



US Army Corps  
of Engineers®  
Engineer Research and  
Development Center

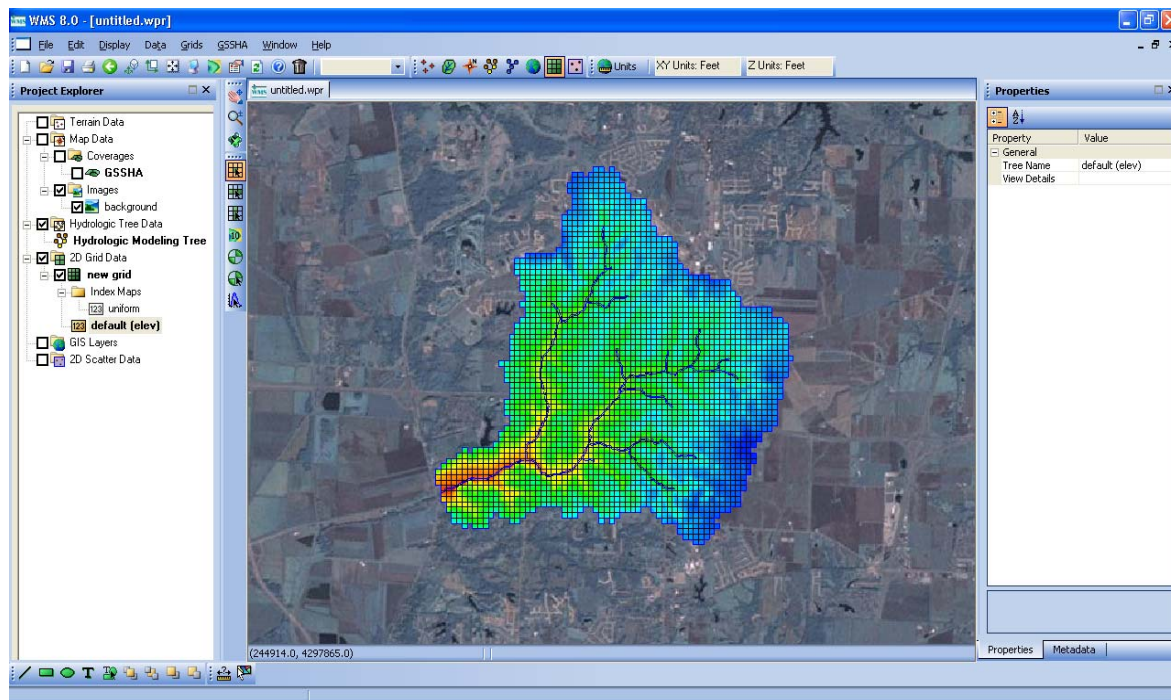
System-Wide Water Resources Program

## GSSHA™ Tutorials

For use with GSSHA™ version 3.0 and WMS version 8.0

Aaron R. Byrd, Ryan Billen, Darla McVan, Mark Jourdan, Ph.D.

April 2008





# **GSSHA™ Tutorials**

For use with GSSHA™ version 3.0 and WMS version 8.0

Aaron R. Byrd, Ryan Billen, Darla McVan, Mark Jourdan, Ph.D.

*Coastal and Hydraulics Laboratory  
U.S. Army Engineer Research and Development Center  
3909 Halls Ferry Rd  
Vicksburg, MS 39180*

Final report

Approved for public release; distribution is unlimited.

Prepared for U.S. Army Corps of Engineers  
Washington, DC 20314-1000

Under Work Unit 56C24C

**Abstract:** This documents contains tutorials on how to set up hydrologic simulations for use with Gridded Surface Subsurface Hydrologic Analysis™ using the Watershed Modeling System. The tutorials cover delineating basins, running and visualizing overland flow, simulating and visualizing stream flow, simulating infiltration, setting up long-term simulations, calibrating simulations, and a brief overview of using Microsoft® Excel® to format data needed.

**DISCLAIMER:** The contents of this report are not to be used for advertising, publication, or promotional purposes. Citation of trade names does not constitute an official endorsement or approval of the use of such commercial products. All product names and trademarks cited are the property of their respective owners. The findings of this report are not to be construed as an official Department of the Army position unless so designated by other authorized documents.

**DESTROY THIS REPORT WHEN NO LONGER NEEDED. DO NOT RETURN IT TO THE ORIGINATOR.**

# Contents

<b>Figures and Tables</b> .....	<b>iv</b>
<b>Preface</b> .....	<b>vi</b>
<b>Unit Conversion Factors</b> .....	<b>viii</b>
<b>1 Initial Overland Flow Model Setup</b> .....	<b>1</b>
Initial Basin Setup .....	1
Initial Grid Setup.....	3
Job Control Setup .....	6
Uniform Index Map Setup .....	6
Roughness Table Setup .....	9
Setting the Uniform Precipitation .....	10
Save the GSSHA™ Model.....	11
Running the Model.....	13
<b>2 Visualizing Overland Flow Results</b> .....	<b>14</b>
Working with Results.....	14
Creating a Movie .....	15
<b>3 Fixing Digital Dams</b> .....	<b>18</b>
Working with Digital Dams.....	18
Manually Adjusting Cell Elevations.....	20
Using Cleandam™ to Fix Digital Dams .....	23
Using Topaz to Fix Digital Dams.....	24
Comparing Topaz and Cleandam™ Results .....	25
<b>4 Using the Mapping Table and GIS Data</b> .....	<b>32</b>
Using Land Use Data.....	32
<b>5 Stream Flow</b> .....	<b>36</b>
Creating GSSHA™ Stream Arcs .....	36
Smoothing the Stream Arcs.....	39
<b>6 Visualizing Stream Data</b> .....	<b>43</b>
<b>7 Break-point Cross Sections</b> .....	<b>46</b>
Adjusting the Stream Course.....	48
Adding an Embankment .....	50
Adding Structures.....	54
<b>8 Infiltration</b> .....	<b>56</b>
Index Map Setup .....	56
<i>Importing the Shapefile as a GIS Layer</i> .....	56

*Cleaning up the Soil Type Coverage*..... 58

*Setting up the index map*..... 60

Visualization ..... 61

**9 Long-Term Simulations ..... 65**

Precipitation ..... 65

*Using the Format Precip Macro*..... 65

Hydrometeorological Data ..... 66

*Using the Format Hmet Macro* ..... 66

Evapotranspiration ..... 67

**10 Using Microsoft® Excel® to format GSSHA™ data..... 69**

Splitting an Excel® column..... 69

*Note*..... 70

Using the AutoFilter ..... 70

Find And Replace ..... 71

General Information ..... 72

**11 Manual Calibration..... 74**

**Report Documentation Page**

# Figures and Tables

## Figures

Figure 1. DEM contours of the area. ....	2
Figure 2. A close up of the outlet area shown in Figure 1. ....	2
Figure 3. The watershed boundary after delineation by WMS and TOPAZ. ....	3
Figure 4. The Create Grid dialog in WMS. ....	4
Figure 5. The regular grid created for the GSSHA™ simulations. ....	5
Figure 6. The Drainage coverage changed type to the GSSHA™ coverage. ....	5
Figure 7. The GSSHA™ Job Control dialog. ....	6
Figure 8. The GSSHA™ index map dialog. ....	7
Figure 9. The Data Calculator is used to create index maps containing uniform values. ....	8
Figure 10. Notice the new index map is placed in the index map folder. ....	9
Figure 11. The Mapping Table editor. ....	10
Figure 12. The GSSHA™ Precipitation dialog. ....	11
Figure 13. The Save GSSHA™ Project File dialog. ....	12
Figure 14. The digital dams are marked. ....	18
Figure 15. The Display Options dialog controls what is shown. ....	19
Figure 16. The contour options dialog controls how the data sets are displayed. ....	19
Figure 17. 3D view of how cells are represented in GSSHA™. ....	20
Figure 18. The 2D grid, contoured with color fill contours and the showing the digital dams. ....	21
Figure 19. Close up of cells to be adjusted for digital dams. ....	22
Figure 20. Elevations and flow directions for the adjusted grid. ....	23
Figure 21. Using the Data Calculator to look at differences in the original elevation and Cleandam™ derived elevation data sets. ....	26
Figure 22. Using the Data Calculator to look at the differences between the original elevation and the TOPAZ derived data sets. ....	27
Figure 23. Differences in modified elevation data sets. ....	28
Figure 24. A histogram of changes made by Cleandam™. ....	28
Figure 25. A histogram of changes made by TOPAZ. ....	29
Figure 26. Results of the TOPAZ data set. ....	30
Figure 27. The results of Cleandam™ data set. Only one digital dam remains. ....	30
Figure 28. Creating a new coverage in the data tree. ....	33
Figure 29. Stream Link numbers. ....	37
Figure 30. Stream link numbers. ....	40
Figure 31. Initial soil types in the watershed. ....	59
Figure 32. Changing a soils type mapping. ....	60

Figure 33. Digital dam next to an embankment. .... 63  
Figure 34. The GSSHA™ observations dialog..... 75

**Tables**

Table 1. GSSHA™/WMS typical file extensions..... 12  
Table 2. New elevation values for the cells shown in Figure 19. .... 22  
Table 3. USGS Land Use codes. [ref]..... 33  
Table 4. Roughness values corresponding to Land Use ..... 35  
Table 5. Initial stream shape and roughness parameters for all streams..... 38  
Table 6. Stream shape and roughness values for individual streams..... 38  
Table 7. Culvert parameters. .... 55  
Table 8. Soils type classifications. .... 57  
Table 9. Soil properties. .... 61  
Table 10. Values for the Evapotranspiration process. .... 67



## Preface

This document and the software GSSHA™ are products of the Watershed Systems Group, Hydrologic Systems Branch, Coastal and Hydraulics Laboratory, Engineer Research and Development Center. For more information about GSSHA™ contact:

Barbara Parsons  
Hydrologic Systems Branch  
Coastal and Hydraulics Laboratory  
Engineer Research Development Center  
3909 Halls Ferry Rd.  
Vicksburg, MS, 39180

[Barbara.A.Parsons@erdc.usace.army.mil](mailto:Barbara.A.Parsons@erdc.usace.army.mil)

<http://chl.erdc.usace.army.mil/GSSHA>

**Disclaimer:** GSSHA™ is a reformulation and enhancement of the hydrologic model CASC2D. The CASC2D hydrologic model is Copyright 1995, 1996, 1997, 1998 by Fred L. Ogden, 1995 by P.Y. Julien and B. Saghafian. No part of this documentation may be reproduced without complete citation. The GSSHA™ code is continuously being improved. Changes in the source code and input/output requirements of GSSHA™ may be made by the authors at any time, without notice. No claims are made regarding the suitability of GSSHA™ for any purpose. The model GSSHA™ is written for research and educational purposes. Use GSSHA™ at your own risk. GSSHA and Gridded Surface Subsurface Hydrologic Analysis is a trademark of the U.S. Army Corps of Engineers.

Most of the images were produced by the Watershed Modeling System (WMS), which is copyrighted by Brigham Young University, 2008 and used under license. For more information on WMS please refer to <http://www.aquaveo.com>.

Microsoft and Excel are registered trademarks of Microsoft Corporation in the United States and/or other countries. No endorsement is made by Microsoft Corporation of this work or as to the suitability of Excel® for any of the processes described in this document.

## Unit Conversion Factors




Multiply	By	To Obtain
Acres	4,046.873	square meters
acre-feet	1,233.5	cubic meters
Angstroms	0.1	nanometers
atmosphere (standard)	101.325	kilopascals
Bars	100	kilopascals
cubic feet	0.02831685	cubic meters
cubic inches	1.6387064 E-05	cubic meters
cubic yards	0.7645549	cubic meters
degrees Fahrenheit	$(F-32)/1.8$	degrees Celsius
Feet	0.3048	meters
gallons (U.S. liquid)	3.785412 E-03	cubic meters
Hectares	1.0 E+04	square meters
Inches	0.0254	meters
Knots	0.5144444	meters per second
miles (nautical)	1,852	meters
miles (U.S. statute)	1,609.347	meters
ounces (mass)	0.02834952	kilograms
square feet	0.09290304	square meters
square miles	2.589998 E+06	square meters
square yards	0.8361274	square meters
Yards	0.9144	Meters

# 1 Initial Overland Flow Model Setup

## Initial Basin Setup

To start a GSSHA™ model, you must have a digital elevation model and a boundary polygon for the watershed. For this tutorial we will use a DEM as the elevation model and also create the boundary polygon and the streams. WMS uses the DEM to interpolate cell elevations and the boundary polygon to select whether or not a cell is active (inside the basin) or inactive (outside the basin.) Unlike lumped parameter watershed models, the basin should not be subdivided into sub-basins. There should only be one basin boundary. The following steps are the same steps used to begin all watershed models in WMS. For more information, consult the WMS 8.0 Tutorials.

Open the DEM (select the Judys\_branch.hdr file in the DEM folder. At the 'Importing NED BIL File' dialog, click 'OK'. When the convert coordinates now dialog pops up, say yes and then turn on the 'Edit project coordinate system'. Convert from the Geographic NAD 83 to UTM NAD 83 and pick the UTM Zone 16.)

1. Click 'OK'.
2. Highlight Watershed icon. 
3. Select DEM | Run Compute TOPAZ Flow Data.
4. Locate the Outlet. The outlet location is shown in the following figures. Using the zoom tool  zoom into the area shown in the following figure and locate the outlet.
5. Create an outlet point using the 'create outlet' feature . Make sure this point is on the 'blue' stream line.

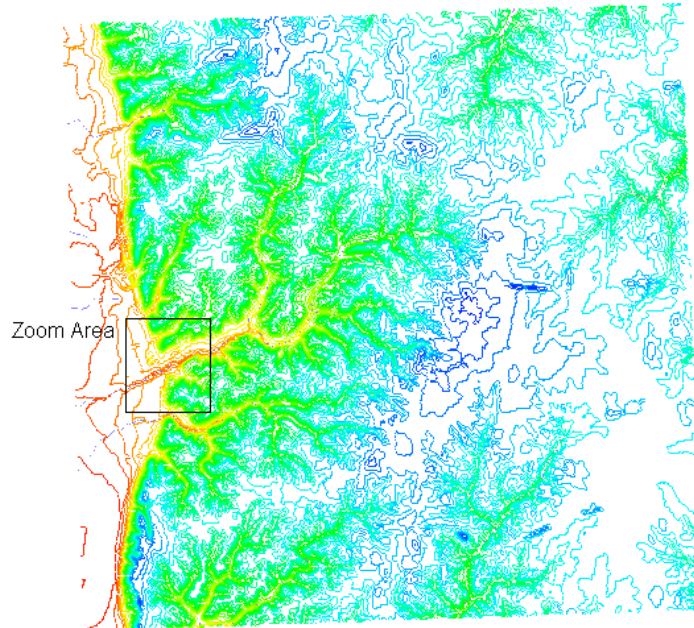


Figure 1. DEM contours of the area. The boxed area shows where the outlet should be located.

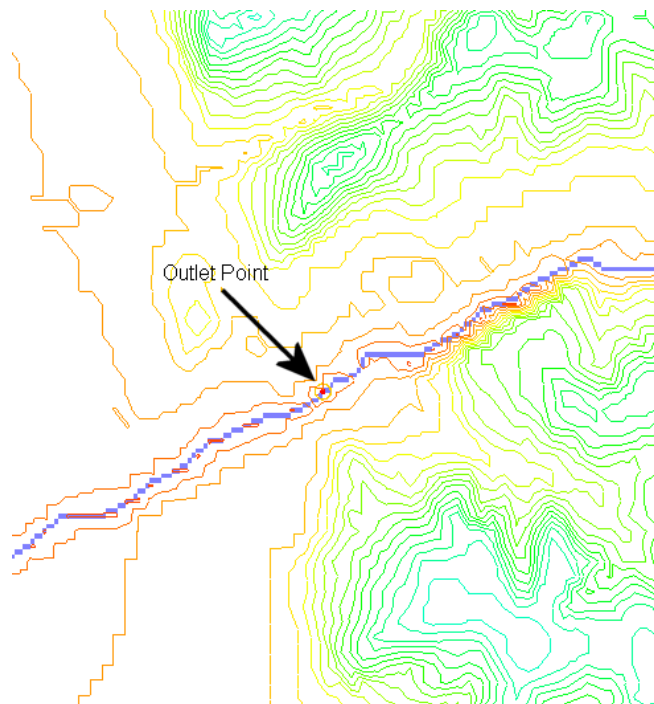


Figure 2. A close up of the outlet area shown in Figure 1. The outlet feature node must be located on a DEM stream cell.

6. Create the Basin and Stream Arcs using the Delineate Basin Wizard. This is found in the DEM file menu. Use a threshold value of 0.2. The finished basin should look like the following figure.

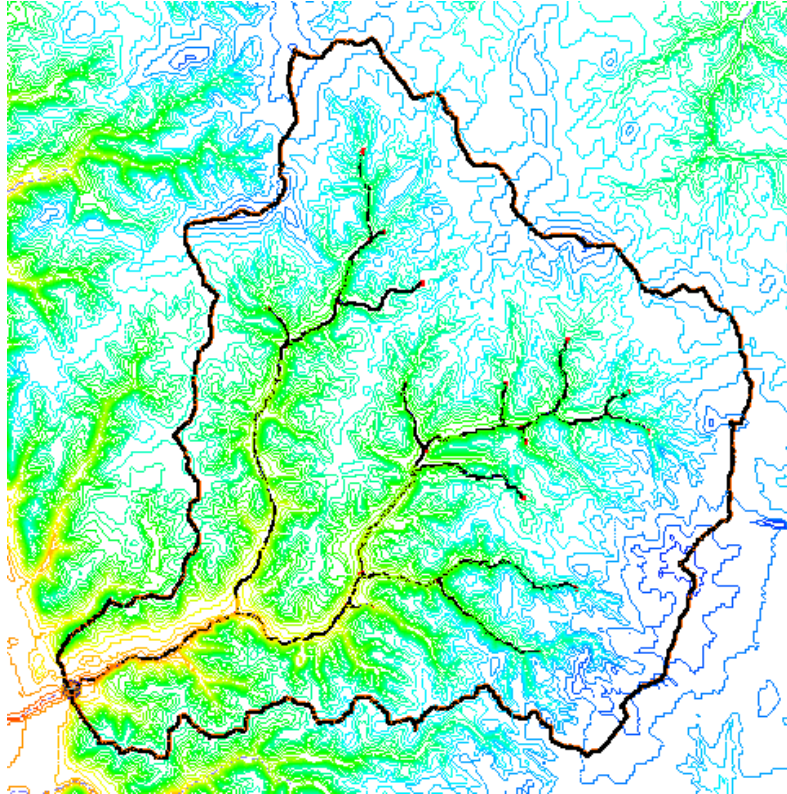


Figure 3. The watershed boundary after delineation by WMS and TOPAZ.



7. Save your project.

## Initial Grid Setup

The basic GSSHA™ model begins with the basin boundary in the Drainage coverage and a DEM. It is helpful to have the streams already set up as well, but not necessary. In this step you are essentially telling WMS to take the boundary polygon and the DEM and create a 2D grid that fits the boundary and has an elevation data set that is interpolated from the DEM. For more information on selecting appropriate cell sizes see the GSSHA™ Primer.

([http://chl.erdcl.usace.army.mil/software/GSSHA™/Primer\\_20/wf\\_njs.htm](http://chl.erdcl.usace.army.mil/software/GSSHA™/Primer_20/wf_njs.htm))

1. Begin from Basin Setup (You can load the delin\_basin.wpr file in the Finished\_tutorials\delin\_basin folder if you are starting from here.)

2. In the Map Module , using the Select Polygon tool , select the basin boundary polygon.
3. Select **Feature Objects | Create Grid...** (Select Yes for GSSHA™ grid.)
4. Select the second toggle, the one for base cell size, and enter 90 (meters.) Select OK.

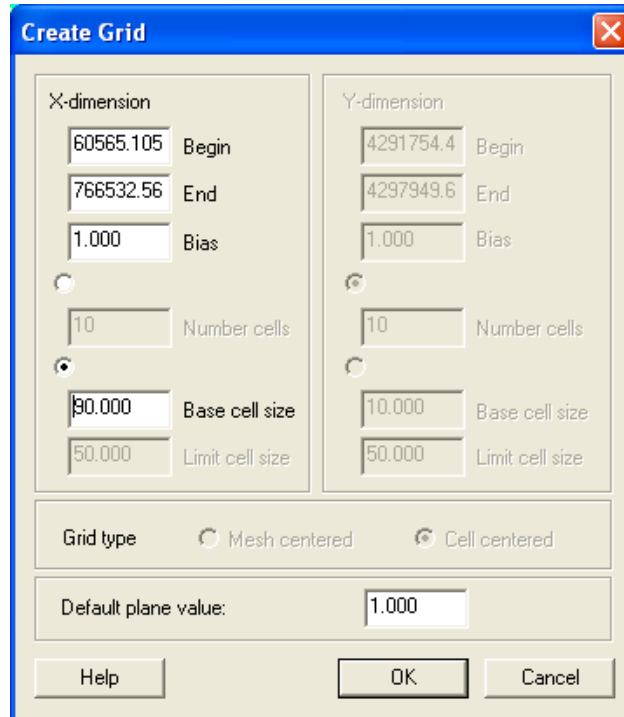


Figure 4. The Create Grid dialog in WMS. Select the second bullet and enter a base cell size.

5. Hit OK on the Background Elev Interpolation dialog. Hit Yes on the Delete Existing Background DEM dialog. The basin should now look like the following figure.

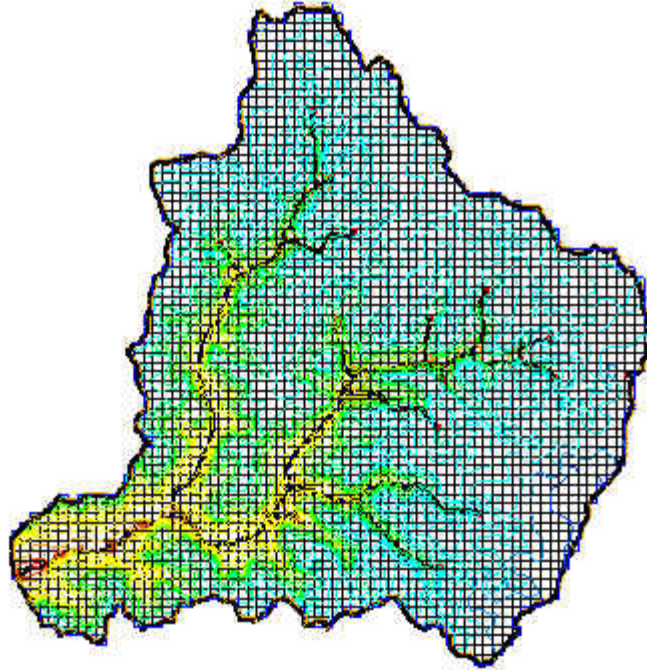


Figure 5. The regular grid created for the GSSHA™ simulations.

You'll notice in the data tree that the Drainage coverage has now changed name (and changed type as well) to the GSSHA™ coverage. A 2D Grid called new grid is also in the tree now.

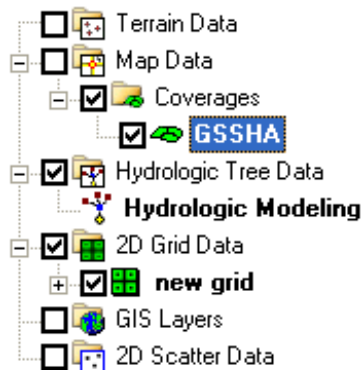



Figure 6. The Drainage coverage changed type to the GSSHA™ coverage and the 2D Grid Data now has a gridded data set group in it..



## Job Control Setup

In the last step, the GSSHA™ Job Control parameters were initialized using the default values, which are mostly zero. It is best to start with some realistic values.

1. In the 2D Grid module  select **GSSHA™ | Job Control ...**
2. Enter an outlet slope of 0.001.
3. Enter a time step of 10 (seconds) and a total run time of 500 (minutes.)

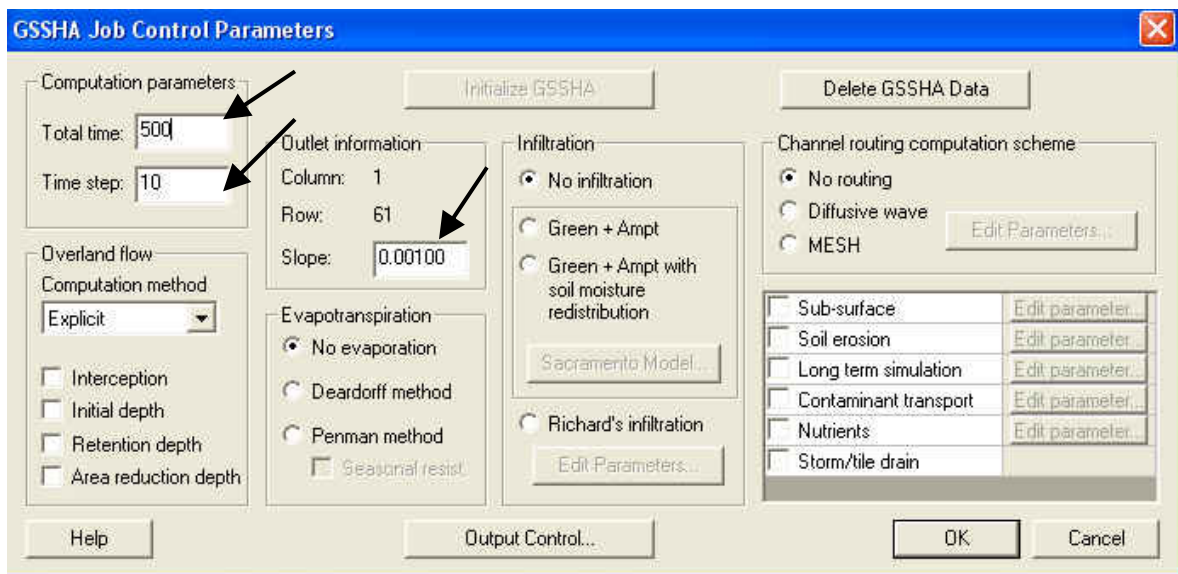


Figure 7. The GSSHA™ Job Control dialog. To begin, you need to enter a total time, time step, and the outlet slope.

4. Select OK.

## Uniform Index Map Setup

Once the Job Control parameters are set to more realistic values, there are two main areas to set up for overland flow. First, the overland flow roughness coefficients need to be set and secondly, the precipitation data needs

to be specified. There are two parts to setting up the overland flow roughness coefficients; first an index map must be set up that describes the spatial variation of the roughness and secondly, the roughness values themselves must be set. We shall create a spatially uniform set of roughness values.

1. Select **GSSHA™ | Index Maps...** (Notice that an Index Map Folder was created when the grid was created.) This will bring up the GSSHA™ Index Maps dialog.

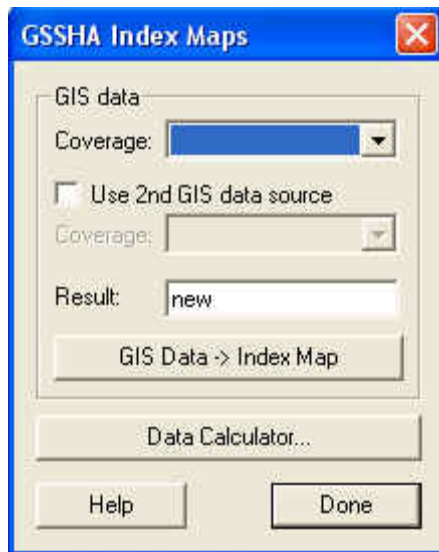


Figure 8. The GSSHA™ index map dialog. Use this dialog to create index maps from GIS coverages.

2. Select Data Calculator. This will bring up the Data Calculator dialog.

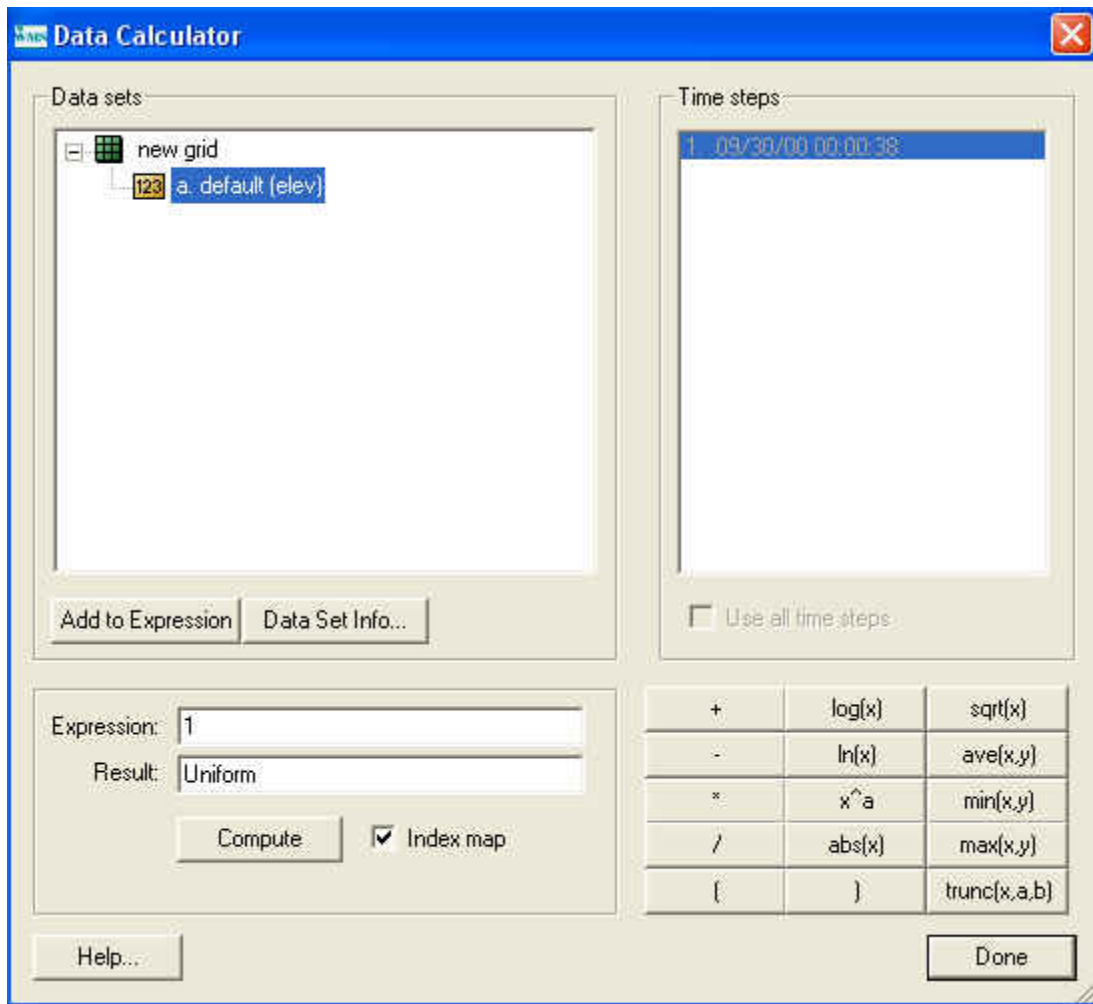


Figure 9. The Data Calculator is used to create index maps containing uniform values.

3. In the Expression box, type 1.
4. In the Result box, type Uniform
5. Check the 'Index map' option
6. Select Compute.
7. Then, select Done. This takes you back to the Index Map dialog.
8. Select Done.

We have just populated an index map (a grid) with the value 1. If you expand the data tree you will notice that our new index map has been added to the index maps folder under the 2D grid folder.

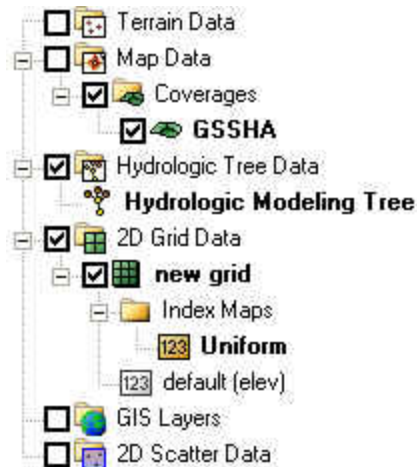


Figure 10. Notice the new index map is placed in the index map folder.

## Roughness Table Setup

Notice that when we made the index map we assigned a value of 1 to the whole map. The 1 is an index number, and we shall now relate the index number to a roughness coefficient. This is done through the mapping table.

1. Select **GSSHA™ | Mapping Tables**.
2. Select the Roughness tab.
3. In the Assigned Index Map combo box select “uniform”.
4. Select Generate IDs.
5. In the ID field, type 1
6. In the Surface Roughness edit field, enter a value of 0.1.

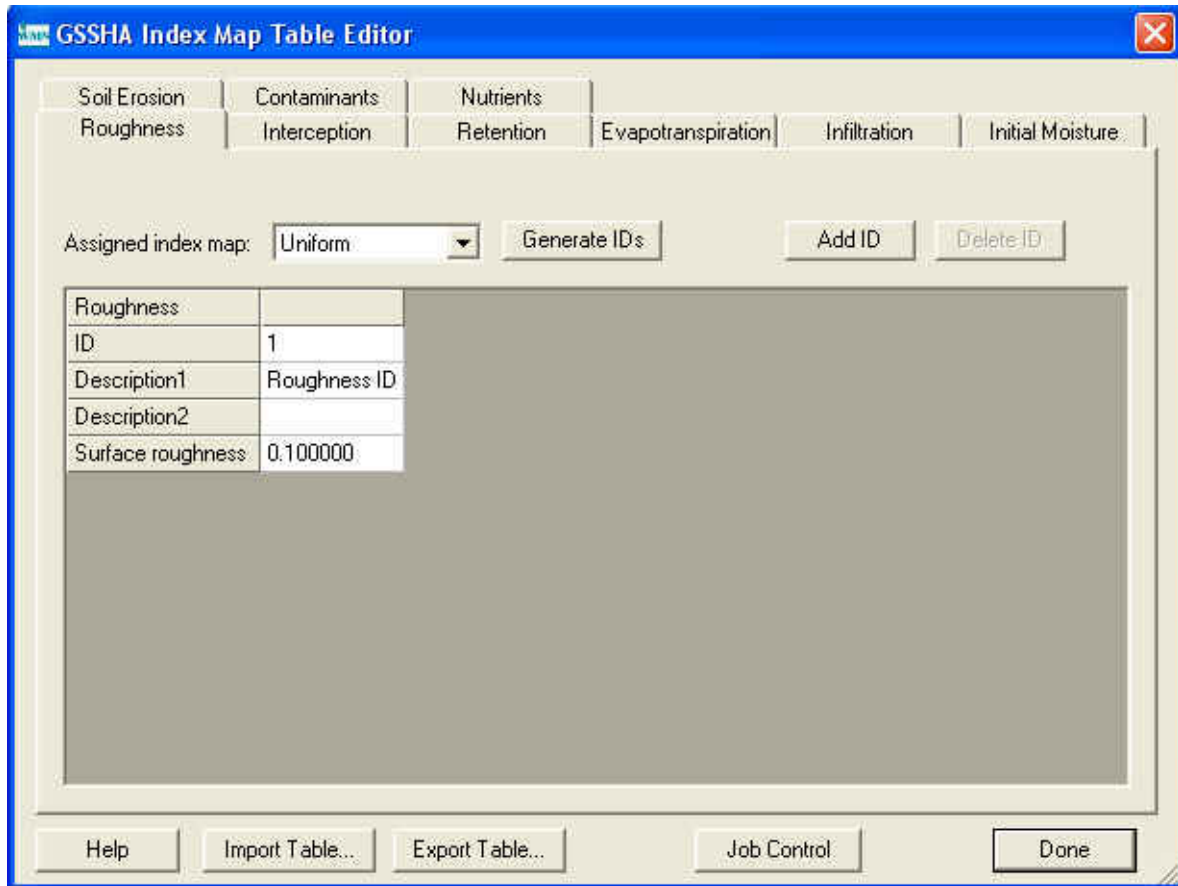


Figure 11. The Mapping Table editor. This dialog sets up the bulk of the GSSHA™ input parameter for each process used.

## 7. Select Done.

The tabs on the Mapping Table dialog list some of the mapping tables that can be set up. We will set up other processes in future tutorials. Through these two steps we have set up the spatial variability of the roughness value (by assigning it the uniform index map) as well as assigned roughness values to the IDs in the map.

## Setting the Uniform Precipitation

Besides the roughness, the precipitation must be set up in order to run the basic model. GSSHA™ can run multiple events in long-term mode only, so for now we will set up a single rainfall event. To simplify the process, we will set up a simple uniform precipitation event for a short duration.

### 1. Select **GSSHA™ | Precipitation ...**

2. Select the Uniform Rainfall toggle.
3. Enter the rainfall intensity of 10 (mm/hr.)
4. Enter the rainfall duration of 60 (minutes.)

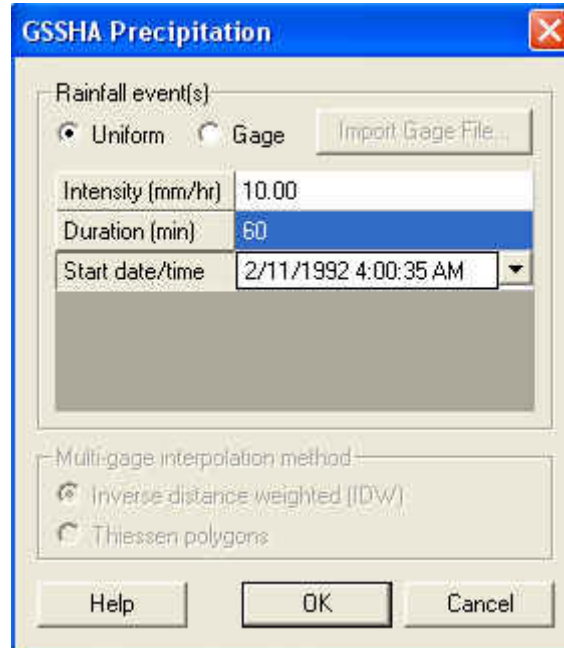


Figure 12. The GSSHA™ Precipitation dialog is used to set up both uniform events and gaged events.

5. You can change the start date/time to be what you like
6. Select OK.

## Save the GSSHA™ Model

It is advisable to create a new folder each time a significant revision is made and save the project in it. Unfortunately, there is no way to make a new folder in the current save project file dialog and this must be done externally. Once you have made the new folder, if desired:

1. Select **GSSHA™ | Save Project File...**
2. On the first line of the upper pane, select browse.
3. Browse to the folder where you wish to save the project. Select OK. (You might want to create the folder in an external browser window first.)
4. In the edit field to the right of the folder name, enter the base file name (e.g. try1).
5. Click in another field (it doesn't matter which one.)

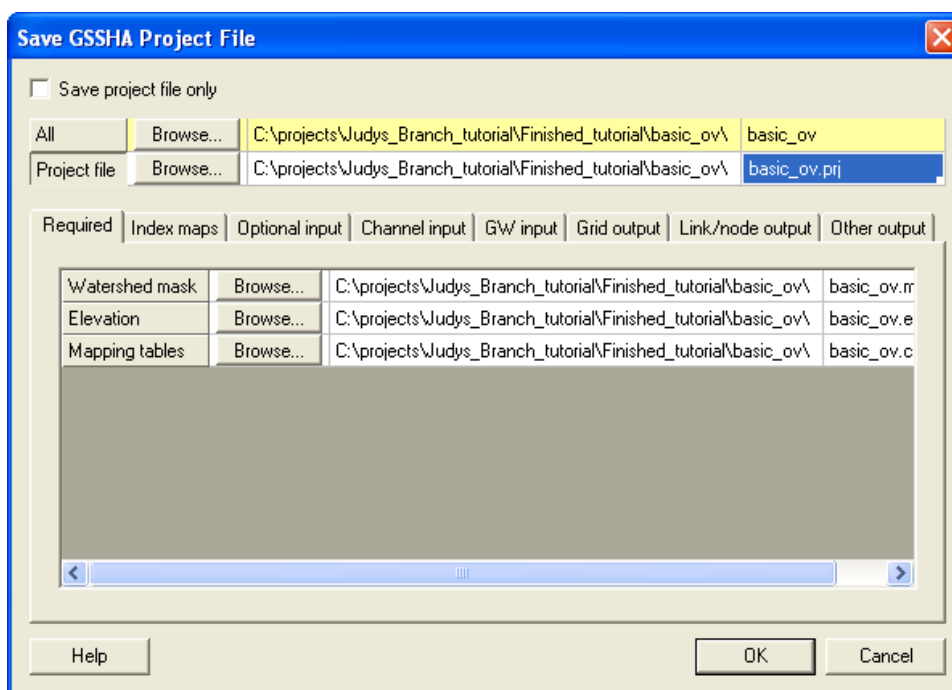


Figure 13. The Save GSSHA™ Project File dialog. Check the file names and paths to make sure the files are being saved out as you want them to be.

6. Check the names, and then select OK.

The All line is used to change the path or root filename of the many files in the project. Typically, most of the files share a similar base file name and only differ in extension. The exceptions to the rule are the index maps, which all have the same extension and different base file names. The file names and extensions may be any name desired; the defaults given in WMS are merely convention, but they do aid in quickly identifying files when you are rummaging through them. The following table lists a few of the extensions used by convention.

Table 1. GSSHA™/WMS typical file extensions. The file extensions do not need to be these, but it helps to quickly identify files if they have a uniform extension convention.

Extension	Description
prj	Project file
ele	Elevation file
msk	Watershed mask
cmt	Mapping table file
cif	Channel input file
gst	Grid-Stream file
idx	Index map
dep	Overland depth map (output)

cdp	Channel depth file (output)
cdq	Channel discharge file (output)
map	WMS map file (not used by GSSHA™)

## Running the Model

1. Select **GSSHA™ | Run GSSHA™**
2. Select OK.

After looking through the output, you'll notice that not a lot of water ran off the watershed. Often at this point in the development the simulation will not run to completion. The problem is usually due to digital dams. Digital dams are artificial depressions in the 2D Grid that cause water to pond. How to fix digital dams is the subject of a following chapter.



## 2 Visualizing Overland Flow Results

### Working with Results

Since we have just run our first simulation, it would be nice to see what happened! You'll notice that after you hit close on the Model Wrapper that WMS automatically read in some files. WMS stores the results of a run together as a solution set (the input data is not a part of the solution, only the output data.) There can be many solution sets in the data tree but they must be for the same grid and streams. For example, you could have solution sets for different roughness parameters or different time steps and then you can compare results across the data sets.

If you are just starting at this tutorial, first load the initial project:

1. In the 2D Grid module , select **GSSHA™ | Open Project File**
2. Browse to the basic\_ov.prj file in the Finished\_tutorial\basic\_ov folder of the GSSHA™ tutorial folder.
3. Hit Open.

In order to open a project you must select a \*.prj file. The project file is the primary file that tells WMS and GSSHA™ what options are set up and which files should be read or written. The project file lists both the input files as well as the output files that GSSHA™ will read and write during a run. Thus the project file doubles as the solution file. In order to open the solution (if you did not run GSSHA™ in the previous chapter or you are starting over):

1. Select **GSSHA™ | Read Solution...**
2. Make sure that the simulation points to the right simulation (basic\_ov.prj in this case.)


Notice that beneath the Select Simulation button WMS has a line that says "Solution file: GSSHA™ Solution files found." What this means is that WMS has opened the project file listed above, read in what should have been the output file names, tried to open those files, and found at least one that exists. If you have not run the simulation yet or GSSHA™ was not able to run, then the dialog will say that the solution files are not found.

3. Select OK.
4. Expand the 'new grid' 2D grid folder in the data tree
5. Expand the 'basic\_ov (GSSHA™)' folder.

Notice that on the 'basic\_ov (GSSHA™)' folder icon, there is an 's' for solution. Now that we have a solution read in we can do many things. First, though, let's look at the summary file.

1. Double-click on Summary File under the solution folder.
2. If WMS asks for your editor just click OK.
3. Look through the summary file. It is good to check things like mass balance and the volume remaining on the surface.
4. When you are done you can close the window.

From the summary file we learned that most of the water remained on the grid instead of running off. There could be a couple of reasons for this. Maybe the simulation did not run long enough, or there are problems with the elevation grid, or both. Let's look at the water depths to determine what happened during the run.


1. In the 2D Grid module  select **Display | Display Options**
2. Turn on the 2D Grid Contours.
3. Select OK.
4. In the data tree, right-click on Depth under the solution folder.
5. Select Contour Options.
6. Under Contour Method toggle to Color fill between contours.

Underneath the data tree a set of time steps appear. Click around on a few. It would be helpful if we knew what the colors represented.

1. Right-click on Depth in the data tree.
2. Select Contour Options.
3. Turn on the legend.
4. Click OK.

## Creating a Movie

Click around some more on the time steps. Try panning, zooming, and rotating. You can also adjust the lighting (**Display | Display Options | Lighting Options**) and the vertical exaggeration (**Display | View | Z Magnification**.) Let's make a movie of the contours. Once you get a view that you like:

1. In the 2D Grid module  select **Data | Film Loop...**
2. Make sure that Create New Film loop and Scalar/Vector Animation are selected. Hit next.
3. Turn on the Depth Scalar Data Set. Hit Next.
4. Hit finish. Wait for WMS to build the movie.

Once the movie is created and playing you can adjust the playback speed, looping, etc.

Now that we have a movie of the runoff depths, it is fairly easy to see that the water has stabilized by the end of the run to sit in many little puddles on the grid. These puddles are the result of the digital dams. Thus our lack of runoff is not due to time constraints (currently, anyways) but due to the presence of digital dams.



### 3 Fixing Digital Dams

#### Working with Digital Dams

The problem with digital dams is that the water gets stuck in a cell where it shouldn't be ponding. There are three methods to fix the digital dams. The first method is to manually adjust the cell elevations. The second method is to use the Cleandam™ tool to smooth the cell elevations. The third method is to use TOPAZ to smooth the cell elevations. Topaz and Cleandam™ are automated tools but you will have to decide which is best and whether or not the results are reasonable. To quickly identify which cells have digital dams:

1. Select Display | Display Options.
2. In the 2D Grid tab, click on Digital Dams.
3. Select OK. The Grid will look something like the following image.

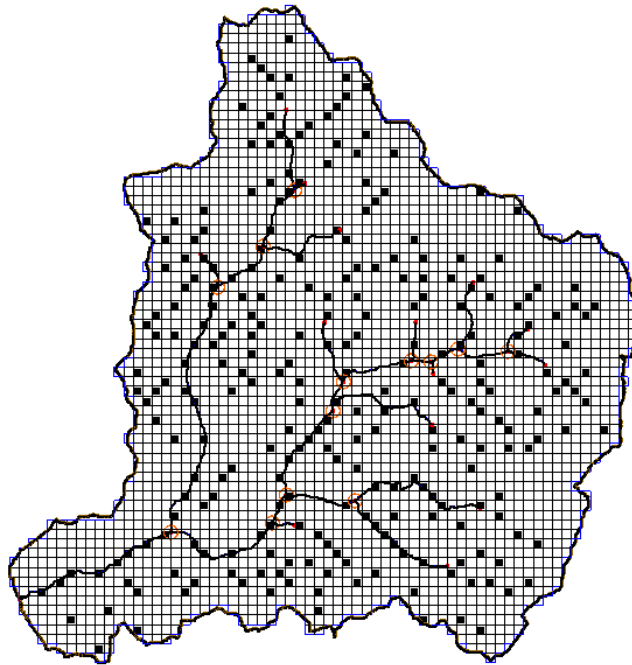


Figure 14. The digital dams are marked.

To better visualize why these cells are digital dam cells:

1. Select Display | Display Options.
2. Under the 2D Grid tab, select the blocked cells toggle.

3. Turn on the 2D Grid Contours.
4. Click on the 2D Grid Contours Options button.
5. Choose Color Fill Contours.

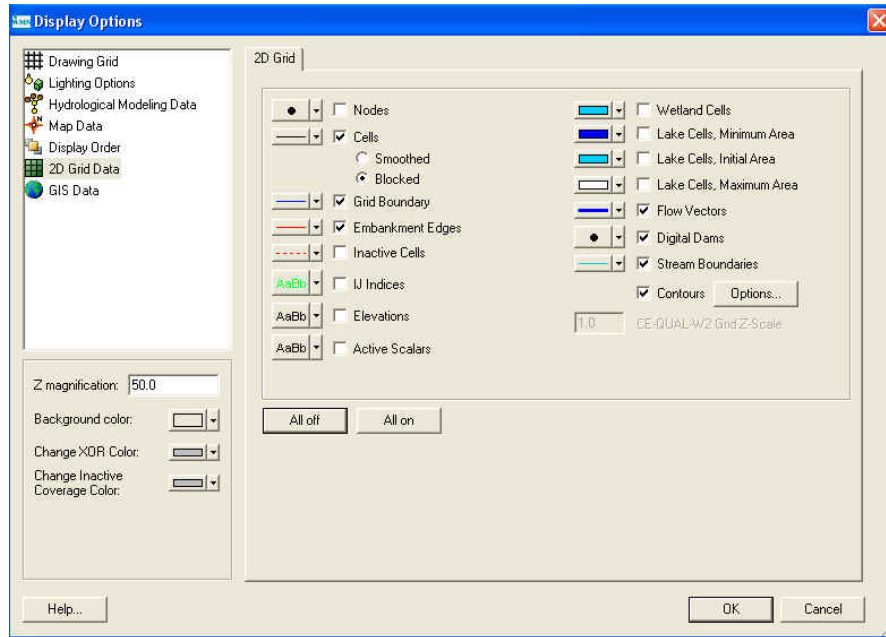


Figure 15. The Display Options dialog controls what is shown.

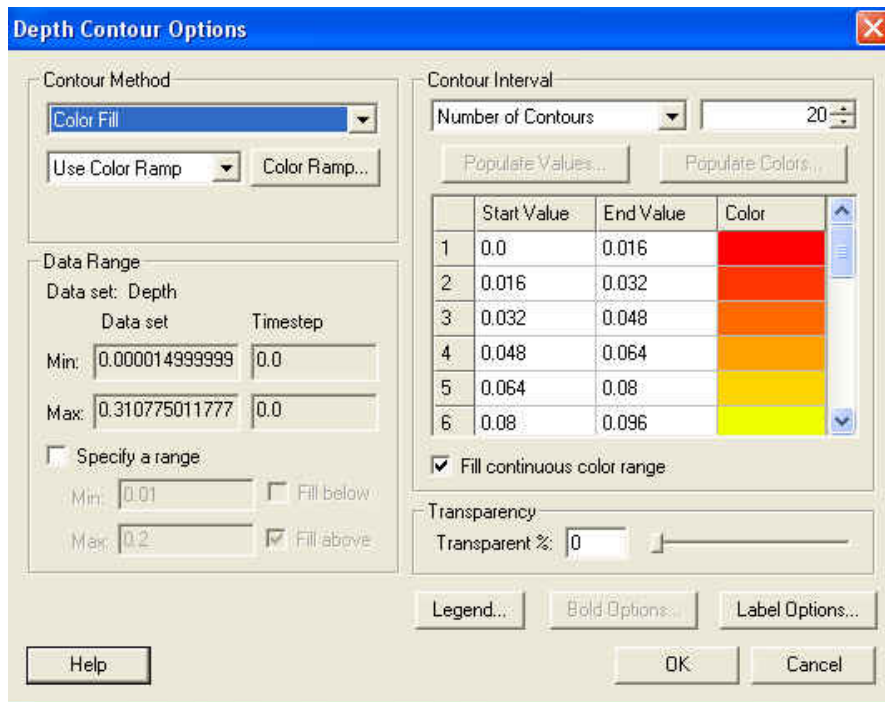


Figure 16. The contour options dialog controls how the data sets are displayed. Each gridded data set has its own set of contour options.

6. Turn on the Flow Vectors.
7. Hit OK.
8. Use the Rotate, Pan, and Zoom tools to look at a digital dam cell.

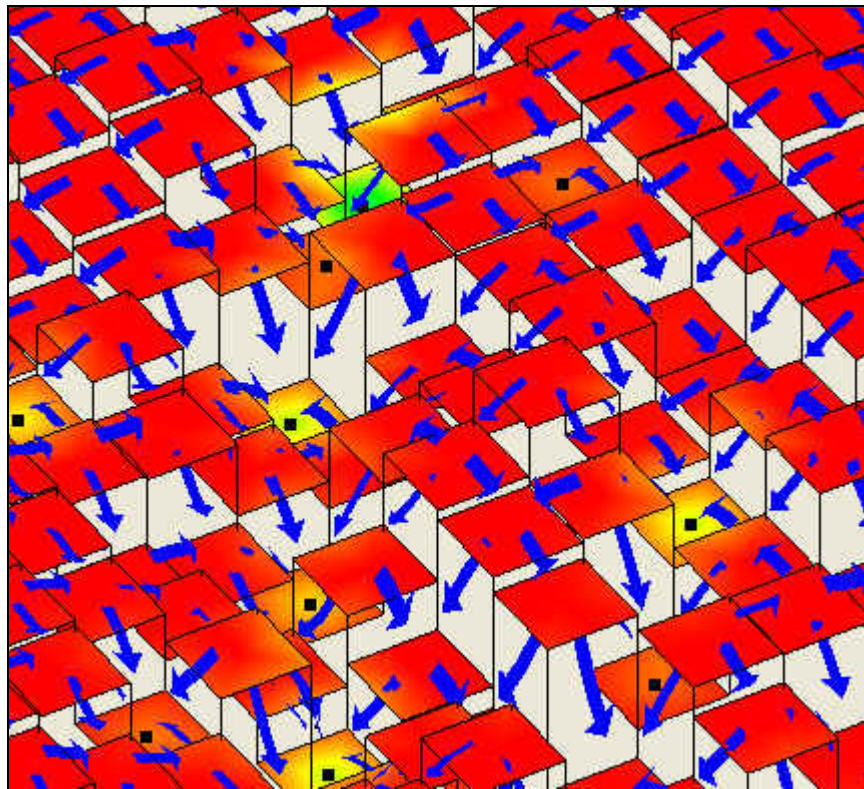


Figure 17. 3D view of how cells are represented in GSSHA™. This display helps to visualize the problems with the digital dams.

The black dots denote the digital dam cells. The arrows point out the downward slope between the cells. The cells with all four sides marked as pointing inward are flagged as digital dam cells. The blocked cells display option helps to illustrate this. You can use the **Display | View | Z Magnification** option to enhance the z scale.

## Manually Adjusting Cell Elevations

The most straight-forward method is to manually adjust cell elevations to make the water flow in an appropriate direction. There are a few tools in WMS to help with this process. In the display options dialog, under the 2D Grid tab, you can turn on elevations, flow directions, and digital dams. Usually the digital dams' option is turned on first and then you would

zoom into a trouble area. Once zoomed in you would turn on the elevations and flow directions to help visualize and understand where the trouble spots are. Occasionally only the digital dam cell has to be adjusted but usually one or more neighboring cells are the source of error. To adjust the elevation of a cell, double click on the cell to bring up the cell properties dialog. In the cell properties dialog there is an edit field with the cell elevation. Once you edit the cell elevation hit ok. Let's try a couple.

1. Hit the Plan View button  and then zoom in to the area shown in the following diagram.

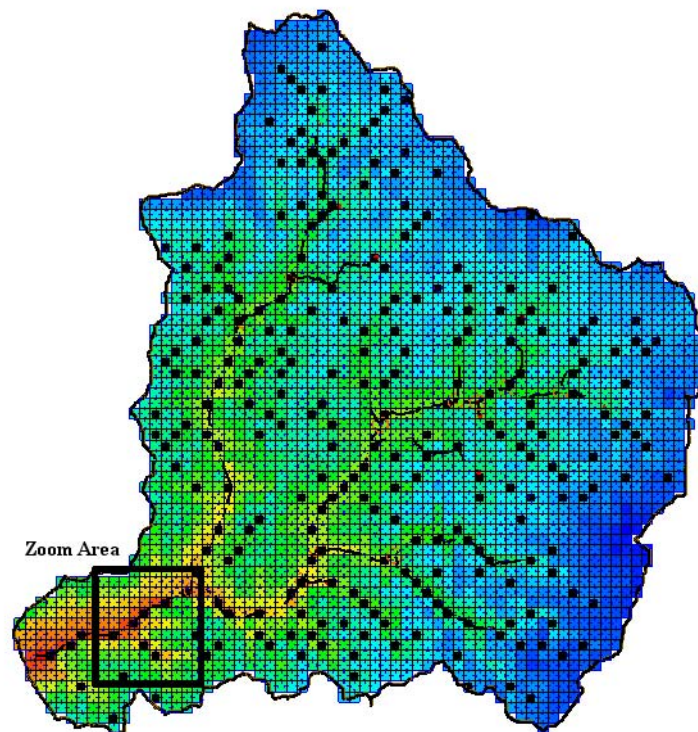


Figure 18. The 2D grid, contoured with color fill contours and the showing the digital dams.

Find the cells marked in the following diagram. You'll notice that the cell marked 1 and the cells above and beneath 4 are all digital dams. From the contours, we recognize that there should be a diagonal channel here. Since GSSHA™ has a four-point flow scheme we will need to adjust some cells to make water flow.



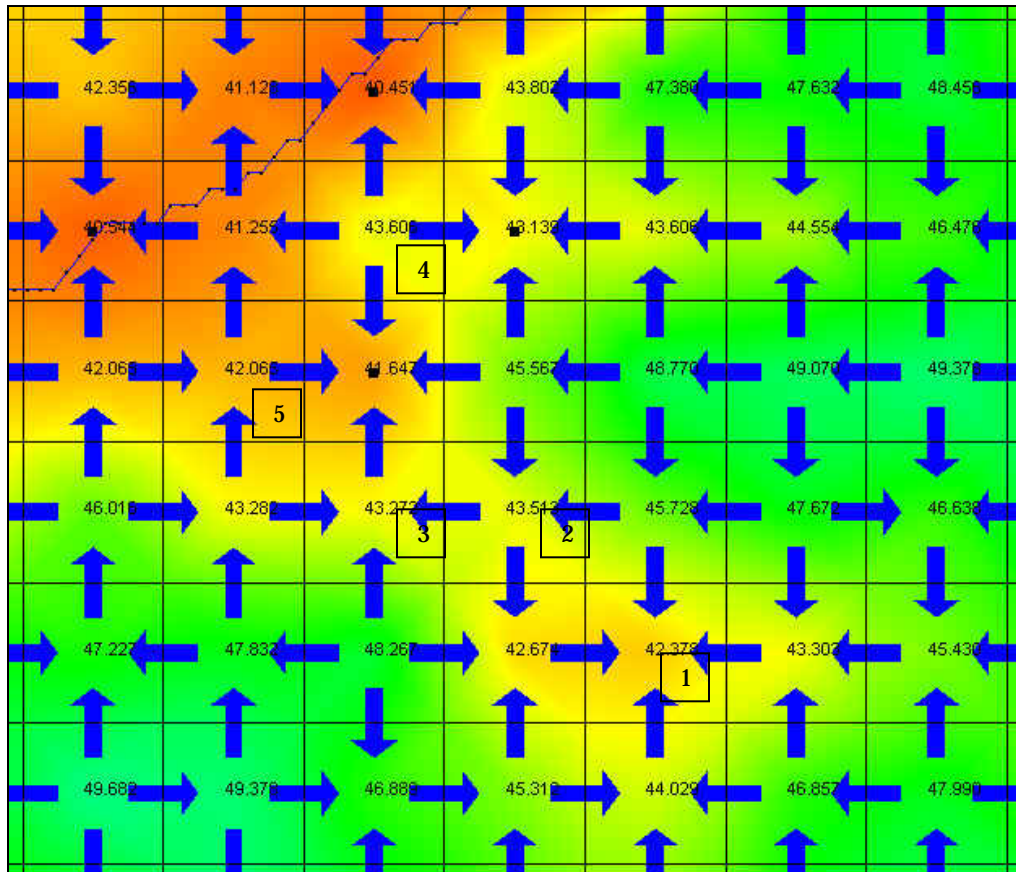


Figure 19. Close up of cells to be adjusted for digital dams.


Using the following table, adjust cells 1, 2, 3, and 4 by double-clicking on the cell using the Select **Grid Cell** tool  and editing the cell elevations to reflect the values in the table. Once the cell elevations have been changed, you will need to right click the **elevation** tab in the **Project Explorer Window** and selecting the **Set as Elevation** option

Table 2. New elevation values for the cells shown in Figure 19.

Cell #	Elevation
1	43.00
2	42.30
3	42.00
4	43.00

5	41.40
---	-------

The grid should look like the following image now.

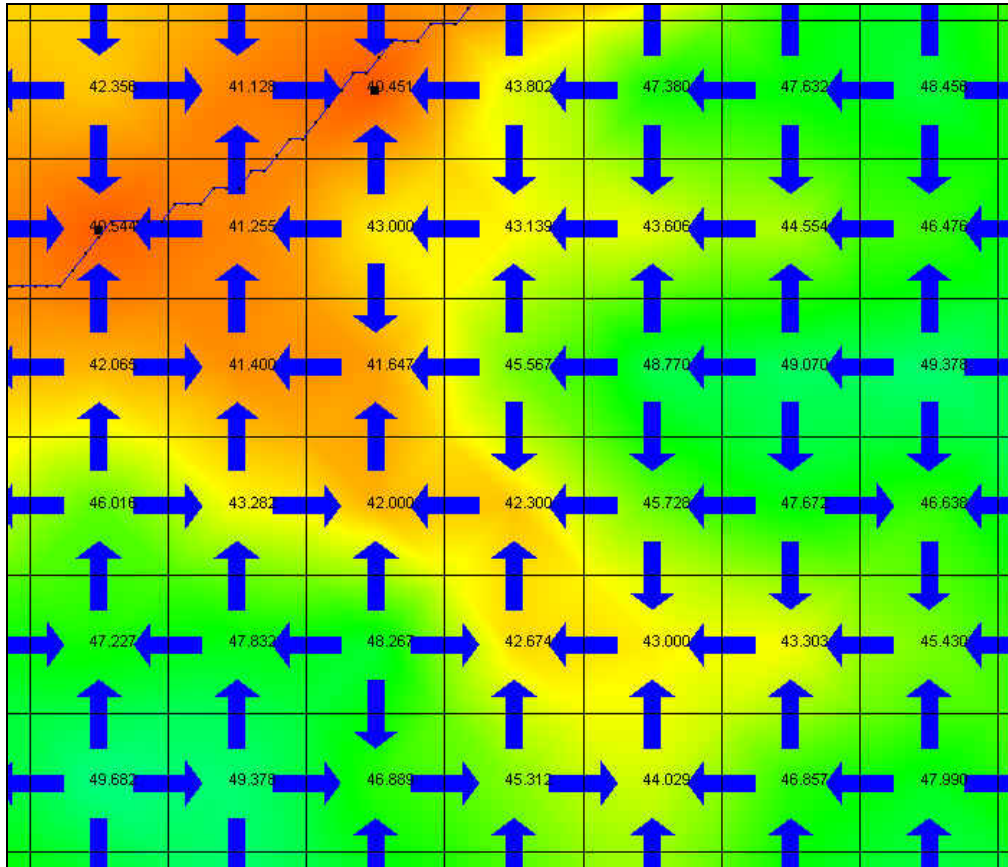


Figure 20. Elevations and flow directions for the adjusted grid. Notice the digital dam in the stream is still there. This one was not adjusted since the stream model will take care of water ponded in this cell.

## Using Cleandam™ to Fix Digital Dams

Manually adjusting cell elevations works fine for small basins with a few digital dams or if there is one or two particularly troublesome digital dams in a larger watershed model. It quickly becomes tedious, however, when there are hundreds or thousands of digital dams. This is why Cleandam™ was created. Cleandam™ uses a stochastic search process to find the best path from the digital dam to a lower elevation. It does this by starting from the digital dam and randomly searching from cell to cell until it finds a

lower cell elevation. A cost function is then calculated which is the difference between the current cell elevations along the path and a linear sloping path from the digital dam and the cell with the lower elevation.

The algorithm does  $n^2$  searches over  $2n \times 2n$  patch sizes. The  $n$  starts out at 8 and if a lower cell is not found then the patch size grows progressively larger until it reaches 256. Once a completed path is found on a patch size, all the completed paths for that patch size are compared to see which one results in the least amount of cutting. The best one is then selected and the cell elevations in the grid are edited to conform to the new path. This process generally favors passing through nearby digital dams as well, so these digital dams are fixed as a by-product of fixing the first digital dam and are called “soft fixes.” You’ll see this as the results of the Cleandam™ program are printed out.

To run Cleandam™:

1. Select **GSSHA™ | Digital Dams | Clean ...**
2. Type in an output file name, e.g. basic\_ov\_clean.ele. Select OK.
3. If WMS can’t find Cleandam™.exe, browse to it in the GSSHA™ subfolder of the WMS folder.
4. You should now see Cleandam™ running in the model wrapper. When it is done look at the output and hit close.
5. You now have the option to create a new data set or replace the elevation data set. For now we will create a new data set. Select Create New.
6. If you expand the data tree you will notice that there is a new gridded data set named after your file name. There are now two elevation data sets. The difference in the two elevation data sets can be seen by clicking on the different data sets.

## Using Topaz to Fix Digital Dams

TOPAZ, as part of its stream routing algorithm and using an 8-point pour method, computes a smoothed ground surface that has no pits or depressions. Using a 4-point pour method, the smoothed grid will still have many depressions.

To run TOPAZ on the original elevation grid:

1. Right-click on the “elevation” data set under the “new grid” 2D grid. (Expand the 2D Grid folder if needed.)
2. Select “Set as Elevation Dataset.”
3. Select **GSSHA™ | Digital Dams | Smooth Grid (TOPAZ)**.
4. If WMS cannot find topaz, browse to it in the WMS80 folder.
5. Once TOPAZ is finished, select “Create New” dataset.
6. You will notice that there is now a new data set in the data tree called “relief – TOPAZ”. It is also currently the elevation data set.

## Comparing Topaz and Cleandam™ Results

In order to understand how TOPAZ and Cleandam™ work we can compute the change in each cell from the original data set. To do this we will use the data calculator. The data calculator works by performing arithmetic and other operations over the data sets on a grid. For example, in the first figure below there is a data set named elevation. The elevation data set is currently in the 'a' position in the data set list. To reference a data set in an expression you refer to its position letter. So, to subtract 5 meters from the elevation data set and create a data set named 'elev\_minus\_5' you would put 'a-5' in the Expression edit field and 'elev\_minus\_5' in the Result edit field and hit compute. WMS would then create a data set named 'elev\_minus\_5' and populate the cell values by running through the cell values of the elevation data set and subtracting 5.

We shall use the data calculator to have WMS compute the difference between the final two 'smoothed' data sets and the original data set. The figure below shows how to set up a difference between data set c ('clndm\_out' in this case) and data set a ('elevation') and call it 'diff\_clndm'.

1. Create the Cleandam™ difference data set by entering 'c-a' in the expression edit field and a suitable name in the result field. Hit Compute.

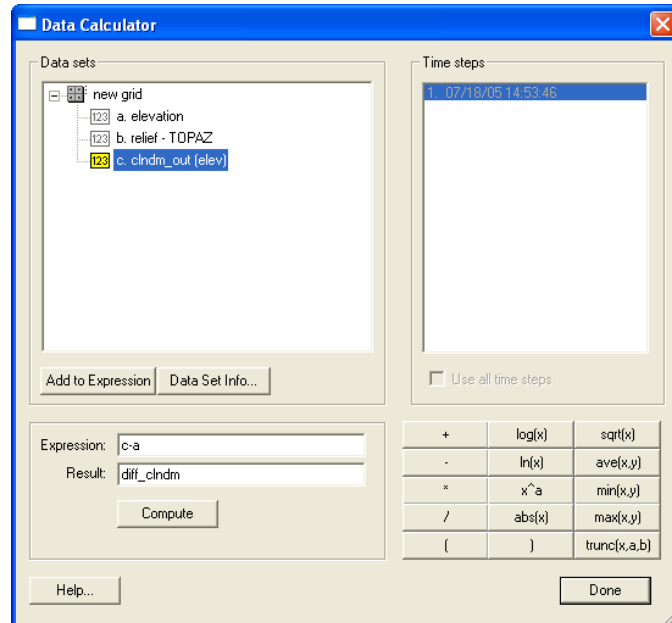


Figure 21. Using the Data Calculator to look at differences in the original elevation and Cleandam™ derived elevation data sets.

2. Create the Topaz difference data set by entering 'b-a' (or an appropriate expression) in the expression edit field and entering a suitable result name. Hit Compute. Hit OK when you are done.

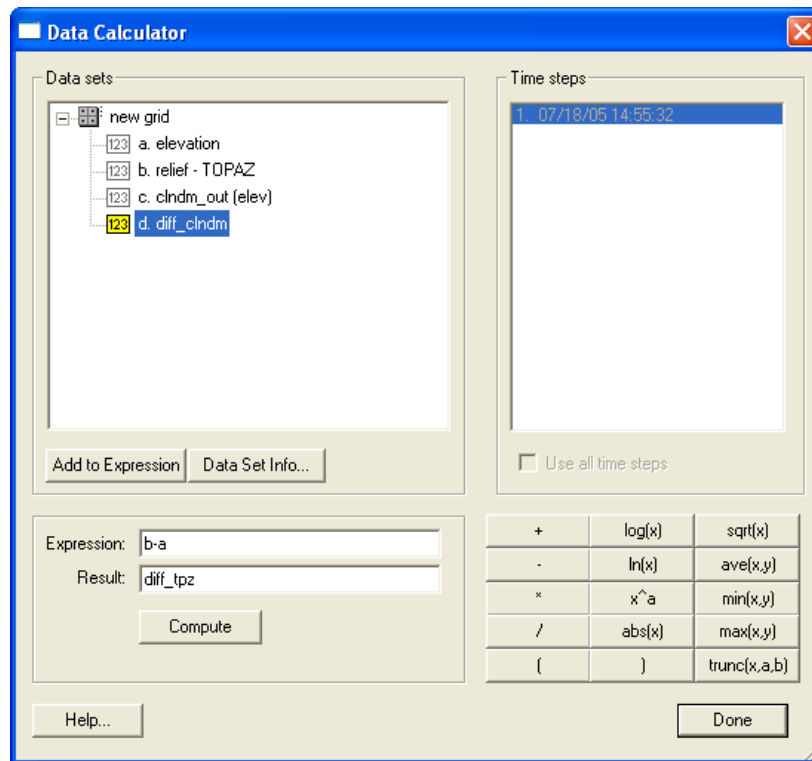


Figure 22. Using the Data Calculator to look at the differences between the original elevation and the TOPAZ derived data sets.

We have now created the two data sets that show us the difference between the original data and the modified data. You can visualize them by turning on the contour display. On the left below is the `diff_tpz` data set and on the right is the `diff_clndm` data set.

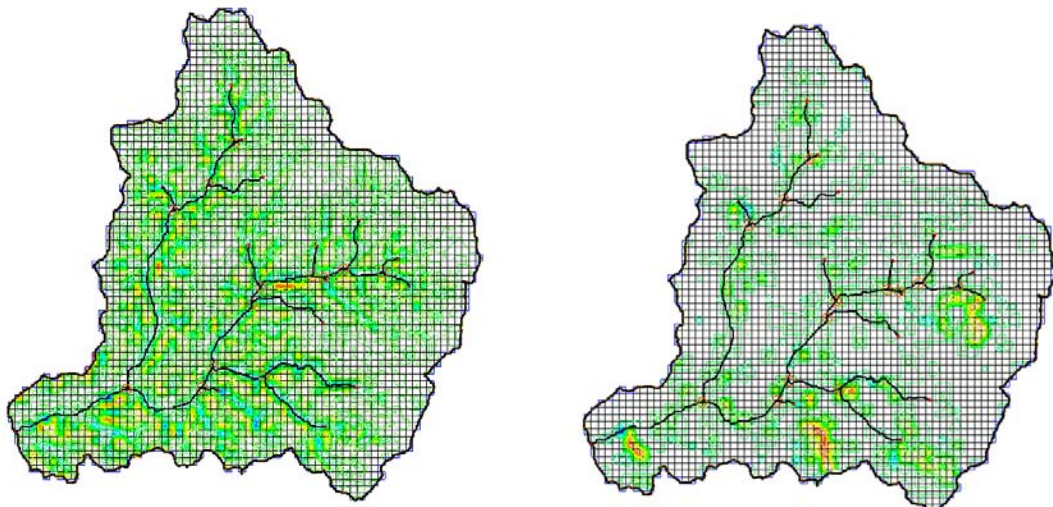


Figure 23. Differences in modified elevation data sets. The one on the left shows the modifications made by TOPAZ and the one on the right shows the differences made by Cleandam™.

Another way to look at the differences in the data set is by looking at the properties of the data set. By right-clicking on a data set you can bring up the data set info dialog. The first image below is for the diff\_clndm data set. The second image is for the diff\_tpz data set. (Hint: you can right-click on the histogram to set display properties.)

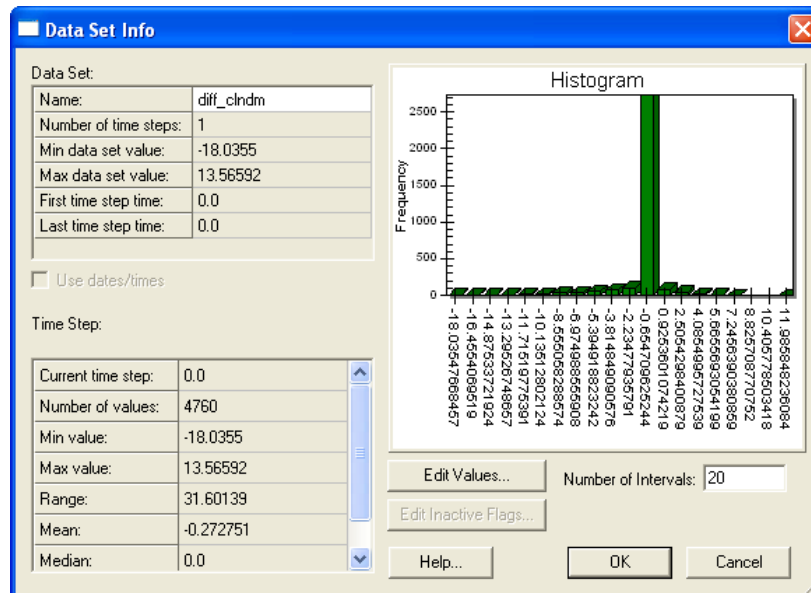


Figure 24. A histogram of changes made by Cleandam™.

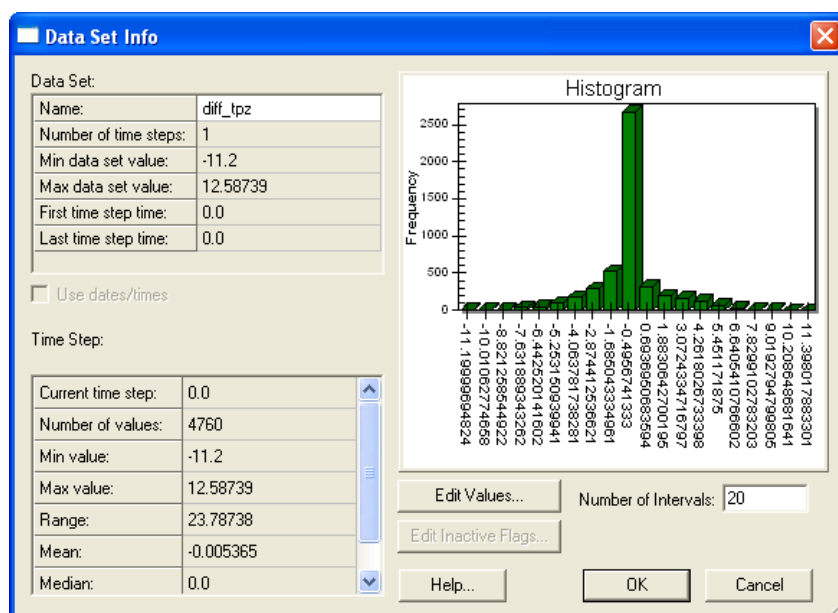


Figure 25. A histogram of changes made by TOPAZ. Notice the difference in shape between this one and the one in Figure 24.

If you look at the two histograms you'll notice that they both have a large bar in the center that is due to the 0.0 values (cells that were not changed) but the bars around the mean do have a different shape (distribution). Topaz clearly changes more of the cells, as we could see from the contours. To set a data set to be the elevation data set:

1. Right-click on the data set.
2. Select Set As Elevations...

First set the Topaz elevation data set to be the elevation data set. Turn on the digital dams display option. Your project should look similar to the following image. Notice that there are still several digital dams present. This is because Topaz uses an 8-point pour method.



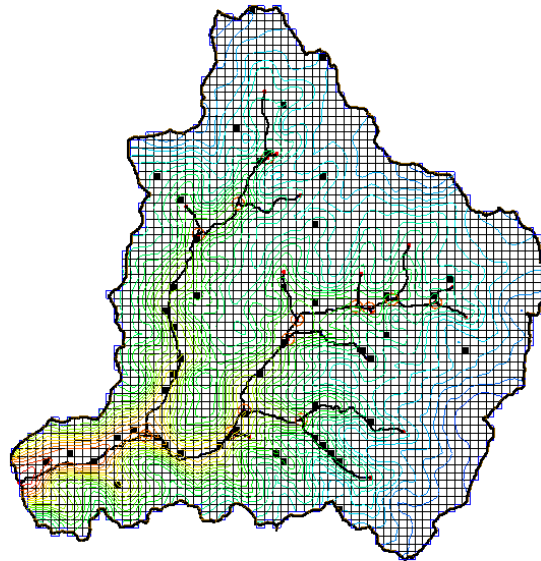


Figure 26. Results of the TOPAZ data set. There are still several digital dams present due to the 8-point pour method used by TOPAZ.

Now set the Cleandam™ data set to be the elevation data set. Your project should look similar to the following image. Notice that there is still one digital dam.

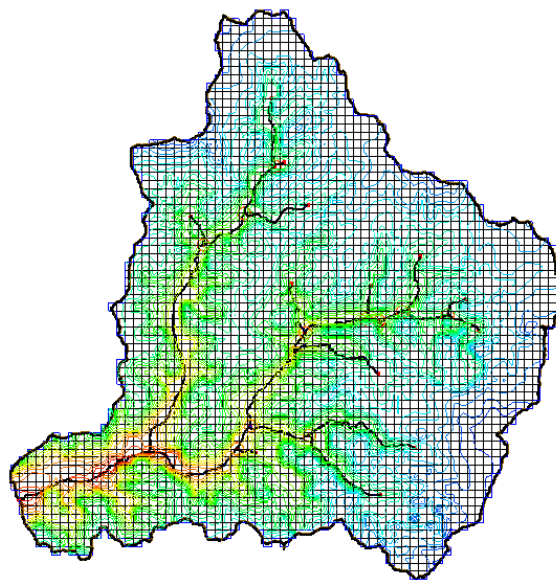


Figure 27. The results of Cleandam™ data set. Only one digital dam remains.

To fix this, the grid must manually be adjusted. This has already been done. To read in the final elevation grid

1. Right-click on new grid in the data tree. Select Import Scalar Data Set.
2. Make sure that Generic Scalar File is selected and hit OK. The Open file dialog will appear.
3. In the Files of Type combo box select All (\*.\*)
4. In the DEM folder select the file final\_ele.dat. Hit Open.
5. Right-click on the final\_ele data set and make it the elevation data set.

Save your project and run the model. Make a movie now to see how well the water runs off.

## 4 Using the Mapping Table and GIS Data

### Using Land Use Data

So far in our model we have set up uniform roughness and precipitation conditions. Setting up non-uniform precipitation will be covered in a later chapter. We shall now look at setting up spatially varied roughness coefficients. Describing the spatial variability of almost all parameters is done through setting up an index map and a mapping table. The index map is a grid of ID numbers. The ID numbers refer to numbers in the mapping table. Index maps are generic and may apply to any number of tables. Each table lists the name of the index map associated with the table and all the IDs that the index map (should or could) have, along with parameters for the IDs. We will be using a land use GIS file to create the IDs for the index map to be used with the roughness table.

1. Start from where the previous chapter ended or you can open up the clean\_ov project in the Finished\_tutorial/clean\_ov folder. (If you starting from here, you will notice in the Coverage that Land Use already exists.)
2. Select **File | Open...**
3. Browse to the Landuse folder in the project folder.
4. Click on jb\_luse\_poly.shp. Click Open. The shapefile will be added as a GIS layer (not a feature object coverage).
5. Right-click on Coverages and select New Coverage.
6. Change the Coverage Type to Land Use. Click Ok.
7. Make sure the Land Use coverage is the current active coverage by clicking on it in the Coverage list.
8. Select the jb\_luse\_poly.shp GIS layer by clicking on it. Notice that this changes the active module to the GIS module.

9. Select Mapping->Shapes to Feature Objects.
10. Click yes on when asked to use all shapes in visible shapefile for mapping.
11. Click Next.
12. Click Next to accept the default attribute mappings.
13. Click Finish.

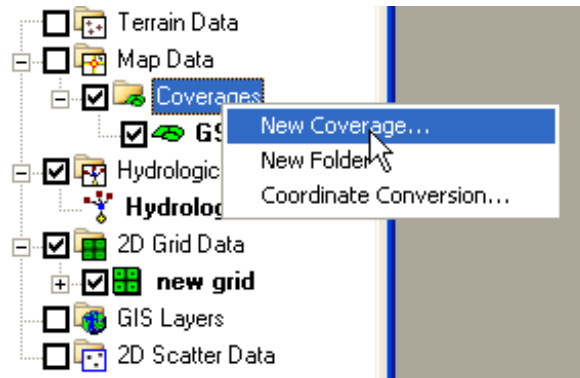


Figure 28. Creating a new coverage in the data tree.


We now have a land use coverage in WMS. Zoom into the GSSHA™ model area. You can turn off the display of the `jb_luse_poly.shp` GIS layer by unchecking the box next to it. Using the Select Polygon tool  double-click a polygon inside the grid. The land use mapping dialog will open. Notice that there are 10 land use types.


Table 3. USGS Land Use codes. [ref]

USGS Land Use Codes	
11	Residential
12	Commercial and Services
14	Transportation, Communications, and Utilities
16	Mixed Urban or Built-up Land
17	Other Urban or Built-up Land
21	Cropland or Pasture
41	Deciduous Forest Land
43	Mixed Forest Land
53	Reservoirs
76	Transitional Areas

Close the land use mapping dialog. To help visualize the land use polygons, we can color-fill them.

1. Select **Display | Display Options...**
2. Turn on Color Fill Polygons.
3. Click on Land Use Display Options. Change any of the colors for the land use IDs that you want. Click OK.
4. Click OK.
5. In the Data Tree, uncheck the box next to the GSSHA™ coverage.
6. Once you are done, turn Color Fill Polygons back off.

Now we can make an index map out of the land use coverage. The index map will be used to describe the spatial variability of roughness values for the simulation.

1. Switch to the 2D Grid Module .
2. Select **GSSHA™ | Index Maps...**
3. For the coverage use the Land Use coverage. Do not use a second GIS data source.
4. Change the result name to Land Use.
5. Click on GIS data -> Index Map.
6. Select Done.

If you expand the 2D Grid part of the Data Tree you'll notice that there are now two index maps listed, uniform and Land Use. Let's visualize them as well.

1. Select **Display | Display Options...**
2. Turn on the 2D Grid Options. Hit OK.
3. Click on the 'uniform' index map in the data tree.

The grid should appear solid red. This is because the uniform map is all ones.

4. Click on the 'Land Use' index map in the data tree.

The grid should now have cells colored in several colors representing the IDs of the polygons that were mapped to the grid cells. Next we will assign

the Land Use index map to the roughness table and set up roughness values for each of the IDs in the Land Use index map.

1. Select **GSSHA™ | Mapping Tables...**
2. Select the Roughness tab if it is not already selected.
3. Under Assigned index map choose the Land Use index map.
4. Click on Generate IDs from map. Click yes when asked to delete the process' existing IDs.

There are four IDs, 12, 15, 22, and 42. If you look at the table above you'll notice that 15, 22, and 42 were not part of the original list of IDs. The trick here is that WMS adds one to the polygon IDs because there cannot be an index map ID of zero whereas this is permissible for polygon IDs. To avoid any uncertainty, WMS adds one to all the polygon IDs when making the index maps. So the ID #12 in the ID list refers to the cells that were covered by the polygon with the ID #11. Likewise, the mapping table ID #15 came from polygon ID #14, etc. However, WMS retains the polygon IDs (or in this case the Land Use ID) and displays them in the Description 1.

5. Fill out the Roughness values according to the table below. To enter a roughness value for an ID, highlight the surface roughness box below the ID then edit the roughness value. (You may also edit the descriptions if you wish, however you should leave the original polygon ID description as is.)

Table 4. Roughness values corresponding to Land Use

ID	Description	Roughness
12	Land ID #11, Residential	0.08
15	Land ID #14, Highway	0.07
22	Land ID #21, Cropland and Pasture	0.35
42	Land ID #41, Deciduous Forest Land	0.20


6. Once you are done, select Done.

You can now save the model and run the simulation.

## 5 Stream Flow



### Creating GSSHA™ Stream Arcs

At present the GSSHA™ streams are generic and have no defined geometry. To specify a stream size, we will need to supply dimensions for each stream arc in the Judy's Branch basin. If you are beginning at this tutorial:

In the 2D Grid module  select GSSHA™ | Open Project File. Browse to the Finished\_tutorial\clean\_ov folder and select the clean\_ov.prj file and click Open.

In WMS, the tools for working with 2D grid data are in the 2D Grid module. The tools for working with the stream data are in the Map module. In order to set up the stream model four things must be done. The stream arcs must be set to a GSSHA™ stream arc type, the stream thalwegs refined, the stream geometry defined, and the channel routing job control item set.

Because of the way we set up the GSSHA™ basin, we have in the GSSHA™ coverage a set of stream arcs. These stream arcs, however, are currently set to be generic stream arcs instead of a type that can be used in GSSHA™.

Select the map module of WMS .  
Click on the "Select Branch" tool .

With this tool selected, click on the stream segment labeled “#1” on the following figure to select all streams.

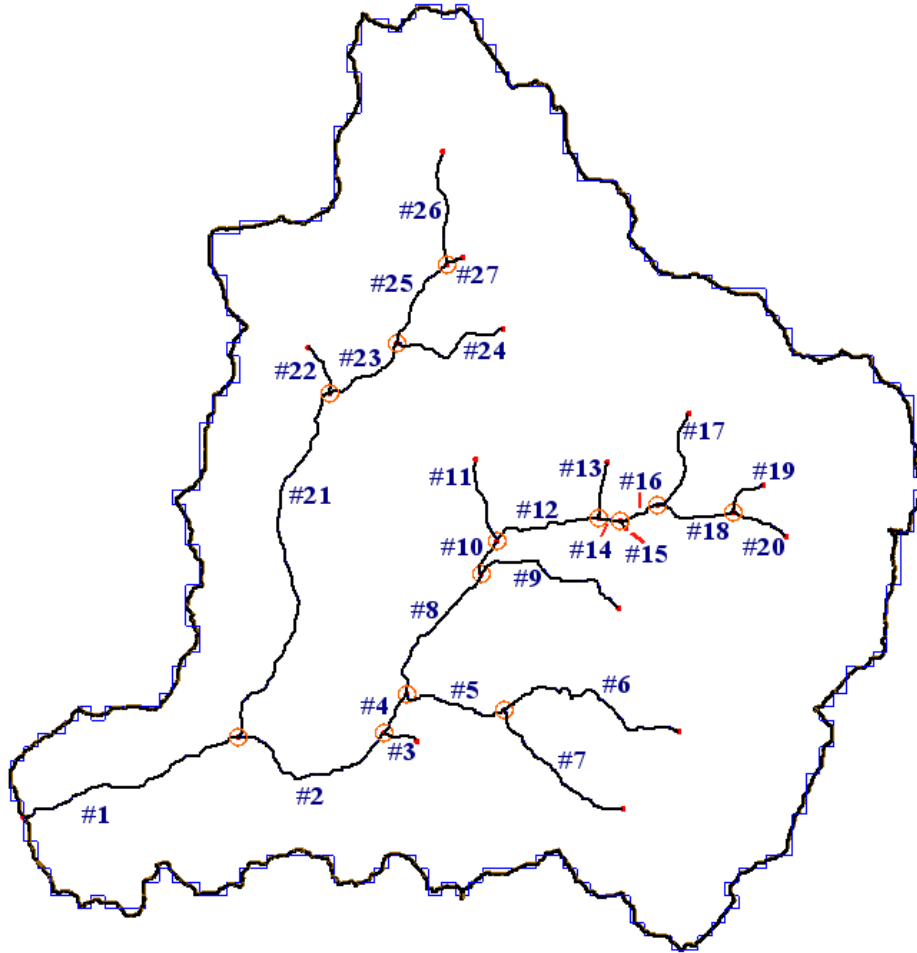


Figure 29. Stream Link numbers.

Select Feature Objects | Attributes...

Toggle on “Trapezoidal Channel” which allows you to define stream geometry for the stream arcs.

Fill out the edit fields according to the following table.



Table 5. Initial stream shape and roughness parameters for all streams.

Parameter	Value
Manning's N	0.07
Bottom Width	8
Channel Depth	8
Side Slope	2

Click OK.

Now all the stream arcs in the basin are defined as trapezoidal channels, and have identical geometries. There are two types of channels that GSSHA™ recognizes, trapezoidal and break-point. It would be nice if we could add some variation to our streams in order to represent the streams narrowing upstream. To do this we will change the dimensions of the up-stream arcs.

Switch to the “Select Arc” tool to modify streams one segment at a time



Double-click stream segment “#2” as shown in the previous figure. Enter the values for depth, width, roughness and slope as shown in the following table.

Repeat steps 9 and 10 for stream arcs 3 through 27.

Table 6. Stream shape and roughness values for individual streams.



Arc Number	Channel Depth	Bottom Width	Man. Rough	Side Slope
2	5	5	.075	2
3	2	2	.08	2
4	5	5	.075	2
5	3.5	3.5	.08	2
6	2	2	.08	2
7	2	2	.08	2
8	5	5	.075	2
9	2	2	.08	2
10	5	5	.075	2
11	2	2	.08	2
12	5	5	.075	2
13	2	2	.08	2
14	3.5	3.5	.08	2
15	2	2	.08	2
16	3.5	3.5	.08	2

17	2	2	.08	2
18	3.5	3.5	.08	2
19	2	2	.08	2
20	2	2	.08	2
21	5	5	.075	2
22	2	2	.08	2
23	5	5	.075	2
24	2	2	.08	2
25	3.5	3.5	.08	2
26	2	2	.08	2
27	2	2	.08	2

The stream arcs now have defined geometries and are ready to be smoothed.

## Smoothing the Stream Arcs

Now that the geometry for each stream arc has been defined, it is important to ensure that each arc is flowing downhill. Due to inaccuracies in the DEM data, stream arcs will occasionally flow uphill, which causes digital dams and other problems. To alleviate this dilemma, we will look at the stream profiles and modify the point elevations until each arc is flowing downhill. First, however, we need to redistribute the vertices on the arcs to a more reasonable spacing.

1. Select the map module .
2. If the Model combo box on the X, Y, Z edit bar is not set to GSSHA™, change it to GSSHA™.
3. Click on the Select Branch tool .
4. Click on the stream segment labeled “#1” on the figure below to select all streams.

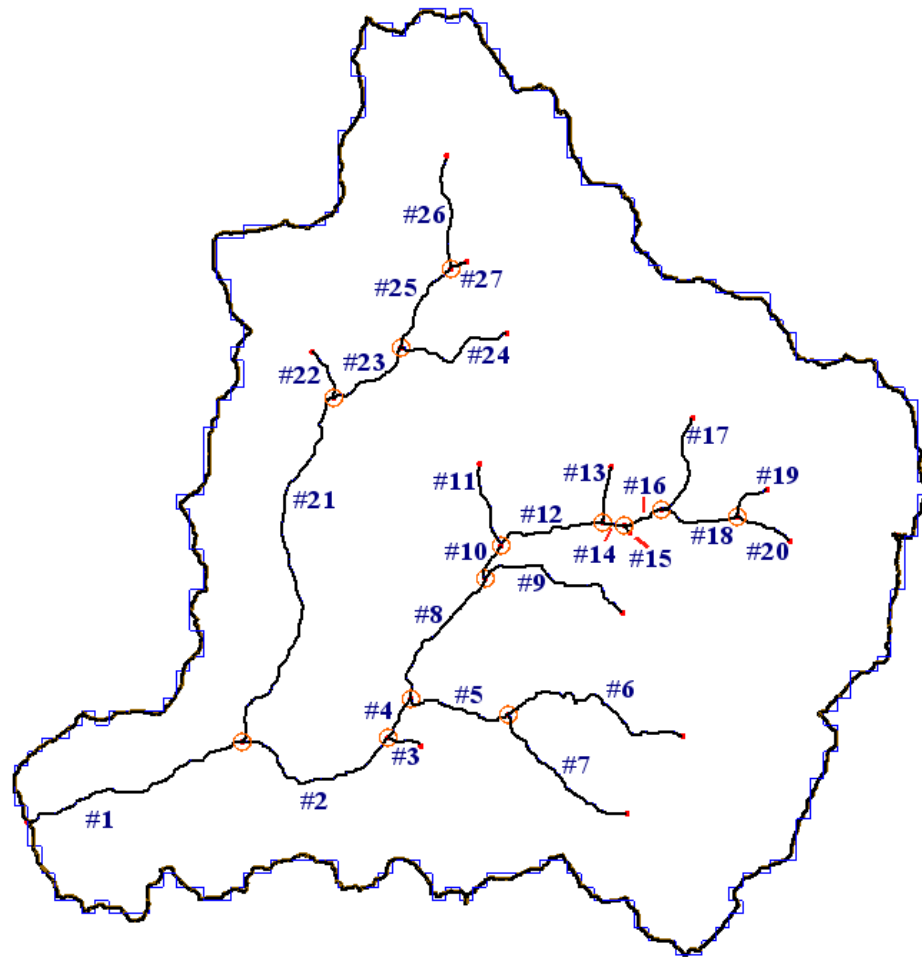






Figure 30. Stream link numbers.

5. Select the **Feature Objects | Redistribute**.
6. Type 90 in the box next to Spacing and click OK.
7. Pick the Select Arc tool .
8. Select arc # 1 as shown on the figure above, and while holding down the shift key, select stream arcs “#2” and “#4”
9. With these streams selected select **GSSHA™ | Smooth Stream Arcs...**

In the Smooth GSSHA™ Streams dialog you will see a profile of the arcs you have selected. Notice that while the segment has a general downward trend, in some places the streambed is significantly adverse. While GSSHA™ is able to handle adverse slopes, it is not desirable that adverse slopes should be in the model where they do not exist in life. We will miti-

gate this problem by making slight changes to the vertex and node elevations along the segment.

- In the pop-up window, click on the “Interpolate stream elevations” buttons as many times as needed to generate a smooth stream segment with no uphill flow, then click OK.
- If uphill flow cannot be eliminated in this manner, you can edit individual points by selecting the “select a point” tool , then dragging the point to a new position or editing the value in the box next to “Stream elevation.” Be especially careful to make sure the nodes next to the outlet are not adverse.
- Use the zoom tool  to zoom in on individual points and modify them.
- You can also use the pan  tool to move a zoomed-in screen along the stream segment.

Once the stream segment you have selected is smooth, select a new stream segment or combination of segments to smooth. Repeat the smoothing process outlined in steps 8 through 11 until all arcs in the basin are smooth. Your streams are now ready for use in the GSSHA™ model.

There is one last step before saving the model. The stream routing option must be turned on in the Job Control dialog.

1. In the 2D Grid Module  Select **GSSHA™ | Job Control...**
2. Under Channel Routing toggle on Diffusive Wave.
3. Click OK.

Save the model and run it.




## 6 Visualizing Stream Data

While the streams are connected to the overland flow plane they are separate models each having their own input files and data formats. The streams output data in a format is called the link/node data set format. These files hold a data value for every link and node at the same time step as that for the gridded output data. The two most common files in this format are the channel depth file (\*.cdp) and the channel discharge file (\*.cdq.)

The link/node data set files are read in with the other solution files and the data tree will place a reference to them in the solution folder under the 2D grid. Since they are not data to be visualized on the 2D grid there are a couple of steps to visualize the data. The link/node data sets are visualized on a TIN. The TIN is made from the stream arcs and the stream boundaries around the streams. The process of making the TIN also copies the link/node data sets to the TIN.

To make the TIN a solution set must be in memory that has valid link/node data sets. If you are just starting this lesson:

1. In the 2D Grid module  select **GSSHA™ | Open Project File...**
2. Browse to the Finished\_tutorial\streams\_trap folder.
3. Select the streams\_trap.prj file and select Open.
4. Select **GSSHA™ | Read Solution**
5. Verify that the Read GSSHA™ Solution dialog has the stream\_trap.prj file chosen and hit OK.

If you expand the streams\_trap solution in the 2D Grid folder you'll see three data sets, Depth, Stream Depth, and Stream Flow, as well as the summary file. We can look at the Depth file using the methods in the second chapter. The Stream Depth and Stream Flow data sets are link/node data sets. You can also see the Stream Depth and Stream Flow in the 2D Scatter Data folder.

In order to visualize these we need to turn on the contours for the 2D Scatter Data.

1. Select either the Stream depth or the Stream flow data set in the 2D Scatter Data folder in the data tree
2. Open the Display Options dialog.
3. Turn on the contours for the 2D Scatter Data.
4. Select a radius and Z Magnification (try a radius of 50 and a Z magnification of 50.)
5. Click OK.

You should be able to see colored circles along the stream links. The color represents the value of the data point. If you rotate into 3D mode, you will see that the circles are actually the tops of cylinders. In the properties folder you will see the time steps. As you select individual time steps the data for that time step both the 2D grid data and the 2D scatter data will be contoured. If you switch to the 2D grid module, you can select Data | Film Loop... and create a film loop of the stream and grid data.







## 7 Break-point Cross Sections

Not only does the GSSHA™ model allow you to define trapezoidal cross sections, it also allows you to import cross sections from survey data. Using cross section data can make your model more realistic, and help you achieve better calibration. The process is relatively simple, and the following exercise will show you how this can be done.

If you are beginning at this tutorial:


1. In the 2D Grid Module  select GSSHA™ | Open Project File.
2. Browse to the Finished\_tutorial\streams\_longer folder.
3. Select the streams\_longer.prj file and select Open.

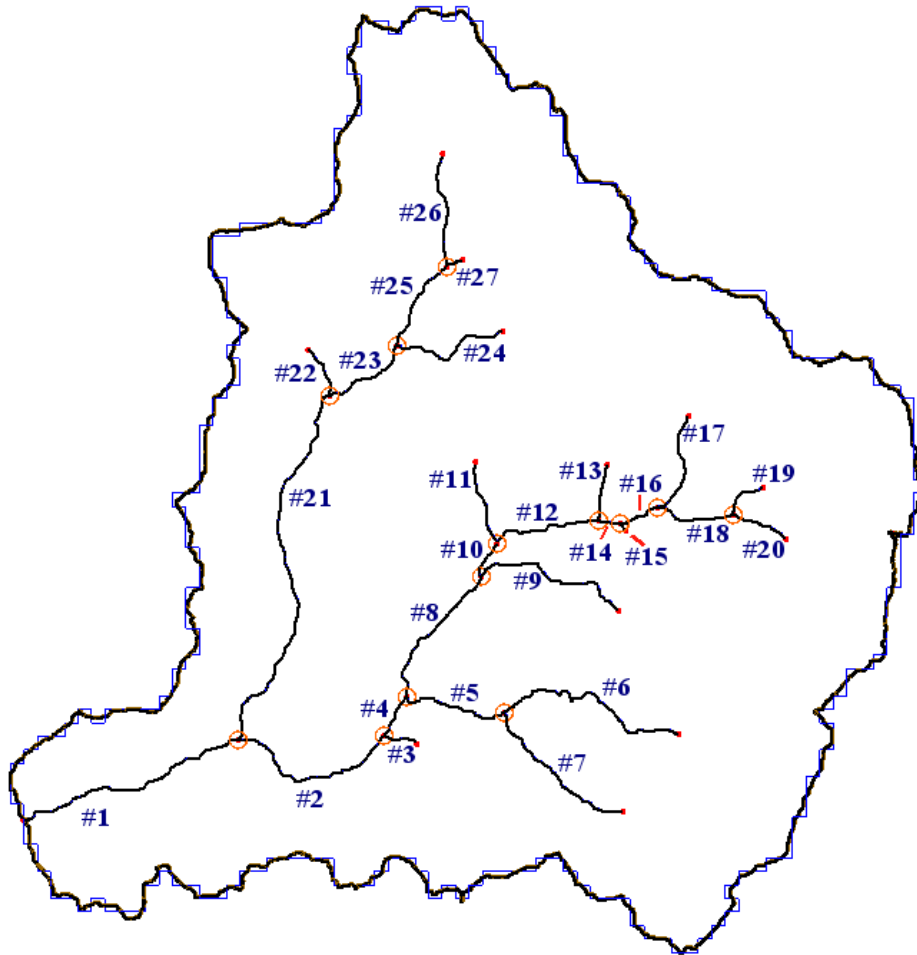
Just as when we set up the trapezoidal cross sections for the streams, all of the stream tools are in the Map Module.

1. Select the Map Module .
2. Click on the Select Arc tool .
3. Double-click on arc “#1” as defined in the following figure.
4. Toggle Irregular cross-section channel to enable the use of survey data.
5. Click on the Define Cross Section Parameter button.

You should now see the XY Series Editor window, which is where you will enter the cross section data (notice the spreadsheet on the left side of the window, with columns labeled “X” and “Y”). The cross section data you

will be entering into this spreadsheet is located in an Excel® file, which you will copy from and paste into the “XY Series Editor”.

1. Open the Excel® file “xsections.xls” in the Xsec\_data folder.
2. Copy the data for cross section #1 located in cells “A2” to “B35” (this data corresponds to cross section #1 in the following figure).
3. Paste this data into the top-left cell of the WMS “XY Series Editor” spreadsheet by right-click on the cell and selecting “paste”.
4. Select OK, then OK again.
5. Double-click on arc “#2” as defined in the following figure.
6. Repeat steps 4 through 9, except this time copy the data from “#2” in the Excel® spreadsheet.
7. Repeat step 11 for the stream arcs #4, #8, #10, #12, #14, #16, #21, #23, #25, and #27 (we do not have survey data for all the streams.)
8. All streams in the basin now have either survey geometry or trapezoidal geometry for their cross sections.
9. Save the GSSHA™ project by selecting the 2-D Grid Module .
10. Click on the **GSSHA™ | Save Project File**.
11. Use the “Browse...” button to navigate to the folder you would like to save the project in, and then click OK.
12. Type your desired project name in the box to the right of the path.
13. Click OK.





## Adjusting the Stream Course

1. If you do not already have the background image displayed, display them now by selecting **File | Open...**, navigating to the Judys\_Branch\_tutorial/Images folder, and opening all 21 carbon.jpg files. (You can filter them by typing \*.jpg in the file name field before you select any.)

If we zoom in around the main freeway interchange we can see where the TOPAZ delineated streams do not follow the actual streams. Additionally, the natural stream course has been altered by the presence of the interchange.




We will adjust the stream course to reflect the true location of the stream.

1. Zoom in around the interchange shown in the above image.
2. In the Map Module  select the Select Feature Vertex tool .
3. Adjust the vertices of the arc to be similar to the following image.



Once the arc has been adjusted we need to make sure that the node spacing is right.

1. Using the Select Feature Arc tool  select the stream arc.
2. Select **GSSHA™ | Smooth Stream**.
3. Select Redistribute.
4. Enter 90 as the value. Select OK.
5. If needed, smooth the stream. Once you are done select OK.


## Adding an Embankment

One unique aspect of the Judy's Branch basin is that it has a freeway bisecting it. This freeway acts as a barrier or embankment, which inhibits overland flow. To accurately model the basin, we need to take the embankment into account. The following exercise will show you how this can be done.

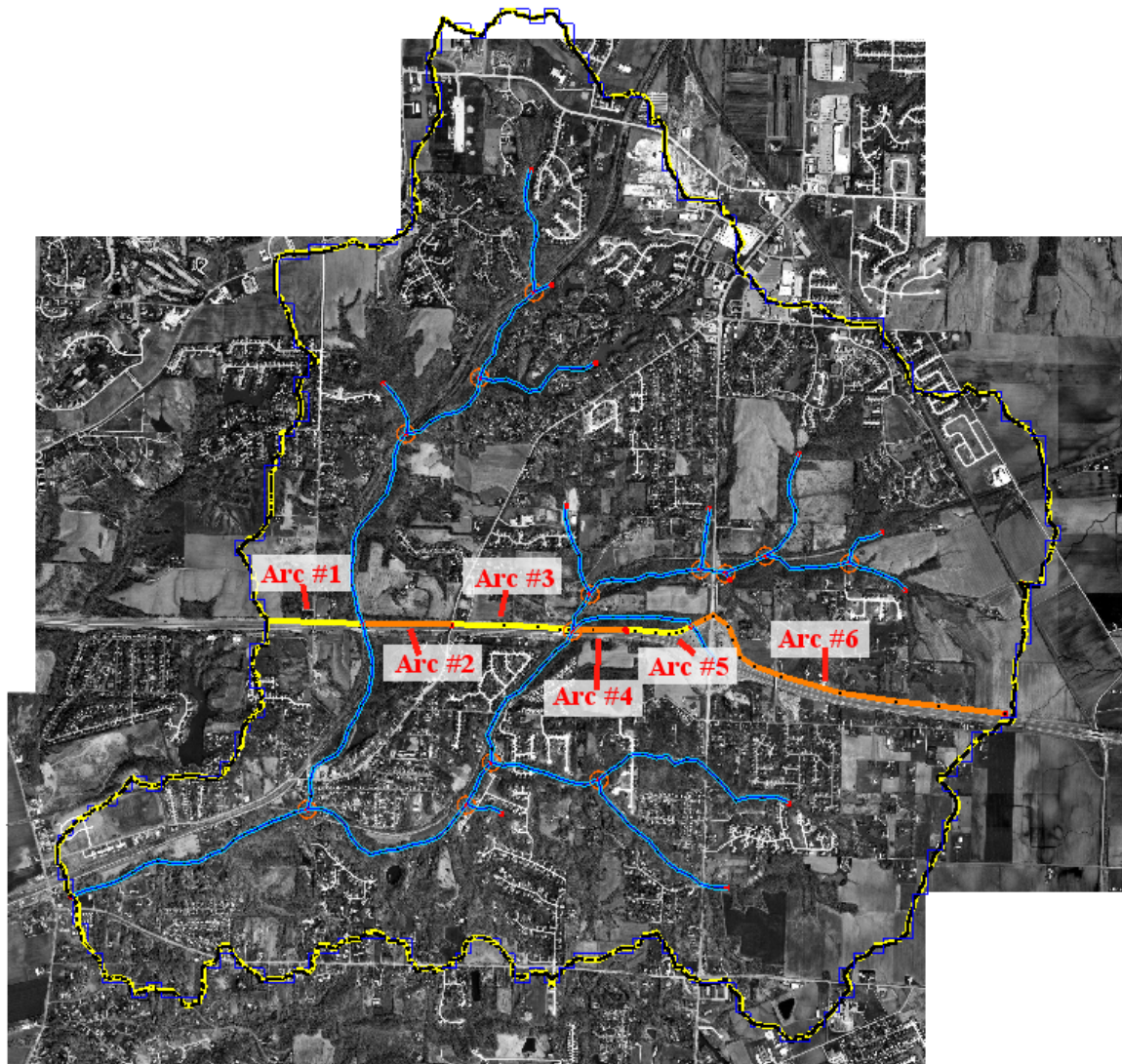
4. Select the map module of WMS .
5. Select **Display | Display Options...**



6. Select the 2D Grid tab in the display options dialog, and uncheck the box next to Cells.
7. Click Ok.

You will now use the freeway that runs through the middle of the basin for reference as you create embankment arcs. You will start on the left side of the basin boundary, and then work your way to the right side creating arcs as you go.

Use the zoom tool  to zoom in on the freeway segment labeled “Arc #1” on the figure below:

\*Note that in the figure the embankment arcs are two different colors, this is to help you distinguish between the arcs, and will not occur on your screen




1. Select the “Create Arc” tool .
2. Select **Feature Objects | Attributes**.
3. Toggle on Embankment. Select OK.
4. Create an arc that follows the north (top) side of the freeway—start at a point that almost touches (but does not touch) the basin boundary arc on the left and then click on the stream segment to end the arc
5. Select the “View Previous” button  to zoom out of the current view.
6. Use the zoom tool to zoom in on the freeway segment labeled “Arc #2” in the previous figure.
7. Select the “Create Arc” tool.
8. Create an arc that follows the north (top) side of the freeway—start by clicking on the stream (where arc #1 ended) then finish by double-clicking in the approximate area that arc #2 ends in the previous figure.
9. Select the “View Previous” button to zoom out of the current view.
10. Use the zoom tool to zoom in on the freeway segment labeled “Arc #3” in the previous figure.
11. Select the “Create Arc” tool.
12. Create an arc that follows the north (top) side of the freeway—start by clicking near the end of arc #2 (but not on arc 2, and make sure the points don’t snap together) then finish by double-clicking on the stream segment shown in the previous figure to end the arc.
13. Select the “View Previous” button to zoom out of the current view.
14. Use the zoom tool to zoom in on the freeway segment labeled “Arc #4” in the previous figure.
15. Select the “Create Arc” tool
16. Create an arc that follows the north (top) side of the freeway—start by clicking on the stream (at the point where arc #3 ended), then finish by double-clicking in the approximate area that arc #4 ends in the previous figure.
17. Select the “View Previous” button to zoom out of the current view
18. Use the zoom tool to zoom in on the freeway segment labeled “Arc #5” in the previous figure.
19. Select the “Create Arc” tool.
20. Create an arc that follows the north (top) side of the freeway—start by clicking near the end of arc #4 (but not on arc 4, and make sure the points don’t snap together) then finish by double-clicking on the stream segment shown in the previous figure to end the arc.
21. Select the “View Previous” button to zoom out of the current view.
22. Use the zoom tool to zoom in on the freeway segment labeled “Arc #6” in the previous figure.
23. Select the “Create Arc” tool.
24. Create an arc that follows the north (top) side of the freeway—start by clicking on the stream (at the point where arc #5 ended), then

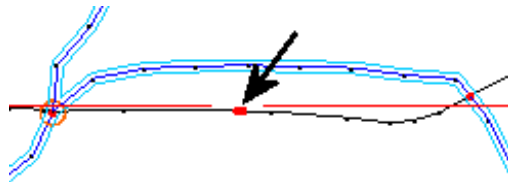
finish by double-clicking near (but not on) the arc that forms the basin boundary.

25. Select the “Frame” tool  to view the entire basin.




Once the embankment arcs have been created, you will need to check their alignment.

1. In the 2D Grid module  select **Display | Display Options...**
2. Turn on embankments and cells. Click OK.
3. Zoom in on the embankment arcs.

You should see the cell edges nearby highlighted in red. These red edges are the actual embankments that GSSHA™ uses. If there are any gaps in the embankment edges adjust the embankment arcs accordingly. Be careful not to connect the arcs to create a circular network. For example, if you see a problem similar to the following image:



It is because the embankment arcs don't extend quite far enough. To fix the problem:



1. In the Map Module , zoom in on the problem spot.
2. Using the Select Feature Node tool  drag one of the vertices past the other a short ways.
3. Select the Refresh button .





## Adding Structures

Due to the embankment we created in the previous exercise, water from the top half of the model will only be able to reach the outlet by flowing through the streams that pass through the embankment. To control the amount of water that passes through the stream at the embankment we will create a structure (in this case a culvert), at a node where the stream arcs intersect the embankment. The following steps outline how this is to be done.

1. Select the map module .
2. Pick the Select Feature Point/Node tool .
3. Using the figures below as a reference, double-click on the point (node) where the embankment arcs #5 and #6 meet (this should be on a stream).



4. In the “Node Type” section of the Feature Node Attributes dialog, click on the drop down box and select Link Break.

5. In the Hydraulic Structure and Curves section of the Feature Node Attributes dialog, click on the Culvert button to add a culvert.
6. Click on the word “Culvert 1” that appears in the text box below “Hydraulic Structures and Curves”.
7. Select the drop-down box next to culvert type and select “Rectangular”.
8. Enter the following values to define the culvert:

Table 7. Culvert parameters.

Culvert #1	
Box width	2.40
Box Height	2.40
Upstream Invert	154.0
Downstream Invert	153.8
Inlet loss coeff	0.50
Loss coeff (rev.)	0.50
Slope	0.02
Length	10.00
Roughness	0.025

9. Click OK.

Once you have finished the culvert, save the model and run it.

## 8 Infiltration

Infiltration is a key loss mechanism in a watershed; no watershed model is complete without it. GSSHA™ has four different infiltration models. During this tutorial you will set up the inputs needed for the Green & Ampt with Soil Moisture Redistribution model.

If you are starting at this tutorial,

1. In the 2D Grid Module  select GSSHA™ | Open Project File.
2. Browse to the Finished\_tutorial\streams\_embank folder.
3. Select the streams\_embank.prj file and select Open.

### Index Map Setup

#### Importing the Shapefile as a GIS Layer

The first step that needs to be done is to set up an index map that describes the spatial variation in parameters needed by the infiltration model. We shall use a soil type shapefile from the SSURGO database of the area to create a soil type index map. Before we do anything with the shapefile in WMS, though, let's look at it so that we know what we are working with.

1. Launch Microsoft® Excel®.
2. Select **File** | **Open**.
3. Browse to the Soils folder of the GSSHA™ tutorial data.
4. Using the control key, select both the soil\_clip.dbf and IDs\_for\_WMS.dbf files. Select open.
5. If soil\_clip.dbf is not the active worksheet change to it by selecting **Window** | **soil\_clip.dbf**.

This is the attribute data that comes in the shape file. Notice that there are four parameters, 'areasymbol', 'spatialver', 'musym', and 'mukey'. What WMS is looking for is a parameter that will serve as the soil type ID. There is not a suitable one in this file.

6. Switch to the IDs\_for\_WMS.dbf worksheet by selecting **Window | IDs\_for\_WMS.dbf**.

This dbf (dbase IV) file has been created to provide the information that WMS needs for the soil\_clip shapefile to be useful. Usually when setting up a GSSHA™ model the soils data base file needs to be manipulated so that it has a number that indicates a soil classification. Notice that the previous symbols are there, although the number of lines of data is greatly reduced. Additionally four parameters have been created, 'newid', 'classifica', 'erosion', and 'descriptio'. Looking at the 'classifica' and 'descriptio' parameters we can see that the soils were grouped according to soil type classification.

Table 8. Soils type classifications.

Classification	Description
1	Urban Land complex
2	Silt Loam
3	Silty clay Loam
4	Fine sand
5	Silty Clay
6	Water

First we want to bring in the soil\_clip shapefile as a GIS layer. You can close Excel® and switch back to WMS.

1. Right-click on GIS Layers in the data tree.
2. Select Add Shapefile Data.

Browse to the Soils folder and open the soils\_clip.shp shapefile. This shapefile is added as a GIS layer; however it is not “officially” part of your project.

It can now be converted into a WMS coverage in a much simpler manner, however before we convert it to a WMS coverage, we want to join the IDs\_for\_WMS.dbf file to it.


3. Right-click on soils\_clip.shp in the data tree.
4. Select Join Table To Layer.

5. Open the IDs\_for\_WMS.dbf file.
6. Under Shapefile Join Field select mukey.
7. Under table data field select classifica.
8. Select OK.
9. Right-click on soils\_clip.shp in the data tree.
10. Select Open Attribute Table.
11. Make sure that the classifica field was added to the table and hit Ok.

The next step is to create the soil type coverage in WMS to receive the polygons.


12. Right-click on Coverages in the data tree.
13. Select New Coverage...
14. Change the Coverage Type to Soil Type.
15. Hit OK.

Now we can convert the shapefile to a coverage.

1. Make sure the Soil Type coverage is the current active coverage by clicking on it in the Coverage list.
2. Select the soils\_clip.shp GIS layer by clicking on it. This will change the active module to the GIS model. 
3. In the GIS Module select, **Mapping | Shapes -> Feature Objects**.
4. Select Yes for use all shapes in visible shapefiles.
5. Select Next.
6. Scroll over to the Classifica column.
7. In the drop-down box that says 'not mapped' change it to be 'SCS Soil type'
8. Hit Next and then Finish.
9. Wait for WMS to convert the shapefile into the coverage.

### **Cleaning up the Soil Type Coverage**

Let's visualize the soil type coverage.

1. In the 2D Grid Module  select **Display | Display Options**.
2. Turn off all of the 2D Grid options except the boundary.
3. Click OK.
4. In the Data Tree, uncheck the land use and GSSHA™ coverages.
5. In the Data Tree, select the Soil Type coverage.
6. Select **Display | Display Options**.
7. Switch to the Map tab.
8. Under Points/Nodes field, turn on the Color Fill Polygons.
9. Select Soil Type Display Options.

10. Set up the colors and patterns to make them more visible. Pick colors similar to the following image. Select OK when you are done.
11. Still in the Points/Nodes field, turn off the verticies.
12. In the Legends field, turn on the Soil Type legend.
13. Select OK.

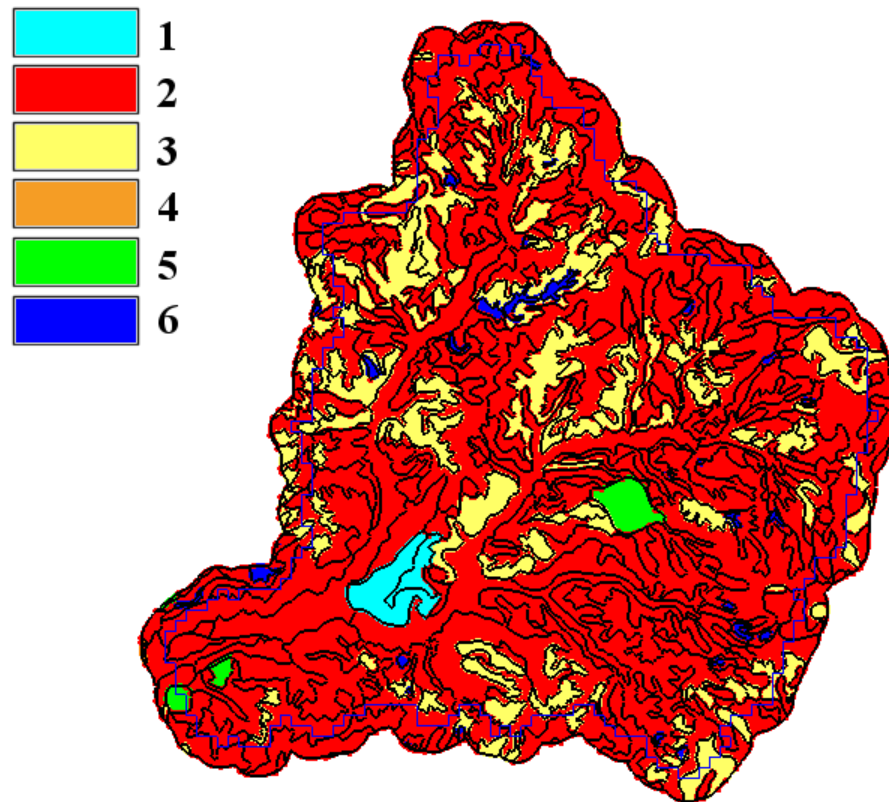


Figure 31. Initial soil types in the watershed.

From our description of the soil types, soil type one is a reworked residential area. Soil type 6 is water. Since we do not know what soils underlie either one, we shall replace them with the values of their nearest neighbor.

1. Using the Pan, Zoom, and Select Polygon tools change the soil polygon IDs for Soil ID 1 (cyan) and Soil ID 6 (blue) to be what their neighbors are. This is accomplished by double-clicking on the desired polygon. This will bring up the Soil type mapping dialog as shown in the figure below. The soil type ID is changed in the WMS soil ID field. Click 'Apply'
2. Repeat step 1.

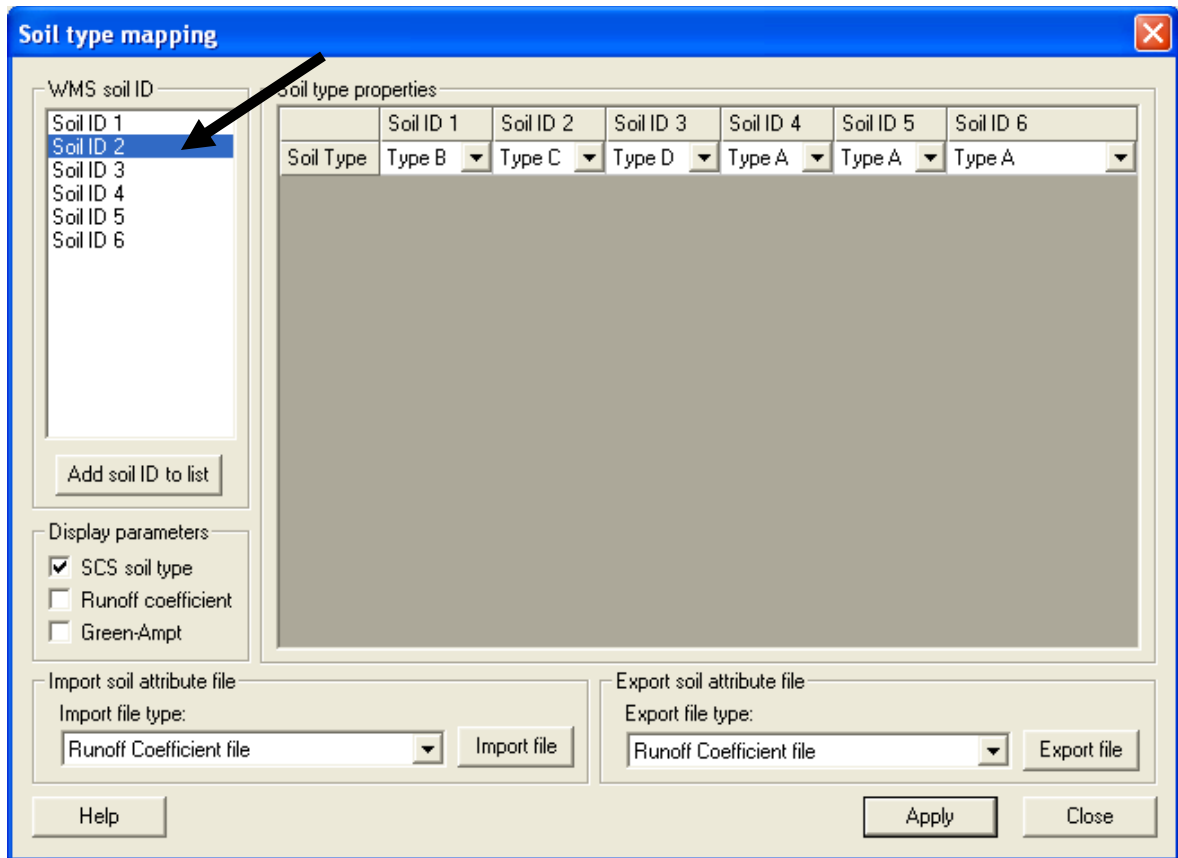



Figure 32. Changing a soils type mapping.

### Setting up the index map


Now we can make an index map out of the soil type coverage. The index map will be used to describe the spatial variability of the infiltration parameters for the simulation.

1. Switch to the 2D Grid Module .
2. Select **GSSHA™ | Index Maps...**
3. For the coverage use the Soil Type coverage. Do not use a second GIS data source.
4. Change the result name to Soil Type
5. Click on GIS data -> Index Map
6. Select Done.

Now we have an index map of the soils shapefile. Now we need to turn on the infiltration parameters in the Job Control.

1. In the 2D Grid module  select **GSSHA™ | Job Control**.
2. In the Job Control dialog change the infiltration option to Green + Ampt with Soil Moisture Redistribution.
3. Click OK.

Now we can set up the mapping tables for infiltration.

1. In the 2D Grid Module  select **GSSHA™ | Mapping Tables**.
2. Select the infiltration tab.
3. In the Assigned Index map box choose Soil Type.
4. Select Generate IDs from Map.

The Generate IDs from Map button created three IDs, 3, 4, and 6. You will recall from when we created the land use index map that these three IDs came from the polygon IDs 2, 3, and 5. So the soil type index map ID #3 represents silt loam soils; the index map #4 represents silty clay loam; and the ID #6 represents silt.

5. Using the following table, enter the values for each parameter.

Table 9. Soil properties.

ID	Description	Hyd. Conductivity	Capillary Head	Porosity	Pore Dist. Index	Residual Saturation
3	Silt Loam	0.68	16.68	0.501	0.234	0.015
4	Silty Clay Loam	0.20	27.30	0.471	0.177	0.040
6	Silty Clay	0.10	29.22	0.479	0.150	0.056

6. Select the Initial Moisture tab in the process window.
7. In the Assigned index map combo box select the uniform map.
8. Select Generate IDs from map.
9. Click on ID #1 in the ID window.
10. Click on Initial Moisture in the Selected ID Property table.
11. Enter a value of 0.3 in the edit box for the initial moisture.
12. Click Done.

You can now save the model and run.

## Visualization

Since no water ran off we can guess that it all infiltrated. But let's check to make sure.



1. In the data tree, under the solution that was just read in, double-click on the summary file.
2. Verify that there are no mass balance errors, check the amount that it rained and the amount that infiltrated, and close the file.

Since we need more water lets increase the precipitation.

1. In the 2D Grid Module select GSSHA™ | Precipitation.
2. Increase the precipitation rate to 25 (mm/hr).
3. Select OK.

Let's also turn on the infiltration output options.

4. Select GSSHA™ | Job Control
5. Select Output Options
6. Turn on Cumulative Infiltration Depth and Infiltration Rate.
7. Select OK.
8. Select OK.

Save and run the simulation. You can now use the same techniques from Chapter 2 to visualize the infiltration data sets.

If you display the depth map (turn on the images and set the depth map contours to color filled, 30% transparency) you'll notice that there is a new digital dam that was created by moving the stream and setting up the embankment. Try and fix it using the manual techniques from Chapter 3. (Hint: make the digital dam cell flow toward the cell with the stream to the right.)

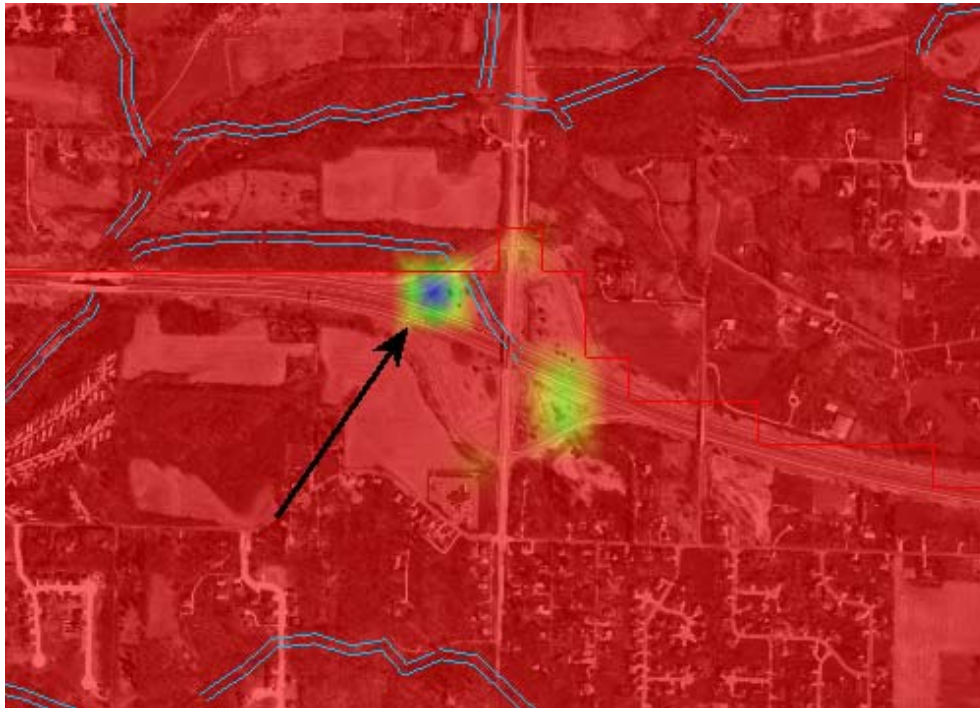


Figure 33. Digital dam next to an embankment.

One of the more interesting movies you can make is of the Infiltration Depth data set. The two different soil types saturate at different rates and make for a pretty impressive movie.



## 9 Long-Term Simulations

Long-term simulations typically involve running several rainfall events along with the evapotranspiration model for weeks to months. There are two key parts to running a long-term simulation. The first is to set up the precipitation file, and the second is to set up the evapotranspiration model with its hydrometeorological (hmet for short) data.

### Precipitation

A long-term event typically consists of multiple rainfall events, often with several rain gages. Currently the multiple-event, multiple-gage file needs to be set up in a text editor external to WMS. Fortunately a handy Microsoft® Excel® spreadsheet has been set up to aid in setting up multiple gage events. If you are only passingly familiar with Microsoft® Excel® it is recommended that you read Chapter 10, “Using Microsoft® Excel® to format GSSHA™ Data” first.

We will start by using a Microsoft® Excel® spreadsheet to set up a multiple-event, single gage precipitation file from some raw data.

#### Using the Format Precip Macro

1. Open the Excel® file “format\_precip\_macro.xls”.
2. Click on the worksheet titled “input\_data”.
3. Select columns “A” through “F”.
4. Right-click on the columns and select “format Cells...”
5. In the “Format Cells” pop-up window select “Text” in the box below the word “Category”.
6. Click “OK”.
7. Open the Notepad file “precip\_raw.txt”.
8. Select and copy the entire text file.

9. Select cell “A1” on the input\_data worksheet, then right-click and pick “Paste”
10. Set up the data to match the format outlined on the “Instructions” worksheet of the spreadsheet (for help look at the instructions, or refer to Chapter 10 Using Microsoft® Excel® to format GSSHA™ Data.
11. Follow the steps on the “Instruction” worksheet. (Use the default values. The coordinate for the gage is found in the Precip\_formatted.txt file.)
12. Once you have setup the data, click on the “Format precip data” button found on the “Instructions” worksheet”.
13. Your formatted data is on the “Output\_data” worksheet.

## Hydrometeorological Data

Hydrometeorological data is used in GSSHA™ to determine how the soil moisture is affected by atmospheric conditions. The hydrometeorological data is used to drive the evapo-transpiration model. In the following exercise we will create a file that contains all the hydrometeorological data for the same period as the precipitation data.

### Using the Format Hmet Macro

1. Open the Excel® file “Format\_Hmet\_macro.xls” from the Formatting\_macros folder.
2. Select the “Instructions” worksheet to learn how the data should be organized before it can be formatted for GSSHA™.
3. To retrieve the raw data, open the Hmet raw data file called “Hmet\_raw.xls” in the Hmet\_data folder.
4. Select the “KBLV\_Scott” worksheet and copy the appropriate columns of raw data to the “input\_data1” worksheet of the format Hmet data macro.
5. Select the worksheet “scott\_radiation\_2001” of the raw data file, and then copy the appropriate columns to the “input\_data2” worksheet of the format Hmet data macro.
6. Make sure that the “input\_data1” and “input\_data2” worksheets are organized as outlined on the “Instructions” worksheet, and then click the “Format Hmet data” button on the “Instructions” worksheet. (Note that the Instructions indicate that the data should be entered in cell A1. This is an error. The first row of data should be in cell A2.)
7. The formatted data will be shown on the “output\_data” worksheet.
8. Save the “output\_data” worksheet as a text file called “hmet.txt” in the folder you created earlier in the precipitation section.

## Evapotranspiration

Now we will go back to WMS and set up the Long-term modeling data. First we need to set up the Job control options to turn on long-term mode.

If you are starting the tutorial from here, open the long\_term.prj file found in the Finished Tutorial folder.

1. Select **GSSHA™ | Job Control...**
2. Check the box next to “Long term simulation” in the lower-left area of the pop-up window, then click “Edit Parameters...”
3. In the pop-up window enter a value of 38.7696 for “Latitude”.
4. Enter a value of 270.05 for “Longitude”.
5. For “GMT” enter a value of -6.00.
6. Enter 0.10 for “Minimum event discharge”.
7. Make the “Soil moisture depth” equal to 0.5.
8. Click on the folder icon to next to “HMET Data File” to browse for the Hmet text file you created with the “Format\_Hmet\_macro” spreadsheet. Navigate to the file and select it.
9. Under “Format”, toggle on WES.
10. Select OK.
11. In the “Evapotranspiration” section of the window toggle “Penman Method”.
12. In the Overland Flow Sim. Type combo box choose “ADE” instead of “Explicit.”
13. Select OK.

Next we need to set up the ET parameters.

1. Select **GSSHA™ | Mapping Tables...**
2. In the Process window click on Evapotranspiration tab.
3. In the drop down box beneath “Assigned index map” select landuse, then click the “Generate IDs from map” button.
4. At this point you will define the albedo and wilting point values for the index map. You may use the values given in the following table, or you can find values from the appendix.

Table 10. Values for the Evapotranspiration process.

ID#	Albedo	Wilting Point	Vegetation Height	Vegetation Transmission Coefficient	Canopy Stomatal Resistance
12	0.15	0.1	0.08	0.7	20
15	0.22	0.1	0.1	0.5	20
22	0.22	0.1	1	0.2	86
42	0.2	0.1	17	0.15	100

5. Click Done.

Next we need to tell WMS to point to our precipitation file.

1. Select **GSSHA™ | Precipitation...**
2. Toggle on the Multiple Gage Rainfall.
3. Click the Open File button that appears below.
4. Browse to the precipitation file, select it, and hit OK.

Since we only have one gage, the rainfall data is spread out uniformly over the watershed. If we had more than one gage we would pick either Theissen Polygons or Inverse Distance weighted here in this dialog.

5. Select Ok.

We are ready to run now, but first we will want to change some output options. We will not want to output the data sets so frequently.

1. **Select GSSHA™ | Job Control | Output Data.**
2. Change the Gridded Write Frequency to 60 (minutes).
3. Select OK, OK.

You are now ready to run a long-term simulation. Save the project, then run GSSHA™.

## 10 Using Microsoft® Excel® to format GSSHA™ data

### Splitting an Excel® column

In this example you will copy a text file from Notepad and paste it into Microsoft® Excel® where it can be manipulated and formatted. Specifically, you will take information from a single column and split it into multiple columns using two different methods.

1. Open the Notepad file titled “date.txt”.
2. Highlight all the data starting with the first row down to the final row.
3. Copy this data by right-clicking on the highlighted data and selecting “copy”.
4. Open a Microsoft® Excel® worksheet and select the entire first column by clicking on the “A” at the top of the column.
5. Right-click on the “A” and select “Format Cells”.
6. In the “Category” box pick text, and then click “OK”.
7. Select cell “A1”, then right-click on the cell, and select “paste”.

You will notice that the data has been pasted so that the year, month, and day values are contained in a single column. To make it so that each value has its own column, you will use the “Text to Columns...” command.

1. Select the entire column by clicking on the letter “A” directly above the column.
2. Select the “Data” menu and click on the “Text to Columns...” command (this will bring up the “Convert Text to Columns Wizard”).
3. Because the data is separated by the character “/”, toggle “Delimited” and select “Next”.
4. Check the box titled “Other” and type the “/” character (without quotation marks) into the box provided to the right of the “Other” delimiter.
5. Click “Finish” to exit the wizard.



You should now see three columns that represent values for year, month, and day

The preceding example shows how data can be separated based on a delimiter such as a comma, space, or other character. However, not all data has a distinct character that divides values. In cases such as these it becomes necessary to divide data based on width. The following steps illustrate how this may be done.

1. Open the Notepad file titled “date2.txt”.
2. Repeat steps 2 – 9 (above.)
3. Toggle “Fixed width” and click “Next” (because the data is not separated by any distinct characters, we divide the column based on width.)
4. In the “Data preview” window, create a break line between the year and the month by clicking between the 2 and the 1 on any row, and then make another break line between month and day by clicking between the 1 and the 0.
5. Click “Finish” to exit the wizard.

You should now see three columns that represent values for year, month, and day.

#### **Note**

When copying and pasting data from Notepad into Excel®, it is useful to format the destination column to be “text” before you paste—the text format will not alter a number or delete zeros that precede a number. Also, make sure the cells to the right of the column you are dividing are empty.

## **Using the AutoFilter**

In this exercise you will be utilizing the AutoFilter option to locate various cells in an Excel® spreadsheet and modify them.

1. Open the Excel® file “filter.xls”.
2. Select column “C” by clicking on the “C” at the top of the column.
3. Click on the “Data” menu and select “Filter”, then select “AutoFilter”.
4. You will see a small box with a black arrow inside (the AutoFilter box) appear in the first cell of the column. Click on the AutoFilter box.

The dropdown menu that appears under the AutoFilter box shows the unique values of all the cells below the AutoFilter box. By selecting one of

the values you are determining which rows will be displayed, and which rows will be hidden.

5. Scroll to the bottom of the AutoFilter dropdown menu and select “(Blanks)” to display the blanks cells in column “C”.

All rows that do not have blank values in column “C” have been hidden. The row numbers for the remaining rows have been changed to blue instead of black.

6. Enter the number “0” in the first blank cell of Column “C”.
7. Repeat step 6 for each cell in column “C” that has a row number written in blue (you can use the AutoFill capabilities of Excel® if you wish).
8. Click on the AutoFilter box again and scroll to the top of the menu, then select “(All)” to show all rows.

Now there are no blank cells in column “C”.

The AutoFilter can be used in a variety of ways, and the following exercise will walk you through additional uses. Let’s say we are only interested in looking at the precipitation values that are larger than 0.1. The following example shows how we can customize the display options of the AutoFilter.

1. Click on the AutoFilter box in the first cell of column “C”.
2. In the AutoFilter dropdown menu select “(Custom...)” to modify which cells you would like to display.
3. Click the top-left drop down box and select “is greater than or equal to” for the AutoFilter to display all rows that are greater than or equal to the value you specify.
4. In the next box over (to the right) type “0.1” (without quotation marks.)
5. Select “OK”.

Now you can view all the cells in column “C” that have values of 0.1 or larger. If you want to delete the cells smaller than 0.1 you would have to change the custom filter to display the cells less than 0.1, then you could select the displayed rows and delete them.

## Find And Replace

The find and replace command can be very useful when formatting data. This tool can be used to delete, replace, or modify cell values. The following exercise will show you how to use the replace command, as well as how it can be applied to formatting data.

1. Open the Excel® file “replace.xls”.

2. Select column “C” by clicking on the “C” at the top of the column.

In column “C” you will notice that the values are hours of the day, followed by a “z” which stands for “Zulu” or Greenwich time. If we want to use these numbers in GSSHA™ we will have to remove the “z” from each cell. To do this we could split the column, however, using the replace command works equally as well.

1. Select the “Edit” menu, and then click “Replace...”
2. In the text box below “Find what:” type “z” (without quotation marks.)
3. Leave the text box below “Replace with:” empty, which means that the “z” in each cell will be replaced with nothing.
4. Select “Replace All” to replace all the z’s in column “C” with nothing.

Instead of replacing the z’s with nothing, you could have replaced it with a letter or a number. If you wish, try repeating the exercise, and at step 5 enter in a number or word or whatever you would like. The replace command can be used to replace words, letters, or numbers, in rows or in columns.

## General Information

The following commands can be useful when formatting data in Excel®

- Saving an Excel® file as a text file
  - Make your active sheet the one you would like to save as a text file.
  - Select the “File” menu, and pick “Save As...”
  - Enter a new name for the file in the box next to “File name”.
  - Select the drop down box for the “Save as type” and pick: “Text (Tab delimited) (\*.txt)”.
  - Then click “Save”.
  - Click “OK” at the prompt.
  - Select “Yes” at the next prompt.
  - The text file will be saved in whatever folder you specified.
- Moving Columns
  - To move an entire column, click on the letter at the top of the column.
  - The entire column will be selected, with two heavier black lines on either side of the column.
  - Put the pointer over one of these heavier dark lines, then when the cursor changes to an arrow, click on the black line.

- While holding down on the left mouse button, drag the column to the destination of your choice.

## 11 Manual Calibration

We shall practice calibrating a simplified model. It does not have ET or long-term turned on. This data set is more to help you become familiar with the WMS interface for manually calibrating your own model than it is to end up with a calibrated model. If you have WMS running, close it and open up a new instance.

1. In the 2D Grid Module, select GSSHA™ | Open Project File...
2. Browse to the Finished\_Tutorials/calibrate/start folder. Open the base\_calib.prj file.
3. Select GSSHA™ | Read Solution... Make sure that the base\_calib.prj file shows up in the dialog. Hit Ok.

This is the project we shall use as the starter project. Let's look at the data.

1. Zoom in around the outlet. You'll notice a short stream that connects the outlet to the rest of the stream network.
2. In the Map Module, using the Select Feature Node tool, double-click the node at the upstream side of that short segment.
3. Click on the Observations button.

This dialog (shown in the following figure) is used to compare solution results with each other and with the observed data. Let's load the observed data.

1. Open up Microsoft® Excel®. Open the observed.xls workbook in the Finished\_tutorials\calibrate folder.
2. Copy cells A2 to B202.
3. In WMS, in the Observations dialog, select the drop-down box under the observed column in the upper pane. Select the Edit field.
4. Paste the data into the XY Series editor. Look at the plot of the data and click OK.

Notice that under the observed data column, in row 1, it says 'Show.' The show and hide commands alter the display of the data in the two plots in the lower part of the Observations dialog. Now let's display our simulated data so that we can see what is happening.

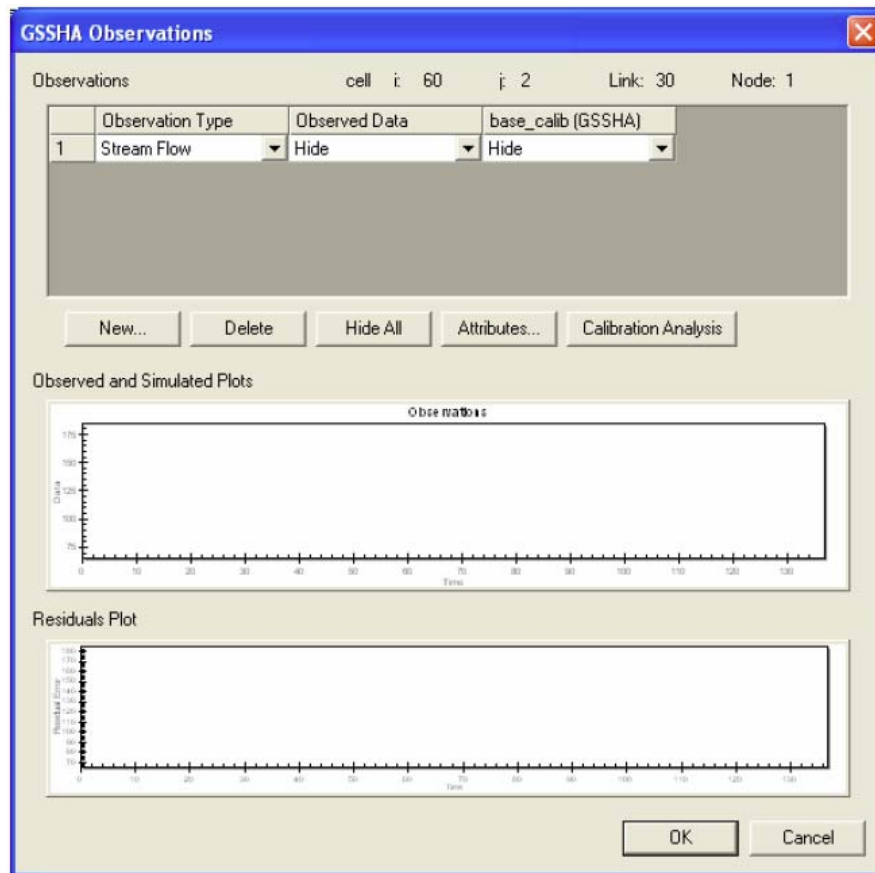


Figure 34. The GSSHA™ observations dialog.

1. Under base\_calib change hide to be show.

The upper plot window now shows plots of the observed and predicted data. The lower plot window takes the difference between the two (the residuals) and plots it. Let's look at how well our simulation matches the observed data.

2. Select the Calibration Analysis button.

The calibration analysis dialog computes a few objective functions for the set of residuals. Once you are done looking at the calibration analysis dialog:

3. Select OK, OK, OK to get back to the main WMS screen.

Your next goal is to create a few more projects so that we can compare data.

1. Select a parameter that you would like to modify and change it. (Try stream or overland roughness values to start with.)
2. After creating a new folder, save the project in the new folder with a different base file name than 'base\_calib.'
3. Run the model.

Once you have run another model you can go back to the same feature node as before to look at what changed in the output and also compare the new output data with the old and with the observed data set.