Geospatial Stream Flow Model (GeoSFM) Training Manual

Version 1.0

Training Center
U.S. Geological Survey
Center for Earth Resources
Observation and Science (EROS)
Sioux Falls, South Dakota, USA

Document accompanying *Geospatial Stream Flow Model Users Manual* and *Geospatial Stream Flow Model Technical Manual*

Revision History

Introduction

Beginning in 1999, scientists at the US Geological Survey's EROS Data Center began developing a streamflow model for monitoring hydrologic conditions over large areas. The activity was initiated with resources from the USAID sponsored Famine Early Warning System Network. While development of the model was progressing, a series of major cyclones hit the Mozambican Coast in Southern Africa in late January and February of 2000. The repeated waves of heavy rainfall, saturated soils and abnormally high reservoir levels combined to generate the flood of record in the lower reaches of the Limpopo basin. In the aftermath of the storm, the need for tool for monitoring streamflow over large areas became evident. The GeoSFM model was selected for the implementation of a flood warning system in the Limpopo Basin. This training manual contains a series of exercises that were developed from the Limpopo Basin application.

GIS software ArcView (version 3.2 or higher) is used a long with the installed spatial analyst extension. Other required data sets and programs are provided in the installation package on the CD-ROM or from the FTP site of the USGS EROS Data Center. While previous knowledge of geographic information systems (GIS) is beneficial, it is not a requirement for using the model and this training guide. There are eight exercises; four are focusing on different aspects of setting up and using GeoSFM for monitoring and forecasting streamflow in a large basin at a daily time step and four are focusing on additional tools that maybe required to complete the modelling. Exercise one introduces the user to GeoSFM and its use in setting up a model of a basin. Exercise two introduces the user to the processing of meteorological data and performing stream flow analysis. In exercise three, the user generates flow statistics and flow hydrographs. The user is provided guidance on how to calibrate the model in exercise four. Exercises five through eight give instruction on how to use the various GeoSFM utility tools.

Many individual have contributed to the development of the GeoSFM model. Dr. James Verdin, the International Project Manager at the EDC first recognized the need for a wide area hydrologic model which uses available remotely sensed data for

parameterization. His persistence in pursuing the resources necessary to get this work under way is exemplary. Dr. Guleid Artan led the team of hydrologists who developed GeoSFM, and he contributed many of the water balance and routing modules in GeoSFM. Dr. Kwabena Asante was responsible for developing the geospatial modules in GeoSFM and for integrating the various modules into a single model. He also led the first field implementation of the model in Mozambique. Dr. Hussein Gadain, Mr. Tamuka Madagzire, Mr. James Kiesler and Dr. Miguel Restrepo were responsible for extensively testing and documenting the model and for making suggestions for its improvement. Contributions by Sr. Rodriguez Dezanove, Sr. Agostinho Vilanculos and Sra. Monica Frederico made the Limpopo basin implementation possible. The difficult task of incorporating the various documents into a single coherent set of exercises was very ably performed by Ms. Jodie Smith and Ms. Debbie Entenman. This training manual would not have been completed without their contributions. Other International Program staff including Mr. James Rowland, the USGS FEWS Net Team Leader, Mr. Ronald Lietzow and Mr. Ronald Smith who process and manage the input data, Dr. Saud Amer and Mrs. Theresa Rhodes who provide technical and administrative support, and the staff of FEWS Net in Mozambique who supported us during various phases of this effort. Our gratitude goes to all of them for their important contributions. Last but not least, the contributions of USAID who provide the funds, other FEWS Net partners including NOAA who process the meteorological data, NASA who process the land cover data and Chemonics international who support the work of our field scientists are much appreciated. We hope the Training Manual will be useful to you in your work.

USGS/FEWS Net Team
EROS Data Center
Sioux Falls, SD 57198
November, 2003

Contents

Geospatial Stream Flow Model

demdata

elevations

raindata

rain_1999
rain_19991
↓
rain_1999365

evapdat

evap_1999
evap_19991
↓
evap_1999111

landcov

usgslandcov

soildata

ks
maxcover
rcn
soildepth
texture
whc

program

geosfm.avx v1.0
geosfm.dll
geosfmcalib.exe
geosfmzip.exe
geosfmpost.exe
geosfmstats.dll
geosfmtar.exe
install.bat

presentation

documentation

GeoSFM Training Manual
GeoSFM Users Manual
GeoSFM Technical Manual

samples

Grids
basins
downstream
elevations
flowacc
flowdir
flowlen
outlets
hilllength
slope
streams
strlinks
traveltime
velocity
Sinks
dem

Shapefiles
basply1.shp
rivlin1.shp
limpbas.shp
gauges2.shp

Project
project.apr

Text files
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

Text files
localflow.txt
logfileflow.txt
logfilesoil.txt
massbalance.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

Above is a list of all data files contained on the GeoSFM CD. The *GeoSFM Training Manual* is accompanied by the *GeoSFM Users Manual* and the *GeoSFM Technical Manual*.

Training Manual for the Geospatial Stream Flow Model
Ex 1: Introduction to the GeoSFM

Contents:

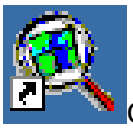
Data and Computer Requirements

1. ArcView version 3.x with the Spatial Analyst Extension installed
2. GeoSFM extension (geosfm.avx and geosfm.dll)
3. Access to the internet or a GeoSFM CD-ROM with input datasets

1.1 Model Installation

To install this version, download or copy from CD all files to your C drive. The 3 files (INSTALL.bat, geosfm.avx, geosfm.dll) are the actual Geospatial Stream Flow Model. The other files will be needed to complete the exercises. In **c:\GeoSFM\Programs** double-click the **INSTALL.bat** file and installation is complete. This will copy all geoSFM files and register the .dll files to the local computer. Create a new directory, **c:\GeoSFM\workspace**, for the ArcView files you will be creating.

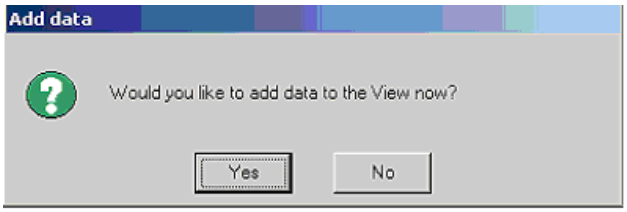
1.2 Opening Project and Loading Extensions



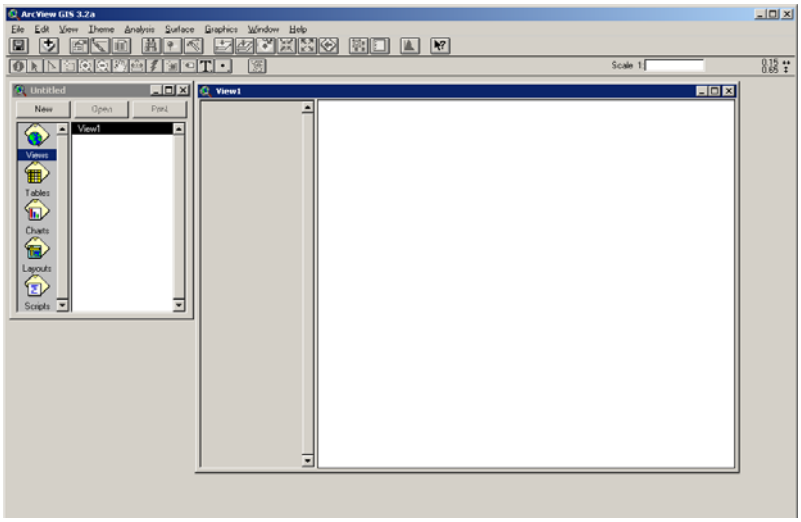
Open ArcView GIS by clicking the shortcut on your desktop or by selecting it from your **Programs** menu. When ArcView opens, the Welcome to ArcView GIS dialog box is displayed. Depending on the setup configuration there are different ways in which to create a new project.

If the above dialog box is displayed - Click the **Create a new project** –**with a new View** radio button, and then click **OK**.

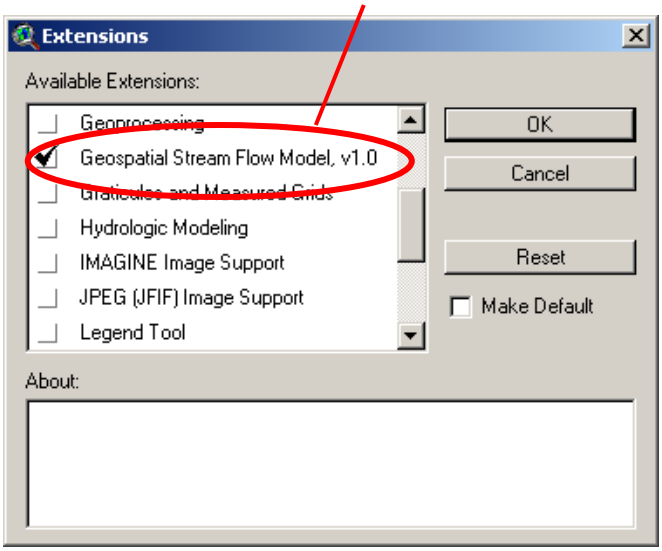
The **Add data** dialog box appears asking if you would like to add data to the **View** now, click **No**.



If the dialog box is not displayed – as below – Click on the **Views** icon and then click on the **New** button in the untitled **Project** window. This will open the **View 1** window, click and drag the bottom right corner to expand view and then position next to the untitled **Project** window.

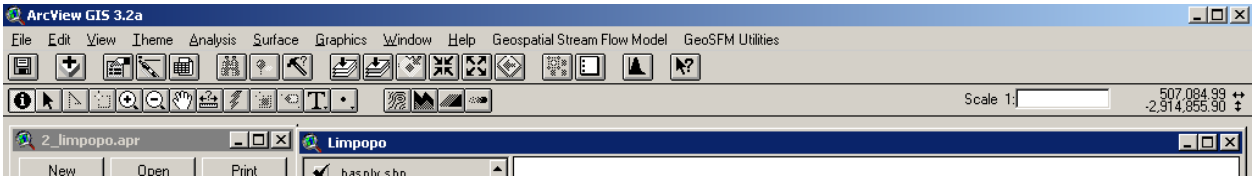


Next, from the **File** menu, select **Extensions….** to load **Geospatial Stream Flow Model** and the **Spatial Analyst**.

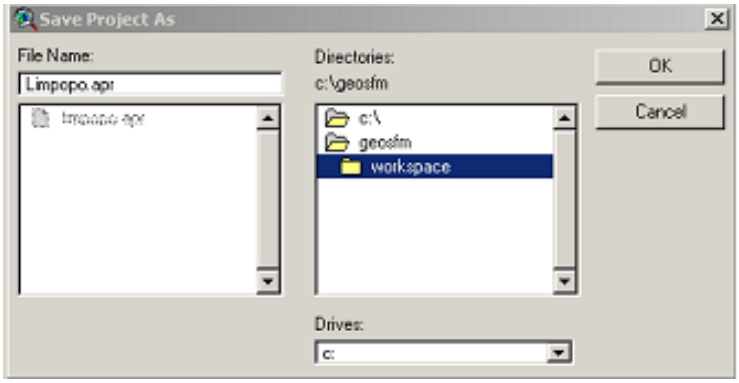


Check the boxes next to the **Geospatial Stream Flow Model** and **Spatial Analyst** to load the extensions to the project, and click **OK**.

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

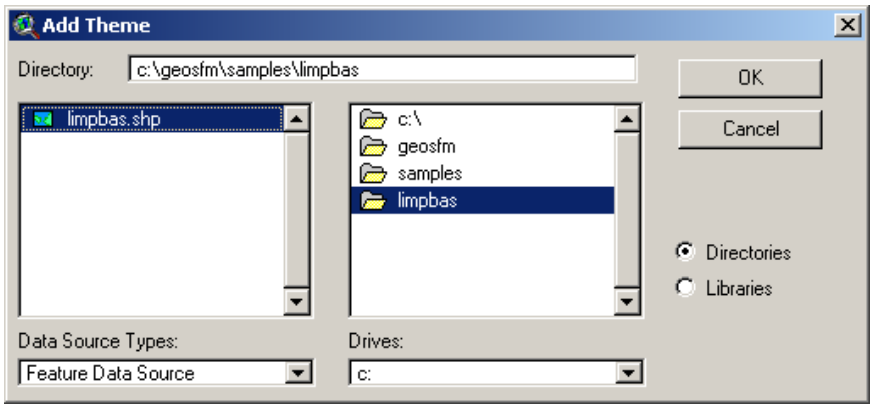


Begin by selecting **File – Save Project As…**from the top menu. Save your project to your workspace **c:\GeoSFM\workspace** with the file name **Limpopo.apr**. The extensions will then be preloaded next time you open the project.



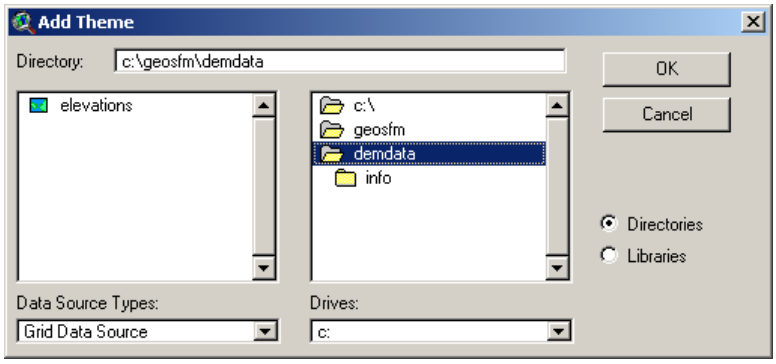
1.3 Processing Elevation Data

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**.

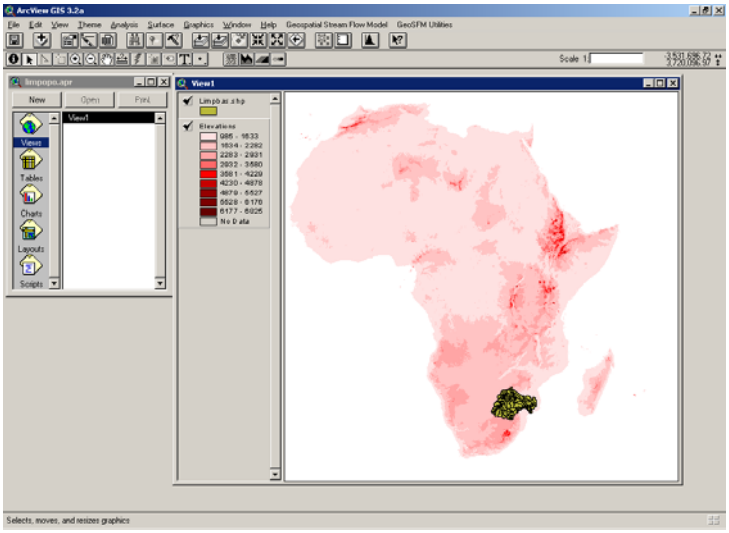


The Geospatial Stream Flow Model uses a digital elevation model for the delineation of hydrologic modeling units.

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

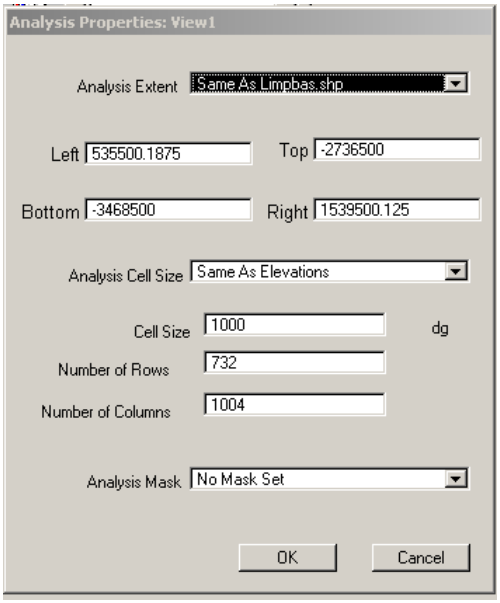


Click and drag the **Limpbas.shp** theme to the top of the table of contents and check the box so that it is visible over the **elevations** grid.



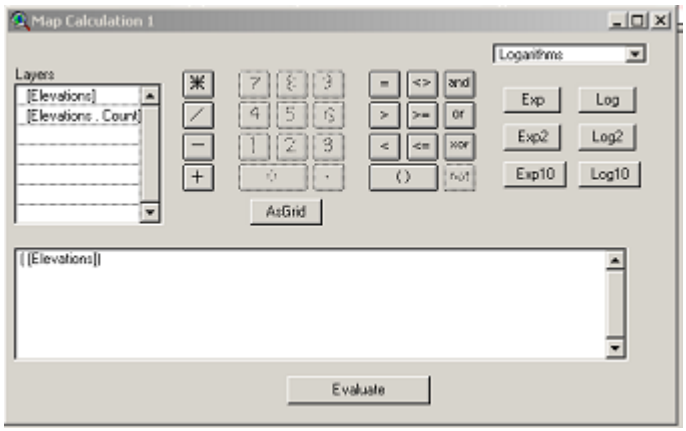
Next, set the analysis environment from the **Analysis** menu by selecting **Properties**. Change the **Analysis Extent** to **Same As Limpbas.shp** and the **Analysis Cell Size** to **Same as Elevations**. All other parameters will adjust themselves.

Click **OK.**



1.4 Performing Terrain Analysis

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**.

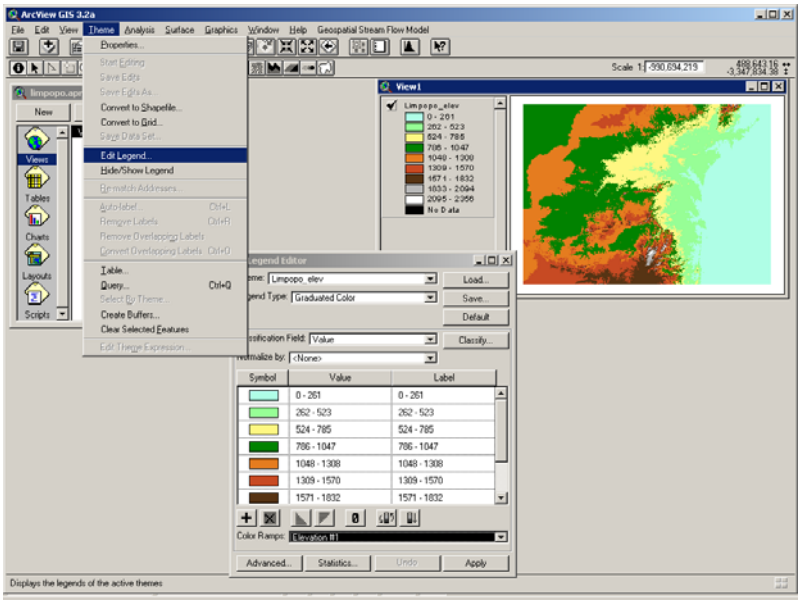


Select **Map Calculation 1** in the table of contents to display theme in a raised box, from the **Theme** menu select **Save Data Set…** In the **Save Data Set**: **Map Calculation 1** Dialog box navigate to the **c:\GeoSFM\workspace** directory and in **Grid Name,** name your new grid extent **Limpopo_elev**. Then 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 on **Limpopo_elev** to add to **View**.

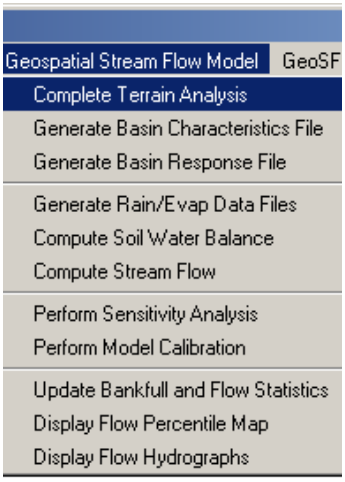
Next, remove all **Themes** except for the new **Limpopo_elev** theme. Select the **Theme** to be removed by clicking on **Theme**, which is now a raised box. In the **Edit menu** select **Delete Themes** to remove selected theme. Continue until all themes are removed except for the **Limpop_elev**. 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 wish to apply an elevations type legend 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**.

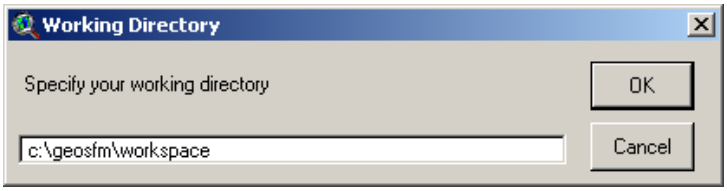


You are now ready to begin running the Geospatial Stream Flow Model!

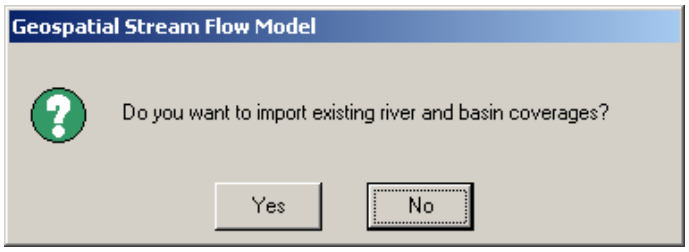
In the **Geospatial Stream Flow Model** select **Complete Terrain Analysis** from the drop down list.



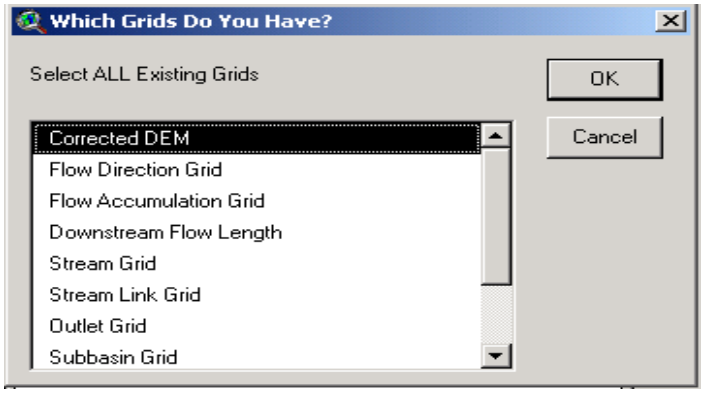
Confirm your working directory as **c:\GeoSFM\workspace** and click **OK**.



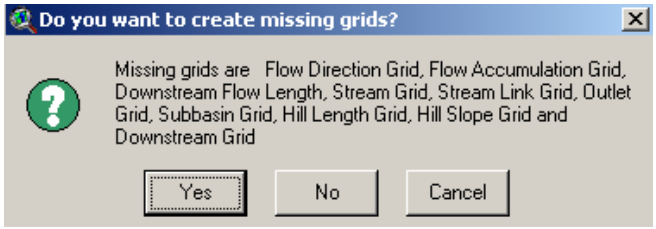
If you have existing river or basin coverages you may add them. For this exercise you will select **NO**.



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



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

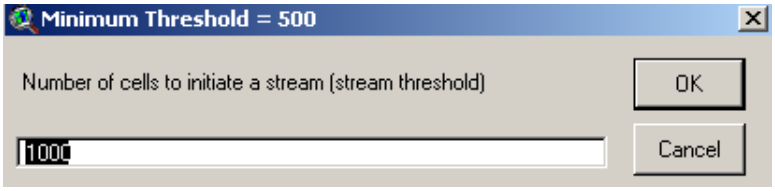


Confirm that the grid called **Limpopo_elev** is indeed the Corrected DEM.

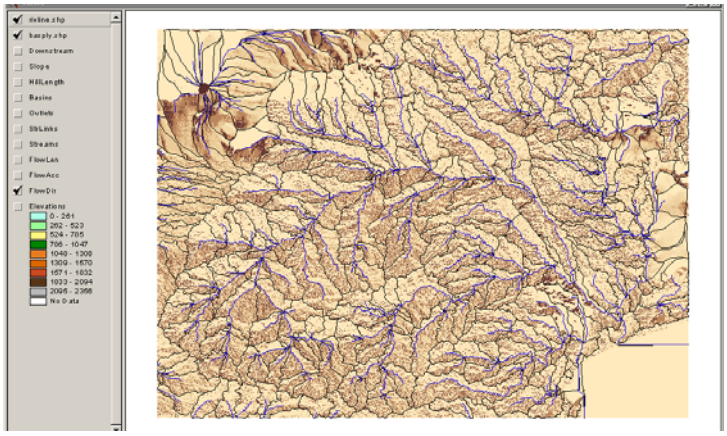
Click **OK.**

The program should begin performing the terrain analysis. After a while 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.

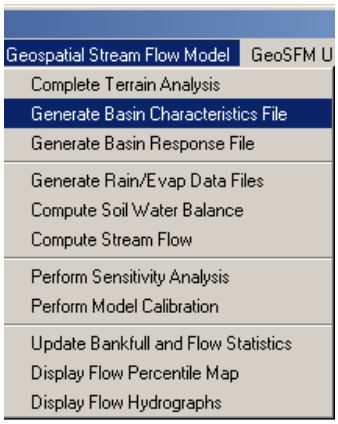


In a few minutes you should get a message telling you the terrain analysis is complete. Click **OK**. (**Limpopo.elev** theme is replaced with **Elevations** Theme during the processing.)



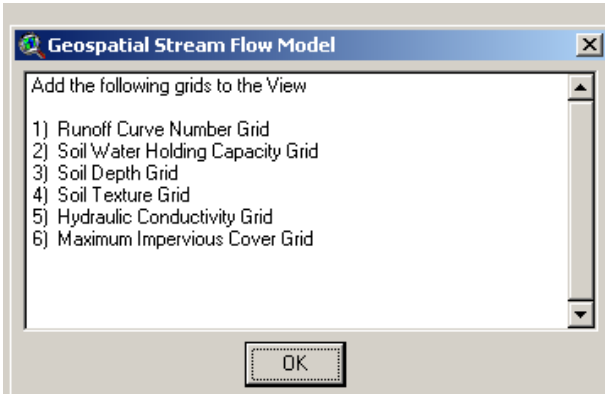
1.5 Creating a Basin Characteristics File

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

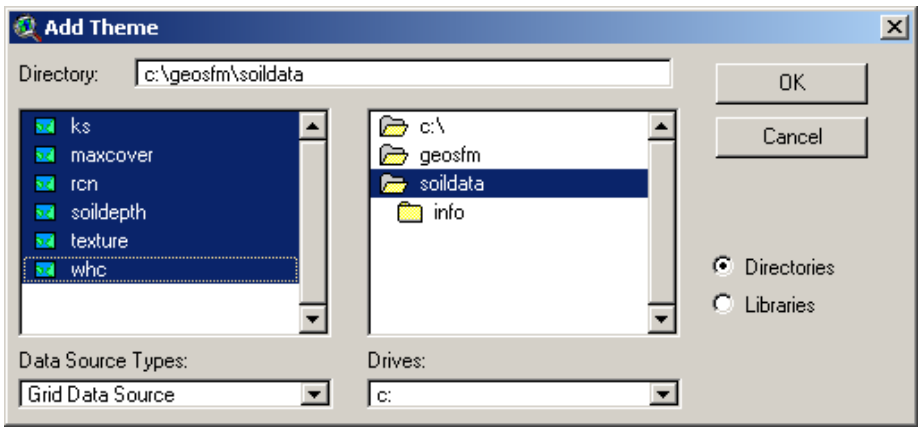


Confirm your working directory as **c:\GeoSFM\workspace.** Select **YES** when presented with the question "**Add Soils & LandCover Data to View?"**

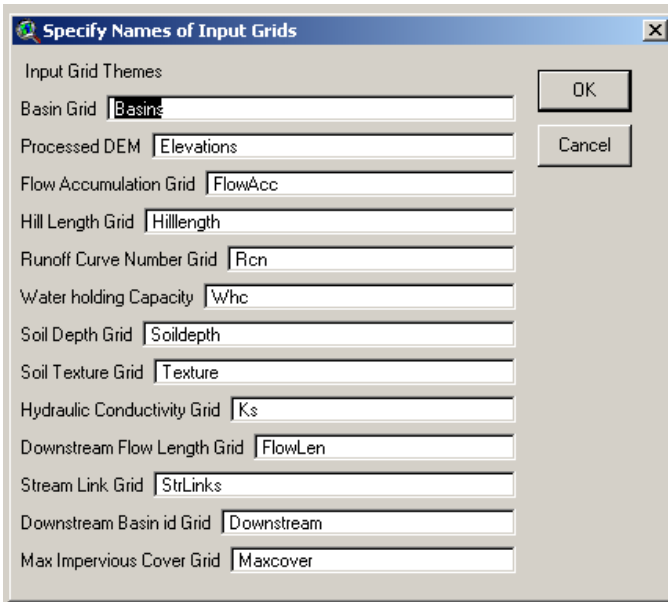
A list of the data sets you need should appear. Click **OK**



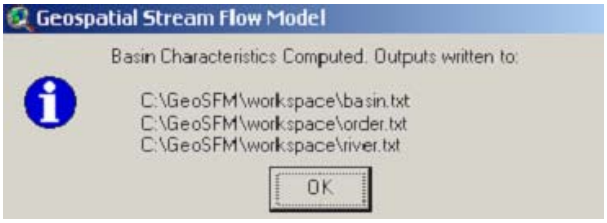
The required data sets are provided for you in the **c:\GeoSFM\soildata** directory. Change the **Data Source Types** to **Grid Data Source**. Hold the **shift key** down to select **all** the grids, (ks, maxcover, rcn, soildepth, texture, and whc.) Click **OK** to add them to the **View**.



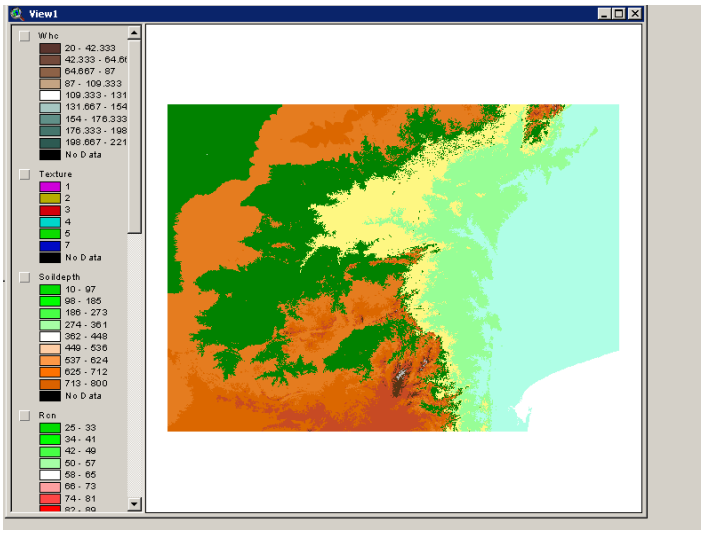
The program will present a new list of your input grids including all the input parameters.



The program will produce 2 files containing the characteristics of each sub-basin and river. It will also produce a file containing the computational order, which is required for subsequent program operations. When it is done processing, it will bring up a message indicating the name and location of the output files. Click **OK**

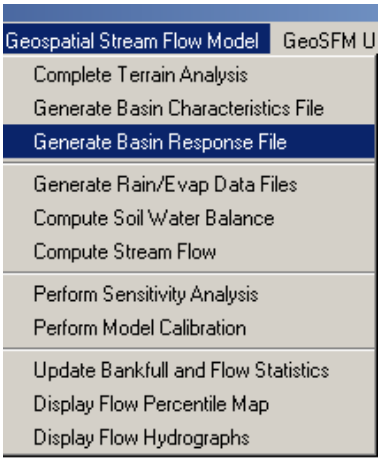


Below all the input grids are added to the table of contents.



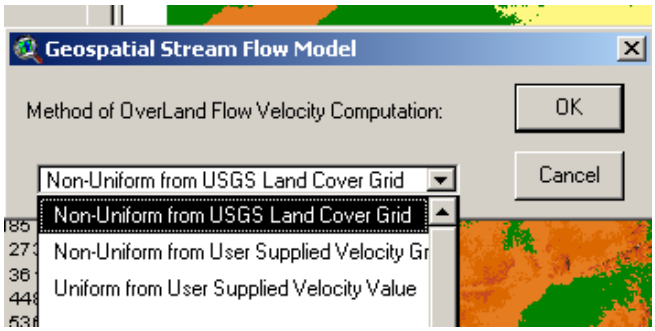
1.6 Creating a Basin Response File

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

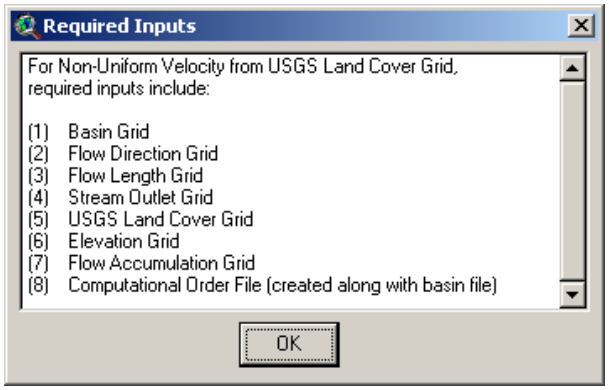


Confirm your working directory and click **OK**.

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

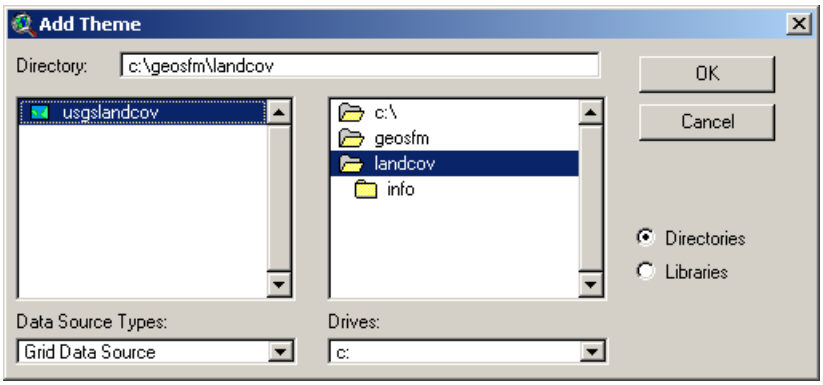


A list of the required inputs is displayed. Click **OK**.

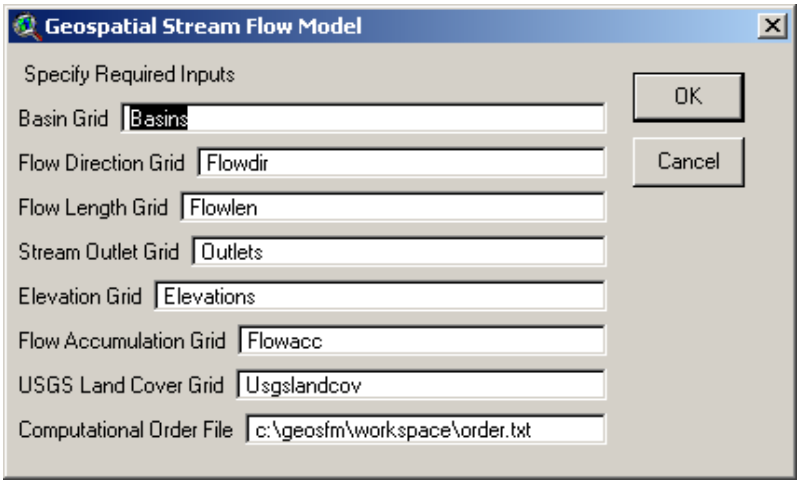


Click **Yes** when asked whether you want to add **USGS Land Cover grid** to the **View**.

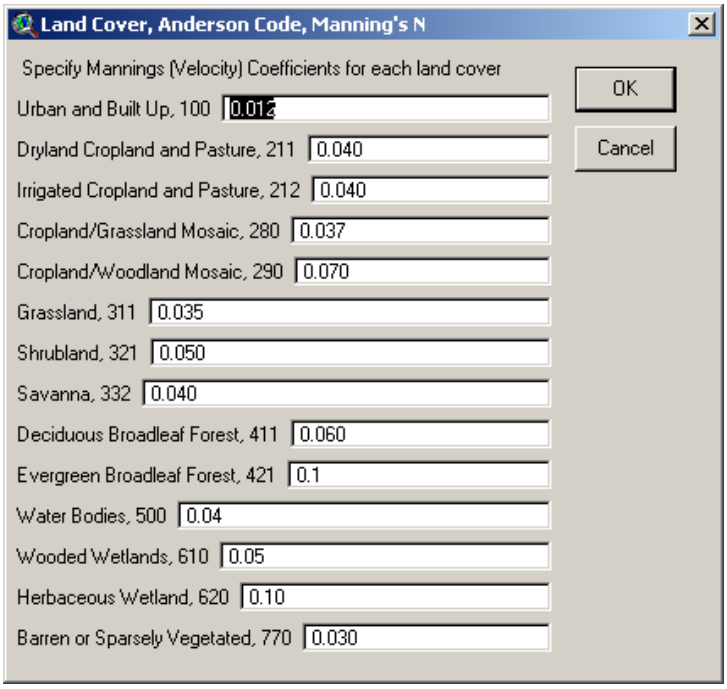
Change the **Data Source Types** to **Grid Data Source**. Select **usgslandcov** from **c:\GeoSFM\landcov** directory and click **OK** to add the land cover grid to **View**.



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

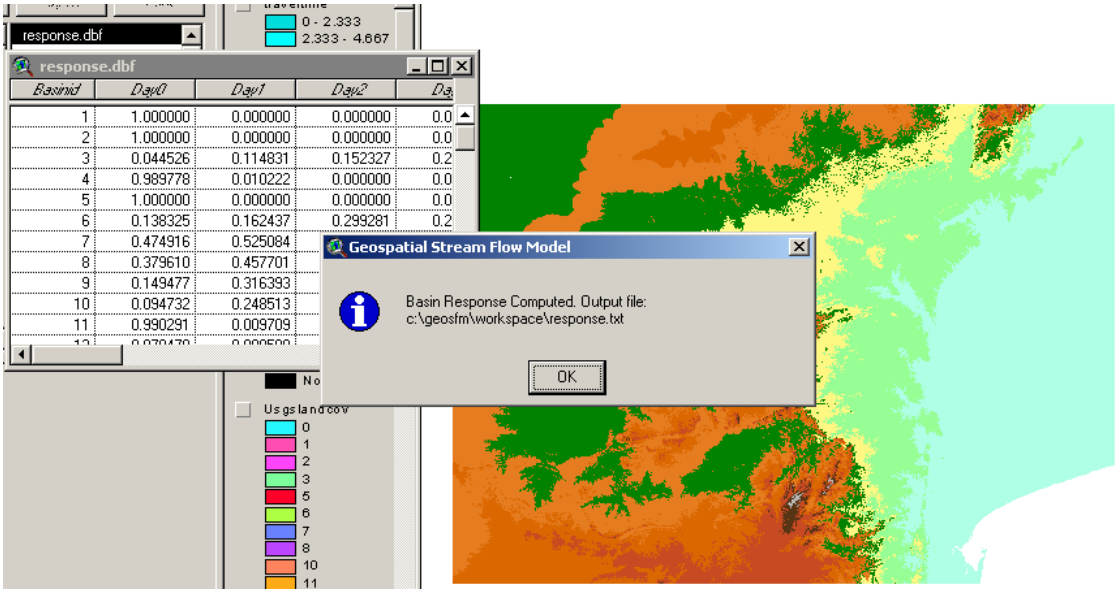


The **Land Cover, Anderson Code, Manning's N** should now be displayed. Use the default Manning's Coefficient values when **Specifying Manning's (Velocity) Coefficients for each land cover**. Click **OK**.



Need to update print screen with added codes

The program will compute a response file (similar to a unit hydrograph) for each sub-basin. 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, displayed is the response table. Click **OK** in the dialog box. You have now finished the terrain analysis; and have generated the basin characteristics file and the response file. Next, you will generate the rain and evapotranspiration files.

This has completed Exercise 1. Save your project. If continuing on to the next exercise you can leave the project open. If you wish to continue with the next exercise at a later time you can close the project now.

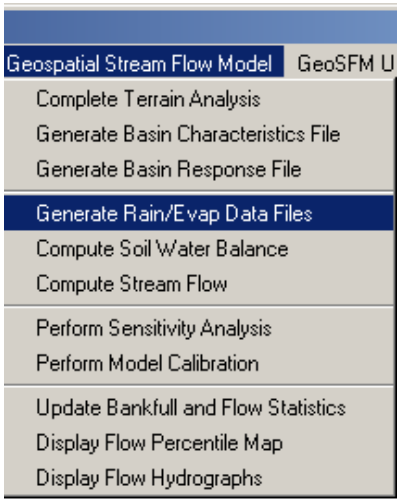
Training Manual for the Geospatial Stream Flow Model

Ex 2: Performing Stream Flow Analysis

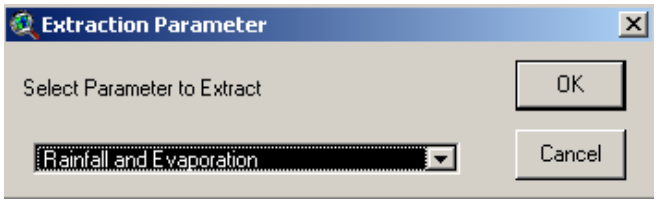
Contents:

2.1 Generating Rainfall and Evaporation files

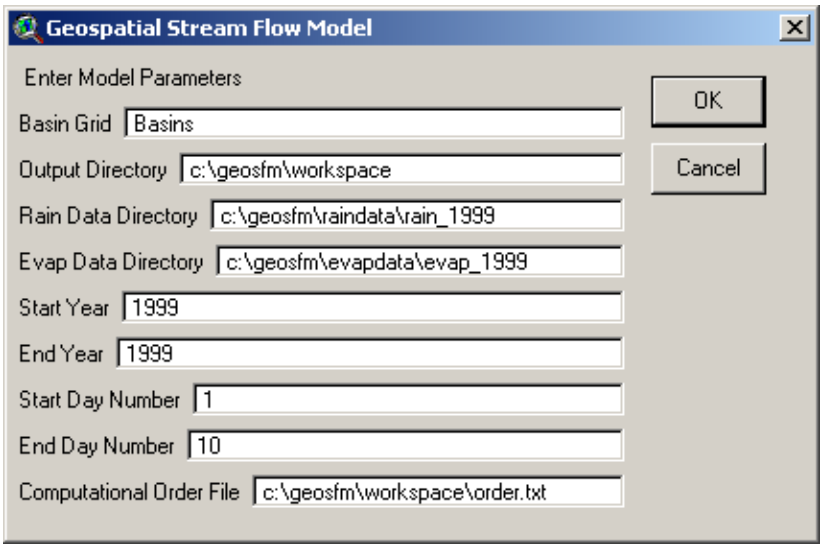
In this next step you will estimate a rainfall value for each sub-basin from the daily rainfall grids. To begin this computation, from the **Geospatial Stream Flow Model** menu select **Generate Rain/Evap Data Files**.



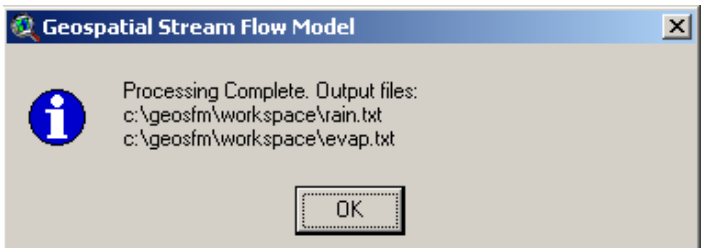
Select **Rainfall and Evaporation** when prompted to **Select Parameter to Extract**. Click **OK**.



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**.



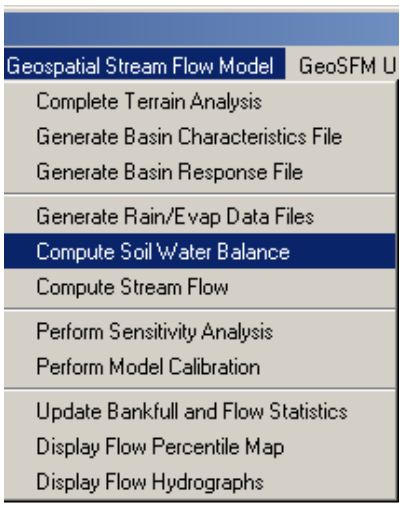
After a few moments a message box will appear indicating that the processing is complete. The message also gives the location of the output files: **rain.txt** and **evap.txt**. Click **OK**



The **rain.txt** file created from this process contains an average rainfall value, in millimeters, for each sub-basin per day. The **evap.txt** file contains a potential evapotranspiration (PET) value, in tenths of millimeters, for each sub-basin per day. The routing program will use these files when you calculate the soil water balance.

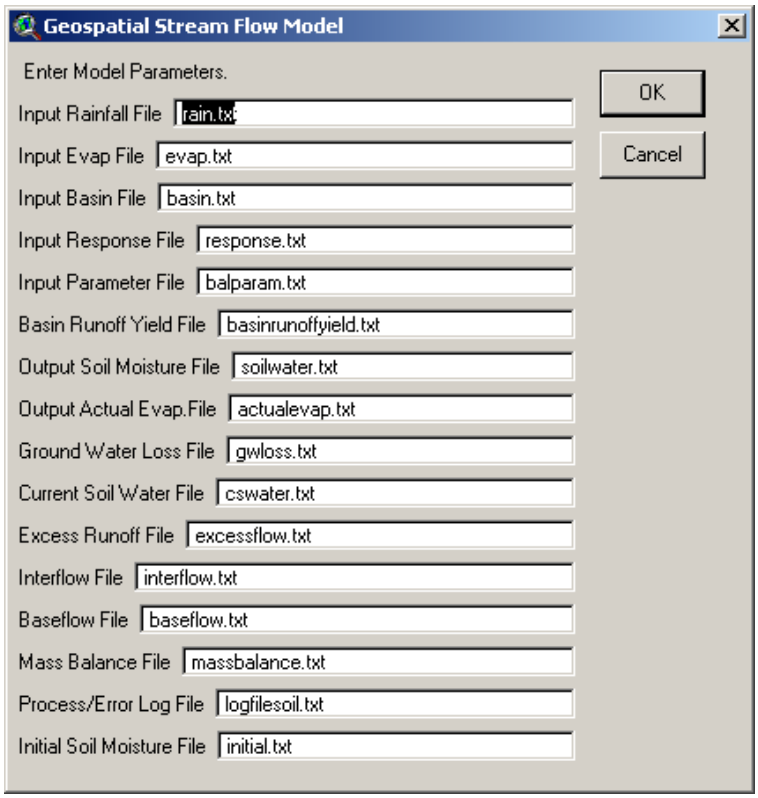
2.2 Computing Soil Water Balance

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

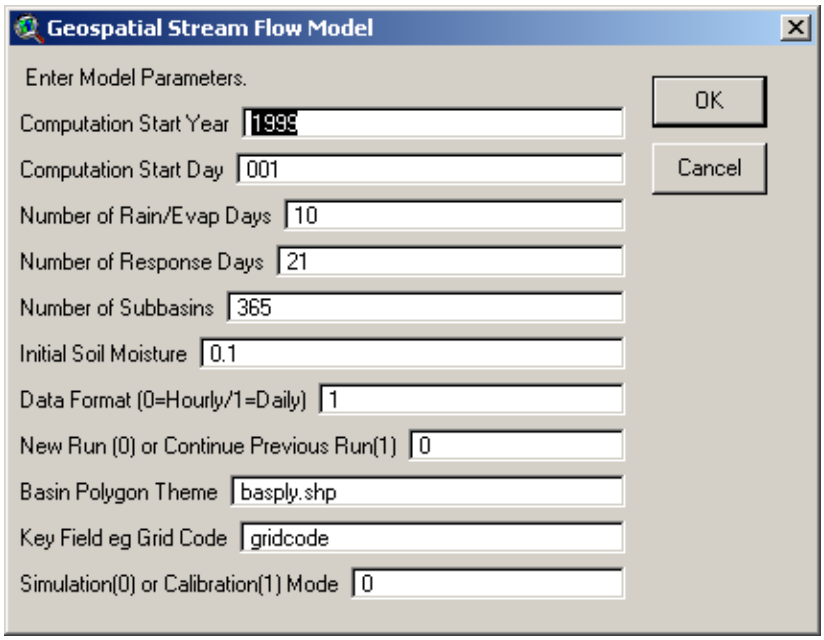


Confirm the working directory as c:\geosfm\workspace. Click **OK**.

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**.



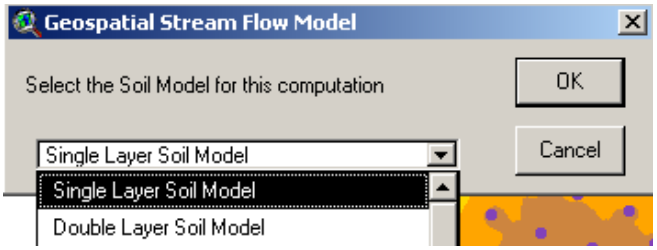
Next, **Enter Model Parameters** as shown below, use the default values. Click **OK**.



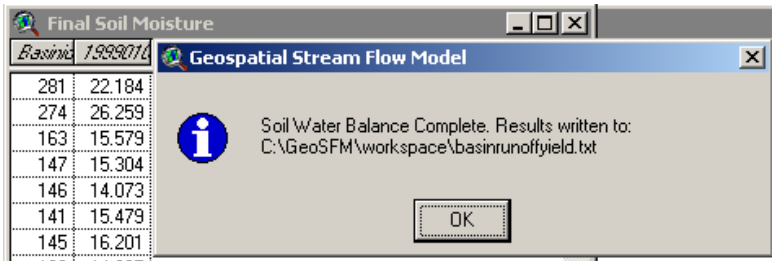
For short runs, 30 days or less, getting the initial soil moisture content correct greatly influences the accuracy of your calculated flows. For this run, you will assume that the

soil is initially dry, containing only 10% of its storage capacity. You report this as a fraction of 0.1. As seen above in the **Initial Soil Moisture** field.

Next, you will be asked to **Select the Soil Model for this computation**. The available choices include: **Single Layer Soil Model** and **Double Layer Soil Model**. For this exercise you will choose the **Single Layer Soil Model**. Click **OK**.

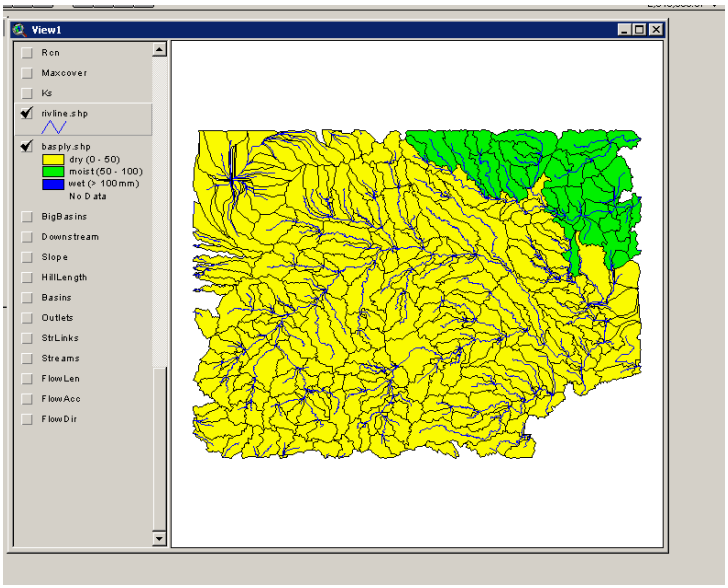


The model computes the soil water balance and indicates the location of the key output file containing the local contribution of each sub-basin to downstream river flow. The **Final Soil Moisture** table is also displayed. Click **OK**.



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.

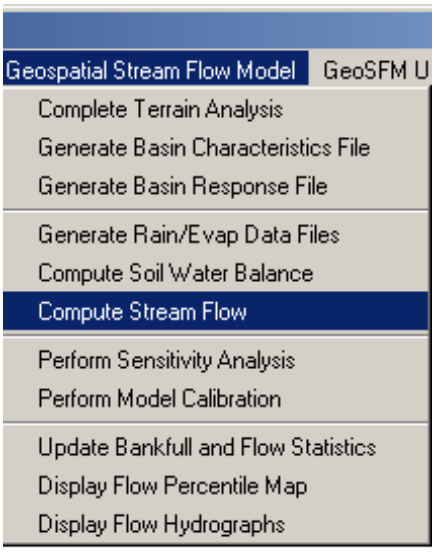
Check the box next to this theme to turn it on and make sure the theme is in a raised box. From the **Theme** menu choose **Hide/Show Legend** to display the legend. You will notice there are three categories for soil moisture, dry, moist, and wet.



2.3 Performing Stream Flow Routing

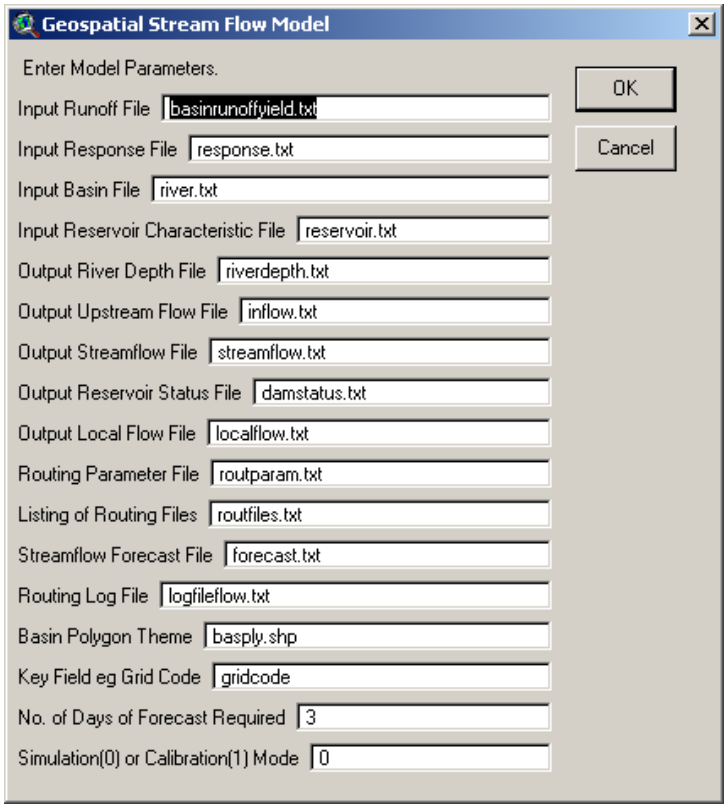
Finally, you get to the part engineers like; moving the water around.

To begin, click on the **Geospatial Stream Flow Model** menu and select **Compute Stream Flow**.

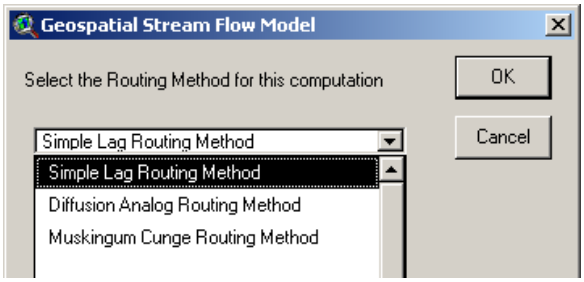


Specify your working directory as usual. Click **OK**.

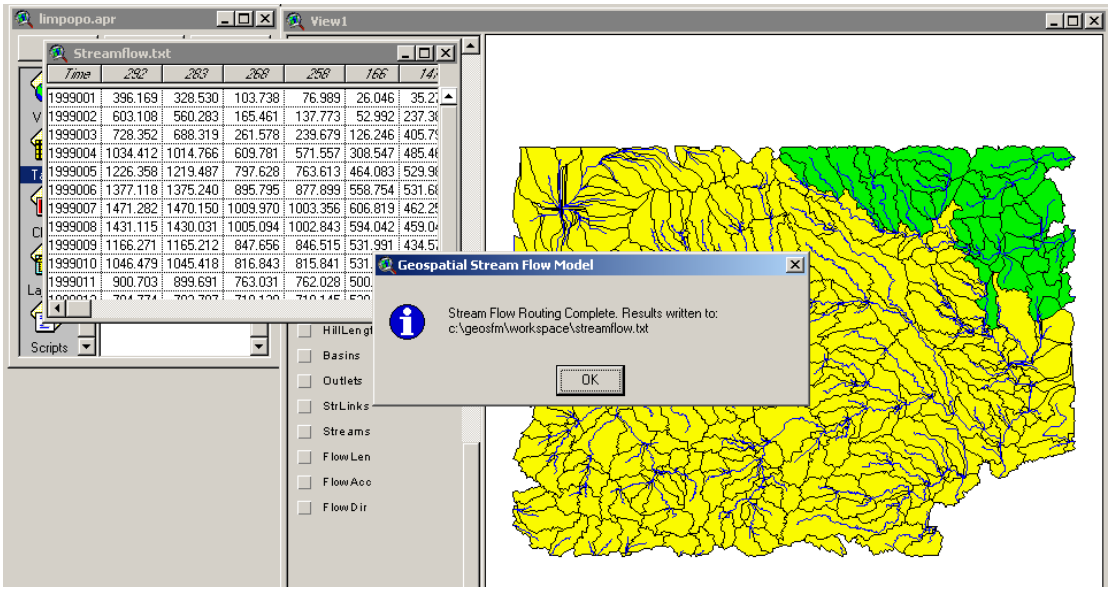
The next dialog box that appears allows you to verify or enter the simulation input files and output files. Pay particular attention to the **No. of Days of Forecast Required** field. In this example the forecast will be for a three day period of time. Click **OK**.



You will then be prompted to select the **Routing Method for this 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 exercise. 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 Results** dialog box.



The **Streamflow.txt** table is also displayed. The **streamflow.txt file** contains a velocity value in cubic meters per second for each stream, each day.

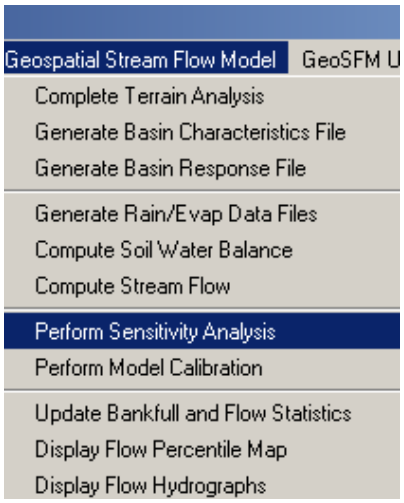
You have now completed Exercise 2. Save your project. If continuing on to the next exercise you can leave the project open. If you wish to continue with the next exercise at a later time you can close the project now.

Training Manual for the Geospatial Stream Flow Model
Ex 3: Calibration

Contents:

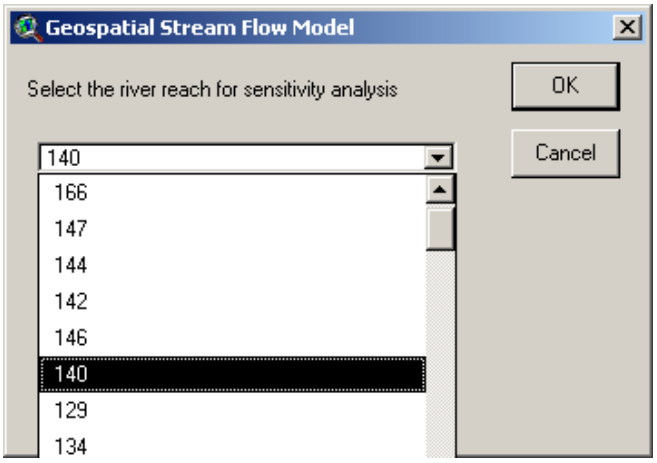
3.1 Perform Sensitivity Analysis

In Exercise three you will familiarize yourself with the calibration functionality within the **Geospatial Stream Flow Model.** The first menu item you will explore is **Perform Sensitivity Analysis**; from the **Geospatial Stream Flow Model** menu select **Perform Sensitivity Analysis**. The sensitivity analysis will test which parameters should be used for calibration as well as analyzing feasible parameter ranges. Sensitivity analysis measures the impact on the model outputs due to changes in the model inputs.

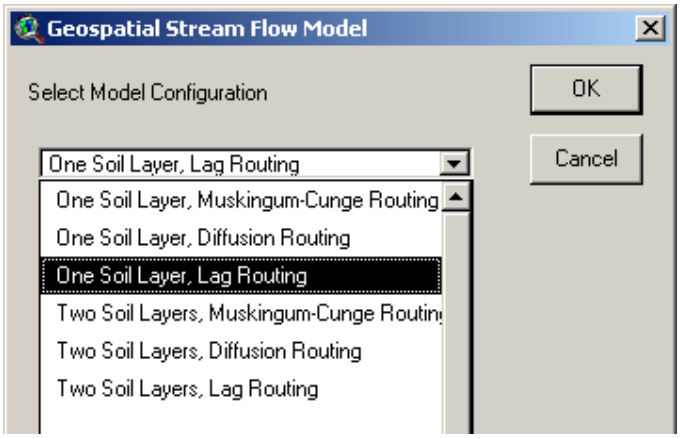


The **Working Directory** dialog box will display, **Specify your working directory**, and click **OK**.

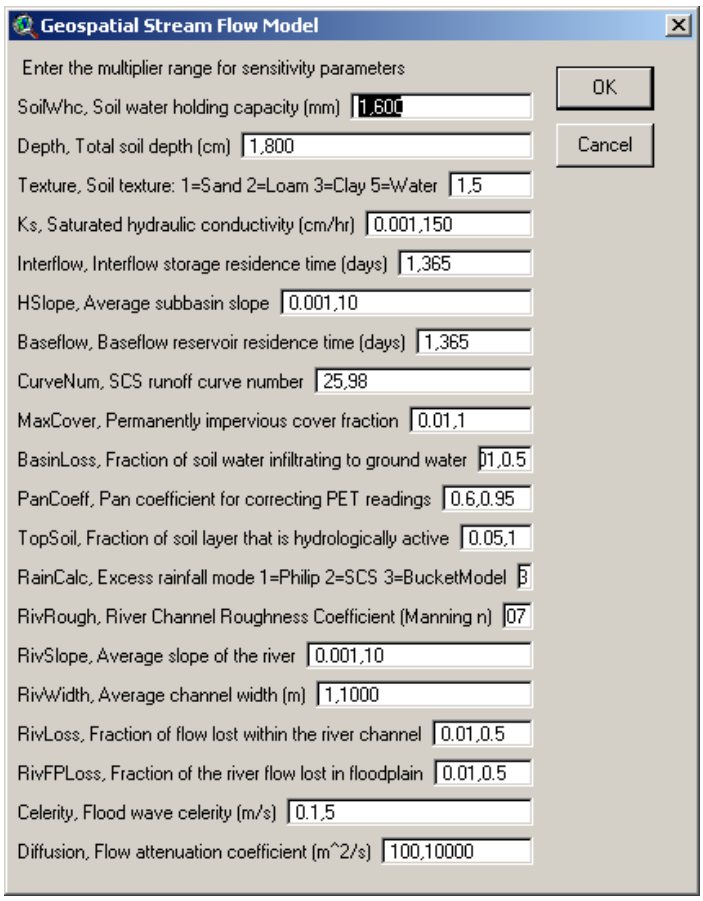
The next dialog box displayed prompts you to; **Select the river reach for sensitivity analysis**. In this exercise you will select basin 140. Highlight/click **140** in the drop down list, and click **OK**.



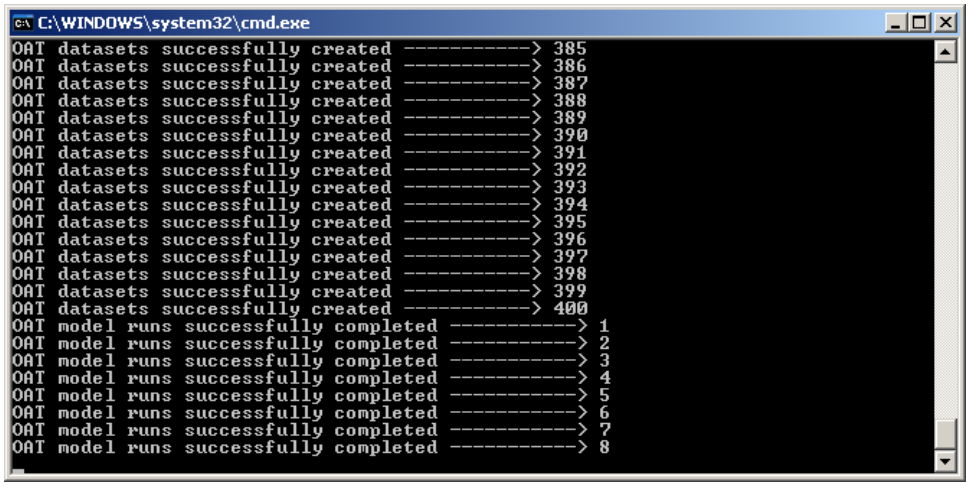
Next, the **Select Model Configuration** dialog box is displayed, select **One Soil Layer, Lag Routing.** This configuration was selected in exercise 2. Click **OK**.



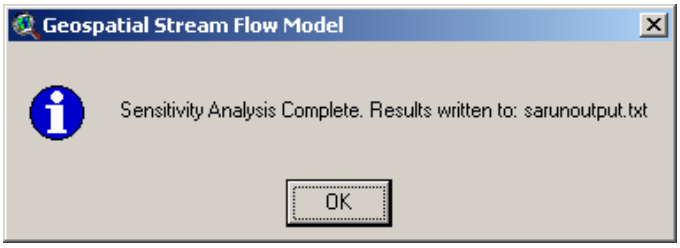
The next dialog box will list the multiplier range for the sensitivity parameters. Use the default values and click **OK**. This is a list of the twenty different parameters that will be tested.



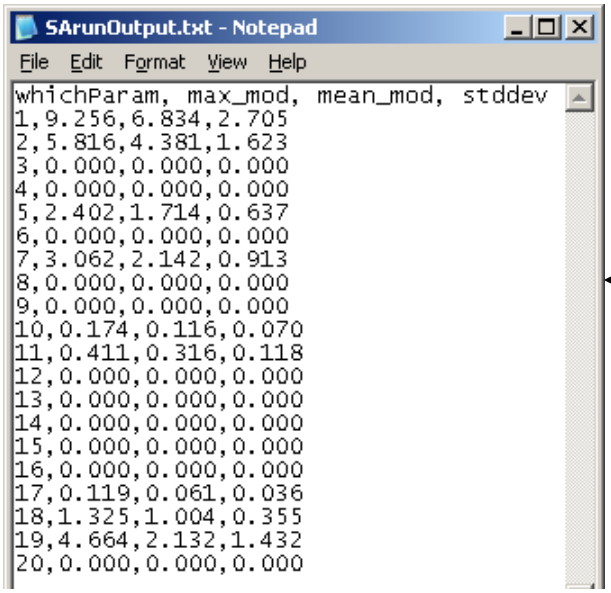
At this point, a window will open and display the number of data sets created, and the number of model runs completed. This method is a one-at-a-time method where one model run will only have one parameter changed with all other parameters held constant. The parameter values are taken at a twenty equal interval sample for twenty different parameters. This results 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**.



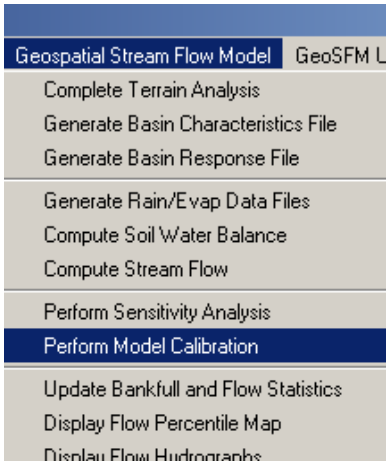
The output file **SArunOutput.txt** will give you the mean absolute difference of test results over the parameter range for each parameter. The greater the differences, the more sensitive the parameter. Sensitivity analysis is important when preparing to calibrate so that resources are not wasted on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



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

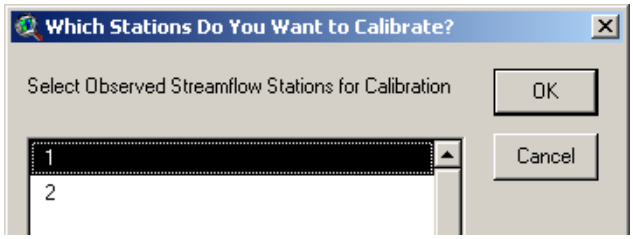
3.2 Perform Model Calibration

The last 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.

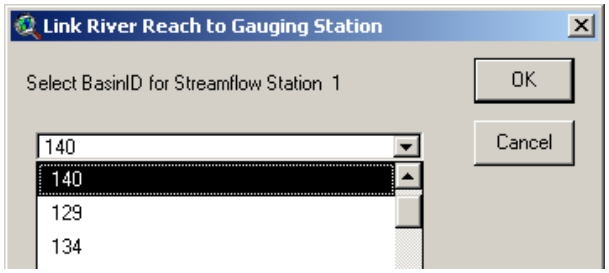


Confirm your working directory. Click **OK**.

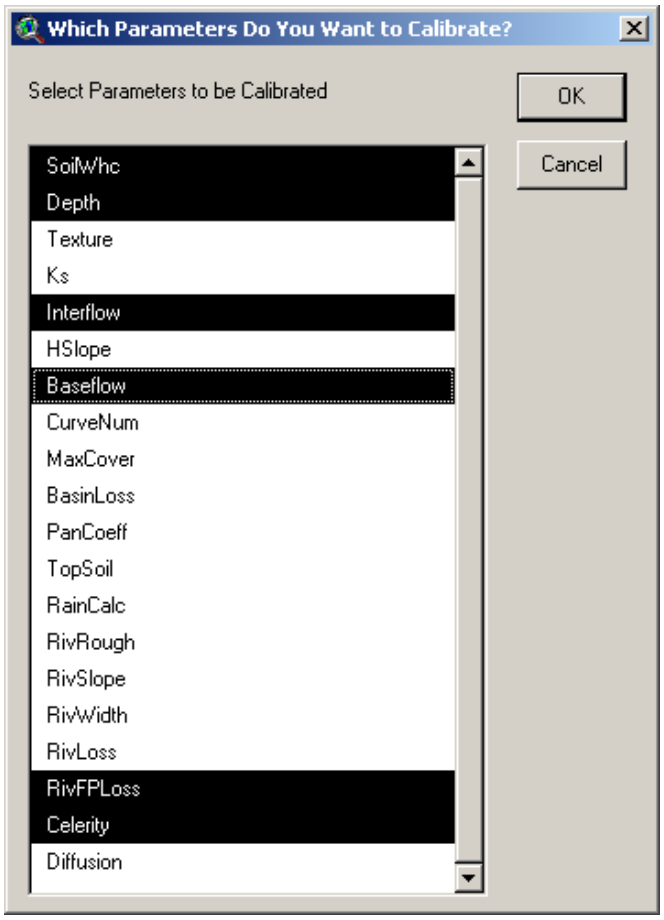
Copy the **observed_streamflow.txt** from the geosfm\samples directory and paste into your working directory –in this example **d:\GeoSFM\workspace**. In the next dialog box you will need to select which observed stream flow stations will be used for calibration. In this example there is only one station; select **1** and click **OK**.



Next, select the basin Id for the stream flow station from the drop down box. The stream flow station used for this exercise is located in basin 140. Select **140** and click **OK**.

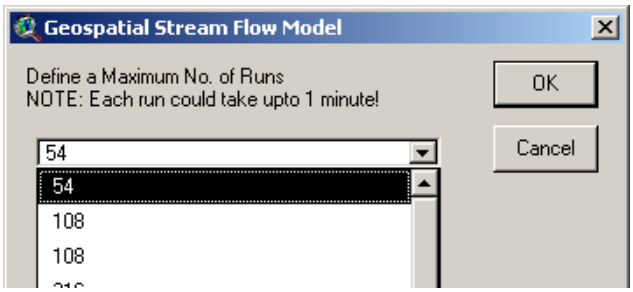


From here you will choose the parameters that you want to calibrate. Hydrographs, the watershed modelled, and sensitivity analysis are just some of the inputs the hydrologist will use to determine what parameters need to be calibrated. In this example six parameters were chosen: SoilWhc, Depth, Interflow, BaseFlow, RivFPLoss, and Celerity. These were the six parameters that were the most sensitive from the sensitivity analysis exercise. Select **SoilWhc**, **Depth**, **Interflow**, **BaseFlow**, **RivFPLoss**, and **Celerity** from the parameter list and click **OK**.

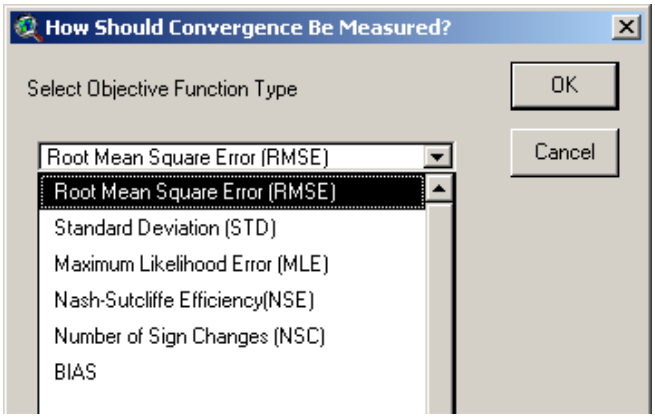


Now, you will **Define a Maximum Number of Runs**. The number of runs is dependent on the length of the streamflow record, the number of parameters being calibrated, and the complexity of the parameter dependent model response for the watershed being tested. In this exercise select **54**, the smallest number of runs from the drop down list, for shorter processing time. For optimum results the more model runs the better. The range will generally be on the order of 5000-10,000 model runs.

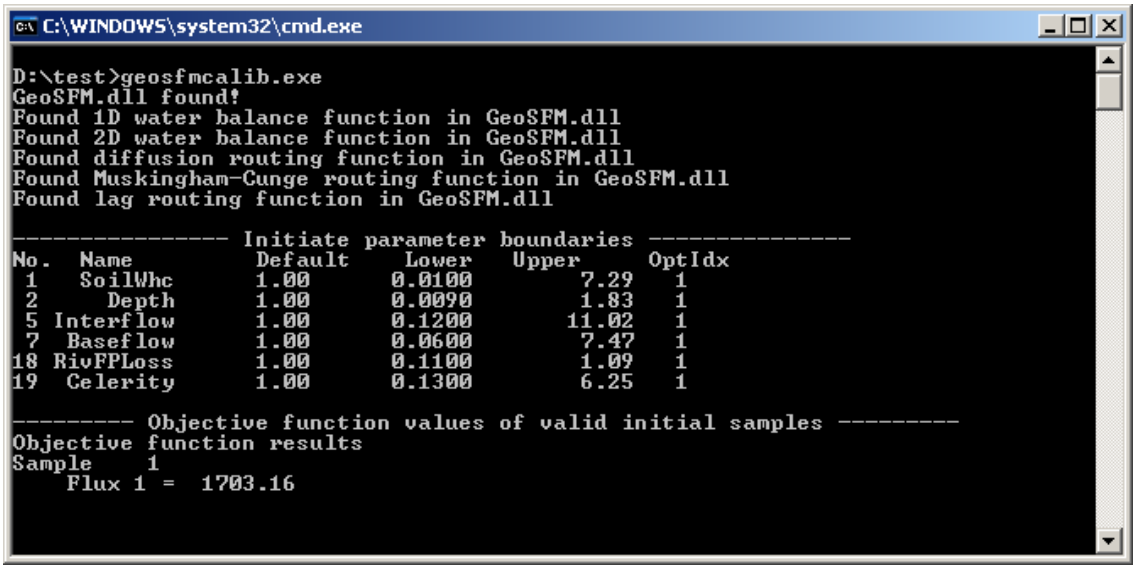
Click **OK**.



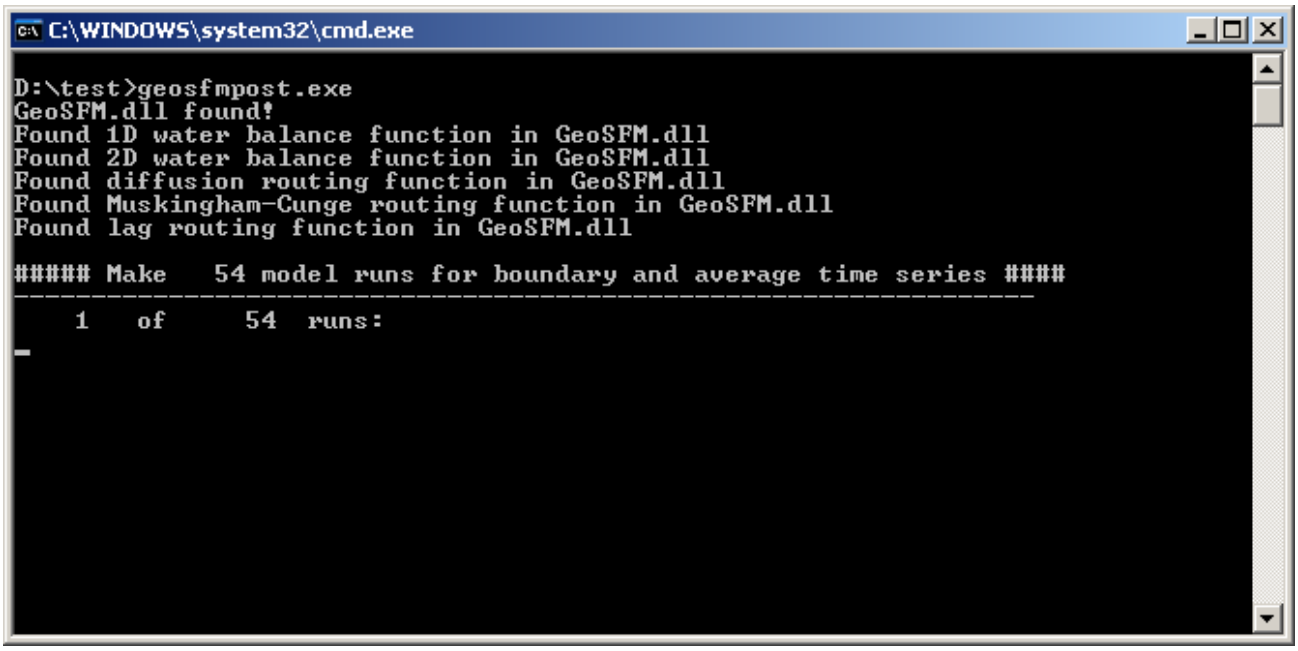
"**How Should Convergence Be Measured?**" dialog box is displayed. **Select Objective Function Type** from drop down list. The statistical test selected is **Root Mean Square Error (RMSE)**. Click **OK**.



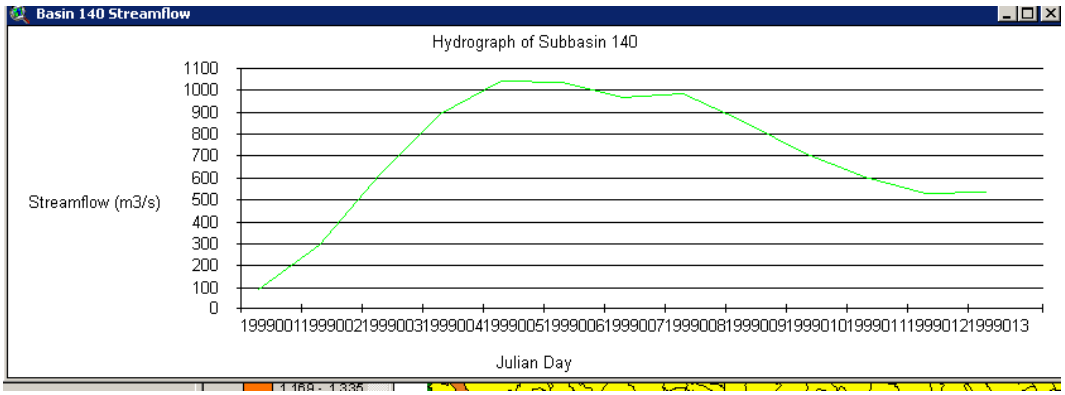
The following window will open and begin the calibration process. Notice the listed parameters. This process will take a few minutes because it needs to run through 54 samples.



The next window will open, this will start the post-processing. This will take a few minutes to finish processing.



You now have completed the calibration process.



First model run

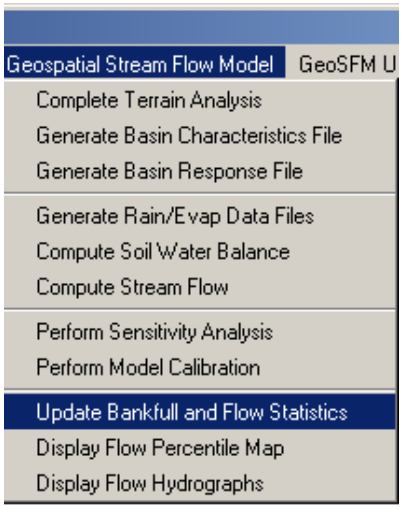
Training Manual for the Geospatial Stream Flow Model

Ex 4: Post-Processing

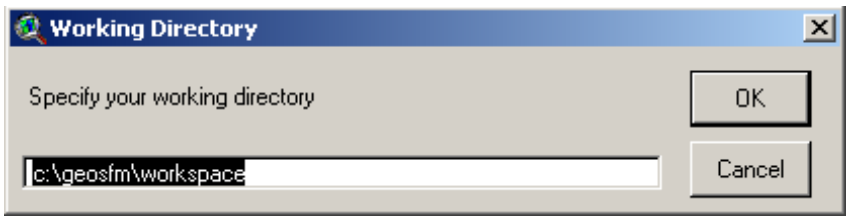
Contents:

4.1 Update Bankfull and Flow Statistics

In Exercise four you will familiarize yourself with the last group of commands in the **Geospatial Stream Flow Model** menu. The first menu item you will explore is **Update Bankfull and Flow Statistics**; from the **Geospatial Stream Flow Model** menu select **Update Bankfull and Flow Statistics**.

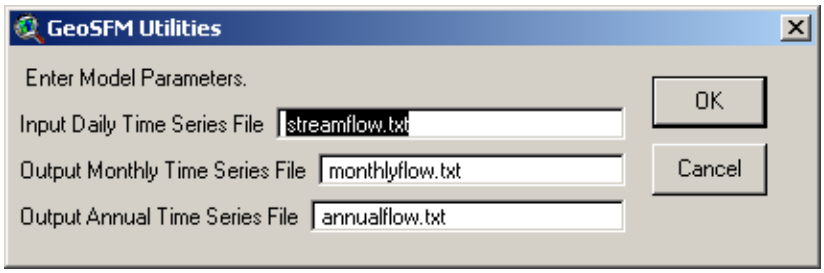


Confirm the working directory as **c:\geosfm\workspace**. Click **OK**.

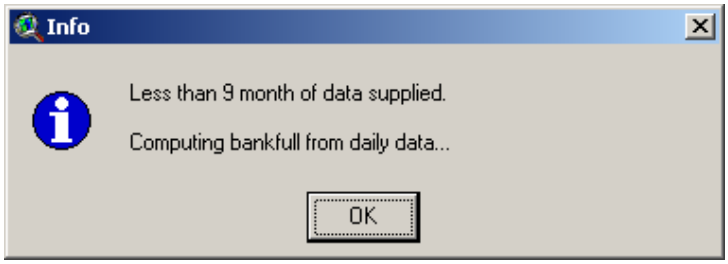


The **Basin Theme** dialog box displays asking you to **Select basin coverage/grid theme** from the drop-down box select **basply.shp**. Click **OK**.

Next, the **GeoSFM Utilities** dialog box opens, for **Enter Model Parameters** the input file should default to **streamflow.txt** and the two output files default to **monthlyflow.txt** and **annualflow.txt**. Click **OK**.

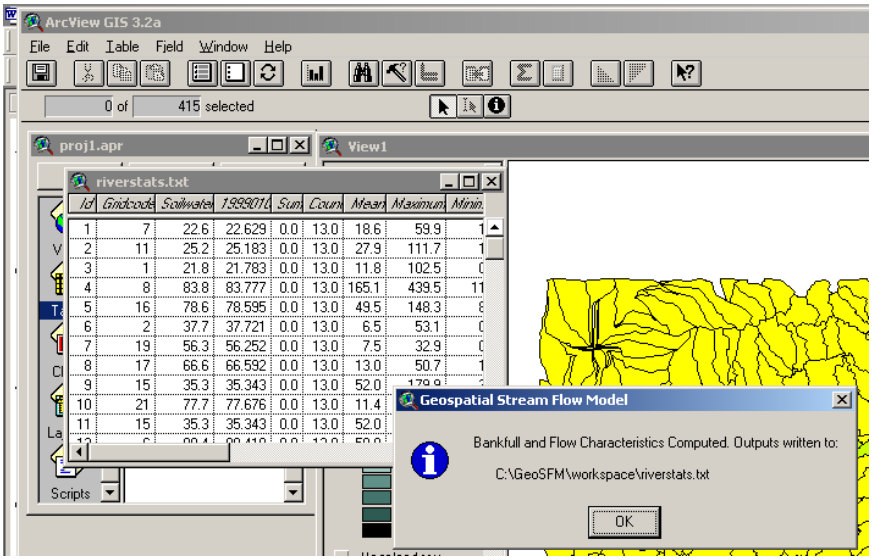


If less than 9 months of data is supplied the following **Info** dialog box will open. Click **OK**.

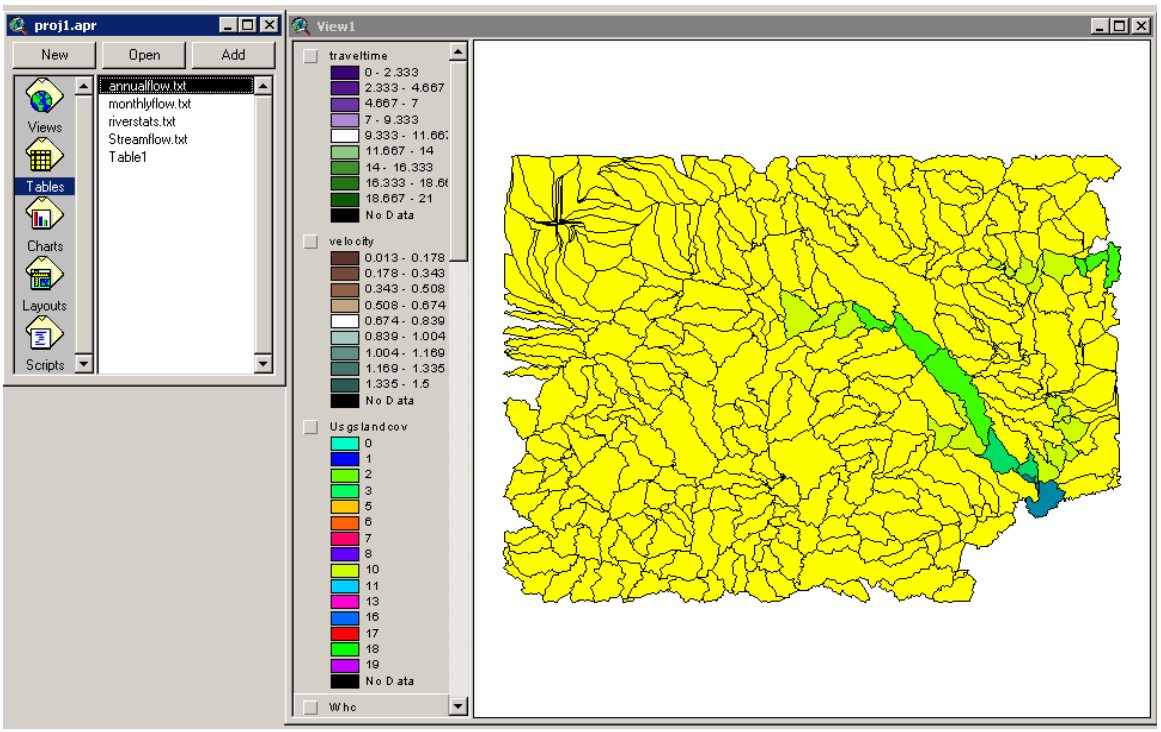


The model will soon indicate that it has finished processing and written the results to the **riverstats.txt** file in the working directory. The riverstats.txt contains the statistical estimate of how much water is needed to fill a river channel (carrying capacity.)

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

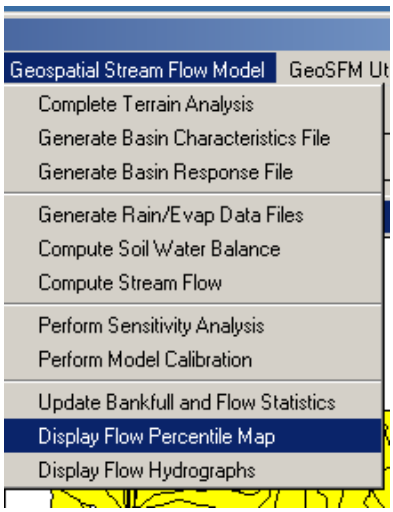


The **View** is updated and the **annualflow.txt**, the **monthlyflow.txt**, **riverstats.txt**, **streamflow.txt**, and **Table 1** are displayed in the **Tables** list.

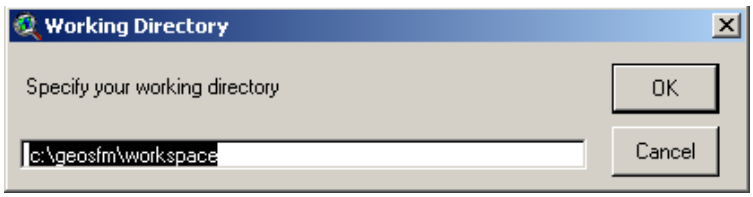


4.2 Display Flow Percentile Map

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

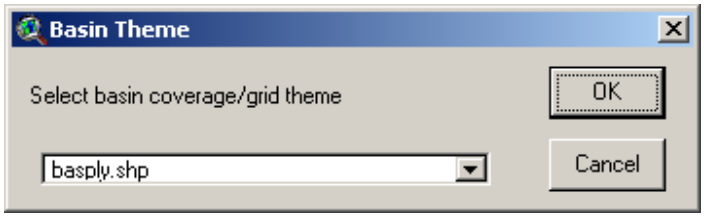


Confirm the working directory as **c:\geosfm\workspace**. Click **OK**.

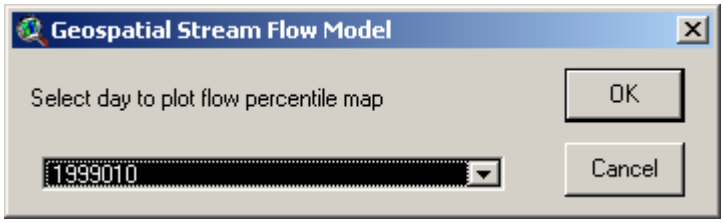


Select **Streamflow.txt** when prompted to **Select Input Data File** in the **Input Flow Time Series** dialog box. Click **OK**.

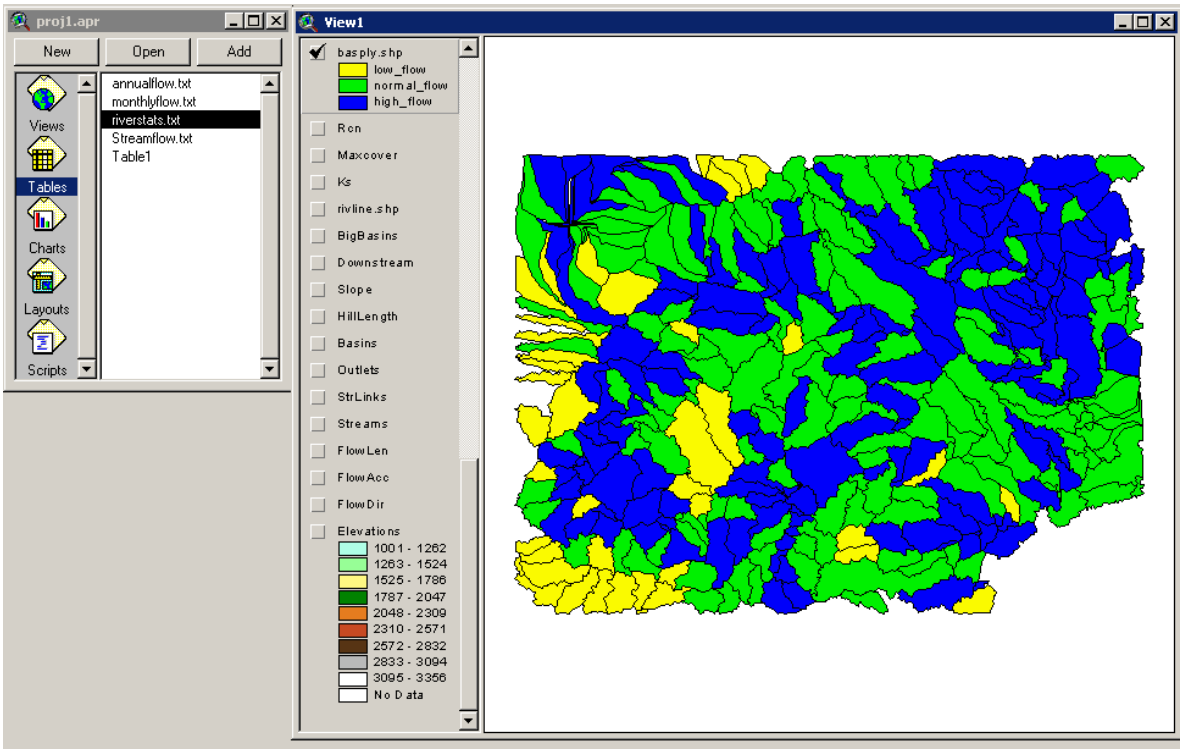
Select **basply.shp** when prompted to **Select basin coverage/grid theme** in the **Basin Theme** dialog box. Click **OK.**



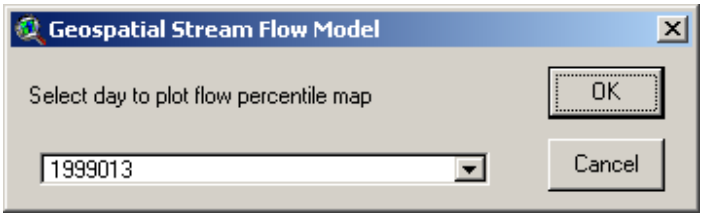
Next, **Select day to plot flow percentile map** from the drop down list. The first selection in this example was made for day 10. Click **OK**.



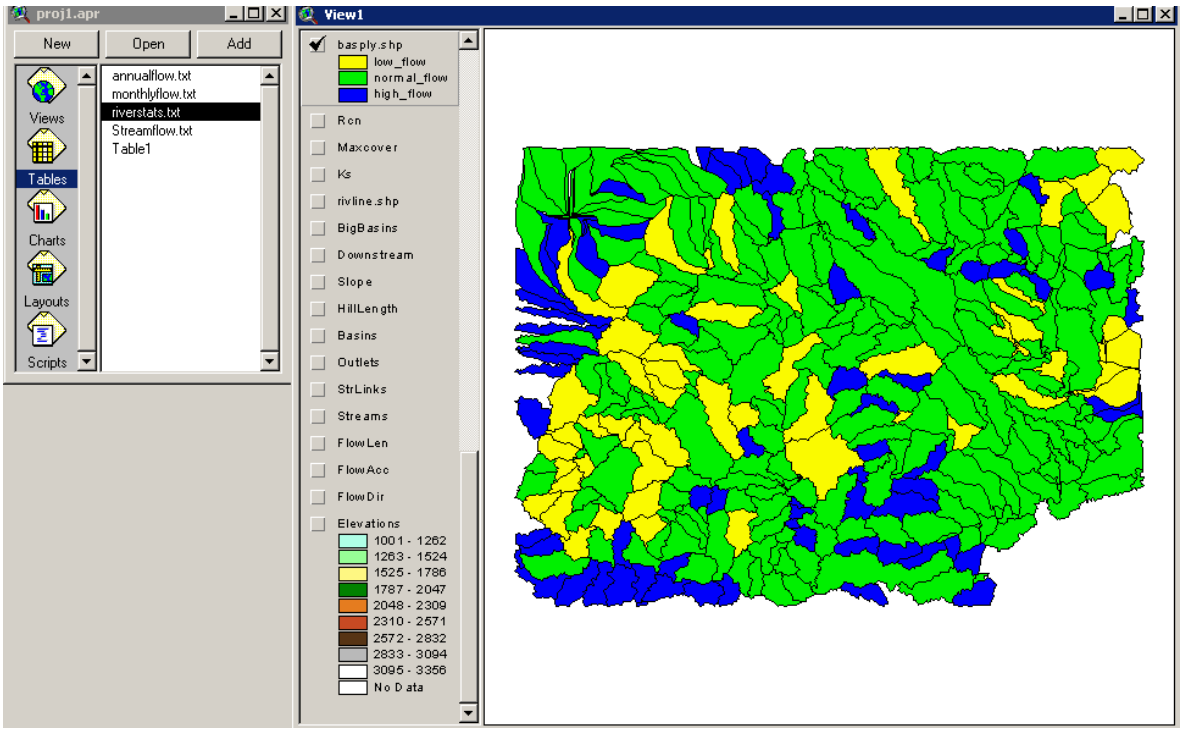
After a few seconds needed for processing your map will display. The basins are now classified into three different categories - low flow, normal flow and high flow.



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



Results for day 13 are seen below.



What changes do you see between day 10 and day 13?

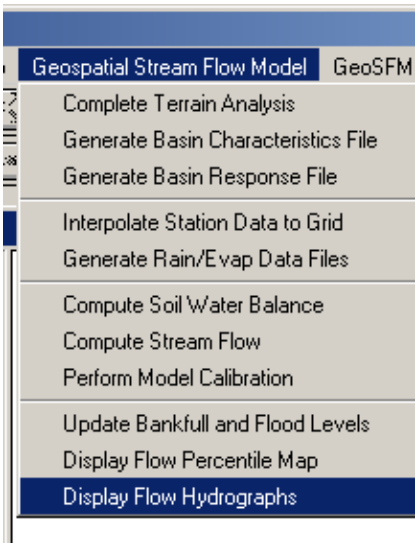
You have displayed flow condition maps, next you will look at hydrographs.

4.3 Display Flow Hydrographs

The **Geospatial Stream Flow Model** provides a tool for plotting and displaying hydrographs. You may also open the resulting file in Excel to plot out the hydrographs; Excel does provide some additional plotting options, it does a good job with displaying a long time series.

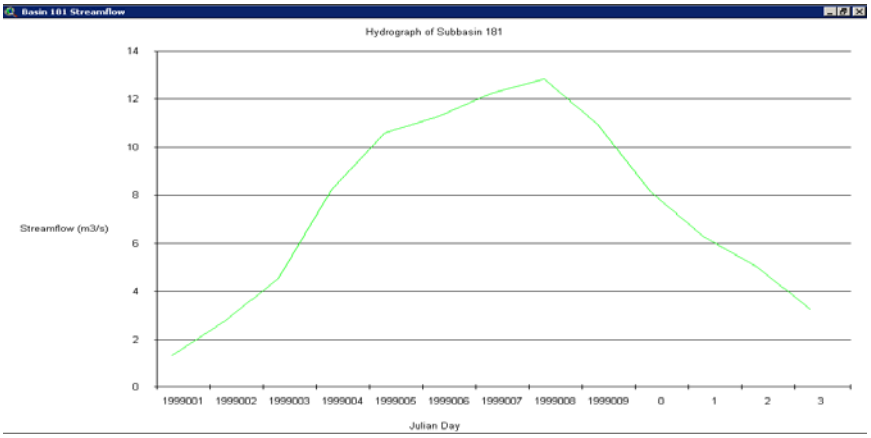
In the **View**, make sure that you have the **basply.shp** selected and displayed in raised box.

To plot the hydrographs in ArcView, go to the **Geospatial Stream Flow Model** menu and select **Display Flow Hydrographs** to active the plotting tool.

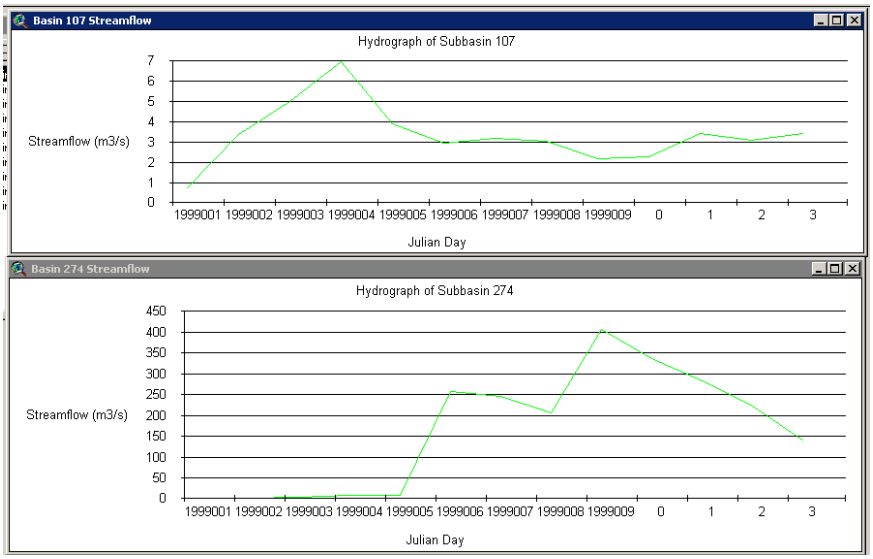


After you select **Display Flow Hydrographs** your curser changes to a "**+**", now click in one of the sub-basins in which you would like to create a hydrograph. The sub-basin is now highlighted to show that it has been selected (the yellow highlight cannot be seen in "low flow" yellow sub-basins) and a window opens with the hydrograph displayed.

An ArcView hydrograph is produced showing the daily variation of flow at the outlet of the selected sub-basin.



Click on a number of different sub-basins to see how different hydrographs compare. See if you can observe the downstream propagation of flow by comparing hydrographs from the headwater sub-basins to those farther downstream.



Does the three-day forecast indicate increasing or decreasing stream flows?

Are the basins with the high soil moisture also the same basins with the most stream flow?

You have now completed Exercise 4. Save your project. If continuing on to the next exercise you can leave the project open. If you wish to continue with the next exercise at a later time you can close the project now.

GeoSFM Utilities

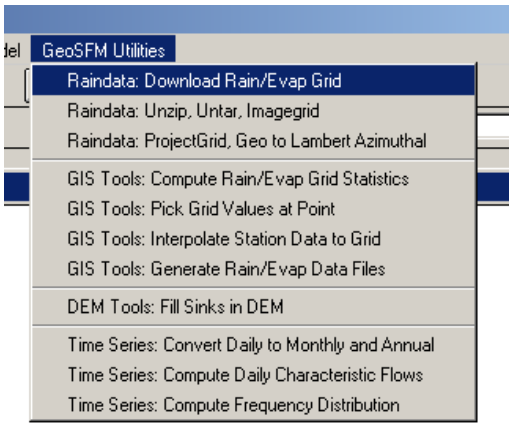
Training Manual for the Geospatial Stream Flow Model

Ex 5: GeoSFM Utilities: Raindata

Contents:

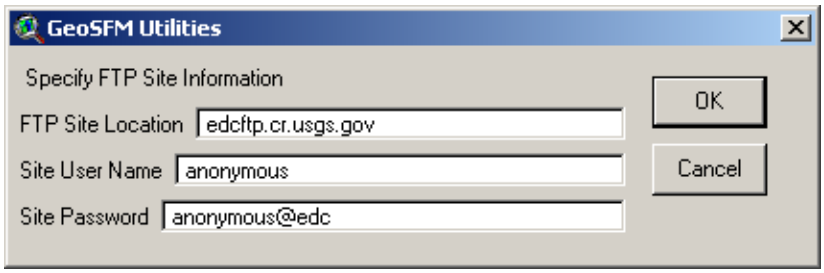
5.1 Download Rain/Evap Grid

In exercise six, you will familiarize yourself with the Raindata selections found in the GeoSFM Utilities extension. The first **Raindata** option is **Download Rain/Evap Grid**. Using the **GeoSFM Utilities** tool, you will download data from USGS/EROS anonymous FTP server. From the main toolbar go to **GeoSFM Utilities**, and select **Raindata:Download Rain/Evap Grid** from the drop down list.

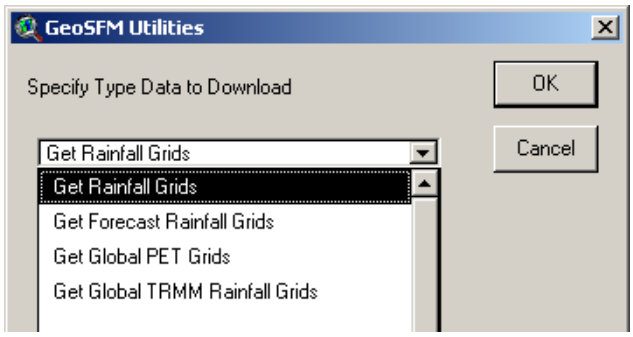


Specify your working directory. Click **OK**.

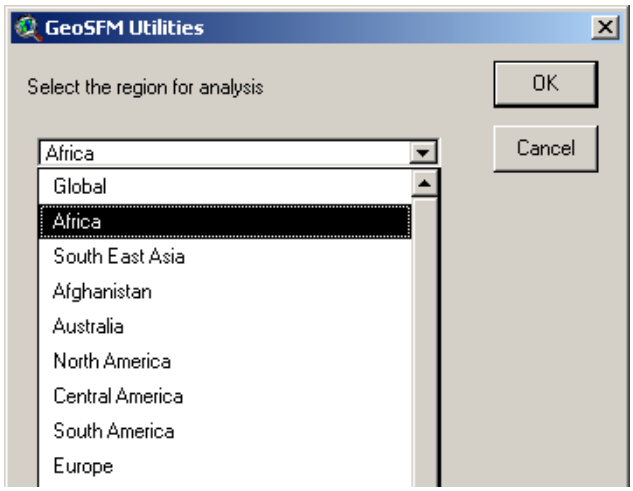
Specify FTP site information –defaults to values below. Click **OK**. To receive data from the FTP site you will need to have an open internet connection.



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

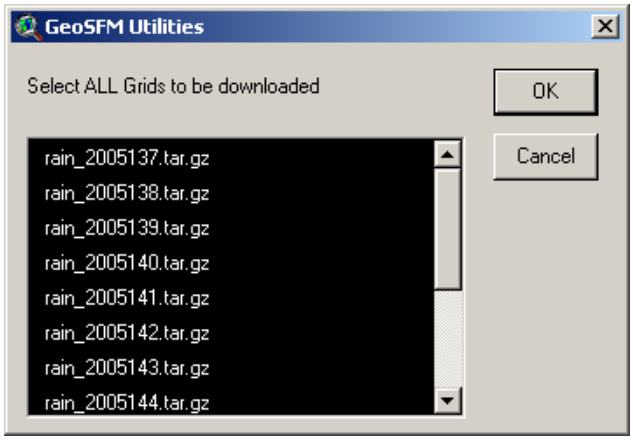


In the **Select the region for analysis** dialog box select **Africa** from drop down list. Click **OK**. The latest available data set will be displayed for the region selected. *See GeoSFM Users manual for detailed list.*

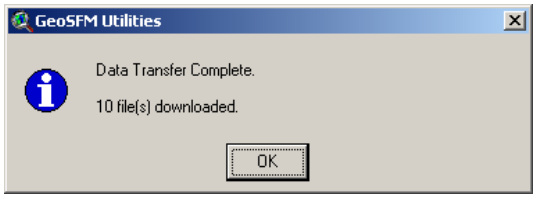


Wait for the FTP transfer to complete (Internet connection must be open). The program writes a local file with the selected region datasets. The available rain/evap data will be displayed for downloading. Generally, sixteen days of data are available for downloading from the FTP site.

Select All Grids to be downloaded by clicking and highlighting each selection that you want downloaded. For exercise two, 10 days of data were used. Click **OK**. (File naming convention for raindata is rain_YYYJJJ.tar.gz ("Y" – year, "J"- Julian day.) File naming convention for evapdata is etcYYMMDD.tar.gz ('Y"–year, "M"–month, "D"-day.)



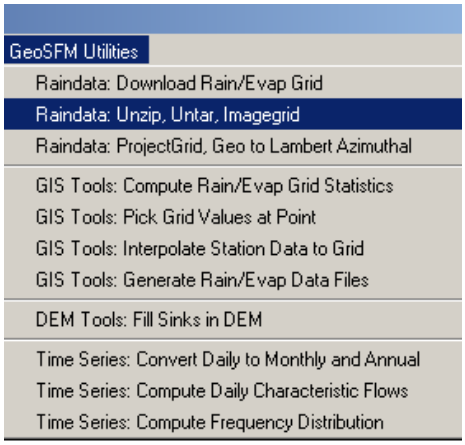
The **GeoSFM Utilities** box appears with the message - **Data Transfer Complete -** with the number of files downloaded. Now, all downloaded files will be in your working directory. Click **OK**. This process can be repeated for downloading PET data.



5.2 Unzip, Untar, Imagegrid

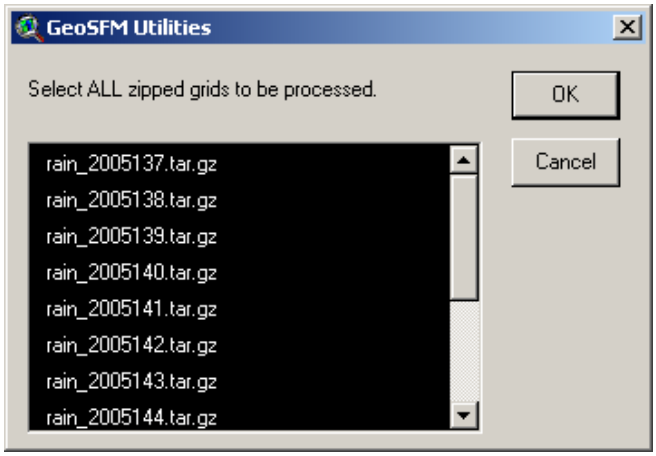
Next, you will prepare the data for analysis. The second selection under **Raindata** is **Unzip, Untar, Imagegrid**. An internet connection is not required for this function.

From the main toolbar go to **GeoSFM Utilities** tool and select **Raindata:Unzip, Untar, Imagegrid** from drop down list.

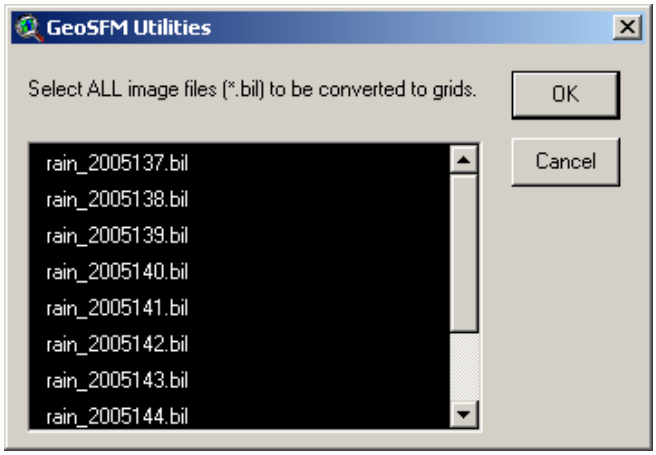


Specify your working directory. Click **OK**.

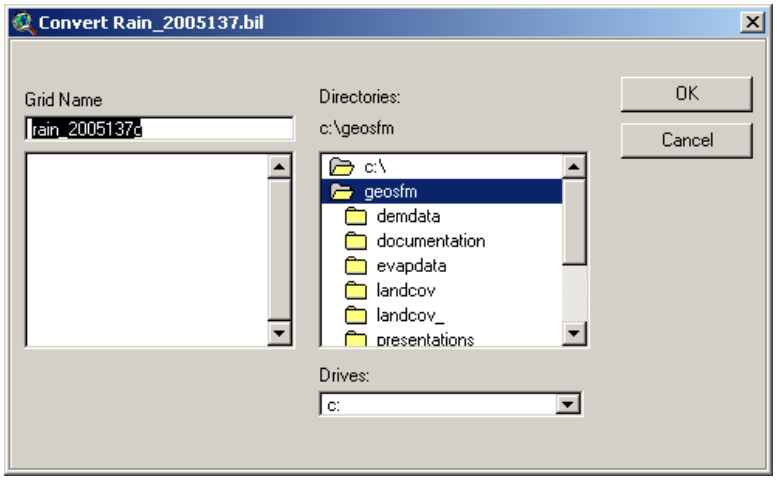
Select All Grids to be processed by clicking and highlighting each selection that you want processed. Click **OK**.



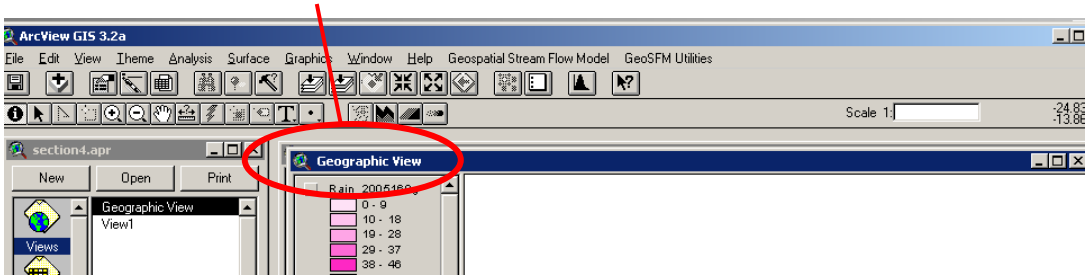
Next, **Select All image files (*.bil) to be converted to grids** by clicking and highlighting each item. Click **OK**.



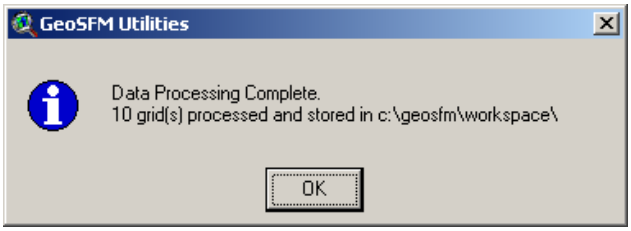
A dialog box will appear for each file, confirm your directory. Click **OK** for each dialog box –one for each rain file downloaded.



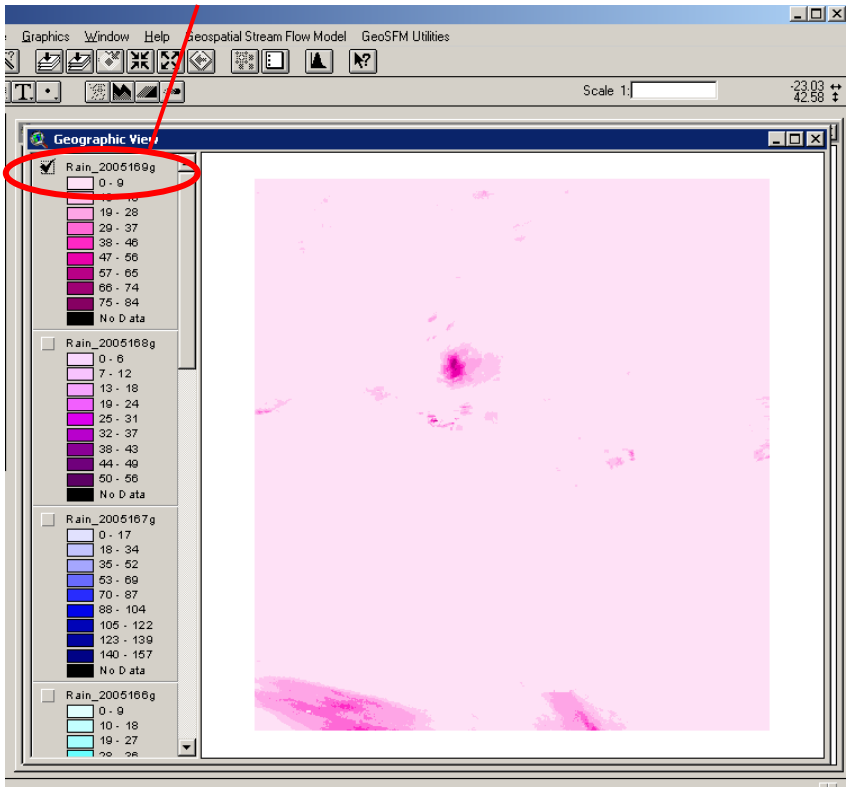
At the same time the rain grids will be added to the table of contents in a new **View** labelled **Geographic View**.



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

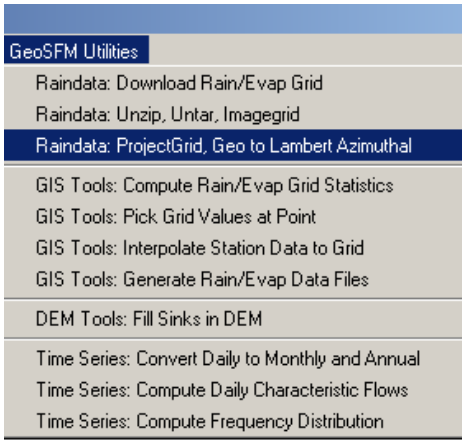


All **Rain_YYYYJJJg** themes are added to the table of contents as seen below. ("g" at the end of grid name –geographic projection)



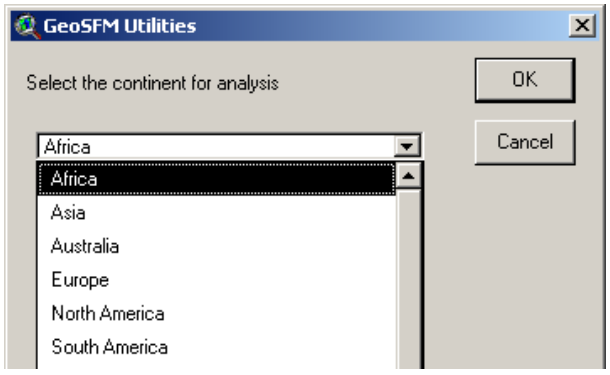
5.3 Geo to Lamber Azimuthal

The last selection in the **Raindata** functionality is **ProjectGrid, Geo to Lambert Azimuthal**. This function will change the projection of the grids from geographic to Lambert Azimuthal. From the main toolbar go to the **GeoSFM Utilities** tool and select **Raindata:ProjectGrid, Geo to Lambert Azimuthal** from drop down list.

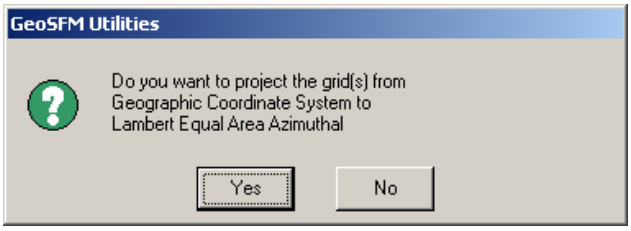


Specify your working directory. Click **OK**.

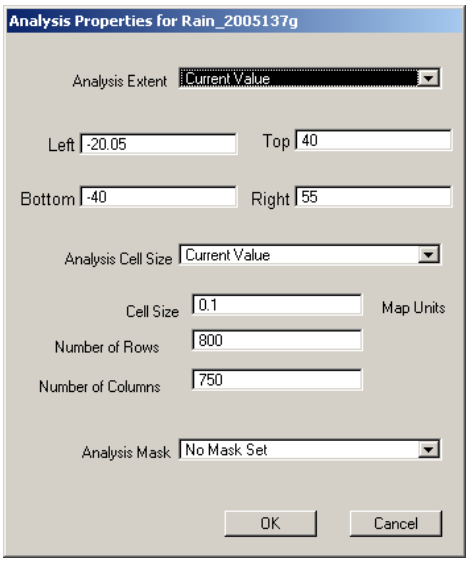
In the **Select the continent for analysis** dialog box select **Africa** from drop down list. Click **OK**.



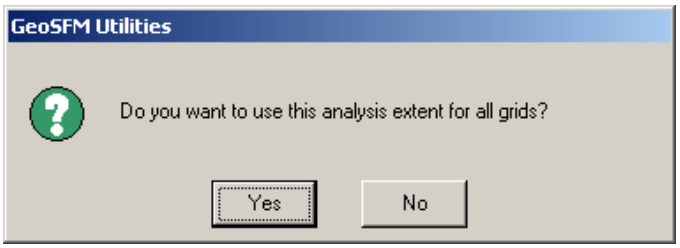
Select **Yes**, in the **Do you want to project the grid(s) from Geographic Coordinate System to Lambert Equal Area Azimuthal** dialog box.



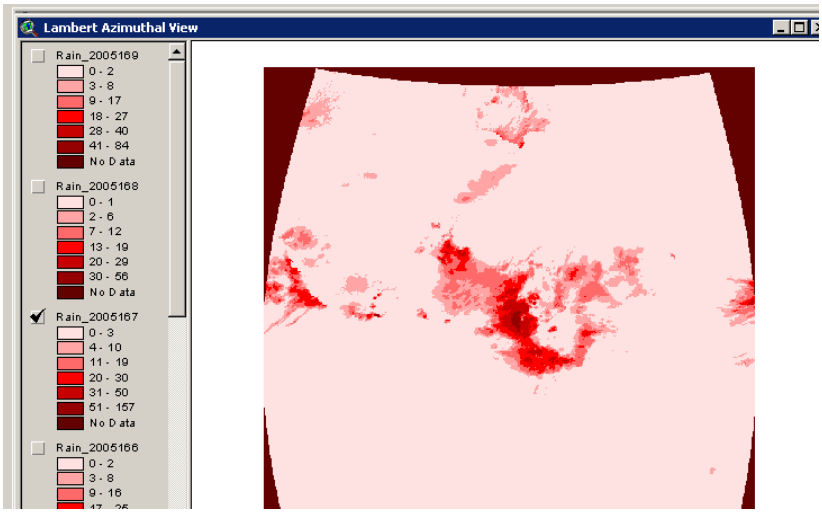
Next, the **Analysis Properties for Rain_YYYYJJJg** is displayed; accept the default values in the **Analysis Properties** dialog box. Click **OK**.



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** process in section 2.1 of this document. All three steps, section 5.1 through 5.3, will need to be repeated for evaporation data.

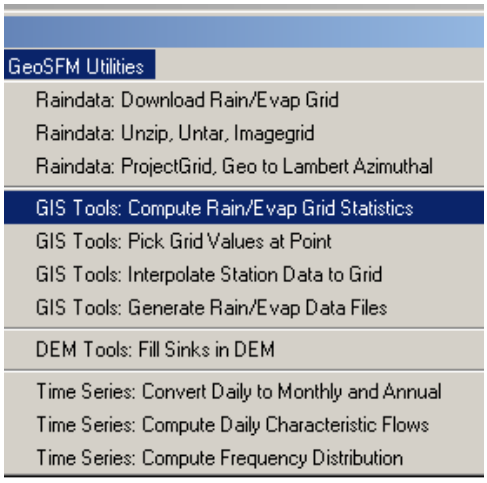
Training Manual for the Geospatial Stream Flow Model

Ex 6: Using GeoSFM Utilities –GIS Tools

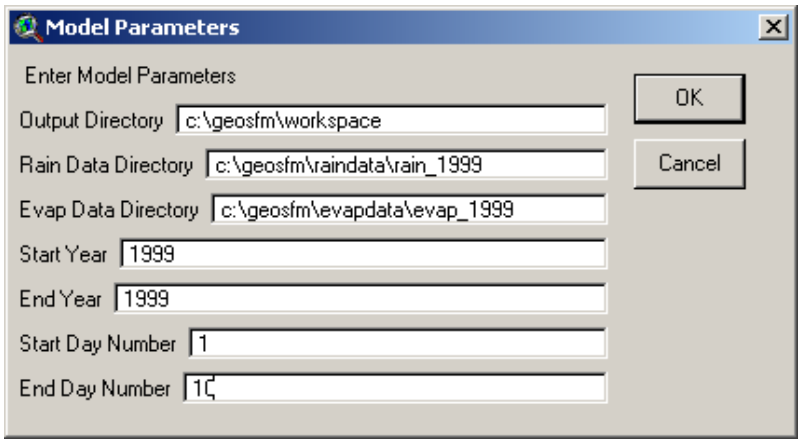
Contents:

6.1 Compute Rain/Evap Grid Statistics

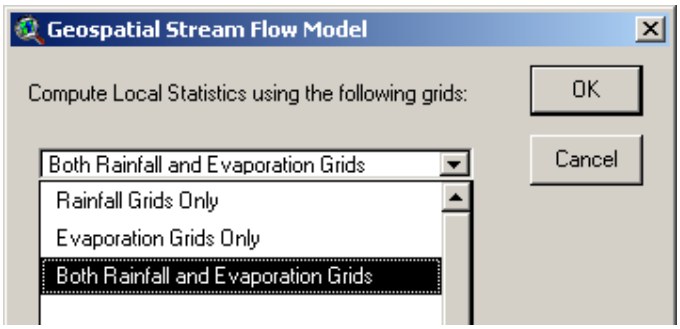
In exercise seven you will familiarize yourself with the GIS Tools found in the **GeoSFM Utilities** extension. The first menu item is **GIS Tools: Compute Rain/Evap Grid Statistics**; from the **GeoSFM Utilities** menu select **GIS Tools: Compute Rain/Evap Grid Statistics.**



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** field 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**.

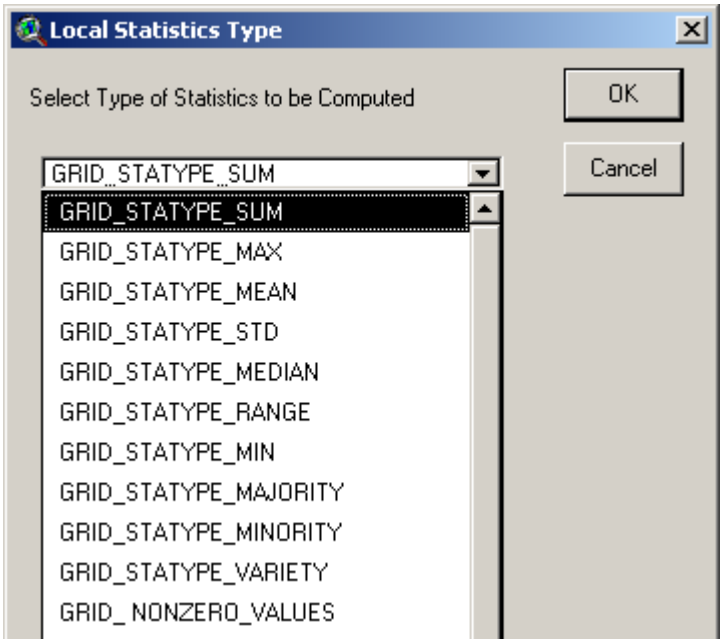


From the **Compute Local Statistics using the following grids**: select **Both Rainfall and Evaporation Grids**. Click **OK**.

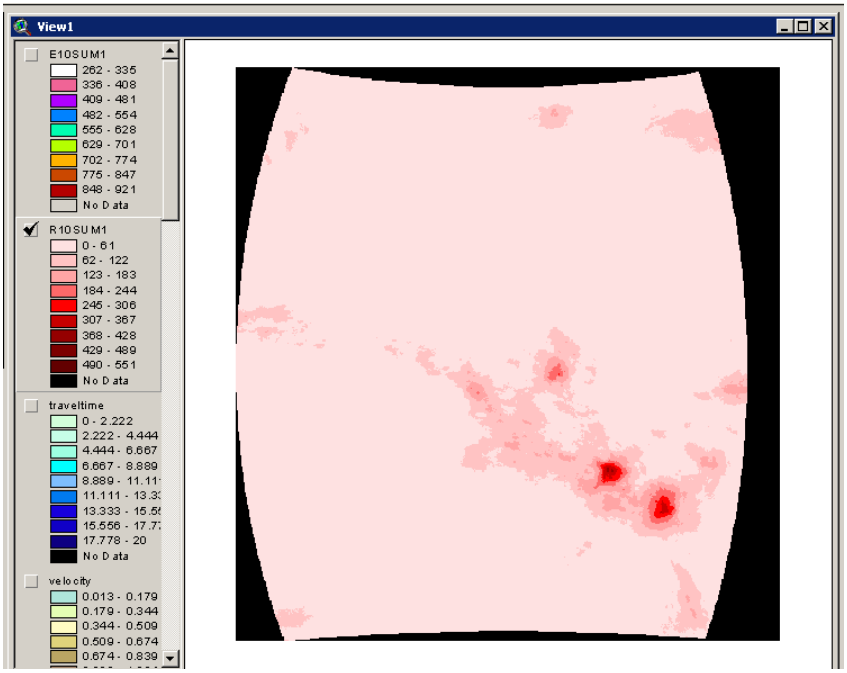


Next, from the **Local Statistics Type** dialog box, **Select the Type of Statistics to be Computed** from the drop down list. For this exercise you will select **GRID_STATYPE_SUM**, which will display the sum rain/evap for the 10 day date range you defined in the **Model Parameters** dialog box.

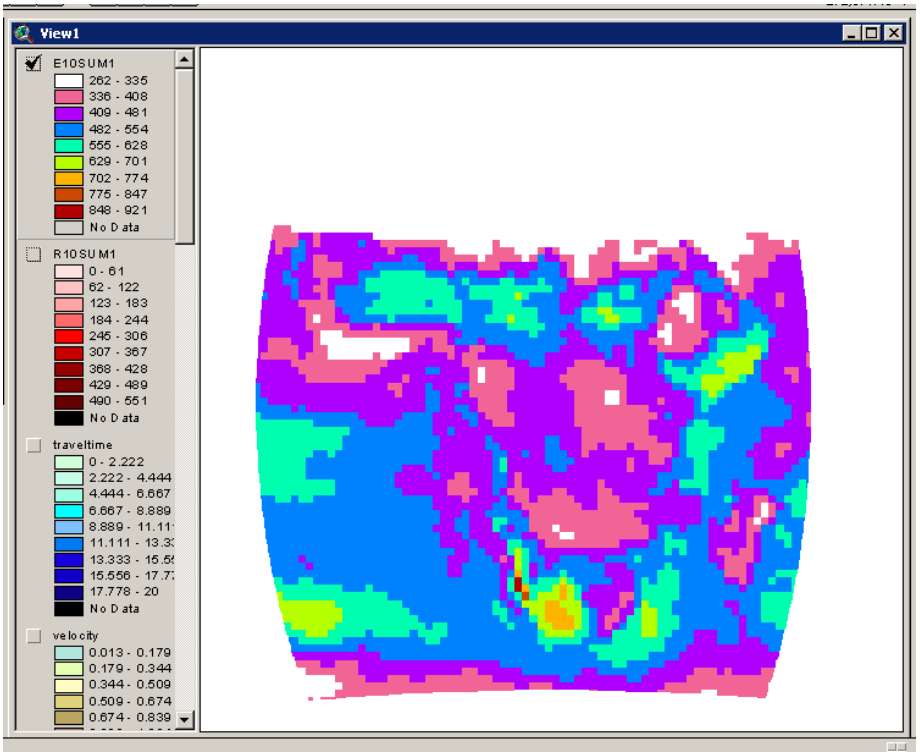
Take note of the list of various statistics that can be computed and displayed. Click **OK**.



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



After a few more seconds the **E10SUM1 Theme** (sum of the evap data for 10 days) is displayed.



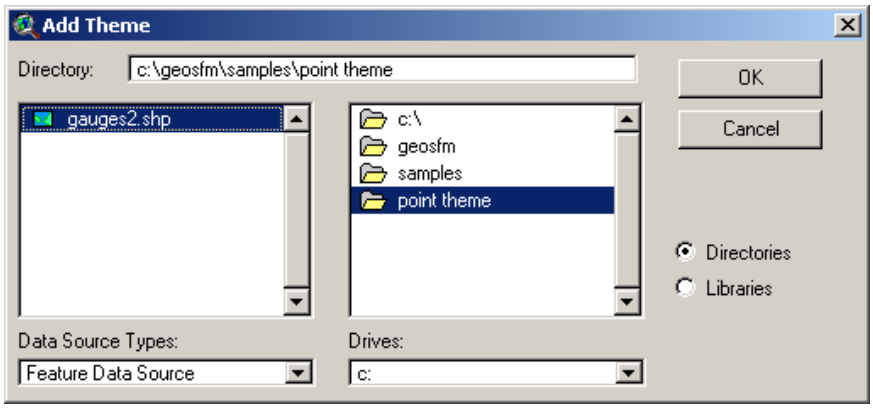
Run through the process again and select a different type of statistic to compute, compare the differences in the displayed data.

6.2 Pick Grid Values at Point

The second menu item is **GIS Tools: Pick Grid Values at Point**. From this menu item rainfall and evaporation data can be converted from polygon data to point data. Through the use of this functionality, you can also compare the rainfall/evap values gathered using satellite products, with observed data gathered from rain gauge stations. To begin this process you will need a point theme, if you need to create a point theme the instructions will follow this section. For completing this exercise the point theme is provided in the **c:\geosfm\samples\point theme** directory.

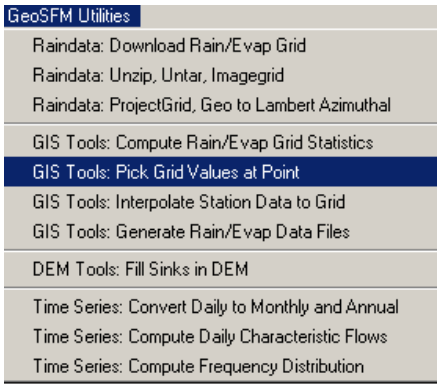
First, you will start by adding a theme, click on the Add Theme button  . From the **Add Theme** dialog box navigate to the **c:\geosfm\samples\point theme**

directory. In **List of File Type** make sure **Feature Data Source** is selected. Click to highlight **gauges2.shp** and click **OK**.



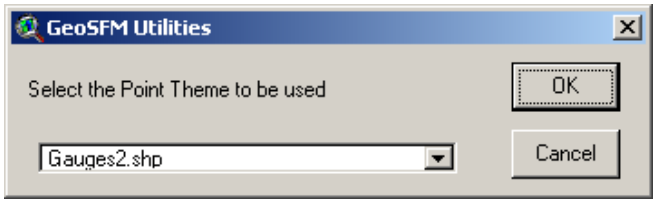
This shapefile contains randomly defined points just for use in this exercise; these points do not reflect actual rain gauge station sites. This is to help you get an understanding of the functionality available to you through the GeoSFM Utilities tool.

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

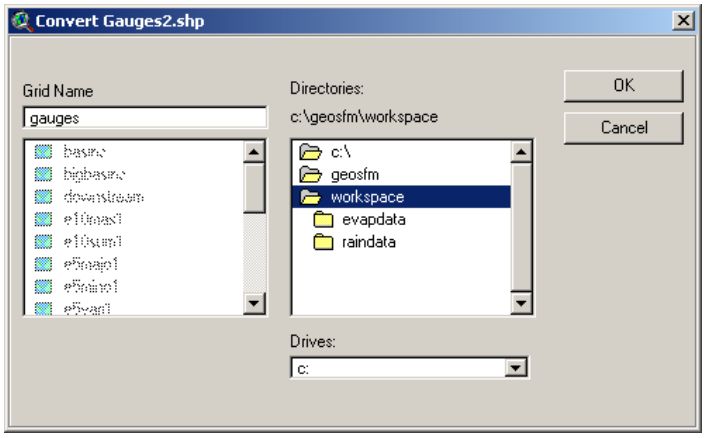


Confirm your working directory. Click **OK**.

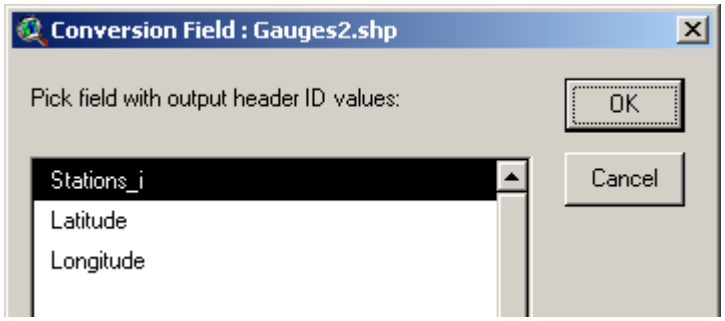
Next, from the **GeoSFM Utilities** dialog box **Select the Point Theme to be used**, by selecting **gauges2.shp** from the drop down list. Click **OK**.



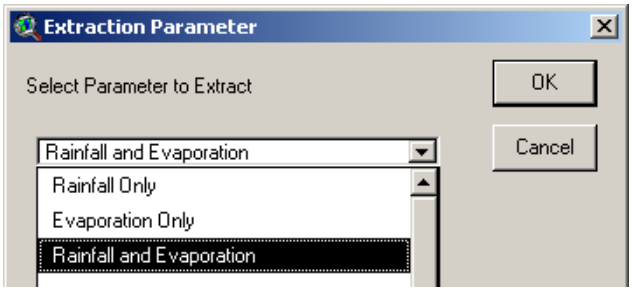
The **Convert gauges2.shp** dialog box opens, now you will convert the shapefile to a grid. Navigate to the **c:\geosfm\workspace** directory. Name your new grid in the **Grid Name** field. Click **OK**.



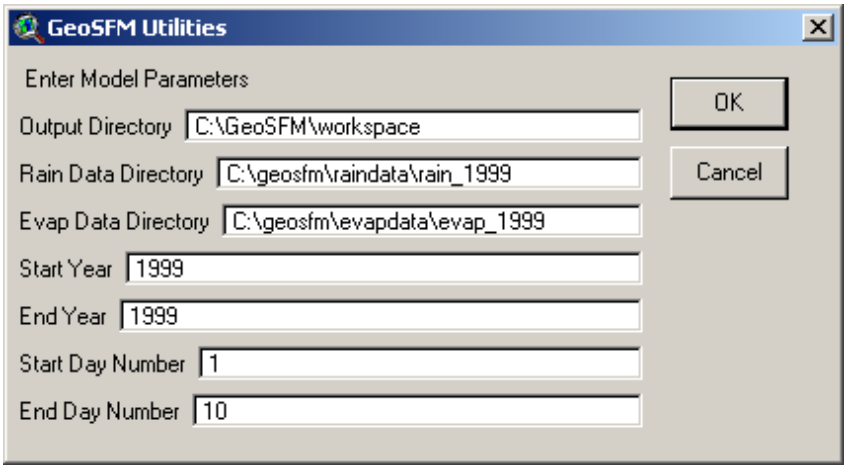
In the **Conversion Field: gauges2.shp** dialog box below **Pick field with output header ID values** select **Stations_i**. Click **OK**.



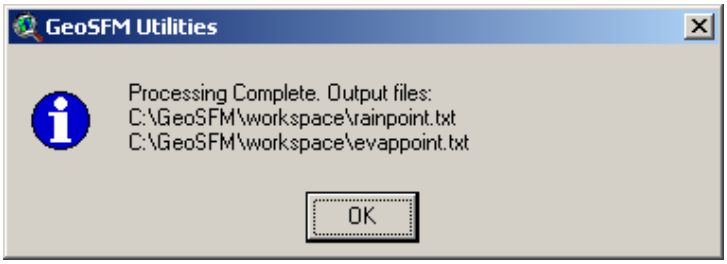
Next, the **Select Parameter to Extract** dialog box will open; for this exercise select **Rainfall and Evaporation.** Click **OK.**



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** field and the **End Day Number** field for the time frame you want computed. 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 ten days in 1999.

6.2.1 Creating a Point Theme

First, you will need to create a table in Excel with site IDs and coordinates for each gauge station site in the basin.

Below is an example of Station IDs along with the latitude and longitude (decimal degrees) coordinates necessary to plot each station point.

Stations_i	Latitude	Longitude
1	-25.3806	28.31667
2	-25.8072	27.91056
3	-25.1278	27.62889
4	-25.7244	26.88611
5	-25.8958	27.93472
6	-25.8936	27.91472
7	-25.5667	27.75306
8	-26.0342	27.845
9	-25.0622	27.52111
10	-25.4667	28.26361
11	-24.1594	27.47972
12	-23.975	27.725
13	-22.95	27.97389
14	-23.9811	28.4
15	-22.935	28.00417
16	-22.5972	28.88639
17	-22.9083	29.61417
18	-22.2256	29.99056

It is important that the latitude and longitude are formatted in decimal degrees; this will be a requirement within ArcView when changing projections. If the data you are working with is in degrees then a conversion will need to be done. This can be accomplished easily in Excel.

The formula for conversion is:

degrees + minutes/60 + seconds/3600 = decimal degrees

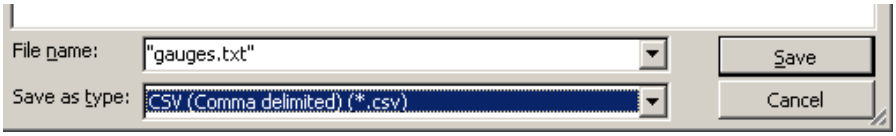
example: lat. 25° 22’ 50”

25 + 22/60 + 50/3600 = 25.38056 (negative for southern latitudes)

When the table is complete select **File** and **Save As…**

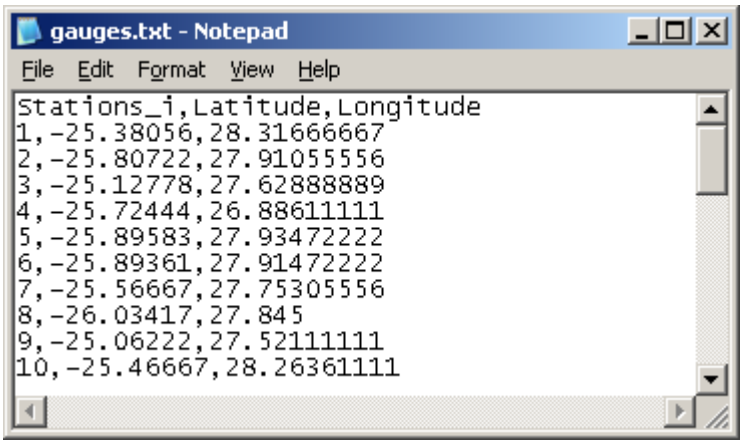
File name: “gauges.txt”

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

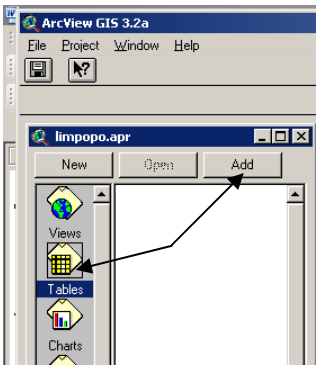


The Excel file has been saved as a comma delimited file and can now be opened in ArcView.

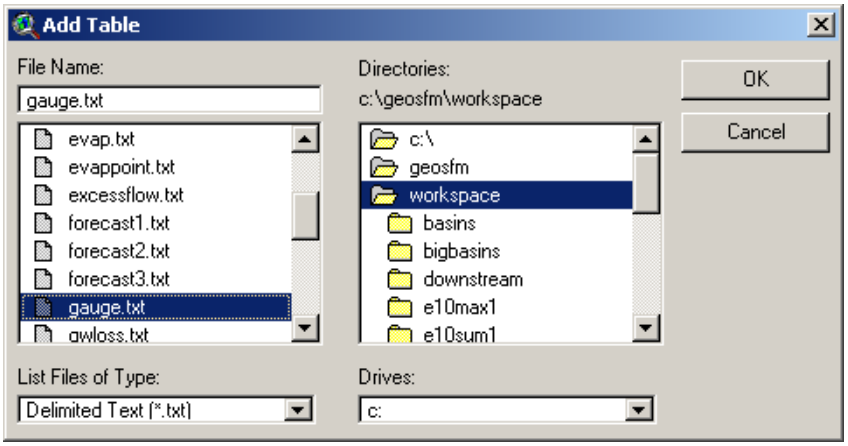
Below is an example of gauges.txt opened in Notepad.



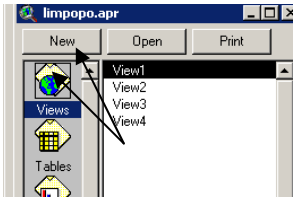
Now, you are ready to add the table to ArcView by selecting **Tables** and clicking on the **Add** button - seen below.



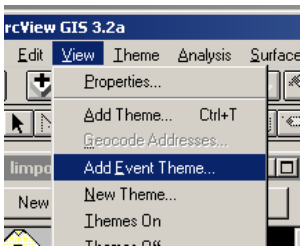
The **Add Table** box will open; navigate to your working directory. Under **List Files of Type**: select file type (this example Delimited Text (*.txt.) Click on the name of your file to populate **File Name**. Click **OK**.



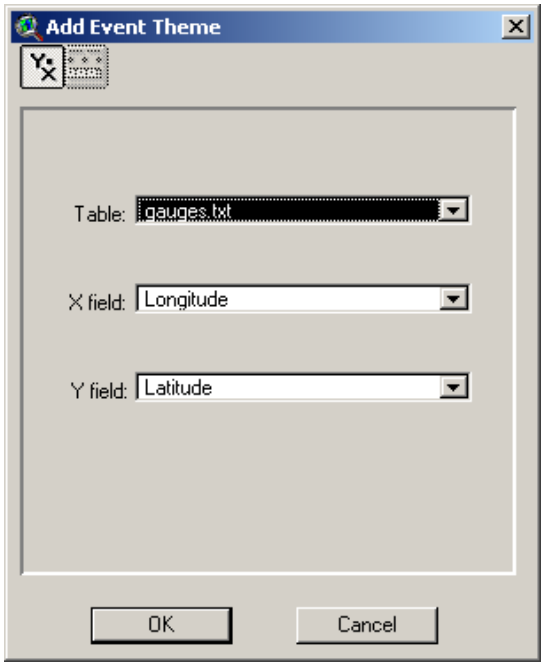
Next, you are going to open a new **View** by clicking on **View** and the **New** button from the project window. You need to open a new **View** because you will be working in a new projection.



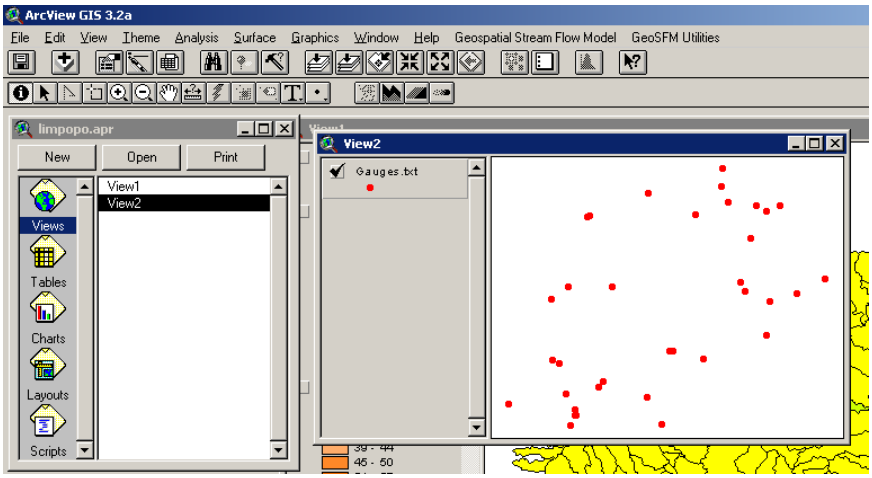
Next, you will add the table to the **View** by using the **Add Event Theme** feature. From the toolbar select the **View** menu and then select **Add Event Theme**.



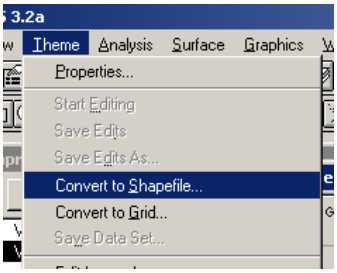
The **Add Event Theme** opens, select your **Table** (gauges.txt) from the drop down list. Select **Longitude** for the **X field** and **Latitude** for the **Y field** from the drop down lists. Click **OK**.



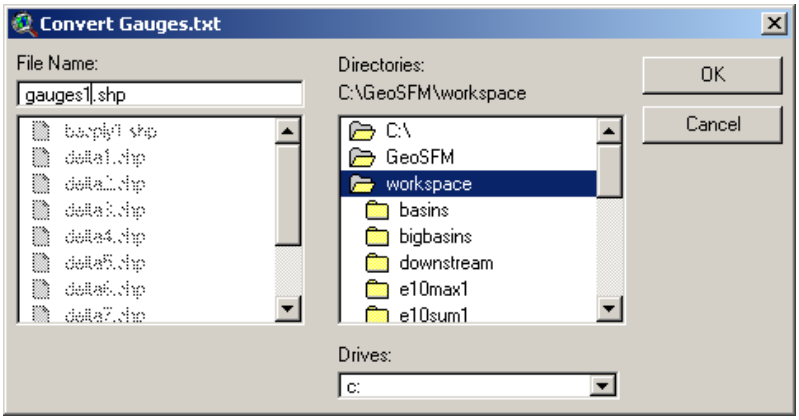
The table is displayed in the **View**.



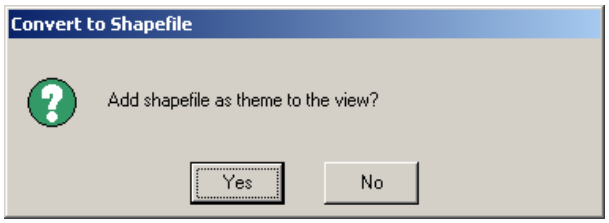
Next, convert the .txt file to a shapefile. From the **Theme** menu select **Convert to Shapefile**.



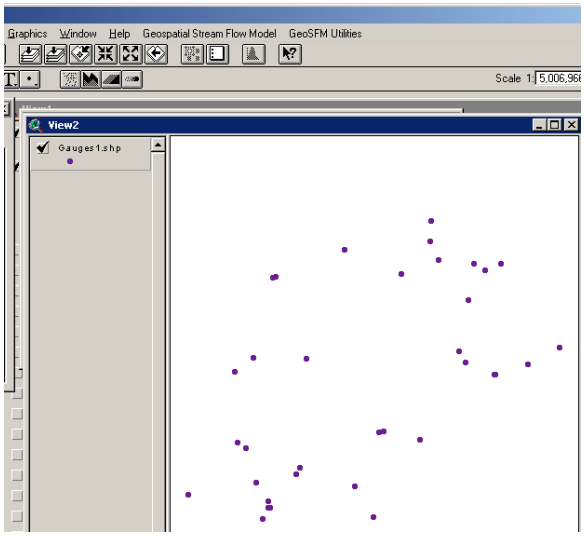
The **Convert Gauges.txt** dialog box opens. Navigate to your working directory. Name your file under **File Name** and click **OK**.



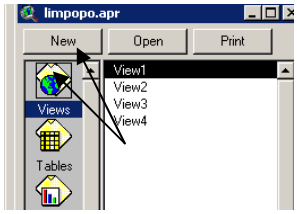
When prompted to "**Add shapefile as theme to the view?"** Click **Yes.**



Displayed shapefile below.



Next, you are going to open a new **View** by clicking on **View** and the **New** button from the project window. You need to open a new **View** because you are going to change the projection of the data to Lambert Azimuth Equal Area projection to match the projection used for all the data in the GeoSFM.



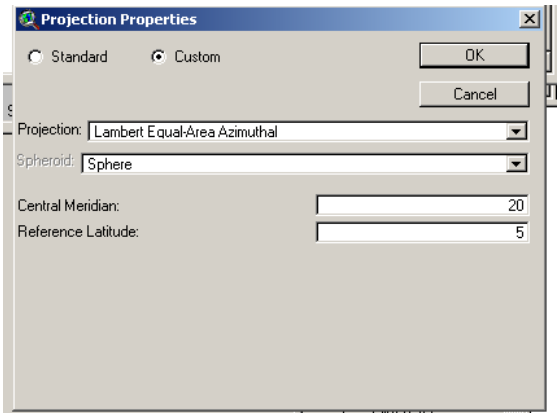
From the **View** drop down menu select **View Properties**. Click on the **Projection** button to open the **Projection Properties** dialog box. Select **Custom** radio button. Populate-

Projection: Lambert Equal Area Azimuthal

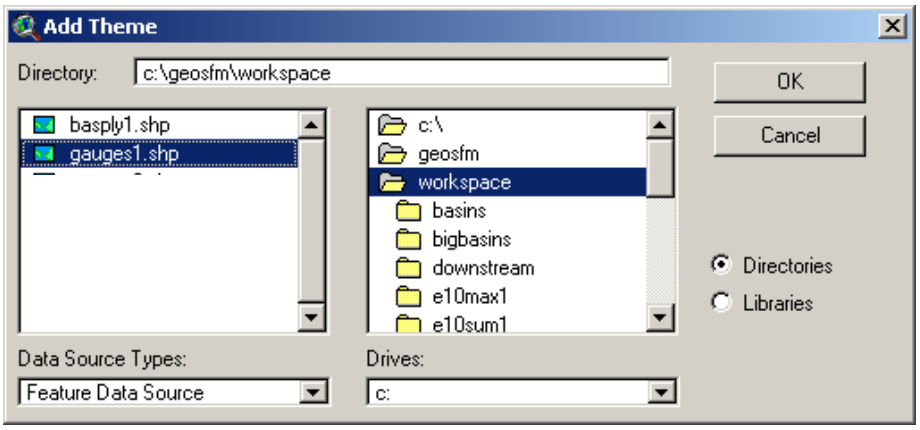
Central Meridian: 20 (parameter for African continent)

Reference Latitude: 5 (parameter for African continent)

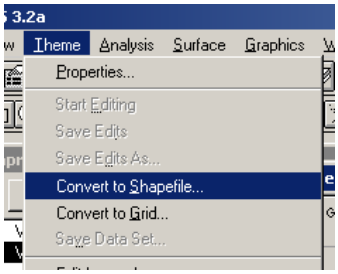
All other fields will default. Click **OK** in both dialog boxes.



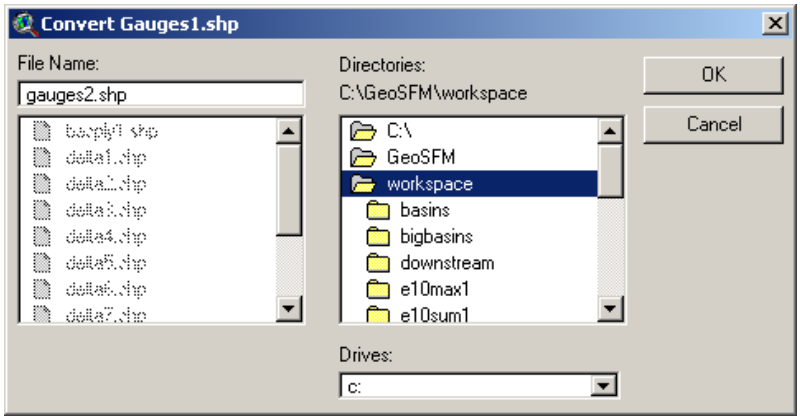
Now add the shapefile into the new View by clicking on the Add Theme button  . From the **Add Theme** dialog box navigate to the **c:\geosfm\workspace directory**. In **List of File Type** make sure **Feature Data Source** is selected. Click to highlight **gauges1.shp** and click **OK**.



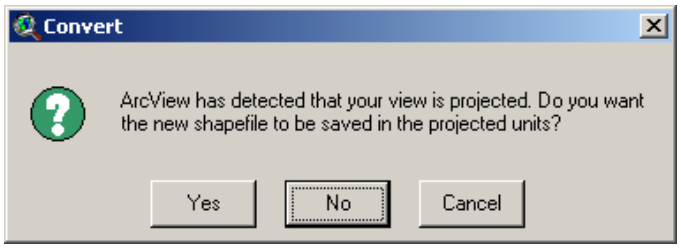
Next, you will convert gauges1.shp to a new shapefile which will contain the new projected properties. From the **Theme** menu select **Convert to Shapefile**.



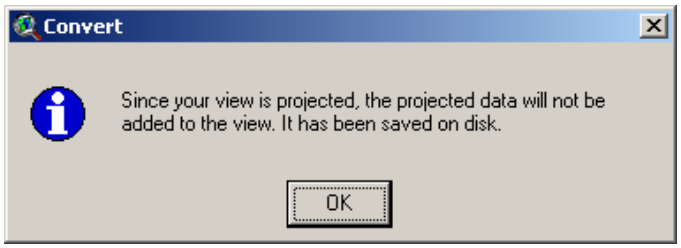
Navigate to your working directory and name your new shapefile. Click **OK**.



The **Convert** dialog box opens asking **Do you want the shapefile to be saved in the projected units?** Click **Yes**.



An informational dialog box opens to inform you that the projected shapefile will not be added to the view. Click **OK**.



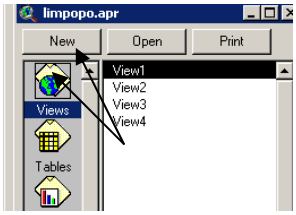
You have created a point theme shapefile, which can be used when working in the **GIS TOOL:Pick Grid Values at Point** in the **GeoSFM Utilities** menu.

6.3 Interpolate Station Data to Grid

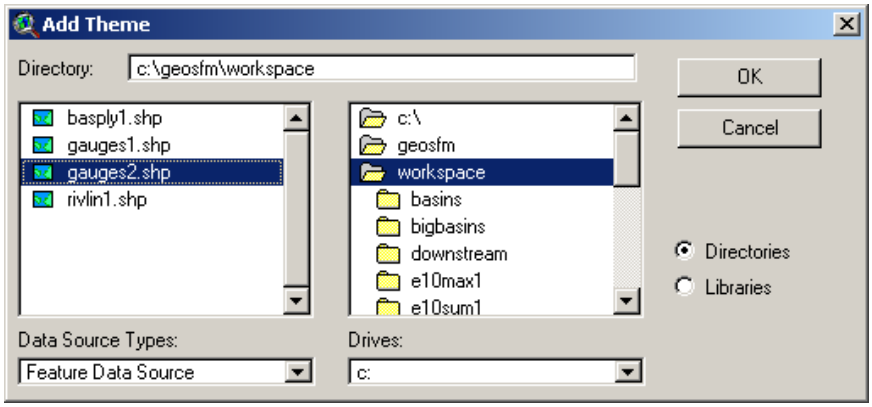
The third **GIS Tools** menu item listed in the **GeoSFM Utilities** is **Interpolate Station Data to Grid**. This function is used when there are areas over which spatially distributed precipitation data is not available. Data interpolation routines convert station readings into a continuous surface. You will need both rainfall and evaporation tables. These tables need to contain station Ids along with daily amounts of rainfall/evaporation for each station. If you need to create these tables the instructions will follow in the next section. For this exercise, tables are provided in the **c:\geosfm\samples\stations** directory.

Create two new folders in your working directory labelled **Raindata** and **Evapdata** this will help in managing the data files that will be created from this process.

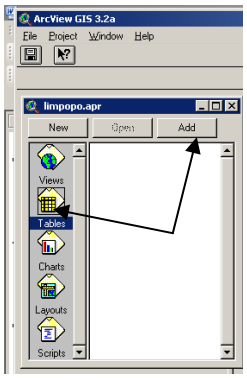
Start by opening a new **View** by clicking on **View** and the **New** button from the project window.



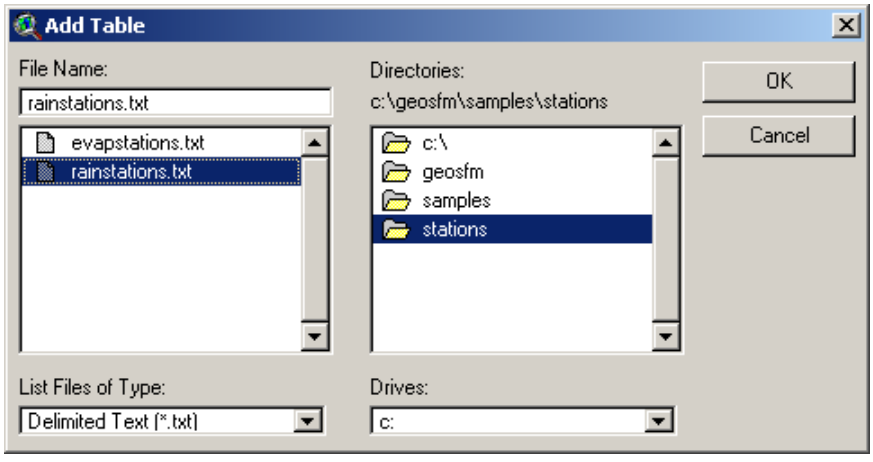
Add the **gauges2.shp** file to the **View** using **Add Theme button**  from the **View** menu. Change the **Data Source Types** to **Feature Data Source**. Click on the **gauges2.shp** file from your working directory and click **OK** to add to the **View**.



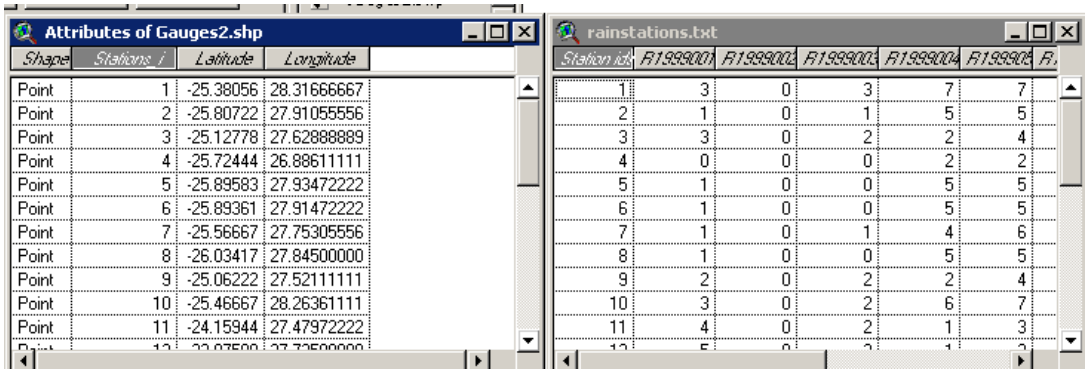
Next, add the rain station table to ArcView by selecting **Tables** and clicking on the **Add** button seen below.



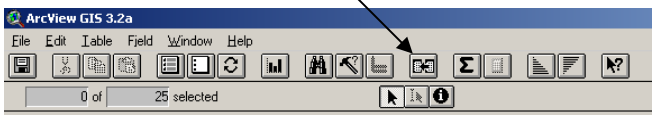
Navigate to your working directory, change **List Files of Type:** Delimited Text (*.txt) and select your .txt file in this example **rainstations.txt**. Click **OK**.



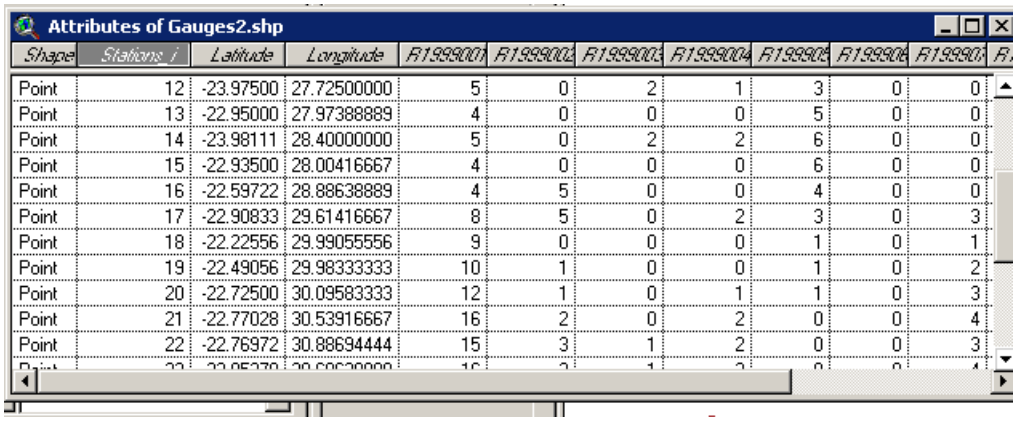
Next, you will join the new rainstations.txt to the attribute table of the gauges2.shp file. Arrange both tables as seen below. Click/highlight the common column headers in both tables –**Station_i** in **Attributes of gauges2.shp** and **Station ids** in **rainstations.txt**. Make the destination table active (adding fields to Attributes of gauges2.shp).



Click on the **Join** icon, the new columns are added to the **Attributes of gauges2** table.

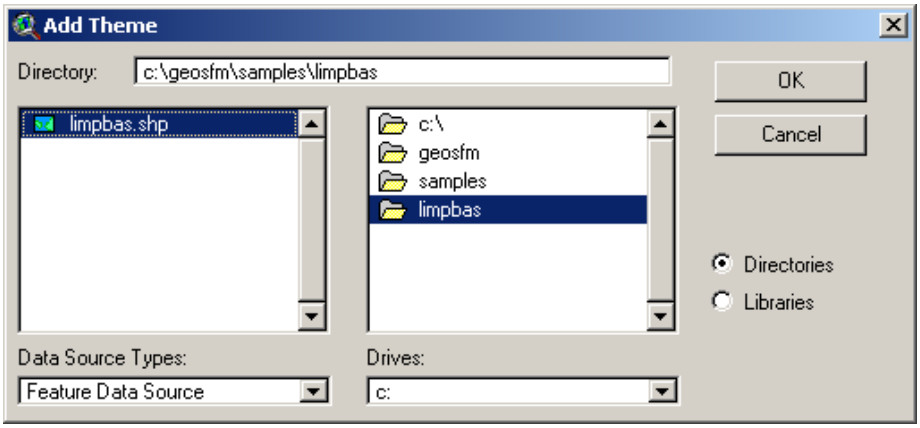


Below new rain data fields added to attribute table.

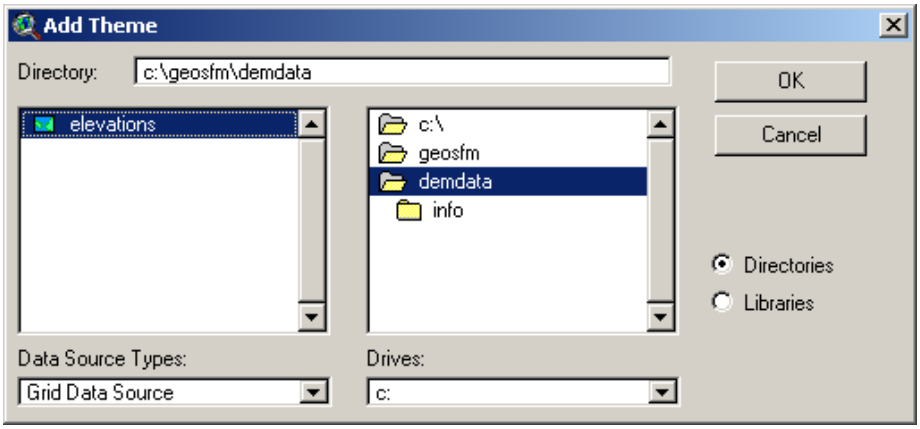


Repeat this process for the evaporation data. You can choose to process rainfall and evaporation data separately or at the same time. The attribute table will need all the necessary data for the selection made during processing.

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**.



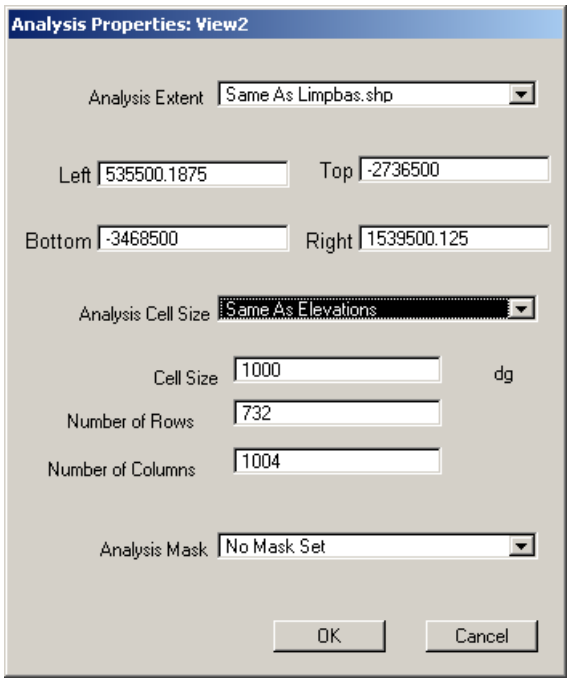
Next, add the **elevations** grid to the **View** using the **Add Theme** button. 
Change the **Data Source Types** to **Grid Data Source**. Click on **elevations** from the **c:\GeoSFM\demdata** directory and click **OK** to add the DEM to the **View**.



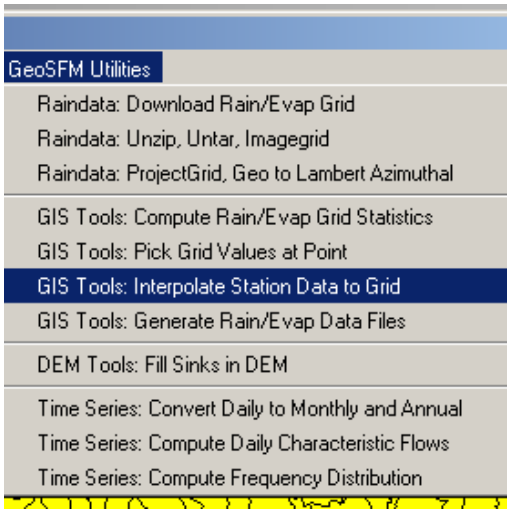
Next, set the analysis environment from the **Analysis menu** by selecting **Properties**. Change the **Analysis Extent** to **Same As Limpbas.shp** and the **Analysis Cell Size**

to **Same as Elevations**. All other parameters will adjust themselves.

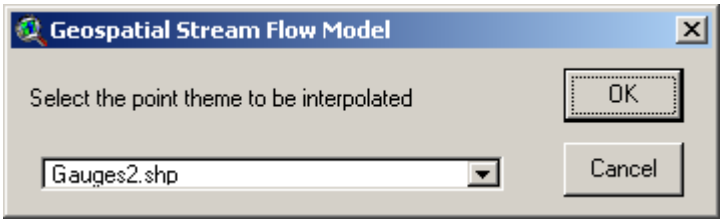
Click **OK.**



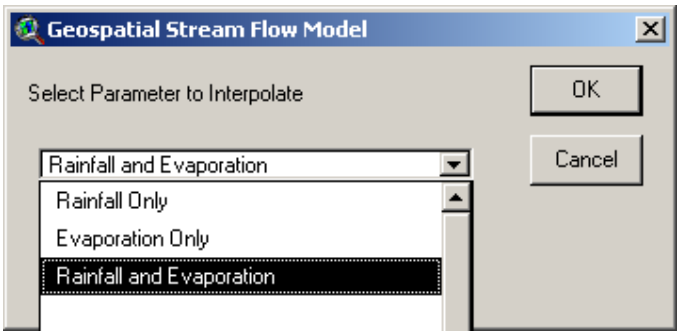
Now you are ready to start the process. From the **GeoSFM Utilities** menu select **GIS Tools: Interpolate Station Data to Grid**.



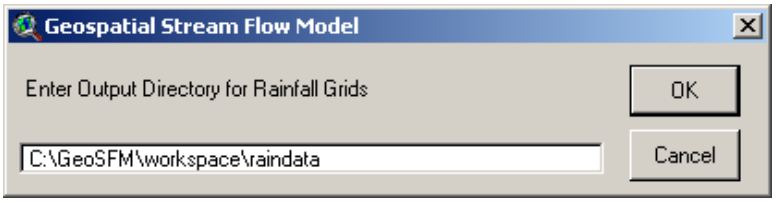
Select the point theme to be interpolated from the drop down list. In this example select **Gauges2.shp**. Click **OK**.



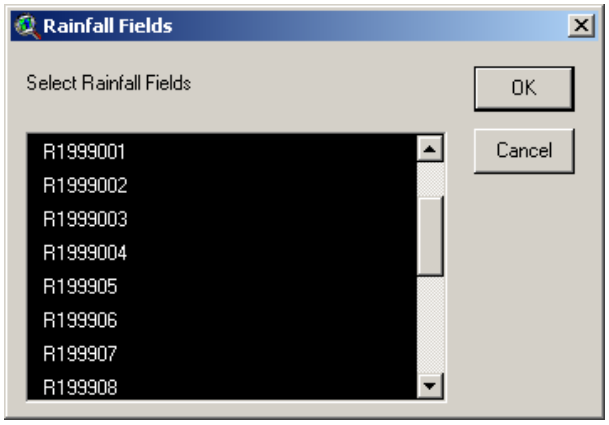
Select Parameter to Interpolate - in this example select **Rainfall and Evaporation**. Click **OK**. If the evapotranspiration data that is being used are already provided as grids they do not need to be interpolated in this exercise. If the evapotranspiration data was received from the rain stations or is in a text type file you will need to interpolate the data.



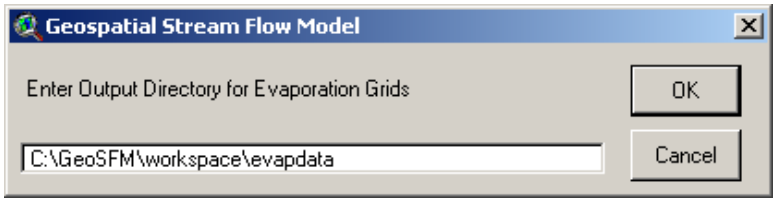
Enter the output directory for rainfall grids –in new raindata folder created in **c:\GeoSFM\workspace\raindata**. Click **OK**.



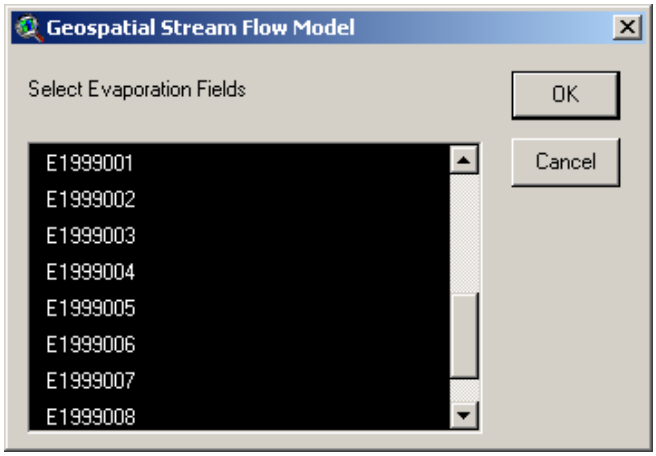
In the **Rainfall Fields** dialog box you are prompted to select the fields that contain the rainfall values. Select the ten days from the list, **R1999001** through **R1999010**. You may select any number of days that you wish, in this exercise you will select all ten listed. Click **OK**.



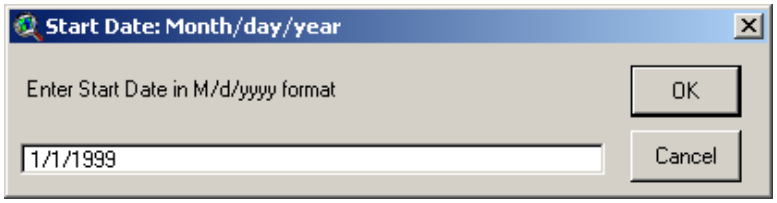
Enter the output directory for evaporation grids –in new evapdata folder created in **c:\GeoSFM\workspace\evapdata**. Click **OK**.



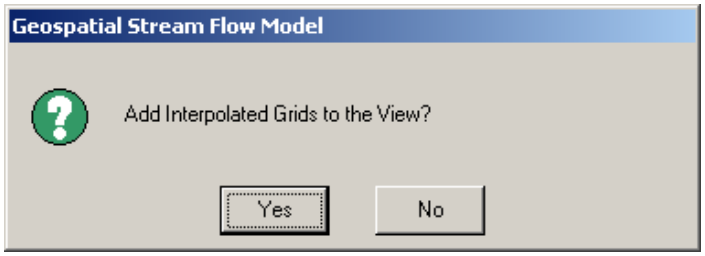
In the **Evaporation Fields** dialog box you are prompted to select the fields that contain the evaporation values. Select the ten days from the list, **E1999001** through **E1999010**. You may select any number of days that you wish, in this exercise you will select all ten listed. Click **OK**.



In the **Start Date** dialog box change the default Start Date to **1/1/1999**, this will match the date of the first rainfall/evap fields selected above. The first field selected **1999001** is equal to **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 –defaults **Stations_i**

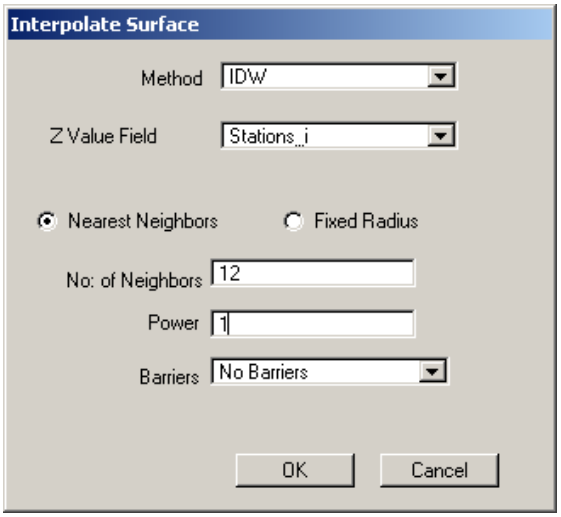
Nearest Neighbors or Fixed Radius –select **Nearest Neighbors**

No: of Neighbors –defaults **12**

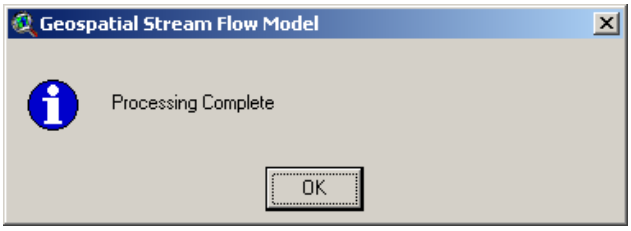
Power –change the **Power** field to **1** to create a smoother surface, defaults 2

Barriers –defaults **No Barriers**

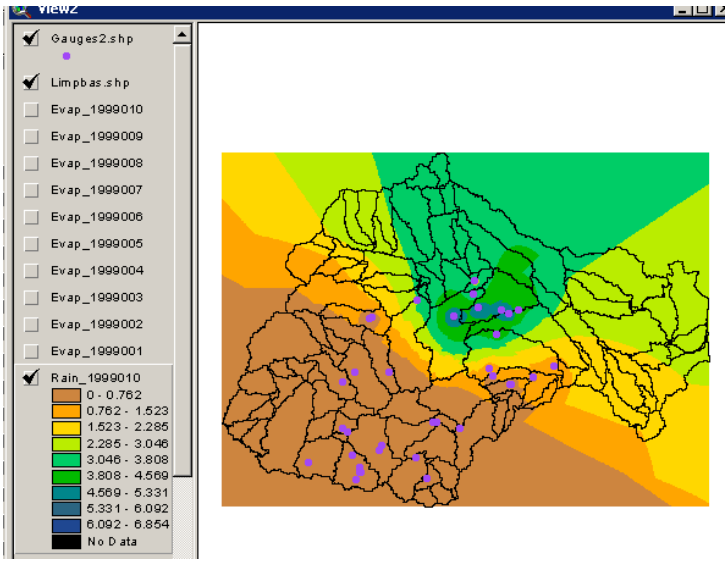
Once parameters are selected click **OK**.



It will take a few seconds to process the rainfall/evap grids. You will be notified when the processing is complete. Click **OK**.



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

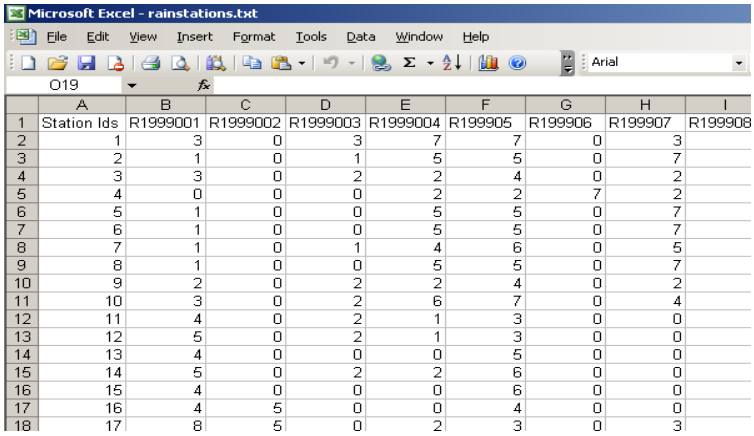


You have now completed all of the **GIS Tool** functions within the **GeoSFM Utilities.**

6.3.1 Creating Rain/Evap Interpolation Tables

To interpolate the rainfall and evaporation data you need to arrange your rainfall and evaporation data in a format similar to the table below. The first column header is the Station IDs with all ID numbers completing column A. The rest of the column header data are a time series of days formatted **R**yyyyddd (rain) and **E**yyyyddd (evap). The values for each cell are rainfall or evaporation amounts captured at each station for the appropriate day. Make sure the column headings are unique "R" for rain and "E" for evap because these tables will be joined to an attribute table in order to complete the interpolation of station data to grids.

rainstation.txt example



	A	B	C	D	E	F	G	H	I
1	Station Ids	R1999001	R1999002	R1999003	R1999004	R199905	R199906	R199907	R199908
2	1	3	0	3	7	7	0	3	
3	2	1	0	1	5	5	0	7	
4	3	3	0	2	2	4	0	2	
5	4	0	0	0	2	2	7	2	
6	5	1	0	0	5	5	0	7	
7	6	1	0	0	5	5	0	7	
8	7	1	0	1	4	6	0	5	
9	8	1	0	0	5	5	0	7	
10	9	2	0	2	2	4	0	2	
11	10	3	0	2	6	7	0	4	
12	11	4	0	2	1	3	0	0	
13	12	5	0	2	1	3	0	0	
14	13	4	0	0	0	5	0	0	
15	14	5	0	2	2	6	0	0	
16	15	4	0	0	0	6	0	0	
17	16	4	5	0	0	4	0	0	
18	17	8	5	0	2	3	0	3	

In this example ten days of data are used. The Excel spreadsheet is limited to 230 columns of data, if you are working with one year of data you will need to create two separate files. When you have the spreadsheet populated repeat the process for the evaporation data.

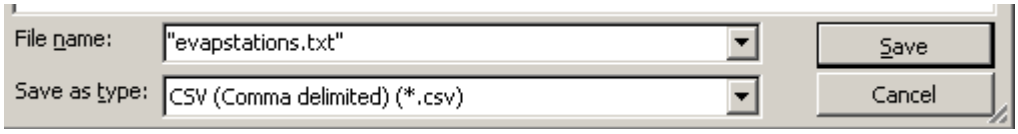
To save the tables select **File** and **Save As…**

File name: “rainstation.txt”

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

File name: “evapstation.txt”

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



You now have two tables that will be used in the **Interpolate Station Data to Grid** process found in the **GeoSFM Utilities**.

Users Manual for the Geospatial Stream Flow Model

Ex 7: Using GeoSFM Utilities –DEM Tools

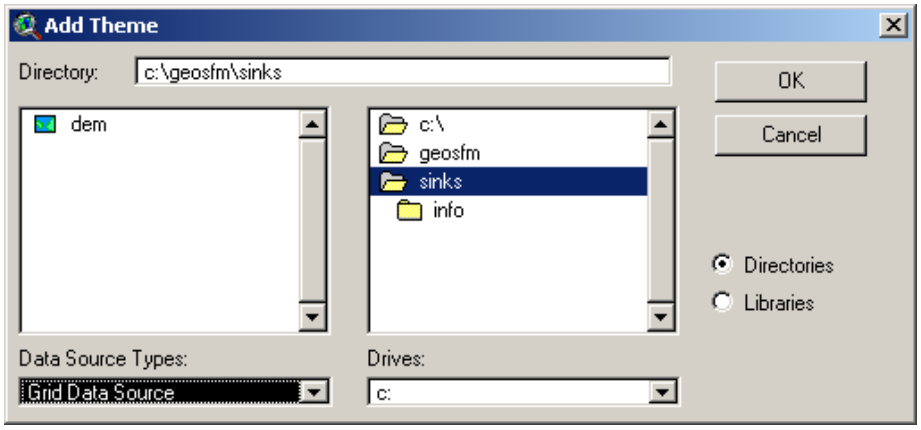
Contents:

7.1 Fill Sinks in DEM

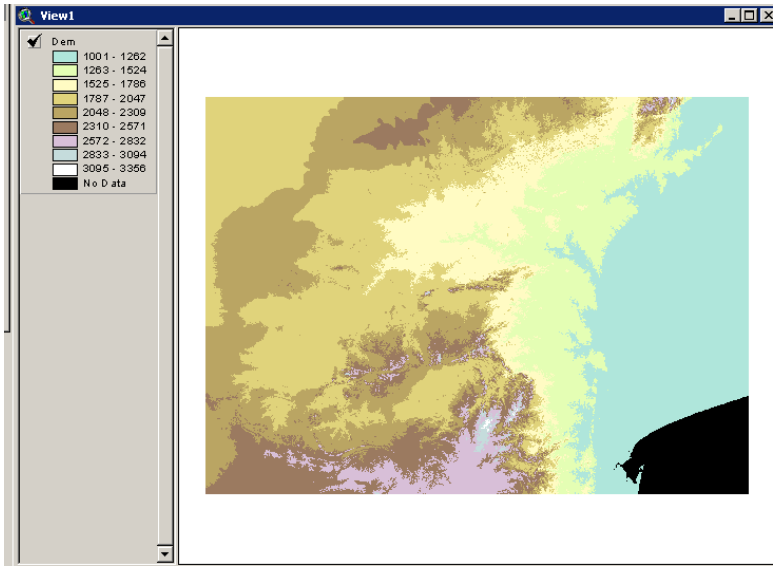
In exercise eight you will familiarize yourself with a **DEM Tool –Fill Sinks in DEM**. This process will fill all spurious sinks, while maintaining sinks that are natural occurrences in the landscape. This time-consuming process yields a DEM, which will properly transport water across its surface. This process is better done on a smaller basin region than for an entire continent. If sourcing the elevation data from HYDRO1K DEM data set, developed at the U.S. Geological Survey's Center for EROS, than this process is not needed. HYDRO1K is a hydrological corrected DEM which implies that it is devoid of spurious pits that interrupt hydraulic connectivity over the land surface.

You will need to be working with a DEM that is in the correct projection, and also completed the calculations needed to produce a corrected elevation grid. (*see Users Manual section 1.2.1*)

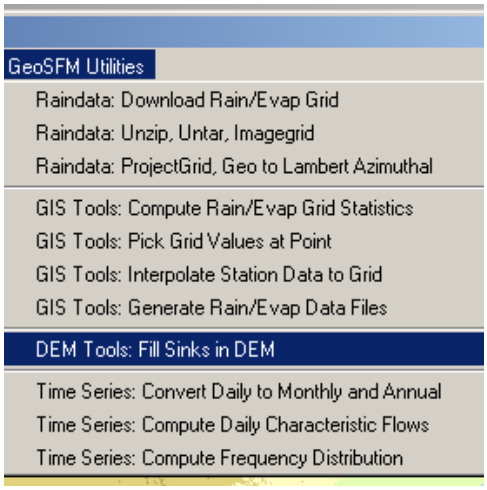
To begin the process to fill sinks within a DEM add the **DEM** grid to the **View** using the **Add Theme** button.  Change the **Data Source Types** to **Grid Data Source**. Click on **dem** from the **c:\GeoSFM\sinks** directory and click **OK** to add the DEM to the **View**.



Below the **Dem** is added to the **View**.

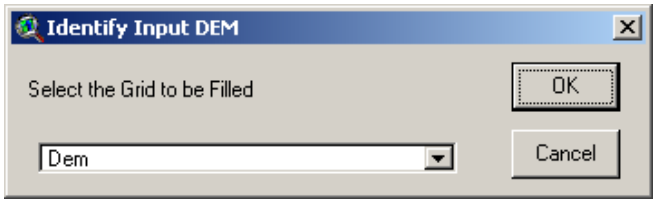


The only menu item under **DEM Tools** is **Fill Sinks in DEM**. From the **GeoSFM Utilities** menu select **DEM Tools: Fill Sinks in DEM**.

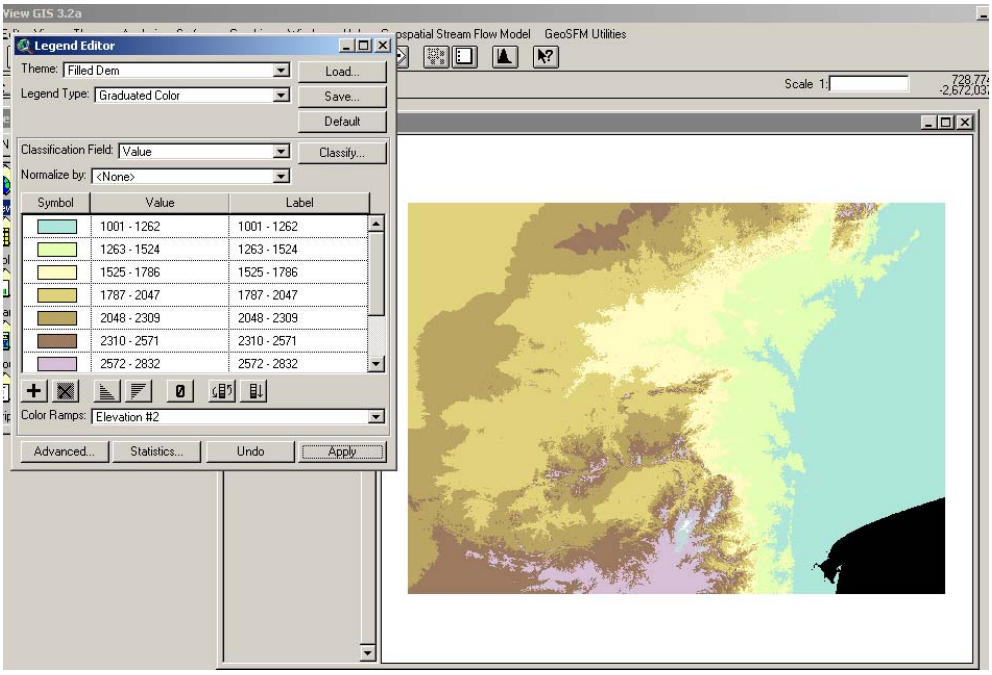


Confirm your working directory. Click **OK**.

Next, the **Identify Input DEM** dialog box is displayed. For **Select the Grid to be Filled** select **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 wish to apply an elevations type legend to the theme. 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**.



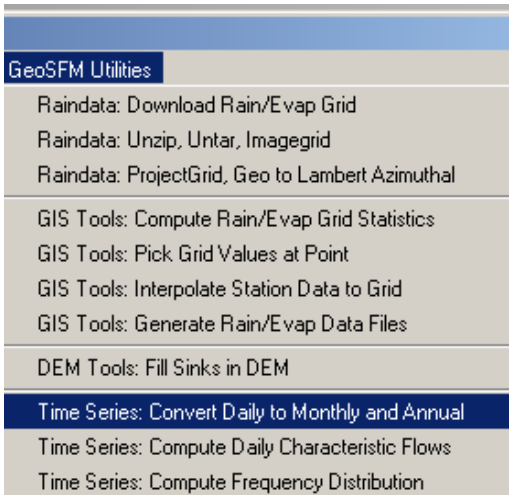
The **DEM** file in the Sinks folder is removed and a new **fill1** grid is added to your workspace directory. You are now ready to process elevation data in section 1.3 of this manual.

Users Manual for the Geospatial Stream Flow Model
Ex 8: Using GeoSFM Utilities –Time Series

Contents:

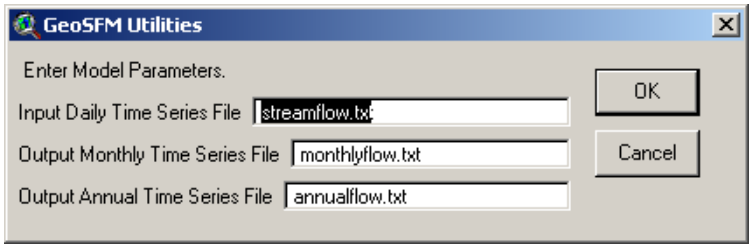
8.1 Convert Daily to Monthly and Annual

In exercise nine, you will look at the **Time Series** functions within the **GeoSFM Utilities** tool. 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 Annual**.

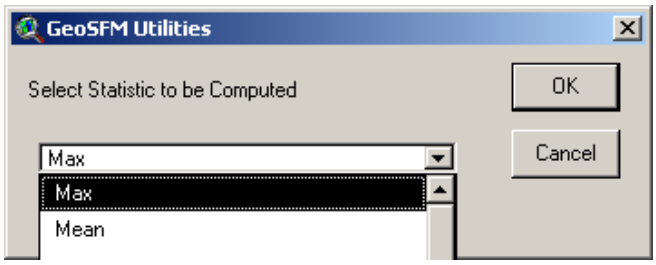


Confirm your working directory. Click **OK**.

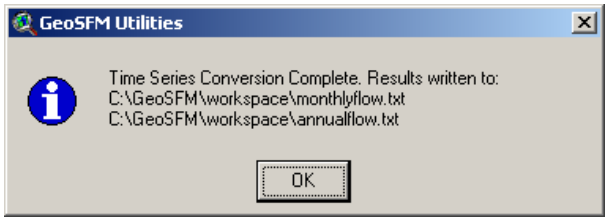
From the **GeoSFM Utilities** dialog box **Enter Model Parameters**. Input file should default to **streamflow.txt** and outputs are **monthlyflow.txt** and **annualflow.txt**. Click **OK**.



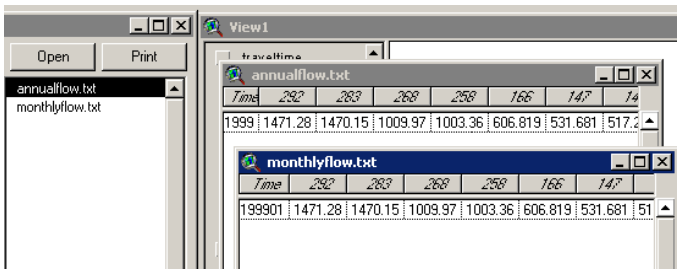
Next, **Select Statistic to be Computed** in this example **Max** was selected. Click **OK**.



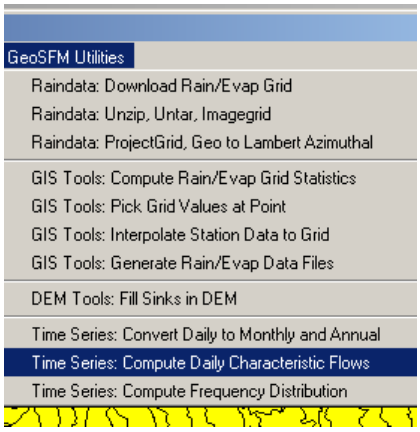
After a few moments of processing the **Time Series Conversion Complete** dialog box will display as seen below. The program will produce two files containing the monthly and annual stream flow amounts for each sub-basin. The message below contains the name and location of the output files. Click **OK**.



Two new tables are shown below the stream flow data was for a few days in Jan.1999.

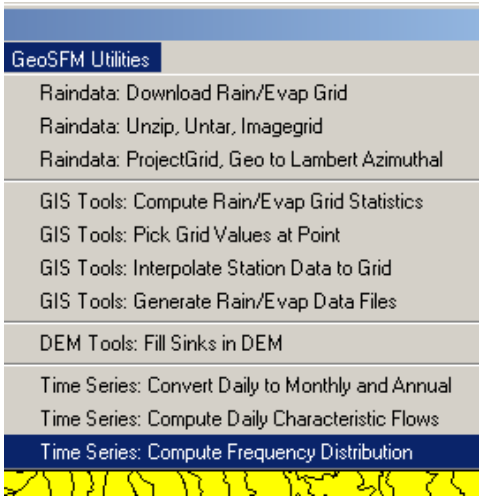


8.2 Compute Daily Characteristic Flows -under development



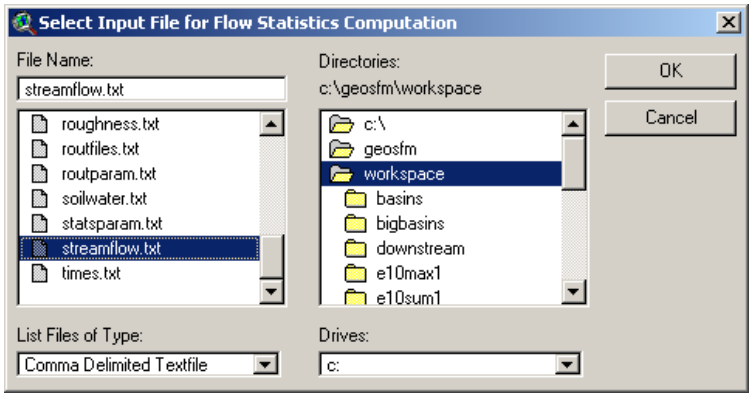
8.3 Compute Frequency Distribution

The last selection in exercise nine is **Compute Frequency Distribution**. From the **GeoSFM Utilities** menu select **Time Series: Compute Frequency Distribution**.



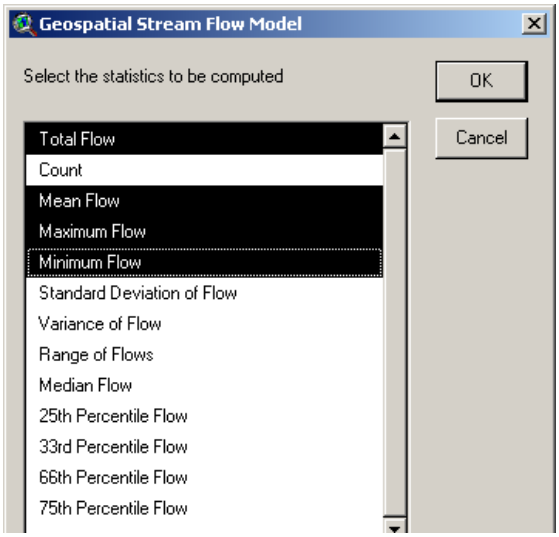
Confirm your working directory. Click **OK**.

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**.

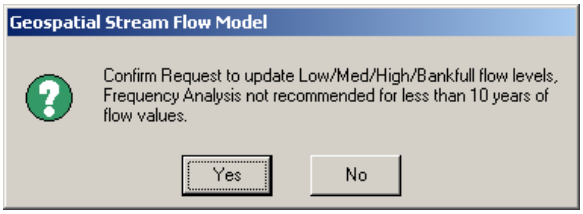


When the **Basin Theme** dialog box is displayed select your basin coverage/grid theme. In this exercise the basin coverage selected is **basply.shp**. Click **OK**.

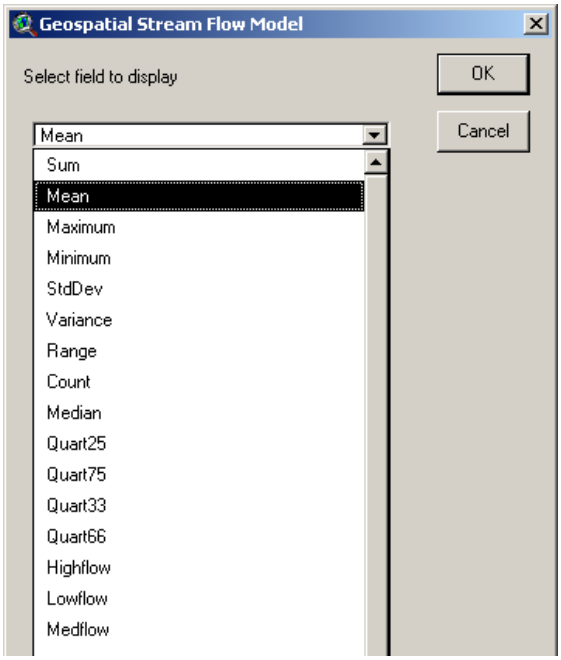
Next, the **Select the statistics to be computed** dialog box opens. In this example **Total Flow, Mean Flow, Maximum Flow and Minimum Flow** have been selected. More than one statistical type can be selected at a time. Notice the list of options. Click **OK**.



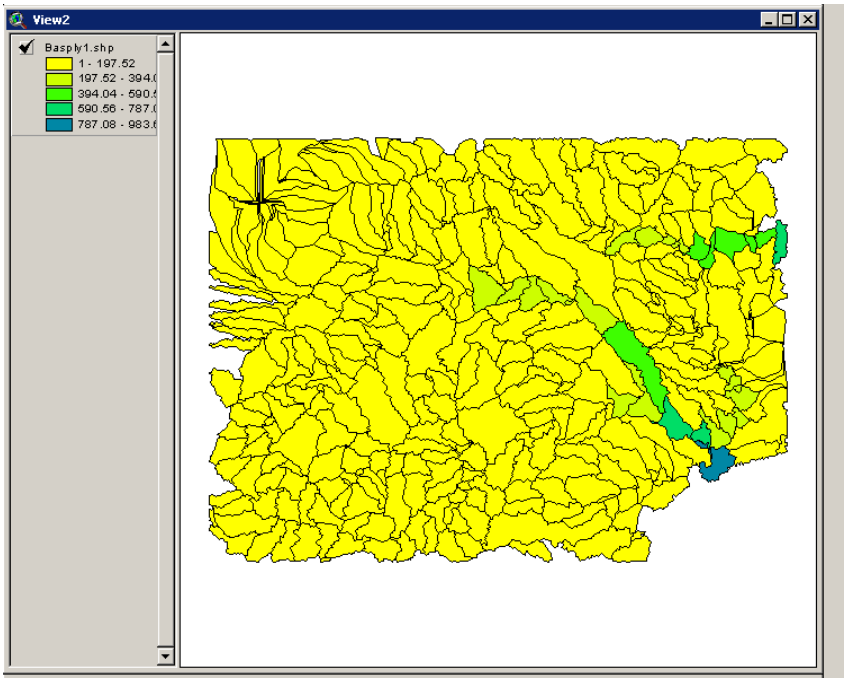
Confirm Request to update Low / Med / High Bankfull flow levels is displayed. Select **Yes**. The analysis is not recommended for less than ten years of data, for this exercise you have been working with ten days of data. The objective is to familiarize you with the functionality available in the GeoSFM Utilities tool.



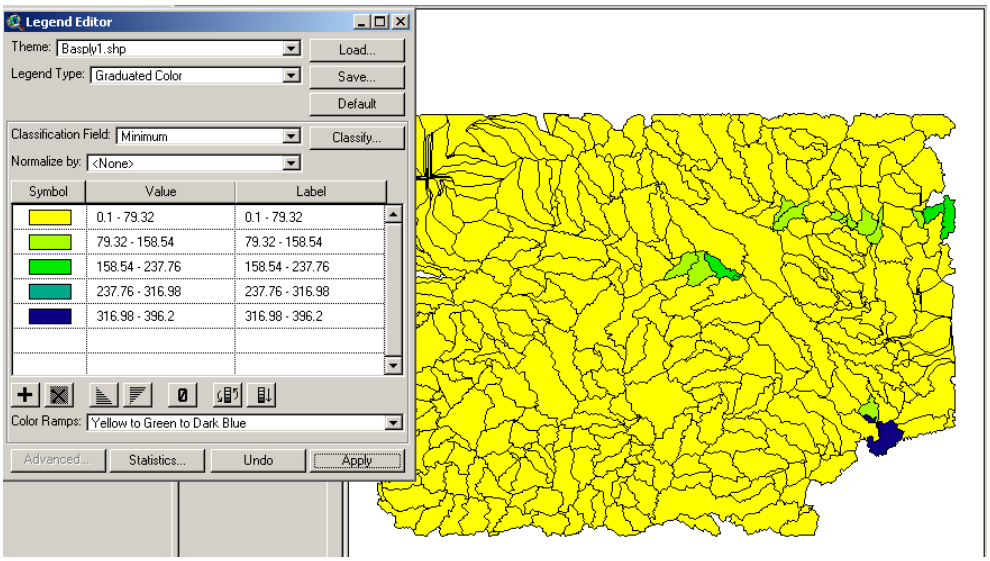
The next dialog box displayed is the **Select field to display**; in this example **Mean** has been selected. Click **OK**. Again, notice the list of options available for displaying the results. The new calculations will be displayed in the **basply.shp** overwriting what is already displayed.



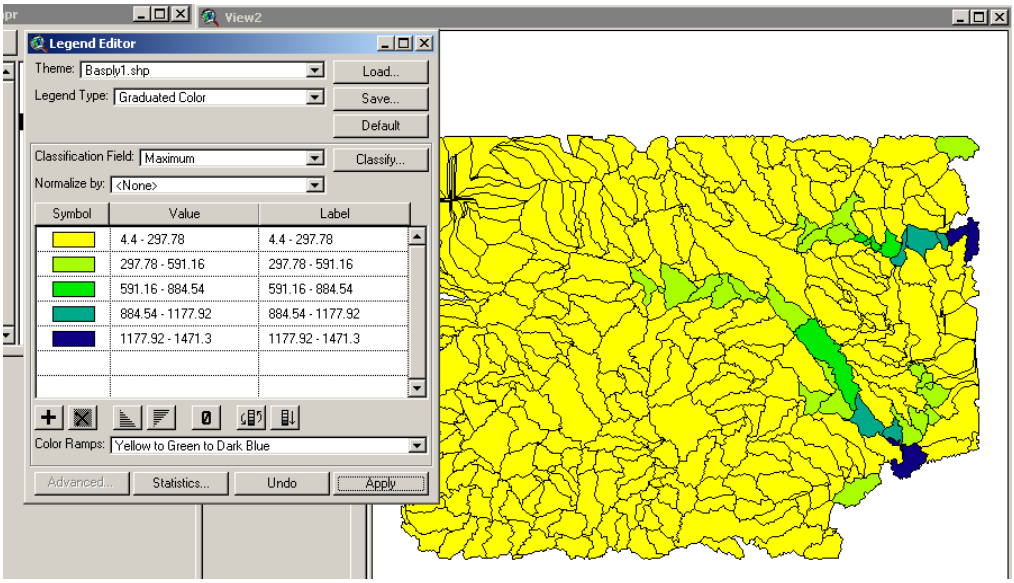
Below is the updated basply.shp with the **Mean** value displayed.



You can change the value displayed from the **Theme** menu and selecting **Legend Editor**. Then changing the **Classification Field** and clicking on the **Apply** button. The **Color Ramp** selected is **Yellow to Green to 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 between the three different classifications.

What are the changes in the sub-basins in these three different scenarios?

You now have completed the exercises for the Time Series selections within the GeoSFM Utilities tool.