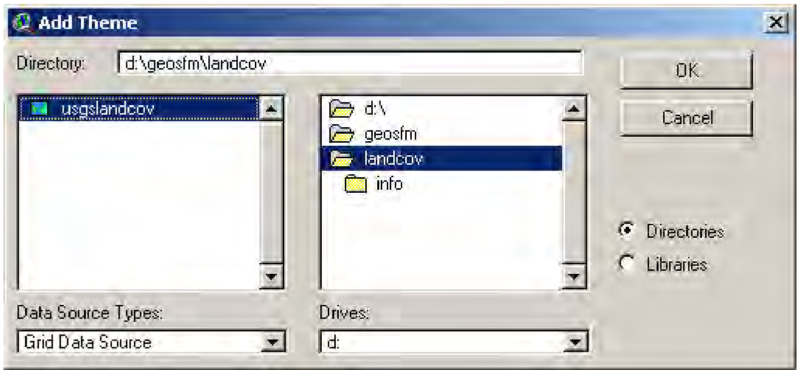
Click **Yes** when asked to **Add grid as theme to the View.**

The new **whc** grid is shown below.

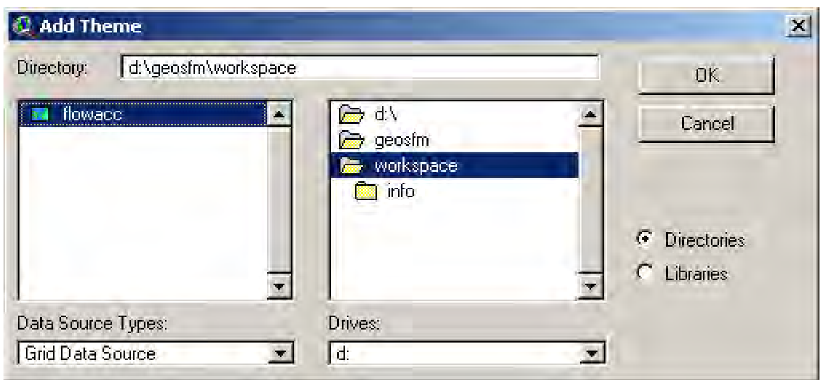


Maxcover

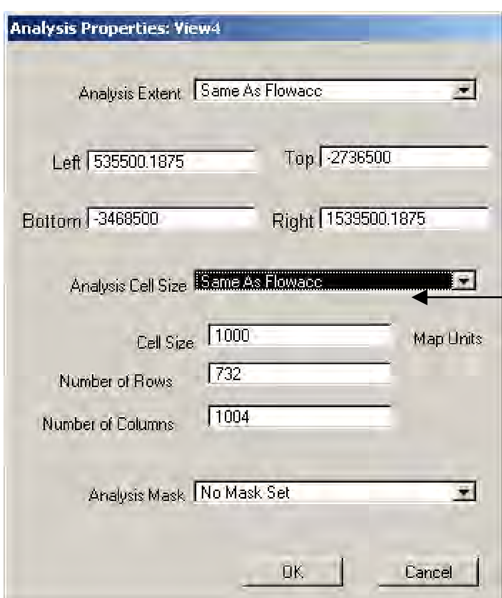
The next required soil characteristic grid is the maximum impervious area, maxcover. It is the area covered by a wetland or water body in the GLCC data or designated as a stream in the flow accumulation grid. Add the grid to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Navigate to the land cover directory and select **usgslandcov**. Click **OK** to add to the View**.**



Next, add the flow accumulation grid to the View. Add the grid to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Navigate to the workspace directory and select **flowacc**. Click **OK** to add to the View**.**



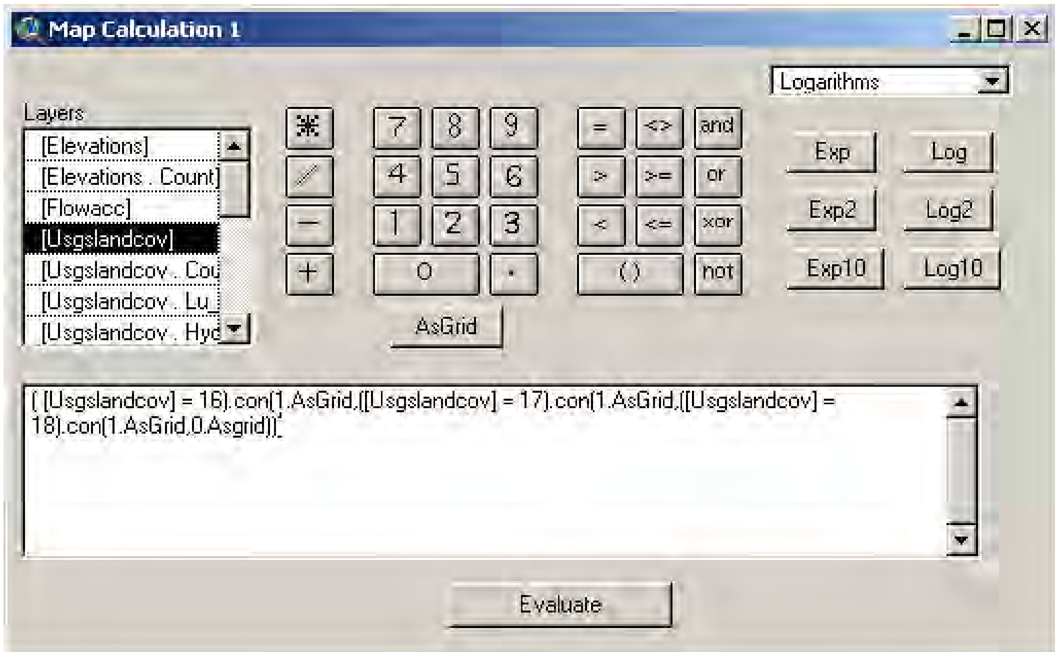
Next, define the analysis extent.



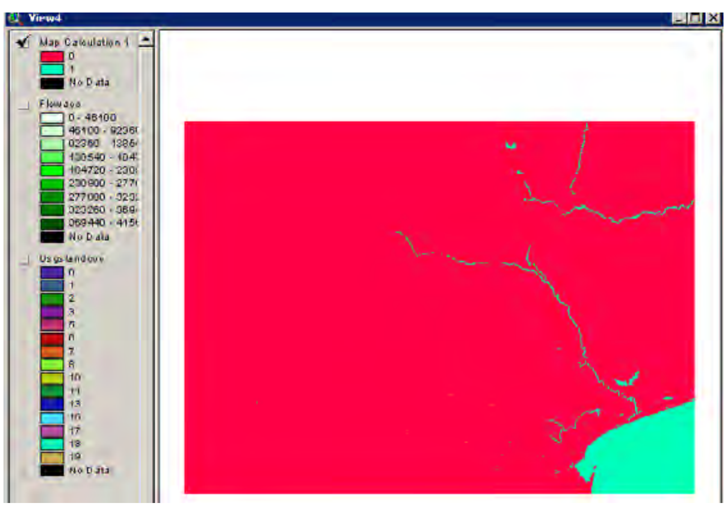
Populate:
Analysis Extent:
Same As Flowacc
Analysis Cell Size:
Same As Flowacc

Click **OK**. Next, select all water bodies and wetland areas from the land cover grid including values 16 (Water Bodies), 17 (Herbaceous Wetland), and 18 (Wooded Wetland). From the **Analysis** menu, select the **Map Calculator**. Type in the following expression exactly as shown below and click the **Evaluate** button:

([Usgslandcov] = 16).Con(1.AsGrid, ([Usgslandcov] = 17).Con(1.AsGrid, ([Usgslandcov] = 18).Con(1.AsGrid, 0.AsGrid)))



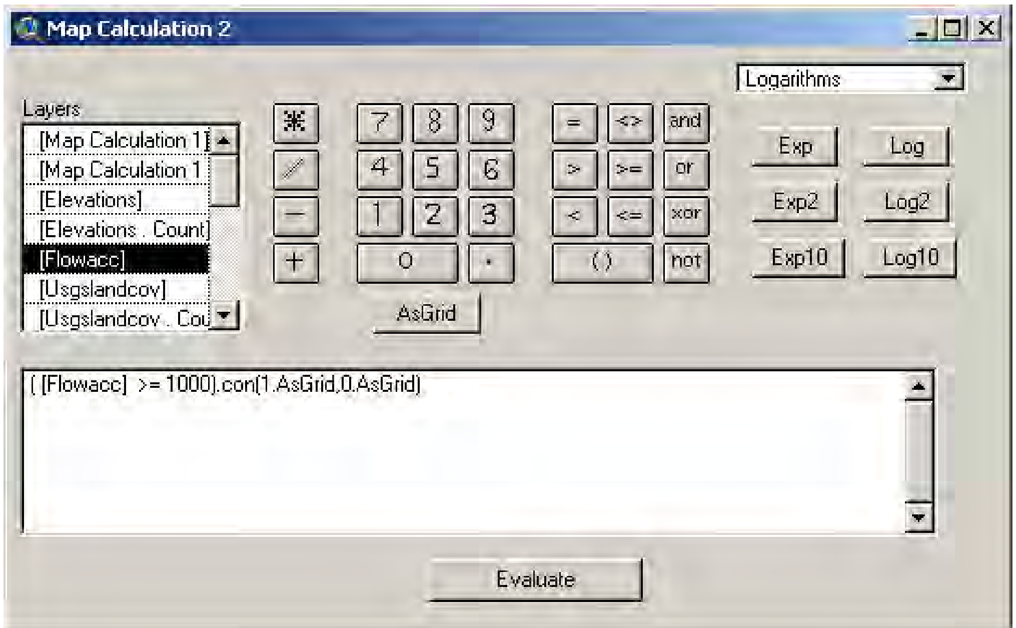
This creates **Map Calculation 1** with all water body and wetland cells assigned a value of 1; all other cells are assigned a value of 0.



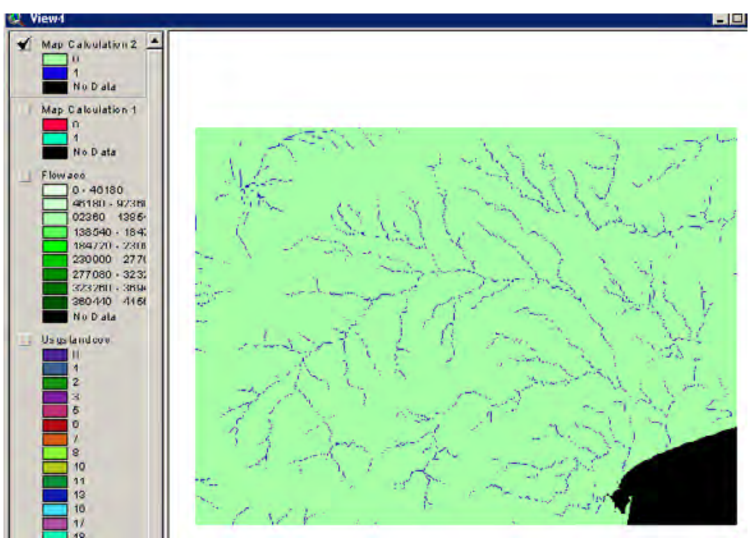
Next, select all cells with a value equal to or greater than 1,000 in the flow accumulation grid. These cells are defined as streams in the flow accumulation grid. From the **Analysis** menu, select the **Map Calculator**. Type in the following expression exactly as shown:

([Flowacc] >= 1000).Con(1.AsGrid, 0.AsGrid)

Click **Evaluate**.



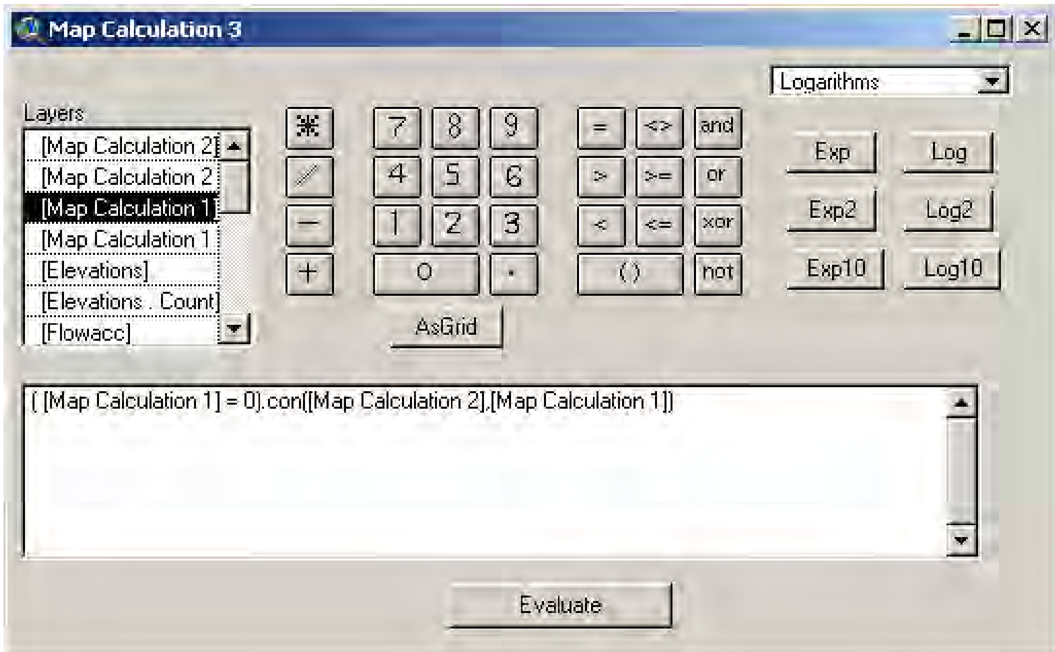
This creates **Map Calculation 2** with all stream cells assigned a value of 1; all other cells are assigned a value of 0.



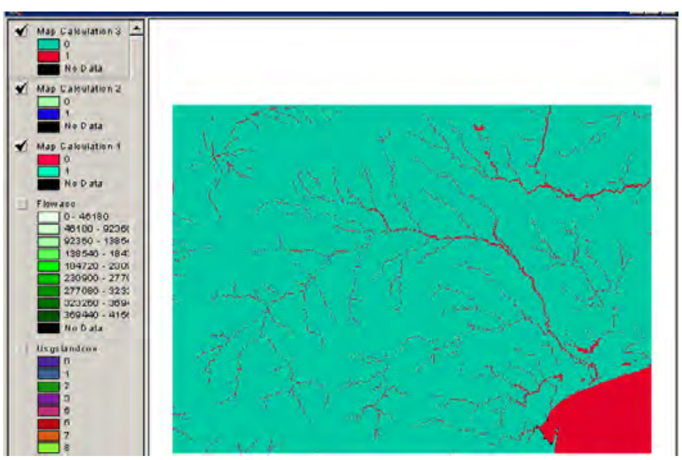
The next calculation combines the water body, wetland, and stream cells into one grid. From the **Analysis** menu, select the **Map Calculator**. Type in the following expression exactly as shown:

([Map Calculation 1] = 0).Con([Map Calculation 2], [Map Calculation 1])

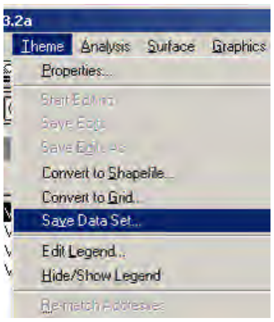
Click **Evaluate**.



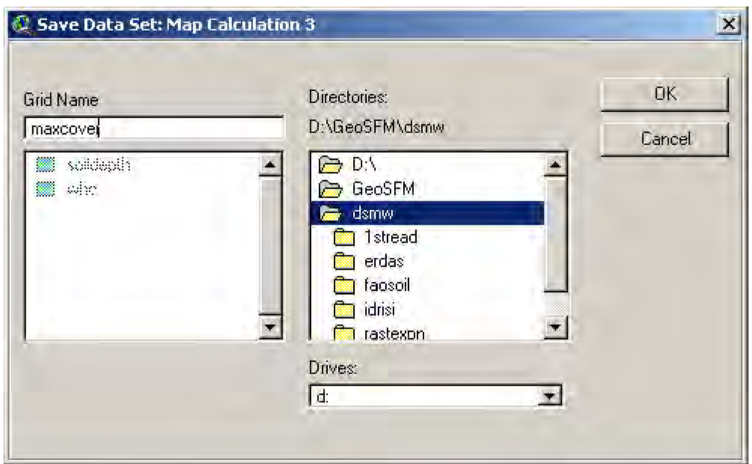
You have created a grid with all water bodies, wetlands, and streams assigned a cell value of 1. All other cell values are assigned a value of 0.



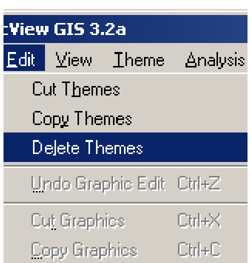
Save the new grid; from the **Theme** menu, select **Save Data Set**.



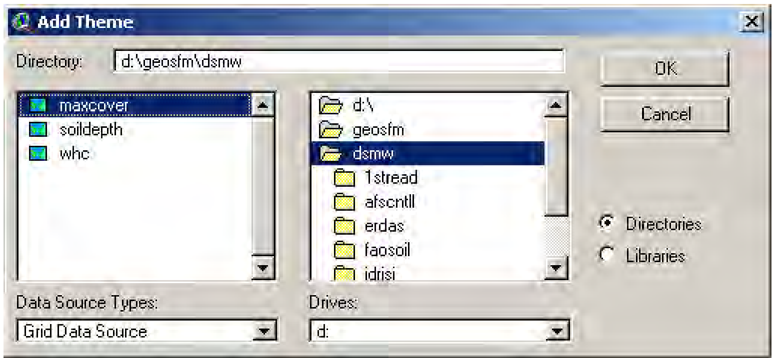
Navigate to your directory; name your grid **maxcover** in **Grid Name**. Click **OK**.



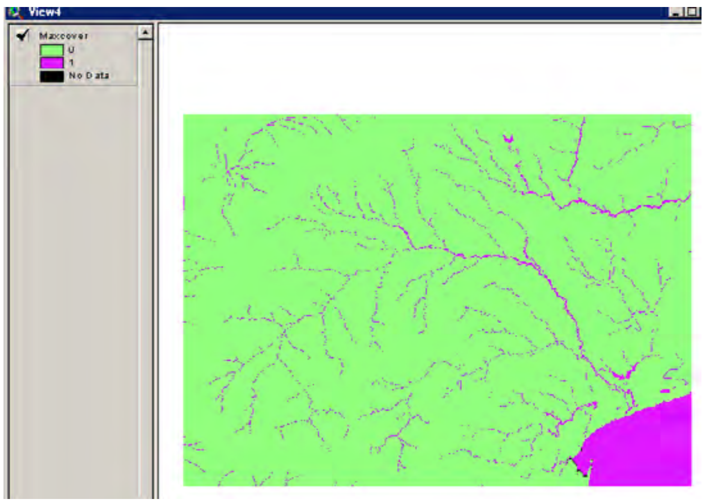
Delete the themes you no longer need from the table of contents. From the **Edit** menu, select **Delete Themes**.



Add the new **maxcover** grid to the View. Add the grid to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Navigate to your **dsmw** directory and select **maxcover**. Click **OK** to add to the View.

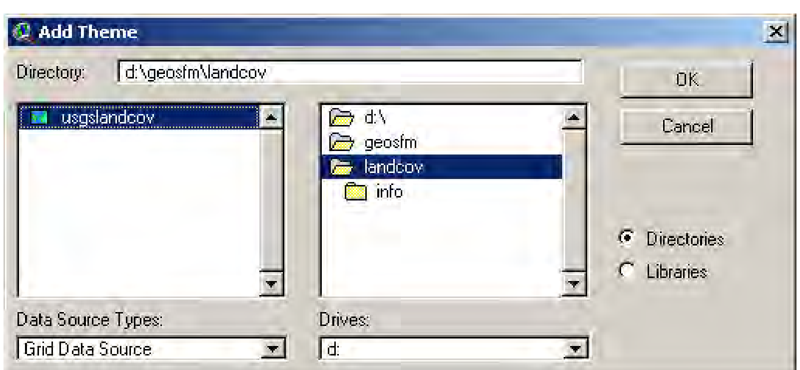


Below is the final **maxcover** grid.

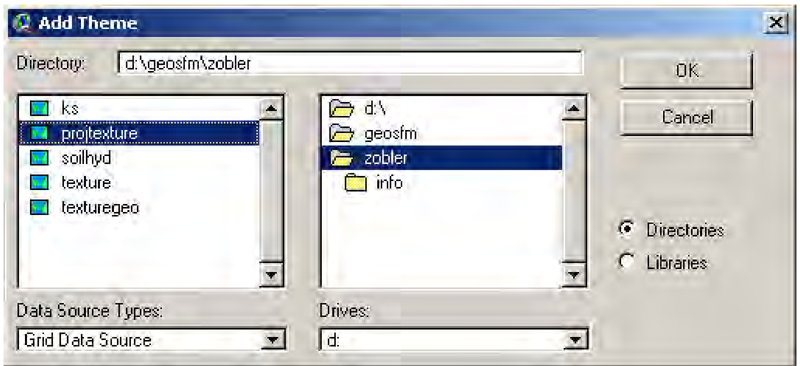


RCN

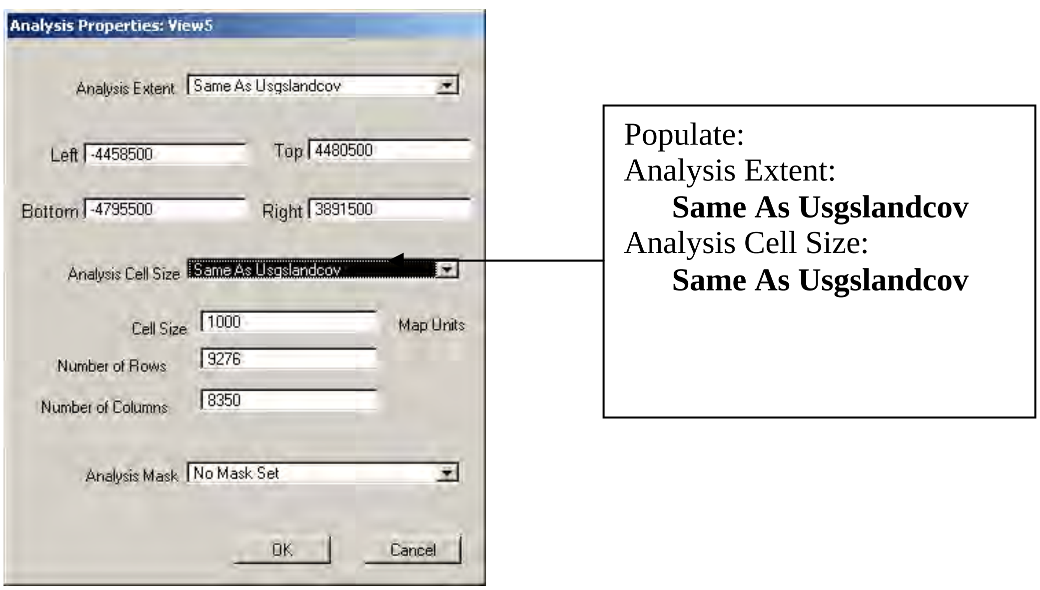
The last required soil characteristic grid is the Soil Conservation Service (SCS) runoff curve number (RCN) grid. SCS is the former name of the U.S. Department of Agriculture Natural Resources Conservation Service (NRCS). The runoff curve numbers are estimated from land cover classes and hydraulic soil groups. First, add the land cover grid. Add the grid to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Navigate to the land cover directory and select **usgslandcov**. Click **OK** to add to the View.



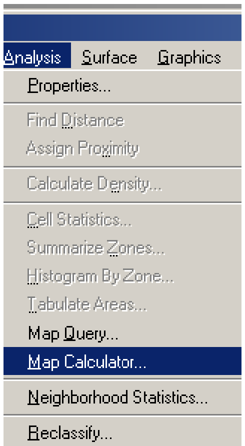
Then add the **protexture** grid. Add the grid to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Navigate to your directory and select **projtexture**. Click **OK** to add to the View**.**



After you have added the landcover and projected texture grids to the View, set the **Analysis Extent**. From the **Analysis** menu, select **Properties**. The **Analysis Properties** dialog box will be displayed.



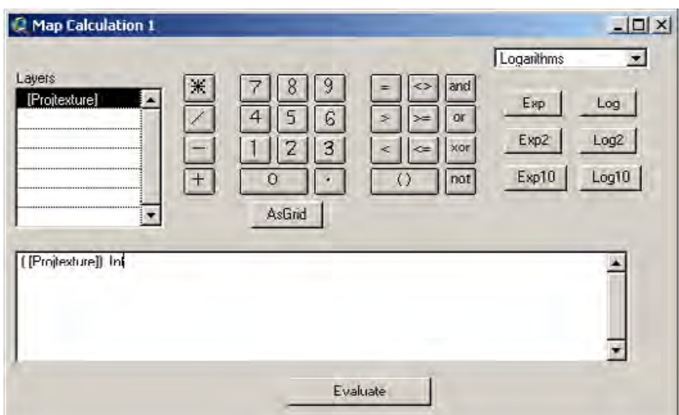
To begin this process you need to work with integers. From the **Analysis** menu, select the **Map Calculator**.



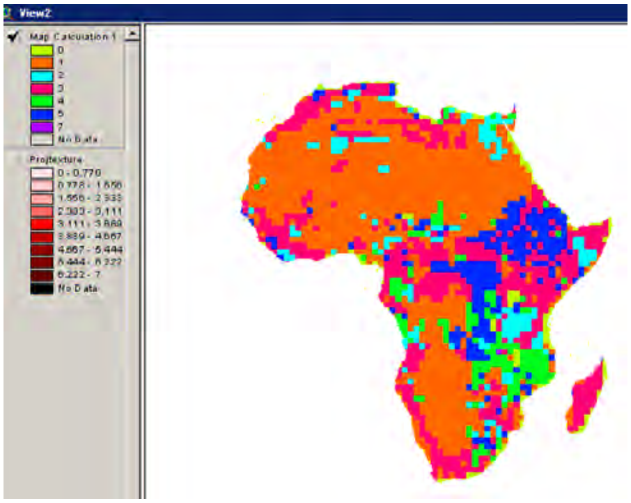
This opens the **Map Calculator** window. Double-click **projtexture** to populate the evaluation window. Type in the following expression exactly as shown:

([Projtexture]) .Int

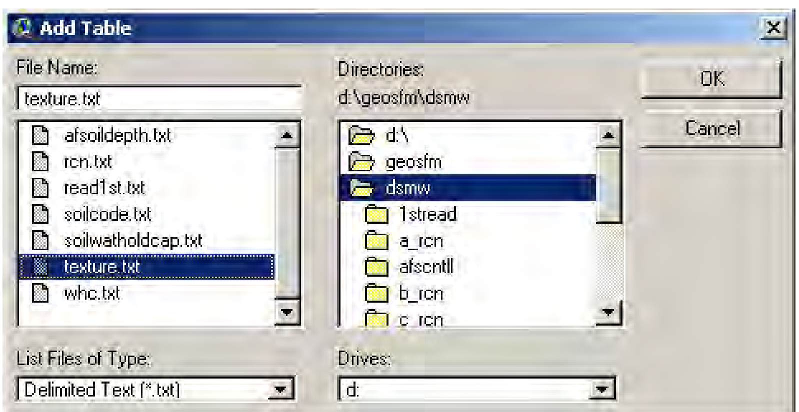
Click **Evaluate**.



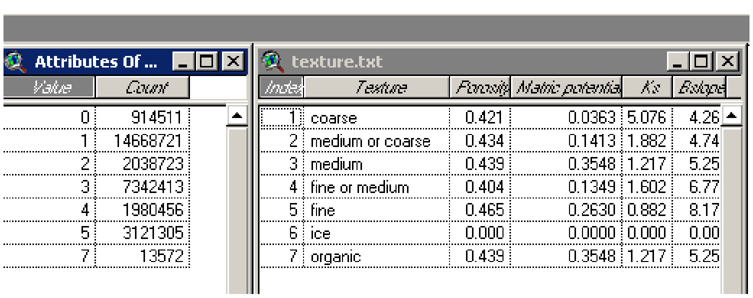
This produces the **Map Calculation 1** grid with values 0–7.



Next, create a soil hydraulic class grid. A text file, "texture.txt," containing the texture classes is required. This file was created earlier in this tutorial. Add the new **texture.txt** to the View. From the project window, select **Tables** and then click the **Add** button. Navigate to your working directory and select **texture.txt**. **List Files of Type** is set to **Delimited Text (*.txt)**. Click **OK**.



Join the **texture.txt** to the attribute table of **Map Calculation 1**. In the View click the **Map Calculation 1** theme so it appears in a raised box, then click the **Open** theme **table** icon. Arrange tables side by side. Highlight the common column headers in both tables—**Value** in **Attributes of Map Calculation 1** and **Index** in the **texture.txt** file. Make the destination table active (adding fields to **Attributes of Map Calculation 1**).



Attributes Of ...

Value	Count
0	914511
1	14668721
2	2038723
3	7342413
4	1980456
5	3121305
7	13572

texture.txt

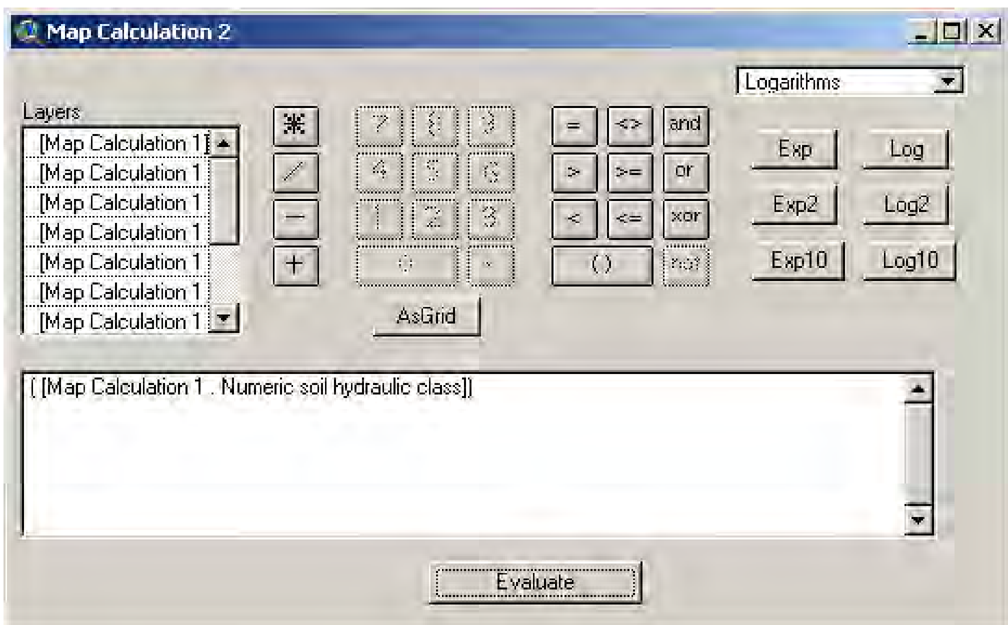
Index	Texture	Porosity	Matric potential	Ks	Bslope
1	coarse	0.421	0.0363	5.076	4.26
2	medium or coarse	0.434	0.1413	1.882	4.74
3	medium	0.439	0.3548	1.217	5.25
4	fine or medium	0.404	0.1349	1.602	6.77
5	fine	0.465	0.2630	0.882	8.17
6	ice	0.000	0.0000	0.000	0.00
7	organic	0.439	0.3548	1.217	5.25

Click the **Join** icon. The new columns are added to the **Attributes of Map Calculation 1** table.

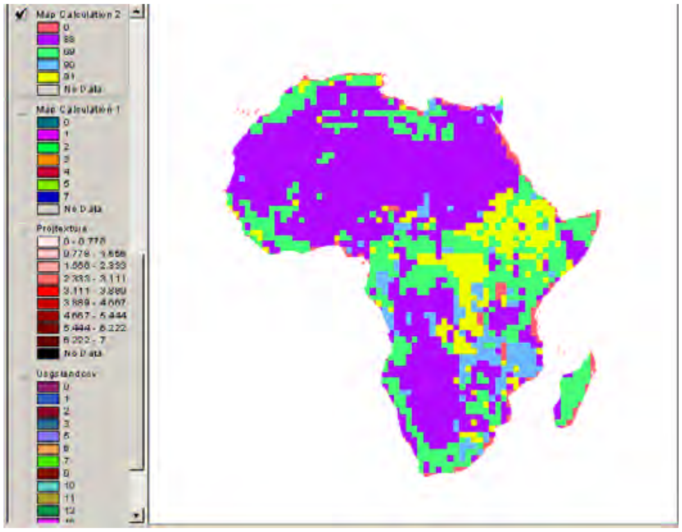
Now you can create a grid of the **Numeric soil hydraulic classes**. From the **Analysis** menu, select the **Map Calculator**. Type in the following expression exactly as shown:

([Map Calculation 1 . Numeric soil hydraulic class])

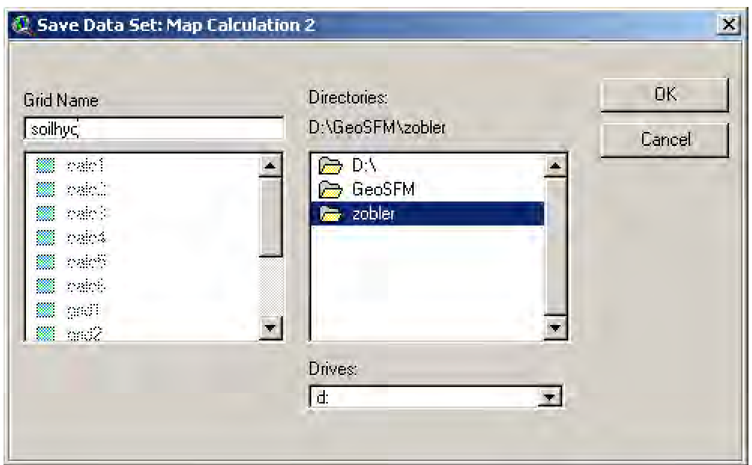
Click **Evaluate**.



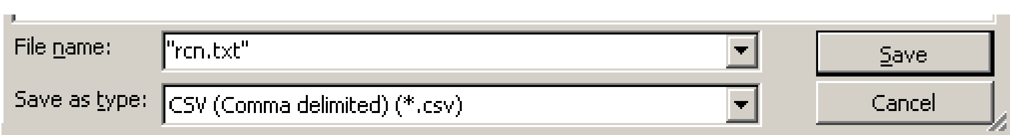
The created grid is displayed below with values of 0, 88, 89, 90, and 91.



Save this data set; from the **Theme** menu, select **Save Data Set**. Navigate to your directory and name your new grid—**soilhyd**. Click **OK**.



Now you can delete all themes except for the **usgslandcover** grid; from the **Edit** menu, select **Delete Themes**. Make sure the themes you want deleted are selected in a raised box. Now add the new **soilhyd** grid to the View. Create four grids for each Soil Hydraulic class (A, B, C, and D) with the RCN values for the landcover classes. The **Hyd_a_mean** field in the **usgslandcov** attributes table indicates the RCN values for soil hydraulic class "**A**," **Hyd_b_mean** for soil hydraulic class "**B**," **Hyd_c_mean** for soil hydraulic class "**C**," and **Hyd_d_mean** for soil hydraulic class "**D**." If the attribute table does not contain these **Hyd_a–d_mean** values, one has been created for you in the **GeoSFM/Samples** directory with the file name "rcn.txt."



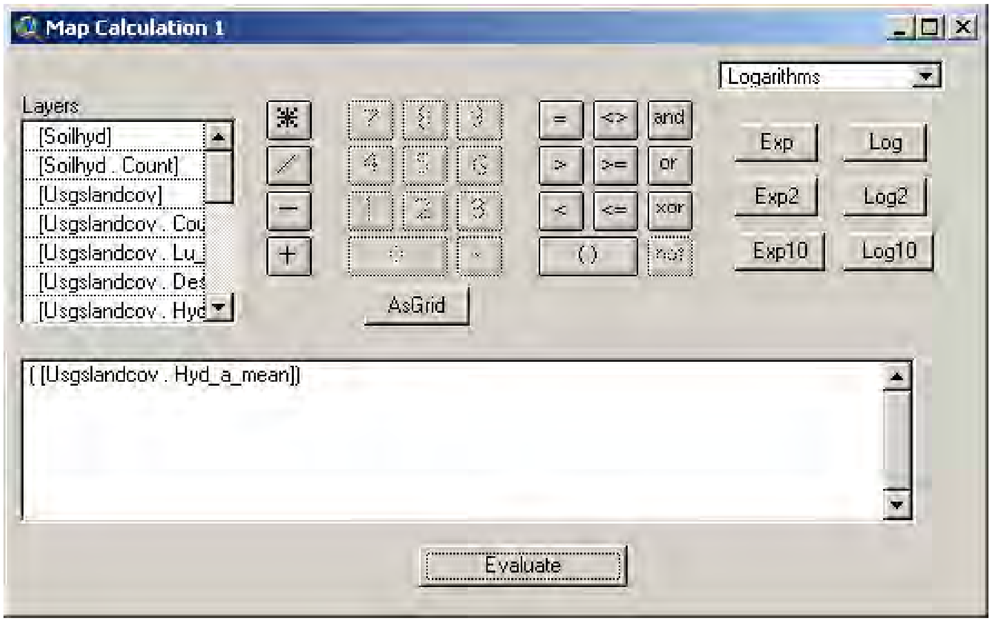
Add the new **rcn.txt** to the View. From the project window, select **Tables** and then click the **Add** button. Navigate to your working directory and select **rcn.txt**. **List Files of Type** is set to **Delimited Text (*.txt)**. Click **OK.**

Arrange the tables side by side. Highlight the common column headers in both tables—**Value** in **Attibutes of Usgslandcov** and **nlucode** in the **rcn.txt** file. Make the destination table active (adding fields to **Attributes of Usgslandcov**). Click the **Join** icon; the new columns are added to the **Attributes of Usgslandcov** table.

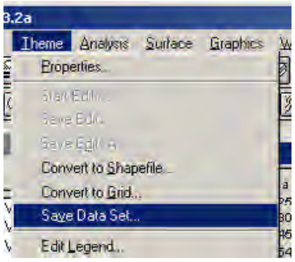
Next, create four separate grids for each of the soil classes. The first grid will capture all the **Hyd_a_mean** values. From the **Analysis** menu, select the **Map Calculator**. Type in the following expression exactly as shown:

([Usgslandcov.Hyd_a_mean])

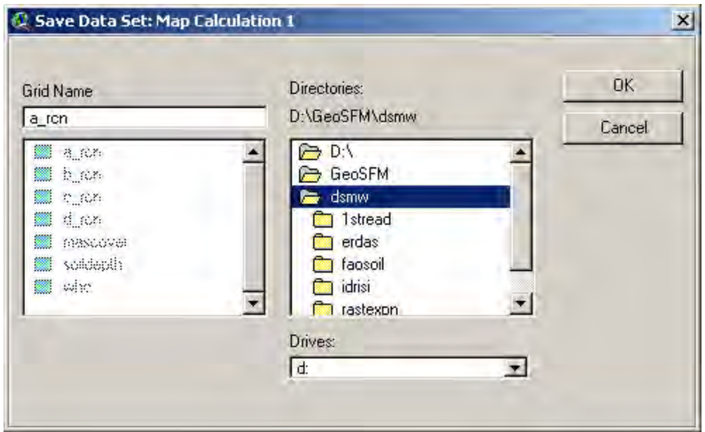
You can double-click the field under **Layers** to populate the evaluation area. Click **Evaluate**.



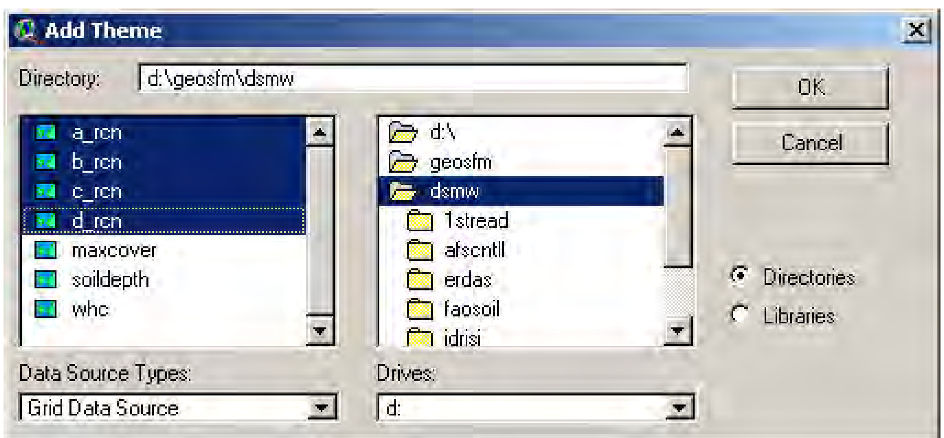
Save your new data set; from the **Theme** menu, select **Save Data Set**.



Navigate to your working directory and type **a_rcn** in the **Grid Name**. Click **OK**.

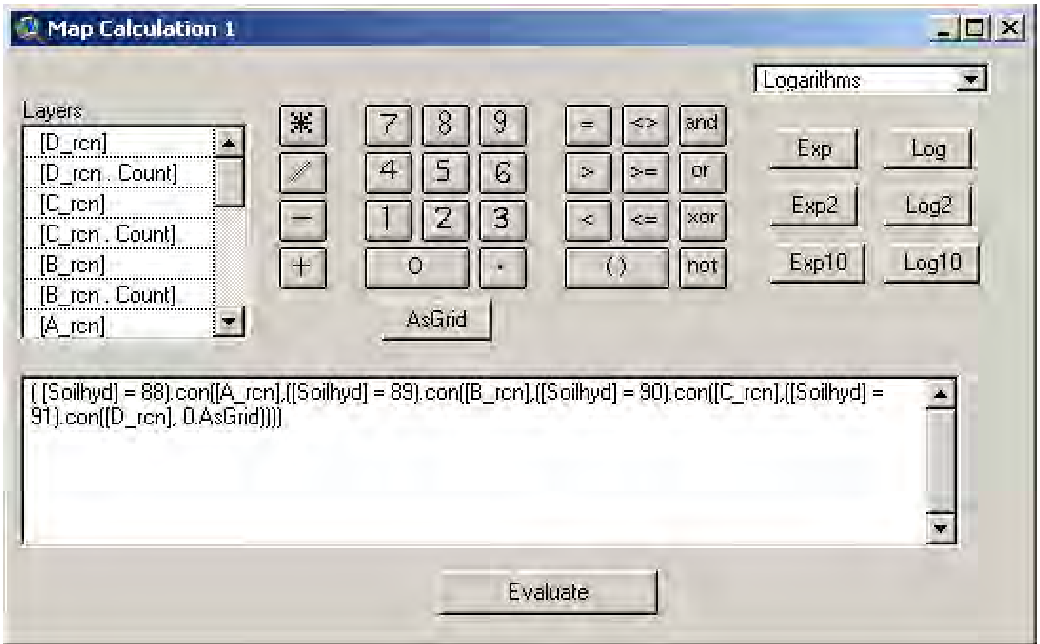


Repeat this process for the remaining three grids. The second grid will capture the **Hyd_b_mean** values and be saved as **b_rcn**. The third grid will capture the **Hyd_c_mean** values and be saved as **c_rcn**. The last grid will capture the **Hyd_d_mean** values and be saved as **d_rcn**. Delete all the **Map Calculation 1–4** grids, and add the newly created **a_rcn, b_rcn, c_rcn,** and **d_rcn** grids. You can select more than one grid by holding down the shift key while selecting. Add the grids to the View using the **Add Theme** button  from the **View** menu. Change the **Data Source Types** to **Grid Data Source**. Navigate to your directory and select the four grids. Click **OK** to add to the View**.**

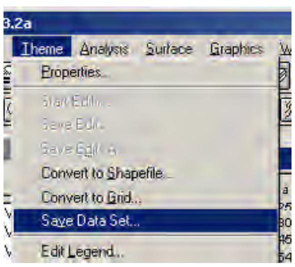


The next step is to combine the grids into one final rcn grid. From the **Analysis** menu, select the **Map Calculator**. Type in the following expression exactly as shown below and click **Evaluate**.

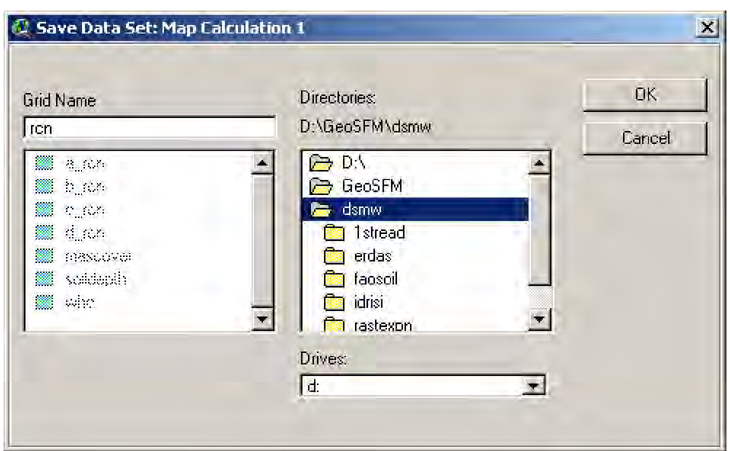
([Soilhyd]=88).con([A_rcn],([Soilhyd]=89).con([B_rcn],([Soilhyd]=90).con([C_rcn],([Soilhyd]= 91).con([D_rcn], 0.AsGrid))))



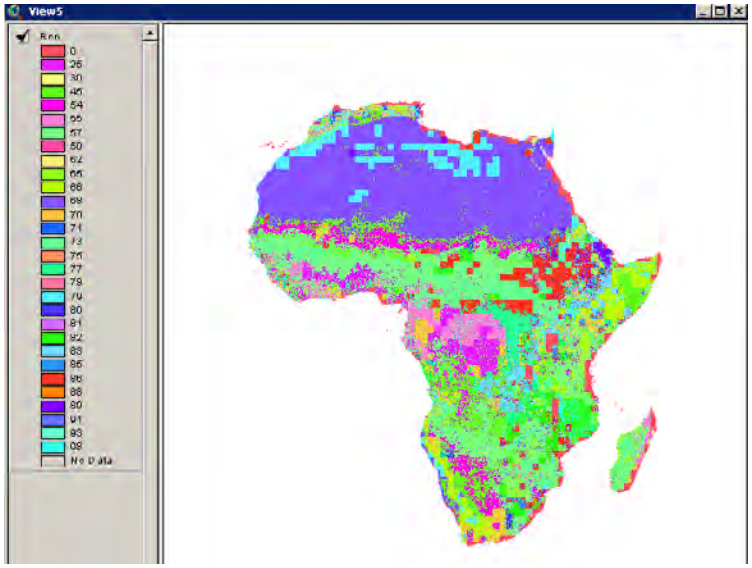
Save your new data set, from the **Theme** menu, select **Save Data Set**.



Navigate to your working directory and type **rcn** in the **Grid Name** box. Click **OK**.



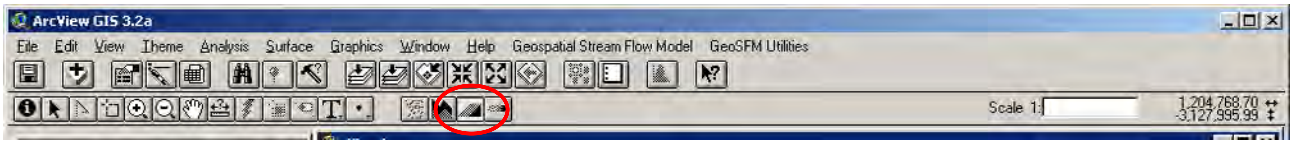
You can now delete all the themes in the View and add your new rcn grid as seen below.



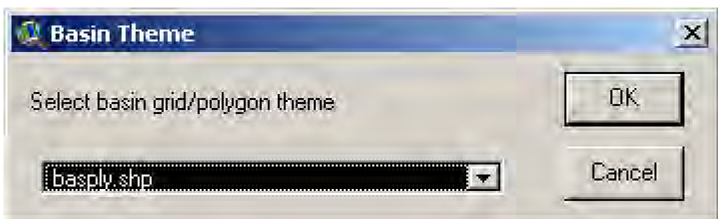
All of the soil characteristic grids have been created. Now the soil grids can be added to the View when prompted during the **Creating Basin Characteristics** process.

Appendix 5: Creating Reservoir and Gauge Rating Files

Reservoir characteristics files (reservoir.txt) are created whenever dams are present on a river or stream within the basin being analyzed. A dam insertion utility is included in GeoSFM for this purpose. On the View Toolbar, click the dam icon to select it.

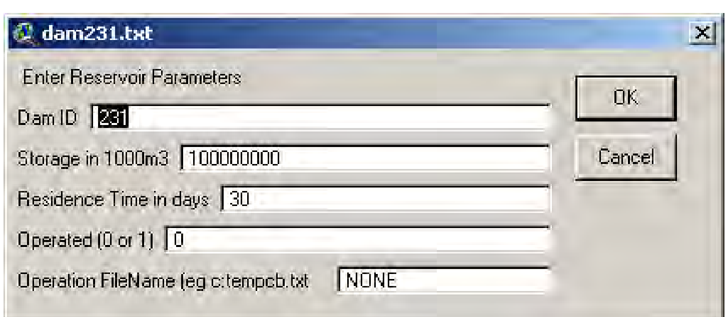


In the View window, position your mouse on the stream containing the dam. Left-click your mouse and a **Dams** shapefile will be added to your project. In the first dialog box displayed, select **basply.shp** from the drop-down list. Click **OK**.

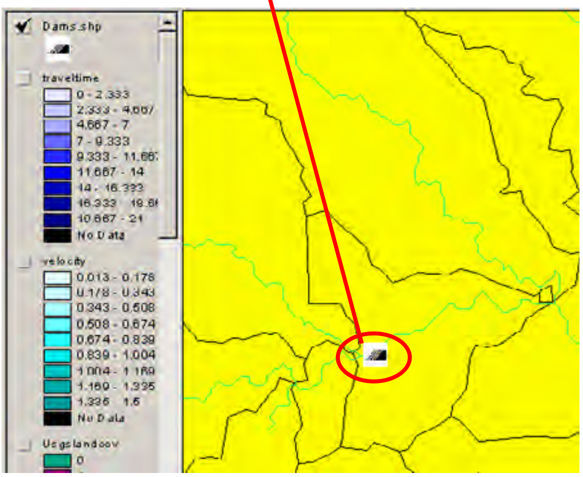


In the **Key Field** dialog box, select **Gridcode**.

Next, enter the reservoir parameters. The **Dam ID** will populate with the Basin ID of the selected point on the stream. The **Storage** value is needed along with the **Residence Time in days**. The **Operated** field is populated with either a 0 or 1 depending on a user-defined file. (0 if no file, 1 if there is a file.) If the file is needed, the file path and name will be populated in the **Operation FileName** field. Click **OK**.

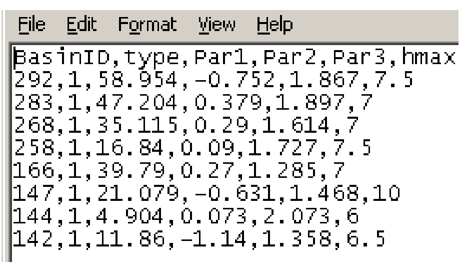


The dam is added to the View. This process will create a **reservoir.txt** file.



It may also be necessary to insert a stream gauge into the model for Muskingum Cunge computations. If you need to input streamflow data from stream gauges, the data will need to be in a format as in the file below. The first column header is BasinID, starting with the most downstream basin. The next column header is Type, which will be populated with either a 1 or 2 depending on the type of rating equation being used. The next three column headers are the three different equation parameters—Par1, Par2, and Par3. The last column is hmax, which is populated with the maximum flow height where the equation is valid. Create the file using Microsoft Excel spreadsheet software.

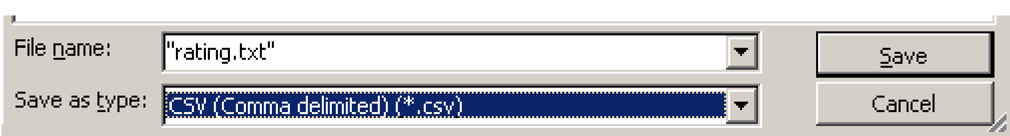
A sample rating file is shown below.



File Edit Format View Help

```
BasinID,type,Par1,Par2,Par3,hmax
292,1,58.954,-0.752,1.867,7.5
283,1,47.204,0.379,1.897,7
268,1,35.115,0.29,1.614,7
258,1,16.84,0.09,1.727,7.5
166,1,39.79,0.27,1.285,7
147,1,21.079,-0.631,1.468,10
144,1,4.904,0.073,2.073,6
142,1,11.86,-1.14,1.358,6.5
```

To save the file, select **File** and **Save As**.
File name: **“rating.txt”**
Save as type: **CSV (Comma delimited) (*.csv)**.



Enclosing the file name (**rating.txt**) in quotation marks ensures that the rating file will retain the specified file name when it is stored as a comma delimited file for subsequent use by the Muskingum Cunge program.