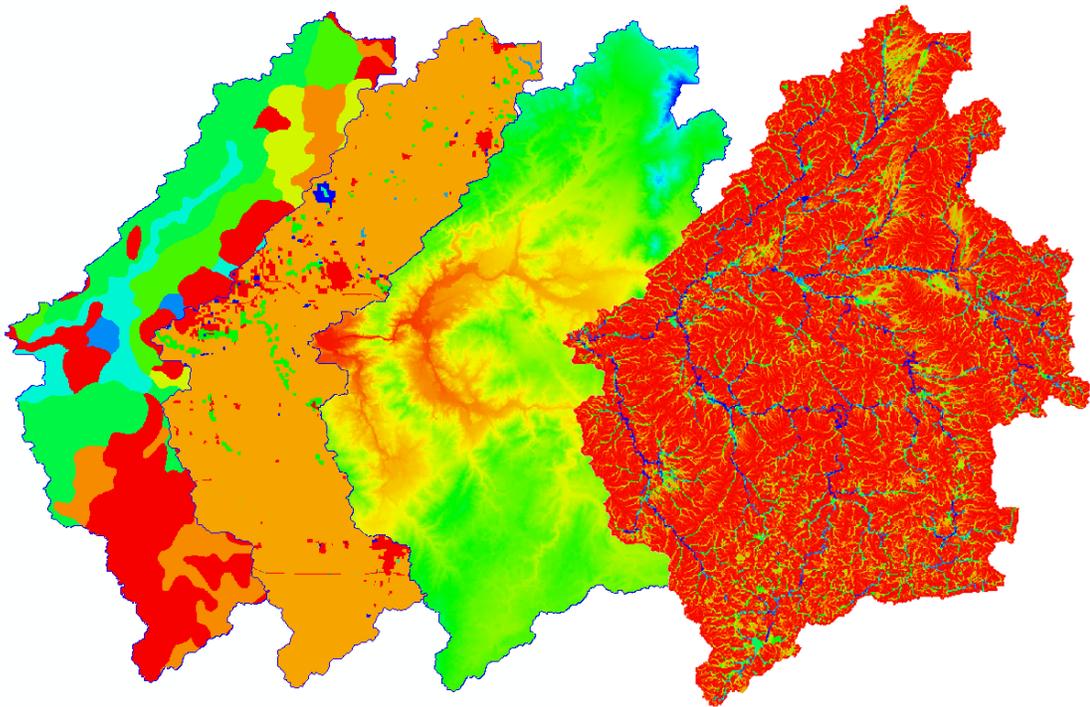


<<draft>>

GSSHA 2.1d Tutorials



Aaron Byrd

Ryan Billen

US Army Corps of Engineers
ERDC-CHL

<<draft>>

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

<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.**

For more information on WMS, including copyright, licensing, and disclaimers, contact EMS-I at <http://www.ems-i.com>.

GSSHA 2.1d Tutorials	1
GSSHA 2.1d / WMS 7.1 Tutorials Overview.....	3
1 Initial Overland Flow Model Setup	5
Initial Basin Setup.....	5
Initial Grid Setup.....	7
Job Control Setup.....	9
Uniform Index Map Setup	10
Roughness Table Setup.....	11
Setting the Uniform Precipitation	12
Save the GSSHA Model	13
Running the Model	14
2 Visualizing Overland Flow Results	16
3 Fixing Digital Dams.....	19
Working with Digital Dams.....	19
Manually Adjusting Cell Elevations.....	22
Using Cleandam to Fix Digital Dams.....	24
Using Topaz to Fix Digital Dams	25
Comparing Topaz and Cleandam Results.....	25
4 Using the Mapping Table and GIS Data.....	31
Using Land Use Data	31
5 Stream Flow	35
Creating GSSHA Stream Arcs.....	35
Smoothing the Stream Arcs	38
6 Visualizing Stream Data	40
7 Break-point Cross Sections.....	43
Adjusting the Stream Course	45
Adding an Embankment	46
Adding Structures	49
8 Infiltration	52
Index Map Setup.....	52
Visualization	56
9 Long-Term Simulations	59
Precipitation	59
Hydrometeorological Data.....	60
10 Using Microsoft Excel to format GSSHA data.....	63
Splitting an Excel column.....	63
Using the AutoFilter	64
Find And Replace	65
General Information.....	66
11 Manual Calibration	67

GSSHA 2.1d / WMS 7.1 Tutorials Overview

These tutorials are designed to guide a new user through the setup of a GSSHA project (for GSSHA version 2.1d) in WMS, version 7.1. GSSHA projects are set up in stages. The overland flow model forms the foundation of the model, and all other models build upon it. The other models that comprise GSSHA include rainfall, stream flow, infiltration, evapotranspiration, groundwater, and sediment and contaminant transport. This tutorial assumes that the user is familiar with how to use WMS and with basic hydrology. For more information on the process models used in GSSHA and a description of data file formats that GSSHA reads, consult the GSSHA Users' Manual.

Throughout this tutorial you will be setting up a GSSHA model of Judy's Branch. Judy's Branch is a small watershed (8.4 square miles) just outside of St. Louis in Illinois. The small town of Glen Carbon is located inside the watershed.



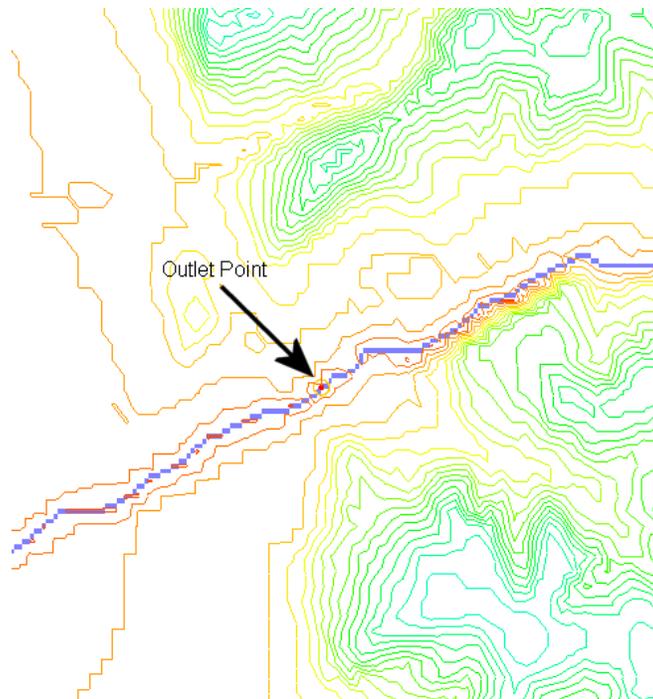
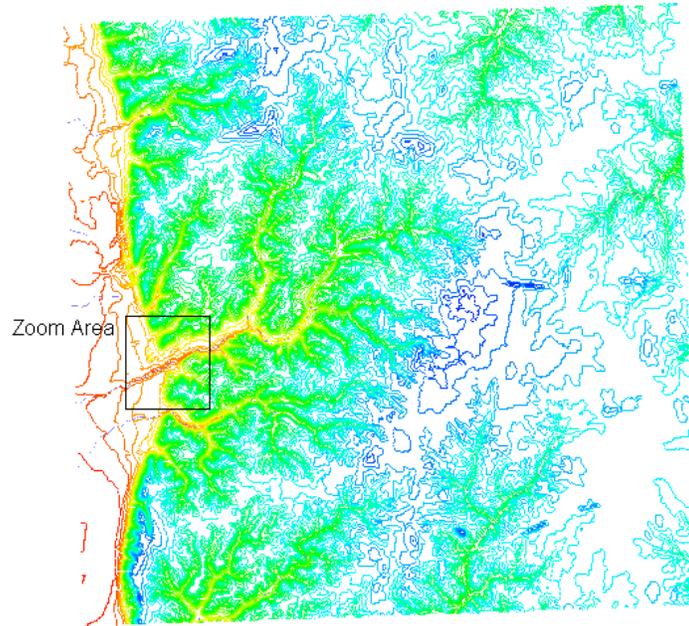
If you are taking the WMS/GSSHA training course, the data for this set of tutorials will be located in the C:\projects\Judys_Branch_tutorial folder. If you want to download the data you can obtain it at <http://chl.erdc.usace.army.mil/gssha>.

1 Initial Overland Flow Model Setup

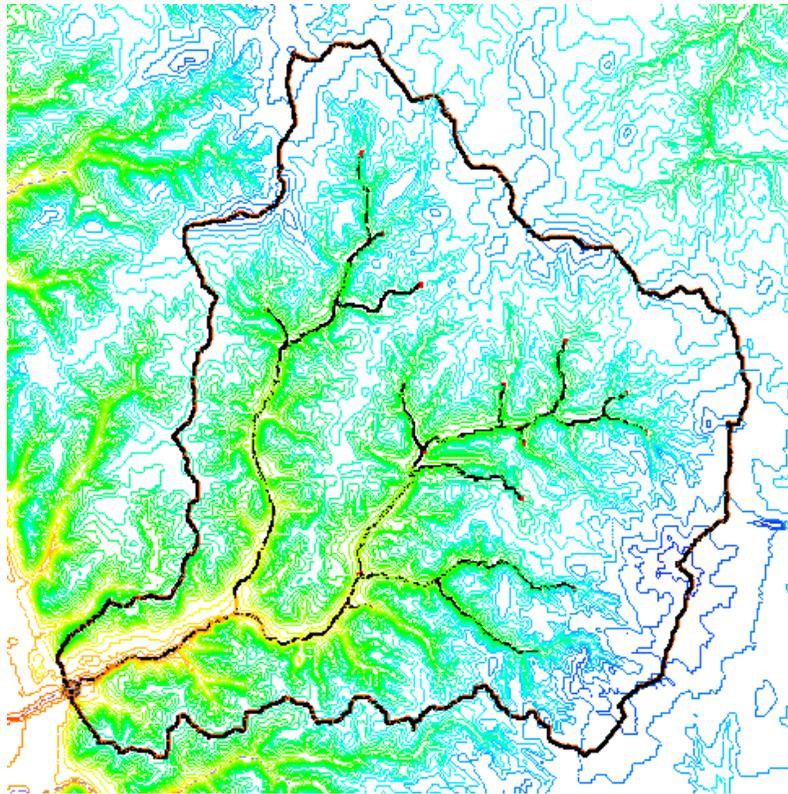
Initial Basin Setup

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 as well as to 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 not (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 7.1 Tutorials.

1. Open the DEM (select the Judys_branch.hdr file in the DEM folder. When the convert coordinates now dialog pops up say yes and convert from Geographic NAD 83 to UTM NAD 83 Zone 16.)
2. Run TOPAZ.
3. Locate the Outlet. The outlet location is shown in the following figures.



4. Create the Basin and Stream Arcs using the Delineate Basin Wizard. (Use a threshold value of 0.2.) The finished basin should look like the following figure.

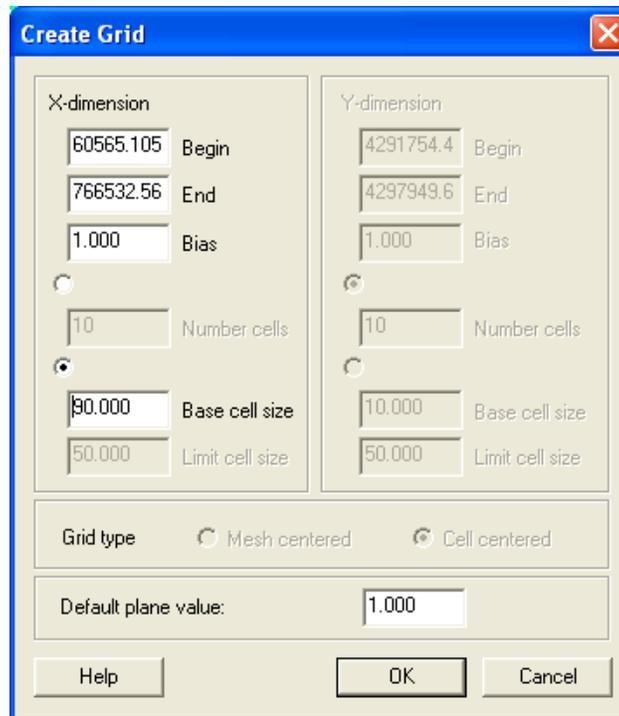


5. Save your project.

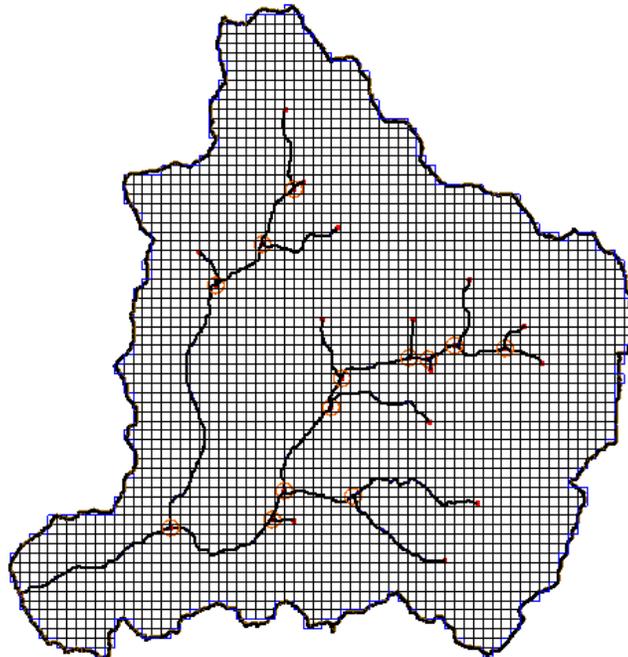
Initial Grid Setup

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.

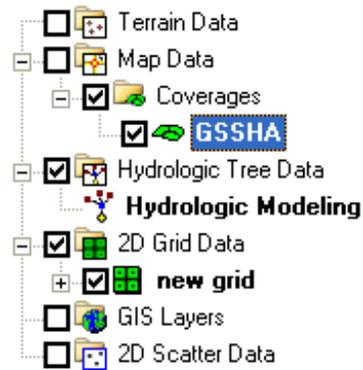
1. Begin from Basin Setup (You can load the delin_basin.wpr file in the Finished_tutorials\delin_basin folder if you are starting from here.)
2. In the Map Module , using the Select Polygon tool , select the basin boundary polygon.
3. Select **Menu | Create Grid...** (Select Yes for GSSHA grid.)
4. Select the second toggle, the one for base cell size, and enter 90 (meters.) Select OK.



5. Hit OK on the Background Elev Interpolation dialog.
6. Hit No on the Delete Existing Feature Object Data dialog.
7. Hit Yes on the Delete Background DEM dialog. The basin should now look like the following figure.



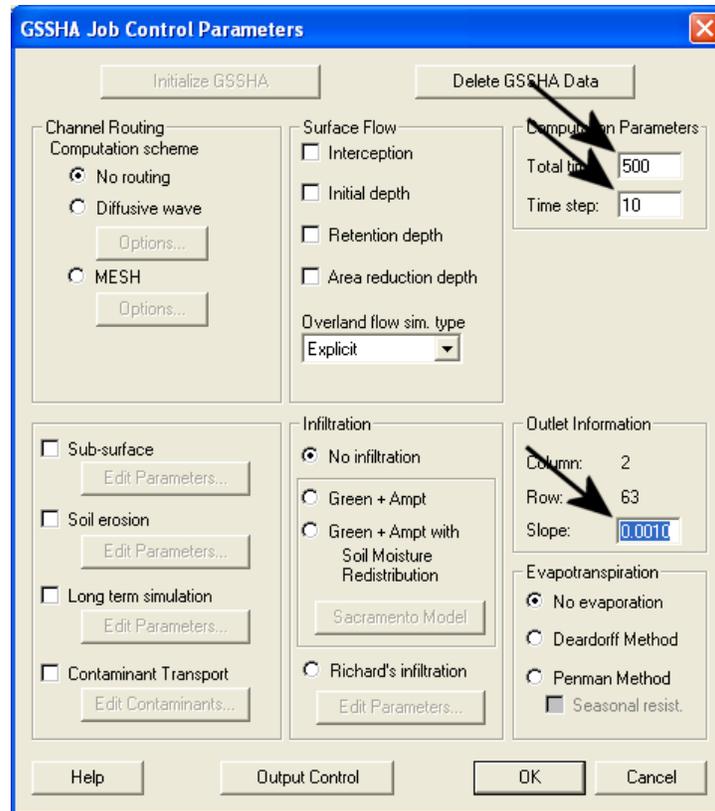
You'll notice in the data tree that the Drainage coverage has now changed name (and changed type as well) to the GSSHA coverage. A 2D Grid called new grid is also in the tree now.



Job Control Setup

In the last step, the GSSHA Job Control parameters were initialized. The Job Control parameters have been set up with default values, which are mostly zero. It is good to start with some better 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.)

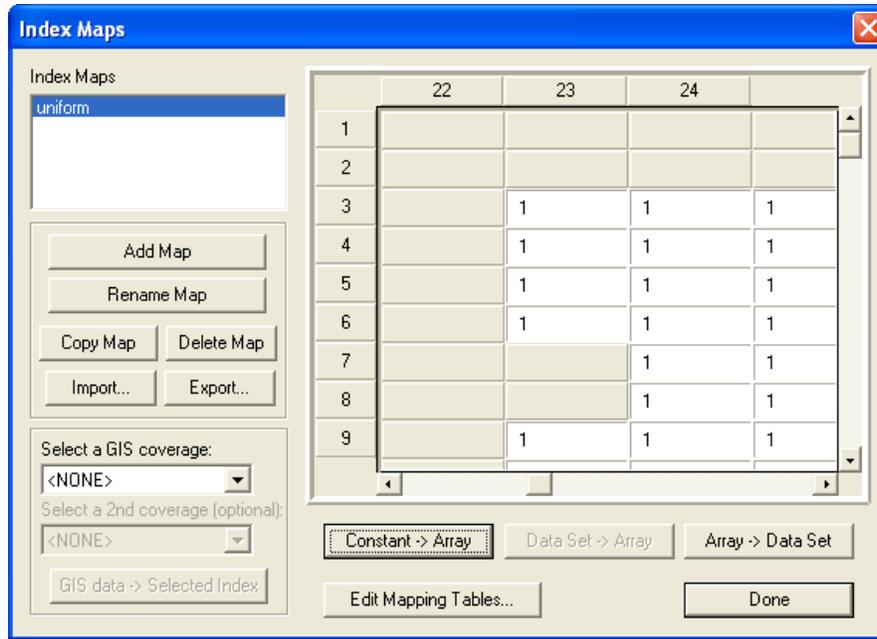


4. Select OK.

Uniform Index Map Setup

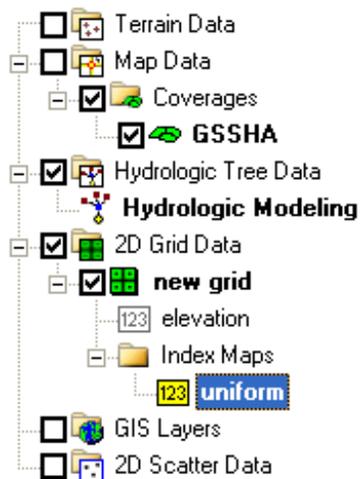
Once the Job Control parameters are set to better values there are two main areas to set up for overland flow. First, the overland flow roughness coefficients need to be set. Secondly, the precipitation data needs to be specified. There are two parts to setting up the overland flow roughness coefficients; 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 the first index map, Index Map – 1, is created.)
2. Select Rename Map.
3. Type in “uniform”, Select OK.
4. Select the Const -> Array, enter 1, Select OK. You can scroll around the grid to see the cell values.



5. Select Done.

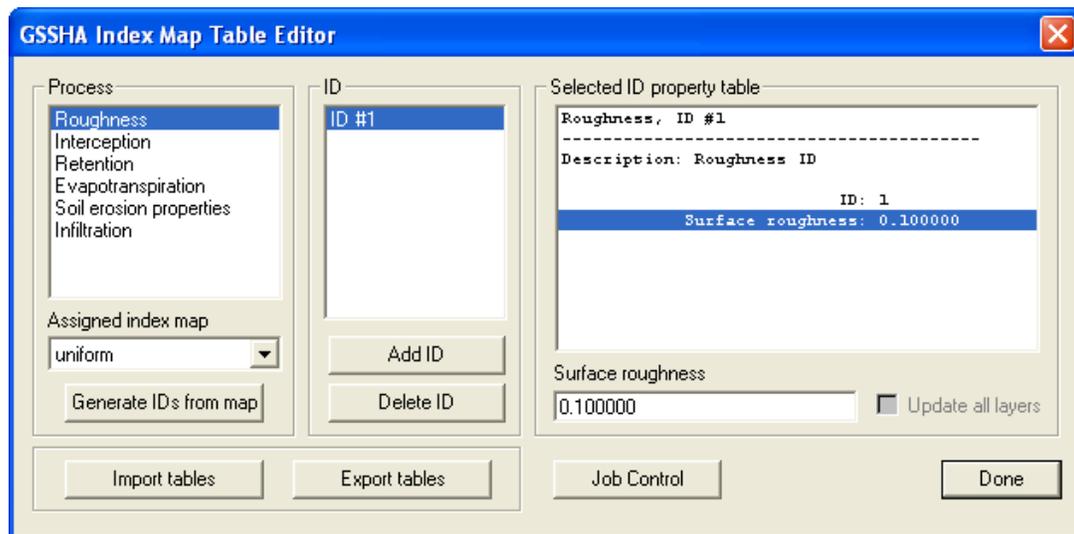
We have just populated an index map (a grid) with the value 1. If you expand the data tree you will notice that our new index map has been added to the index maps folder under the 2D grid folder.



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. In the Process window select Roughness.
3. In the Assigned Index Map combo box select “uniform”.
4. Select Generate IDs from map.
5. In the ID window select ID #1.
6. In the Selected ID Property Table select the Surface Roughness line.
7. In the Surface Roughness edit field enter a value of 0.1.



8. Select OK.

The list of names in the left most pane of the Mapping Table dialog lists some of the mapping tables that can be set up. Which ones appear depend in part on the options set in the Job Control dialog. 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 shall set up a single rainfall event. To simplify the process, we shall set up a simple uniform precipitation event for a short duration.

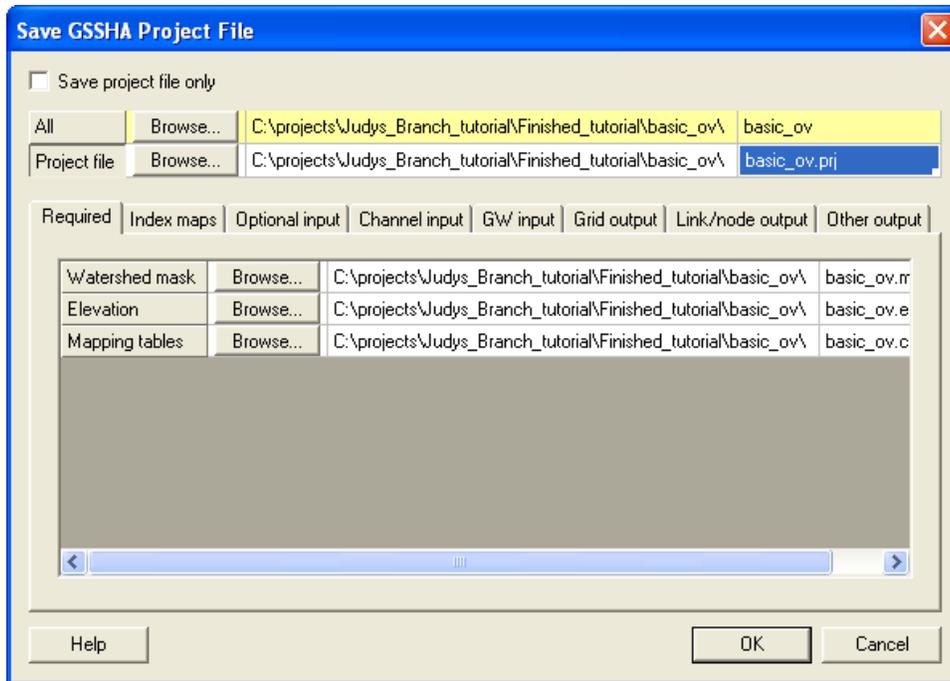
1. Select **GSSHA | Precipitation ...**
2. Select the Uniform Rainfall toggle.
3. Enter the rainfall intensity of 10 (mm/hr.)
4. Enter the rainfall duration of 60 (minutes.)

5. 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.)



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.

<i>Extension</i>	<i>Description</i>
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 tutorial.

2 Visualizing Overland Flow 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 of 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 tutorial 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, lets 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. Lets 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.

Click around some more on the time steps. Try panning, zooming, and rotating. You can also adjust the lighting (**Display | Shading 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 Filmloop 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.

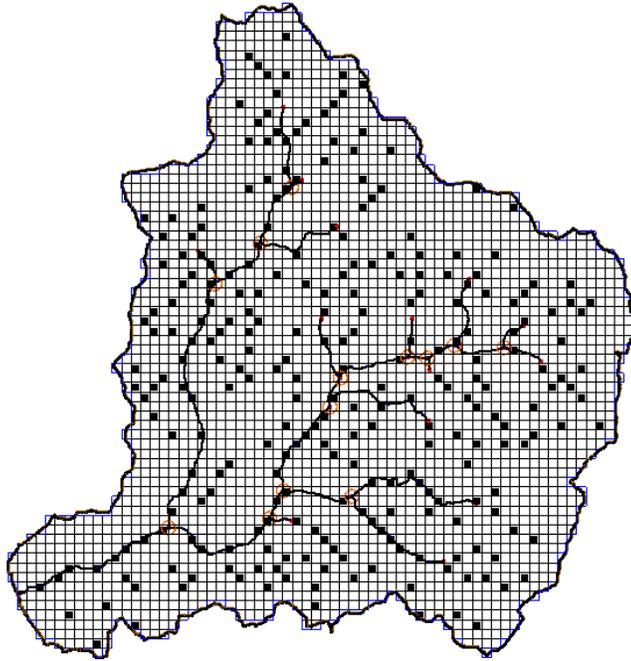
Now that we have a movie of the runoff depths, it is fairly easy to see that the water has stabilized by the end of the run to sit in many little puddles on the grid. These puddles are the result of the digital dams. Thus our lack of runoff is not due to time constraints (currently, anyways) but due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

3 Fixing Digital Dams

Working with Digital Dams

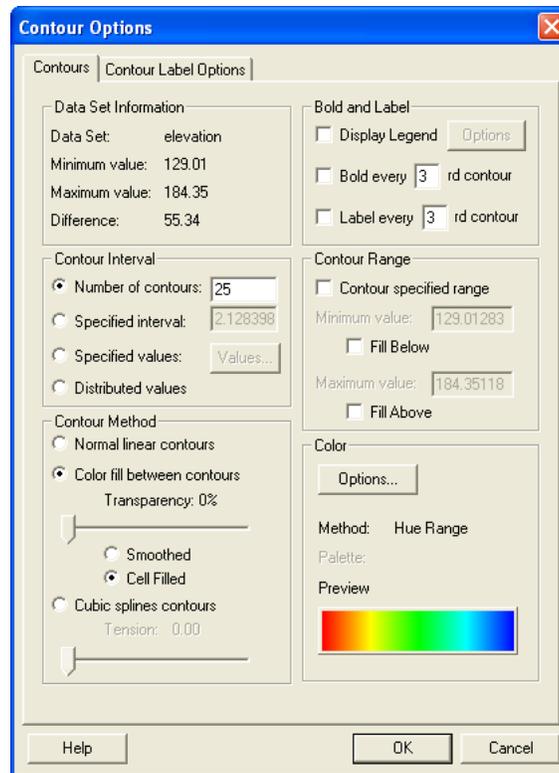
The problem with digital dams is that the water gets stuck in a cell where it shouldn't be ponding. There are three methods to fix the digital dams. The first method is to manually adjust the cell elevations. The second method is to use the cleandam tool to smooth the cell elevations. The third method 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 any good at all. 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.

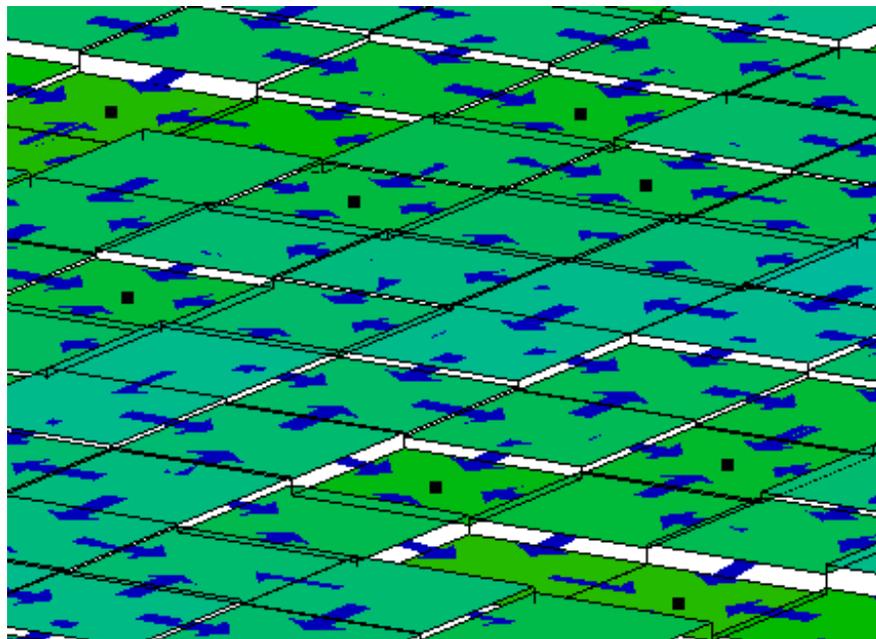


To better visualize why these cells are digital dam cells:

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.
6. Select the Cell Filled toggle.



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

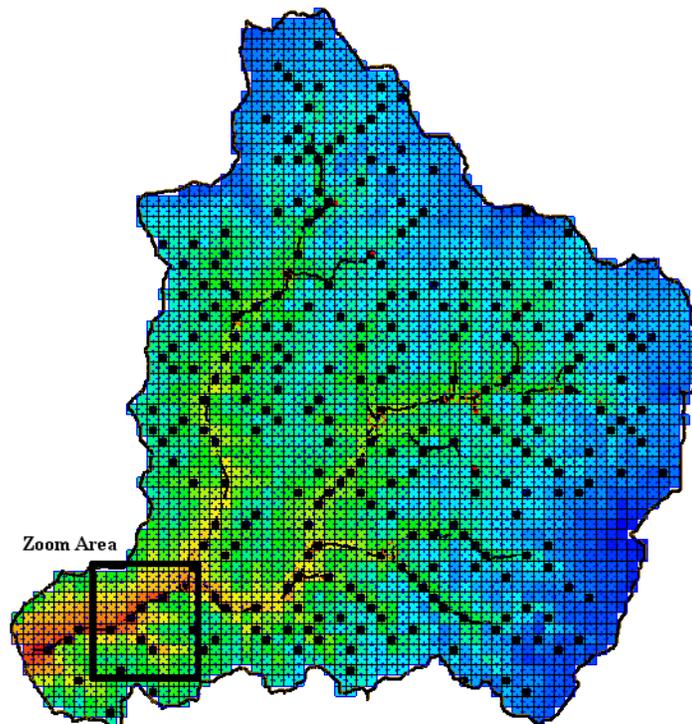


The black dots denote the digital dam cells. The arrows point out the downward slope between the cells. The cells with all four sides marked as pointing inward are flagged as digital dam cells. The blocked cells display option helps to illustrate this. You can use the **Display | View | Z Magnification**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

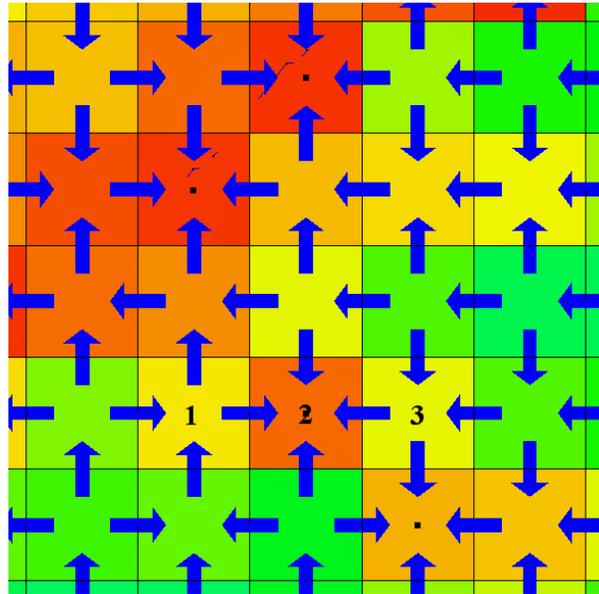
Manually Adjusting Cell Elevations

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.



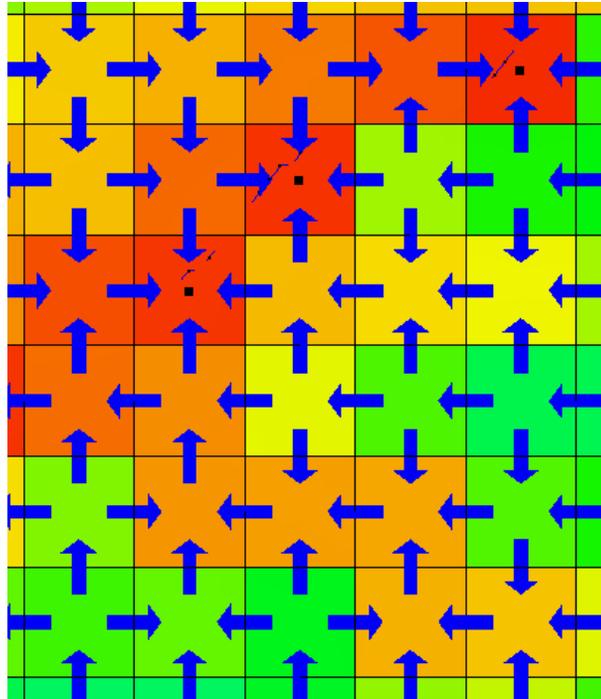
Find the cells marked in the following diagram. You'll notice that the cell marked 2 and the cell beneath 3 are both digital dams. From the contours we can recognize that there should be a diagonal channel here. Since GSSHA has a four-point flow scheme we will need to adjust some cells to make flow happen.



Using the following table, adjust cells 1, 2, and 3 by double-clicking on the cell using the Select Grid Cell tool  and edit the cell elevations to reflect the values in the table.

Cell #	Elevation
1	137.5
2	138.0
3	138.5

The grid should look like the following image now.



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 what is most likely 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. 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 that are fixed as a by-product of fixing the first digital dam are called “soft fixes.” You’ll see this as the results of the cleandam program are printed out.

To run cleandam:

1. Select **GSSHA | Clean Digital Dams...**
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. This is now the elevation data set as well.

Using Topaz to Fix Digital Dams

TOPAZ, as part of its stream routing algorithm computes a smoothed ground surface that has no pits or depressions, from an 8-point pour method perspective. From a 4-point pour method perspective the smoothed grid will still have many depressions. To run TOPAZ on the original elevation grid:

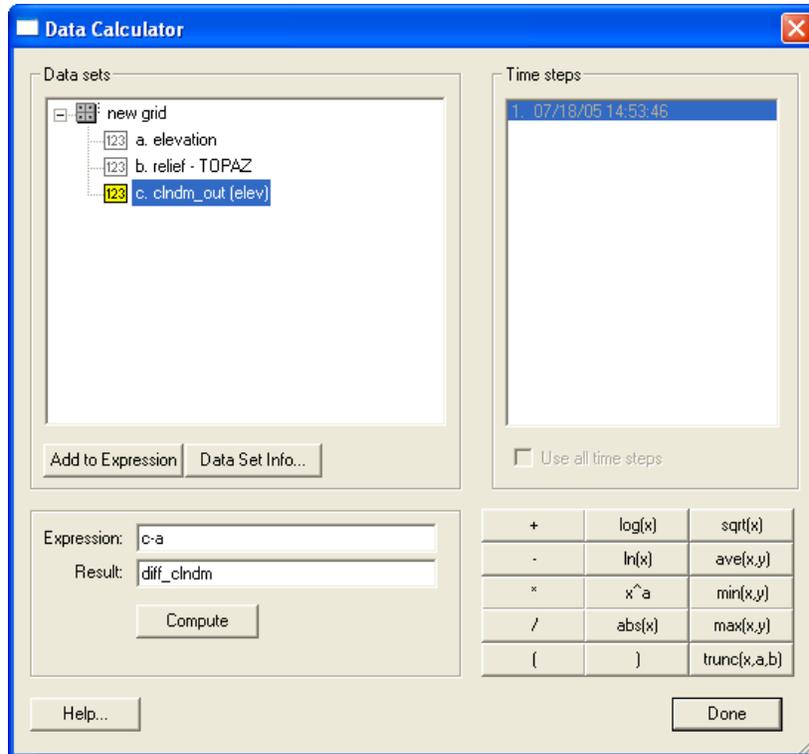
1. Right-click on the “elevation” data set under the “new grid” 2D grid. (Expand the 2D Grid folder if needed.)
2. Select “Set as Elevation Dataset.” (If it does not appear then it is currently the elevation data set.)
3. Select **GSSHA | Smooth Grid (TOPAZ)**.
4. If WMS cannot find topaz, browse to it in the WMS71 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

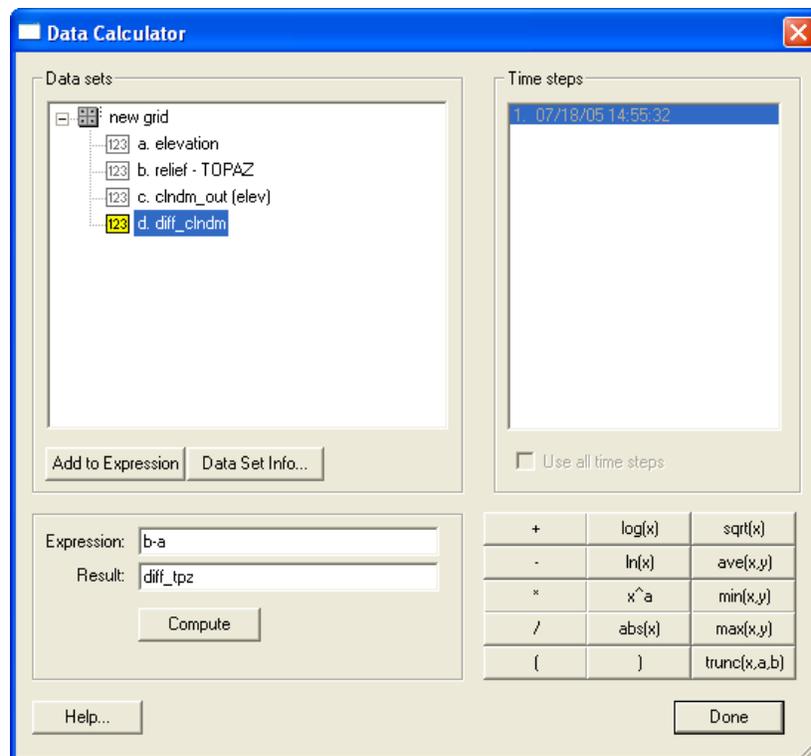
In order to understand how TOPAZ and Cleandam work we can compute the change in each cell from the original data set. To do this we will use the data calculator. The data calculator works by performing arithmetic and other operations over the data sets on a grid. For example, in the first figure below 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’.

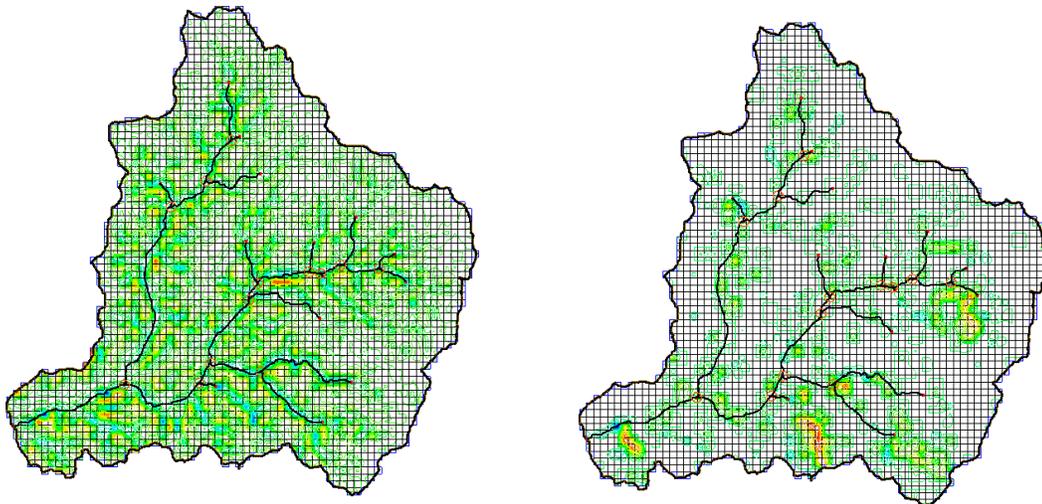
1. Create the Cleandam difference data set by entering ‘c-a’ in the expression edit field and a suitable name in the result field. Hit Compute.



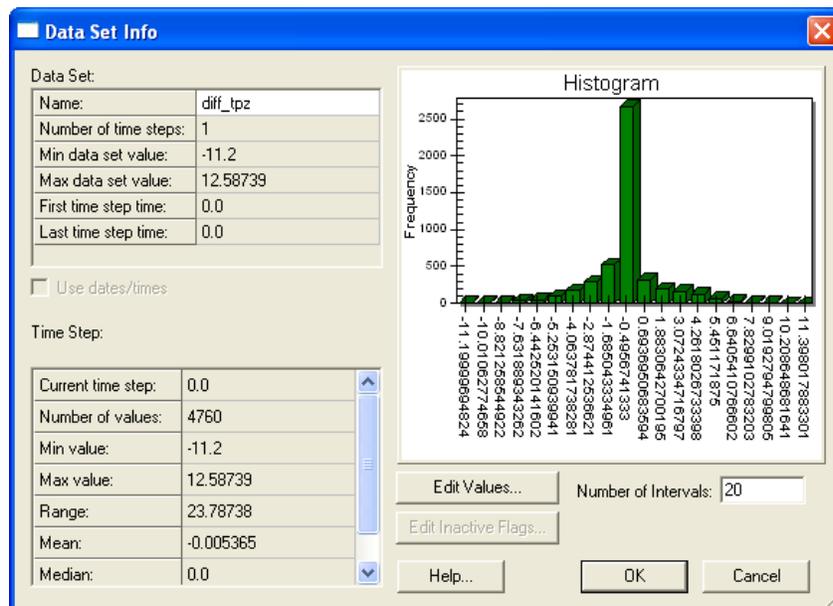
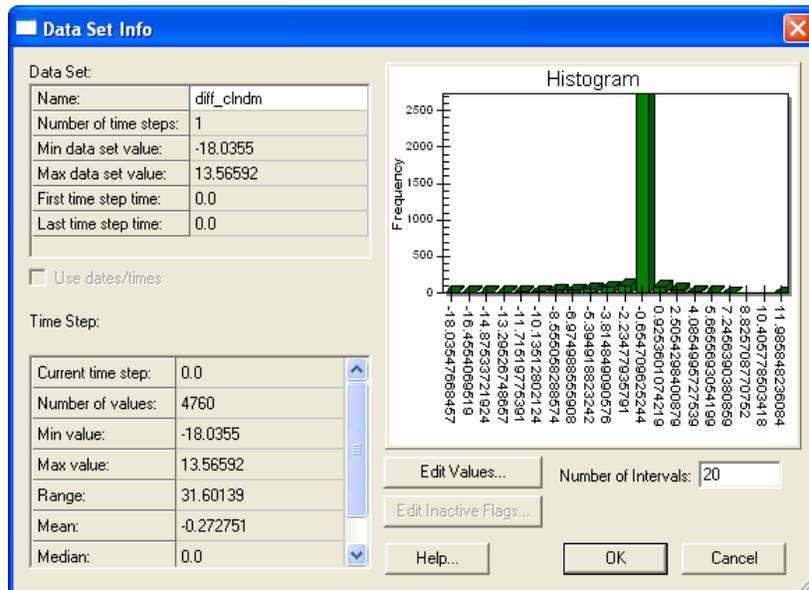
2. Create the Topaz difference data set by entering ‘b-a’ (or an appropriate expression) in the expression edit field and entering a suitable result name. Hit Compute. Hit OK when you are done.



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.



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.)

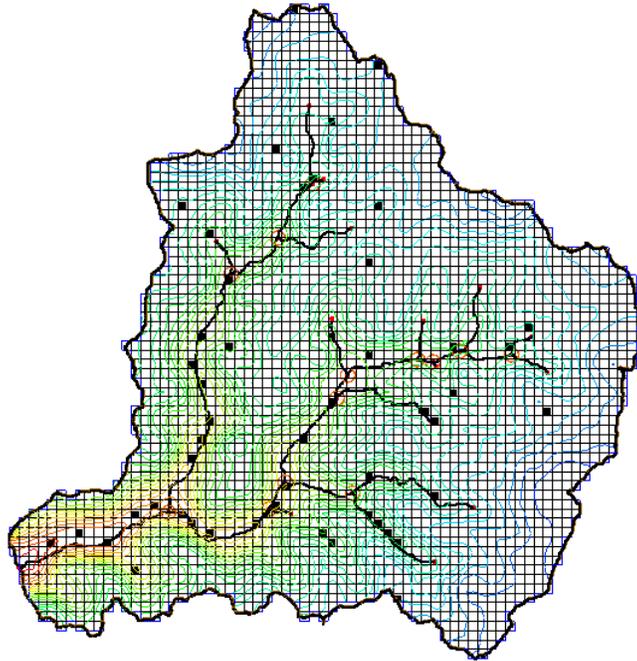


If you look at the two histograms you'll notice that they both have a large bar in the center that is due to the 0.0 values (cells that were not changed) but the bars around the

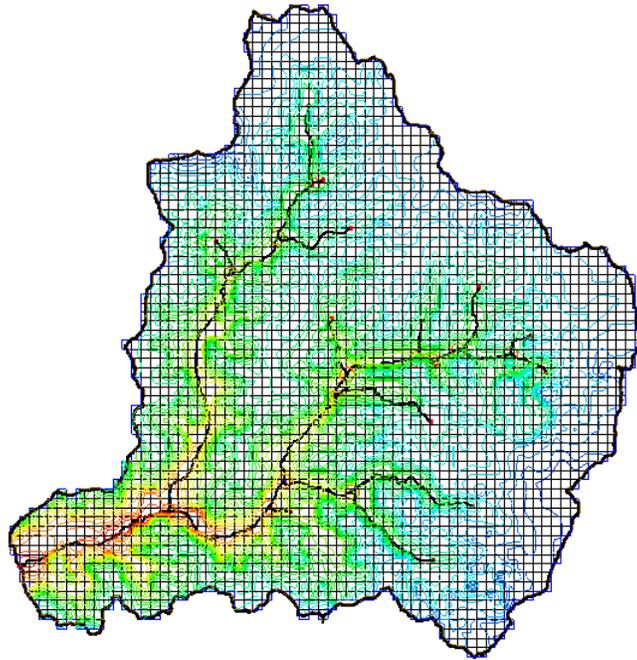
mean do have a different shape (distribution). Topaz clearly changes more of the cells, as we could see from the contours. To set a data set to be the elevation data set:

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.



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.



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.

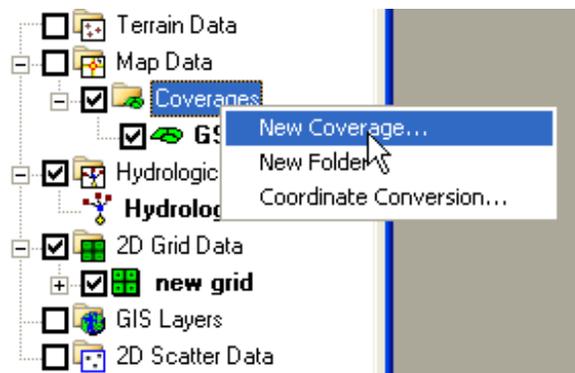
Save your project and run the model. Make a movie now to see how well the water runs off.

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 tutorial. 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 tutorial ended or you can open up the clean_ov project in the Finished_tutorial/clean_ov folder.
2. In the data tree, right-click on the Coverages folder and select New Coverage...



3. Change the Coverage Type to Land Use. Select OK.

Notice that there is now a new coverage in the data tree and that it is a land use coverage. It is also the active coverage (bolded).

4. Select **File | Open...**
5. Browse to the Landuse folder in the project folder.
6. Click on jb_luse_poly.shp. Click Open. The Import Shapefile Data dialog will pop up.
7. Click OK to accept the default attribute mappings.

We now have a land use coverage in WMS. Zoom into the GSSHA model area. 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.

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. Select Add Map.
4. Select Rename Map. Name the map Land Use. Click OK.
5. Under Select GIS Coverage pick the land use coverage. Leave the 2nd coverage at <NONE>.
6. Click on GIS data -> Selected Index.
7. Scroll around the grid to see the numbers. 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. Lets 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 process 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 now 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 while that 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.

5. Fill out the Roughness values according to the table below. To enter a roughness value for an ID, select the ID in the ID window, then select the Roughness line in the Selected ID property table. In the edit field below the property table you can enter the roughness value. (You can also edit the descriptions if you wish.)

<i>ID</i>	<i>Description</i>	<i>Roughness</i>
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

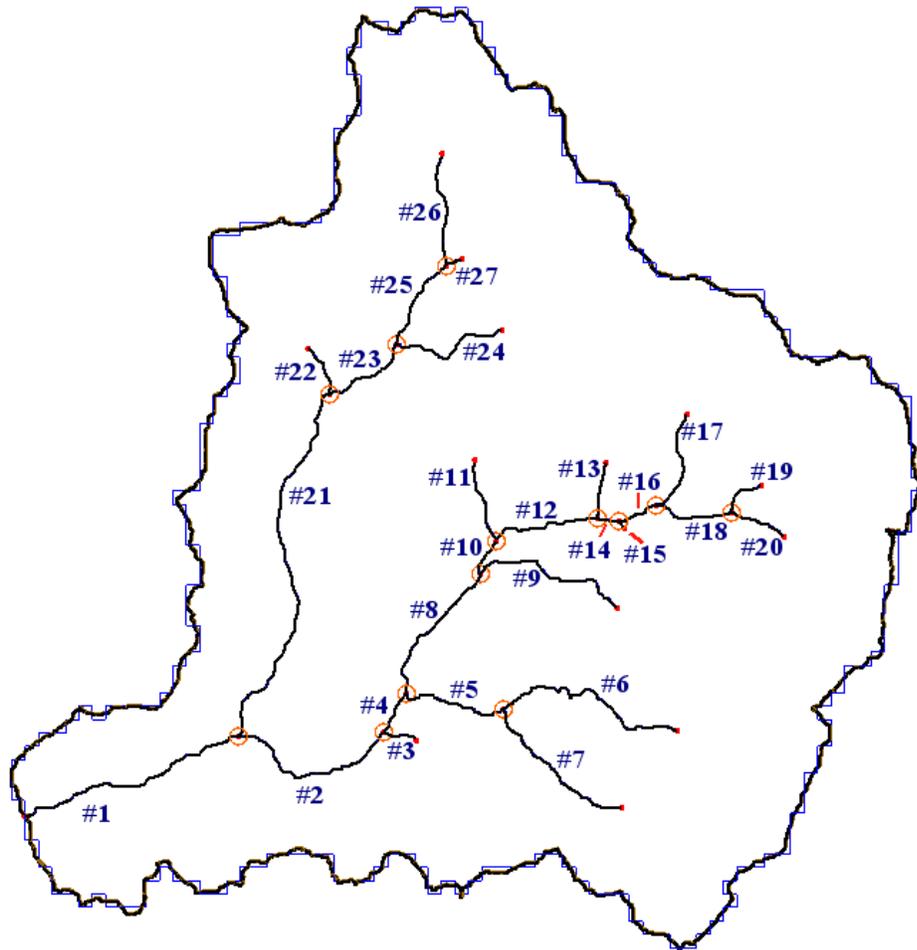
At present the GSSHA streams are generic and have no defined geometry. To specify a stream size, we will need to supply dimensions for each stream arc in the Judy's Branch basin. If you are beginning at this tutorial:

1. In the 2D Grid module  select **GSSHA | Open Project File**.
2. Browse to the Finished_tutorial\clean_ov folder of the project folder.
3. 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.

1. Select the map module of WMS .
2. Click on the "Select Branch" tool .
3. With this tool selected, click on the stream segment labeled "#1" on the following figure to select all streams.



4. Select Feature Objects | Attributes...
5. Toggle on “Trapezoidal Channel” which allows you to define stream geometry for the stream arcs.
6. Fill out the edit fields according to the following table.

<i>Parameter</i>	<i>Value</i>
Manning's N	0.07
Bottom Width	8
Channel Depth	8
Side Slope	2

7. 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 upstream arcs.

8. Switch to the “Select Arc” tool to modify streams one segment at a time .
9. Double-click stream segment “#2” as shown in the previous figure.
10. Enter the values for depth, width, roughness and slope as shown in the following table.
11. Repeat steps 9 and 10 for stream arcs 3 through 27.

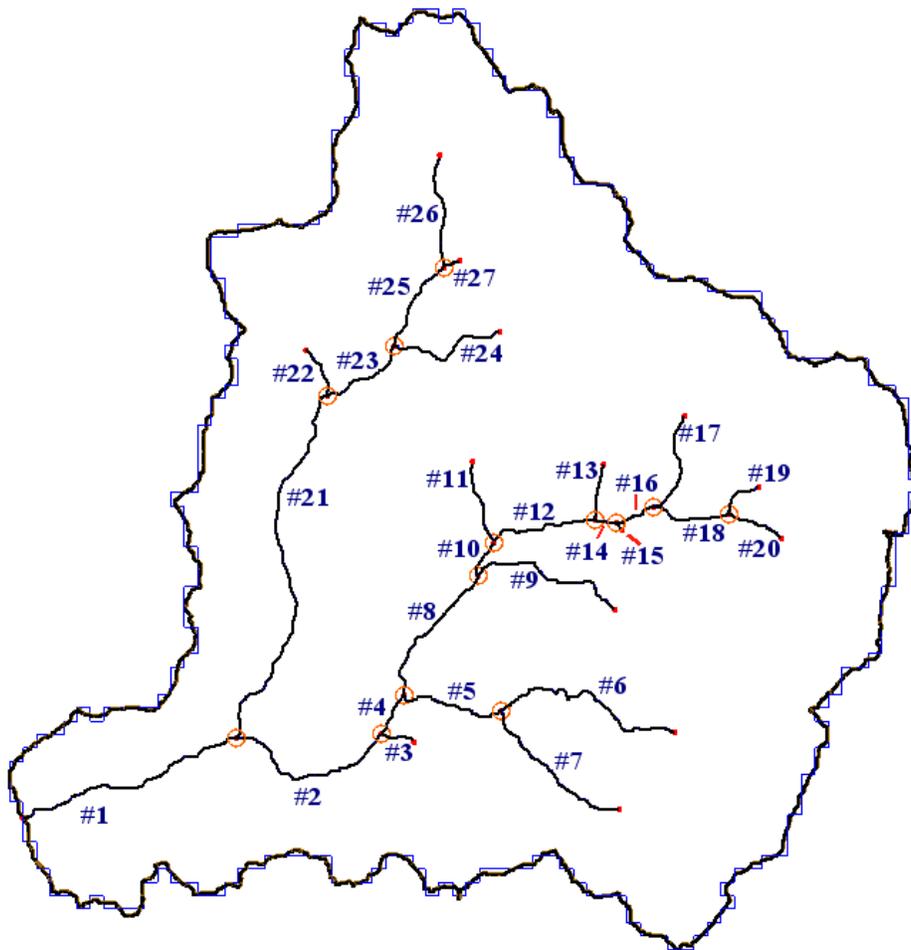
<u>Arc Number</u>	<u>Channel Depth</u>	<u>Bottom Width</u>	<u>Man. Rough</u>	<u>Side Slope</u>
2	5	5	.075	2
3	2	2	.08	2
4	5	5	.075	2
5	3.5	3.5	.08	2
6	2	2	.08	2
7	2	2	.08	2
8	5	5	.075	2
9	2	2	.08	2
10	5	5	.075	2
11	2	2	.08	2
12	5	5	.075	2
13	2	2	.08	2
14	3.5	3.5	.08	2
15	2	2	.08	2
16	3.5	3.5	.08	2
17	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.



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 mitigate 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 on Options under Diffusive Wave.
4. Check the node length to make sure that it is the correct value. Click OK.
5. 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 tutorial. The Stream Depth and Stream Flow data sets are link/node data sets. In order to visualize these we need to make a TIN. This process has been automated.

1. In the data tree select the streams_trap folder.
2. Select **GSSHA | Build Display Tin**.

In the data tree under the Terrain Data folder you'll find a TIN called New tin. If you expand that folder you will find two data sets, Stream Depth and Stream Flow. Let's look at them.

1. In the Terrain Data Module  select **Display | Display Options**
2. If TIN contours are not turned on, turn them on.
3. Click on the Contours button.
4. Change the contour method to Color fill between contours.
5. Turn off triangles and boundaries.
6. Switch to the 2D Grid tab.
7. Turn everything off except the boundary.
8. Click OK.
9. In the data tree, uncheck the box next to Coverages.
10. Select the Refresh button .
11. In the data tree, under New tin, click on Stream Depth.
12. Click on a few of the time steps.

As you scroll through the time steps (or make a movie) you'll notice that the data seems somewhat jumpy. This is because our data output time step is too large. Also, our simulation doesn't finish running the water off the watershed because we terminate too early. To fix the run time problem:

1. In the 2D Grid Module  select GSSHA | Job Control.
2. For the total time put 1000.
3. Click on Output control.
4. Under the gridded data sets, change the 30 (minutes) to be 5 (minutes).
5. Change the Hydrograph Write Frequency to 5 minutes as well.
6. Click OK.
7. Click OK.

The output control dialog is where you can select which output data sets you want GSSHA to produce, as well as how frequently GSSHA should write out the time steps.

Save the project and run it again. Once it is done you will have two solutions in the data tree. If you want to load up the new solution it is in the stream_longer folder.

1. Right-click on the stream_trap folder. Select Delete.
2. Right-click on the New tin. Select Delete.
3. Select the new solution.
4. In the 2D Grid Module  select **GSSHA | Build Display Tin**.
5. Expand the Terrain Data folder in the data tree.
6. Select the Stream Depth data set. Click on a few time steps.

Let's make a movie of the new stream data. Turn on the display options you would like and adjust the image how you would like.

1. In the Terrain Data Module  select **TIN | Data | Film Loop**.
2. Select Next, Next, and Finish.
3. Wait for WMS to build the movie.

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.

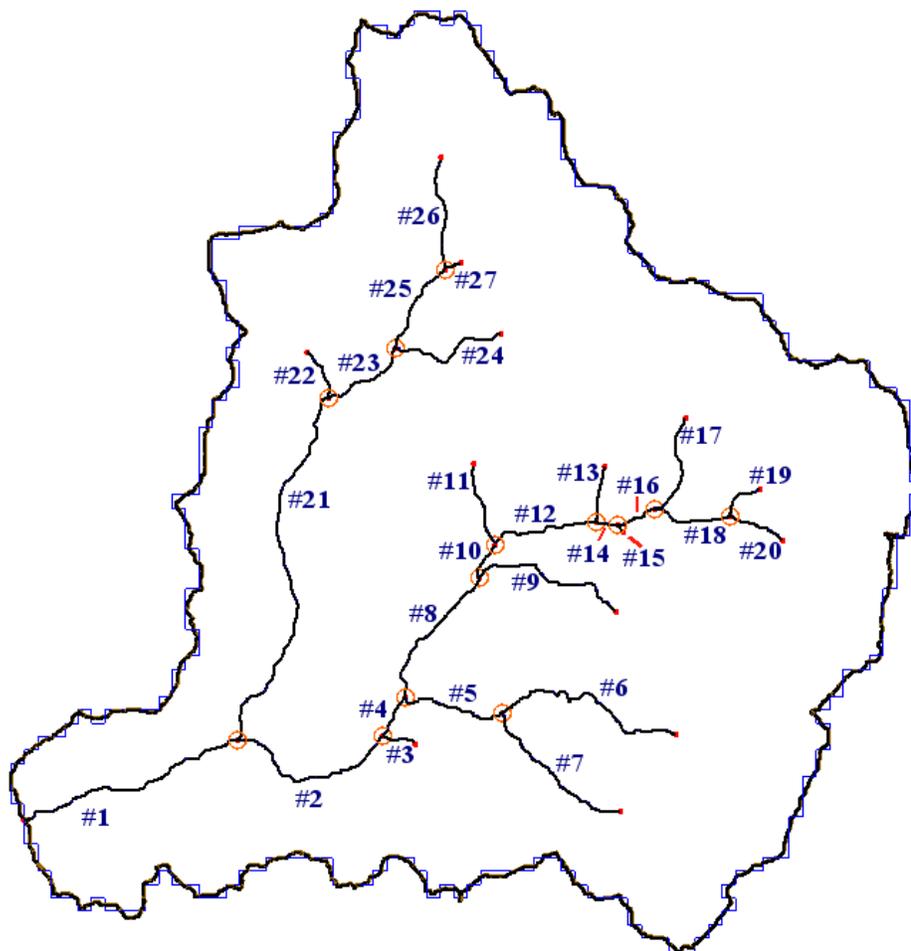
Just as when we set up the trapezoidal cross sections for the streams, all of the stream tools are in the Map Module.

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, 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 the 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

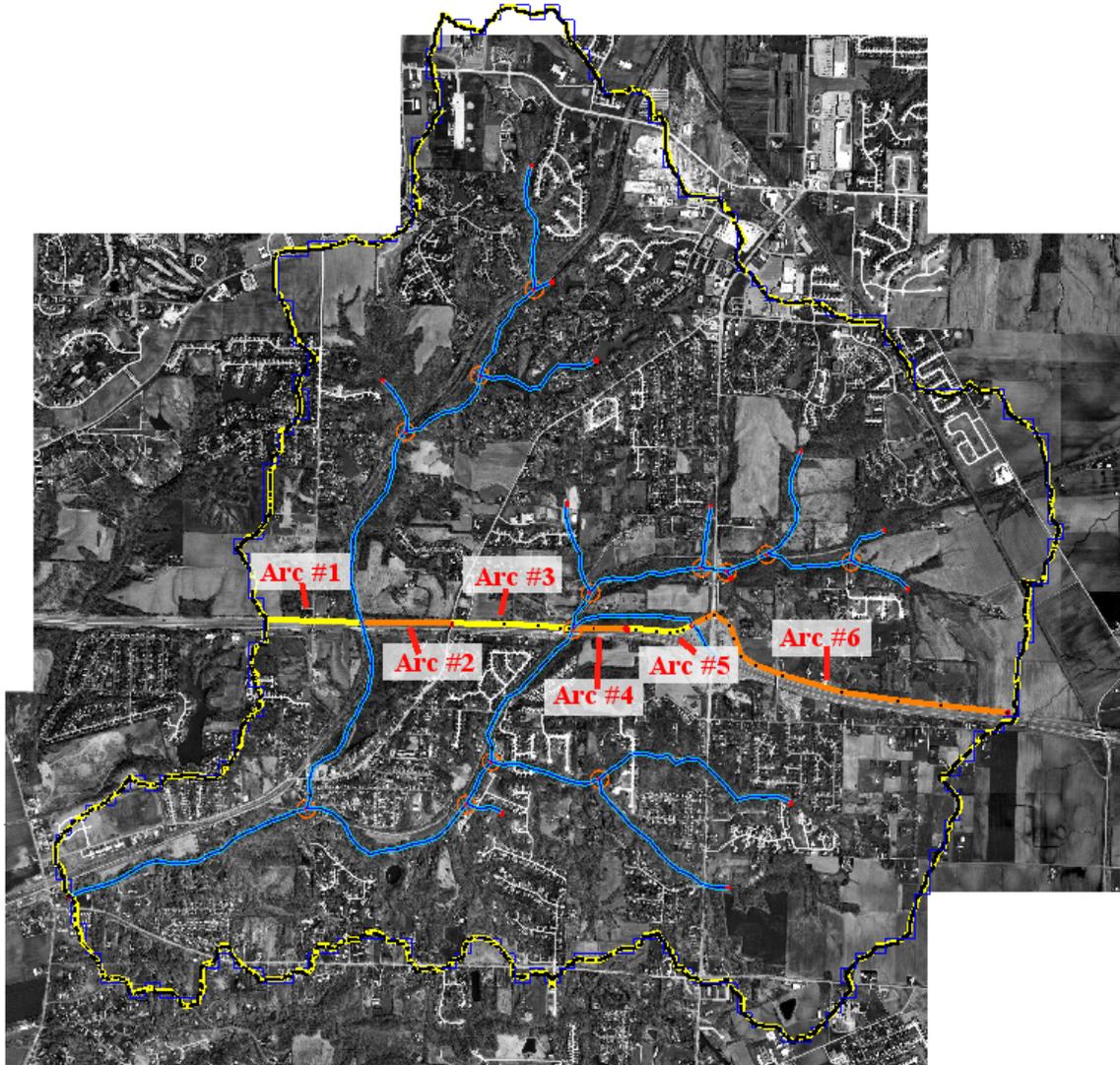
One unique aspect of the Judy's Branch basin is that it has a freeway bisecting it. This freeway acts as a barrier or embankment, which inhibits overland flow. To accurately model the basin, we need to take the embankment into account. The following exercise will show you how this can be done.

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



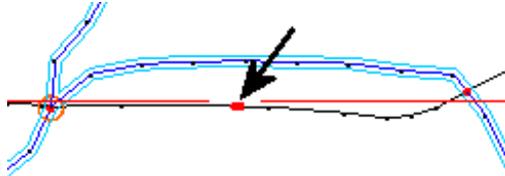
1. Select the “Create Arc” tool .
2. Select **Feature Objects | Attributes**.
3. Toggle on Embankment. Select OK.

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:

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.

- 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.

<i>Classification</i>	<i>Description</i>
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.

- Right-click on GIS Layers in the data tree.
- Select Add Shapefile Data.
- Browse to the Soils folder and open the soils_clip.shp shapefile.

The soils_clip shapefile 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.

- Right-click on soils_clip.shp in the data tree.
- Select Join Table To Layer.
- Open the IDs_for_WMS.dbf file.
- Under Shapefile Join Field select mukey.
- Under table data field select classifica.
- Select OK.
- Right-click on soils_clip.shp in the data tree.
- Select Open Attribute Table.
- 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.

- Right-click on Coverages in the data tree.
- Select New Coverage...
- Change the Coverage Type to Soil Type.

4. Hit OK.

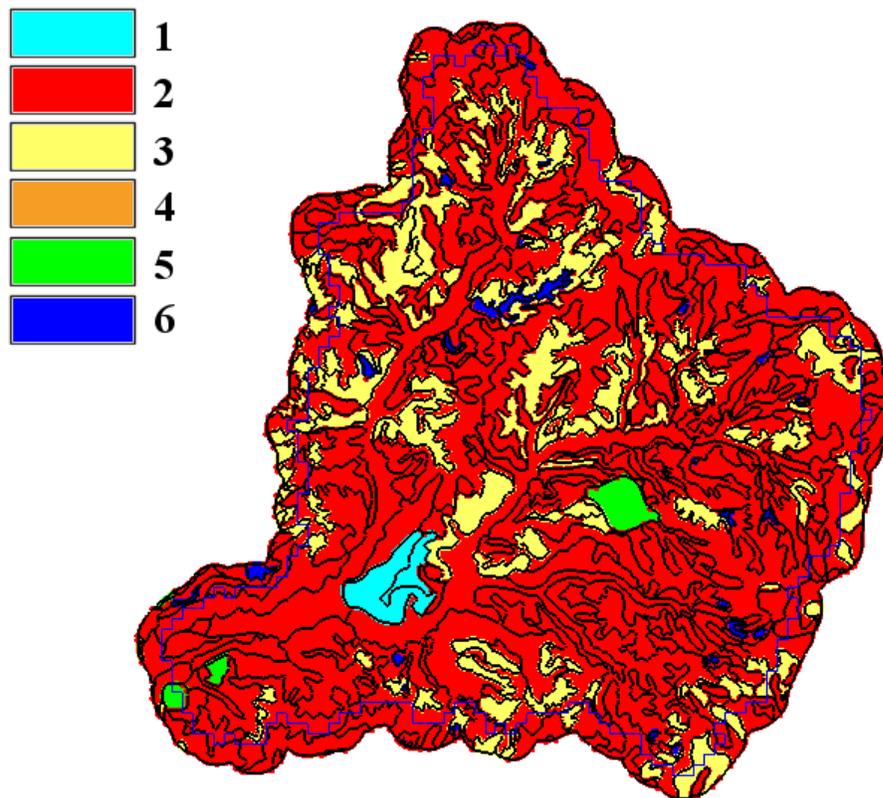
Now we can convert the shapefile to a coverage.

1. Click on the Soil Type coverage in the Data Tree.
2. In the GIS Module  select **Mapping | Shapes -> Feature Objects**.
3. Select Yes for use all shapes in visible shapefiles.
4. Select Next.
5. Scroll over to the Classifica column.
6. In the drop-down box that says 'not mapped' change it to be 'SCS Soil type'
7. Hit Next and then Finish.
8. 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. 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. Under Points/Nodes, turn off the verticies.
12. Switch to the General tab.
13. Turn on the Soil Type legend.
14. Select OK.



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 type 1 (cyan) and 6 (blue) to be what their neighbors are.

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. Select Add Map.
4. Select Rename Map. Name the map Soil Type. Click Ok.
5. Under Select GIS Coverage pick the Soil Type coverage. Leave the 2nd coverage at <NONE>.
6. Click on GIS data -> Selected Index
7. Scroll around the grid to see the numbers. 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. In the process window select infiltration.
3. In the Assigned Index map combo 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.

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 process 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 Ok.

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.

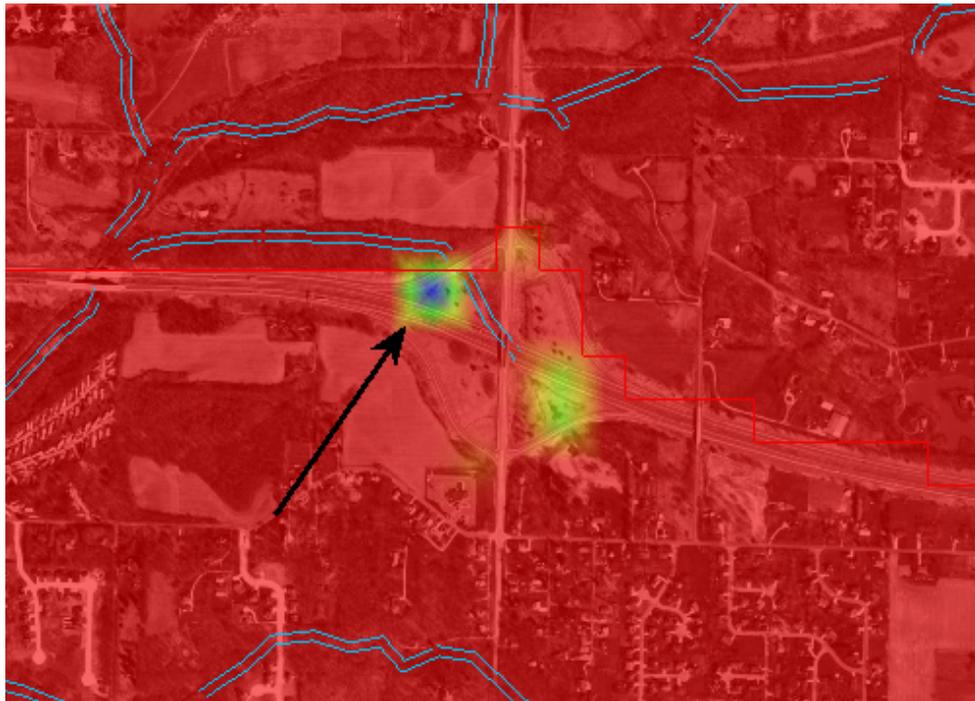
1. In the 2D Grid Module select GSSHA | Precipitation.
2. Increase the precipitation rate to 25 (mm/hr).
3. Select OK.

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 tutorial 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 tutorial 3. (Hint: make the digital dam cell flow toward the cell with the stream to the right.)



One of the more interesting movies you can make is of the InfiltrationDepth 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.

If you are starting here

1. In WMS, select the 2D Grid module and open the “more_rain” GSSHA project in the Finished_tutorial folder/infil_more_rain folder.

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 go over the “Using Microsoft Excel to format GSSHA Data” tutorial.

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” from the Formatting_macros folder.
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” from the Precip_data folder.
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 use the tutorial file “Using Microsoft Excel to format GSSHA data.doc”).
11. Use the values “243038.0” and “4294413.0” for the coordinate values on sheet “scratch2”
12. Once you have setup the data, click on the “Format Precip Data” button found on the “Instructions” worksheet.
13. Your formatted data will be displayed on the “Output_data” worksheet.
14. Set up a folder for the long term project.
15. Save this worksheet as a text file called precip.txt in the folder.

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.
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.

Turning on the Hmet options

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.

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.
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.

ID #	Albedo	Wilting Point	Vegetation Height	Vegetation Transmission Coefficient	Canopy Stomatal Resistance
12	0.15	0.1	.08	0.7	20
15	0.22	0.1	.10	0.5	20
22	0.22	0.1	1.00	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.

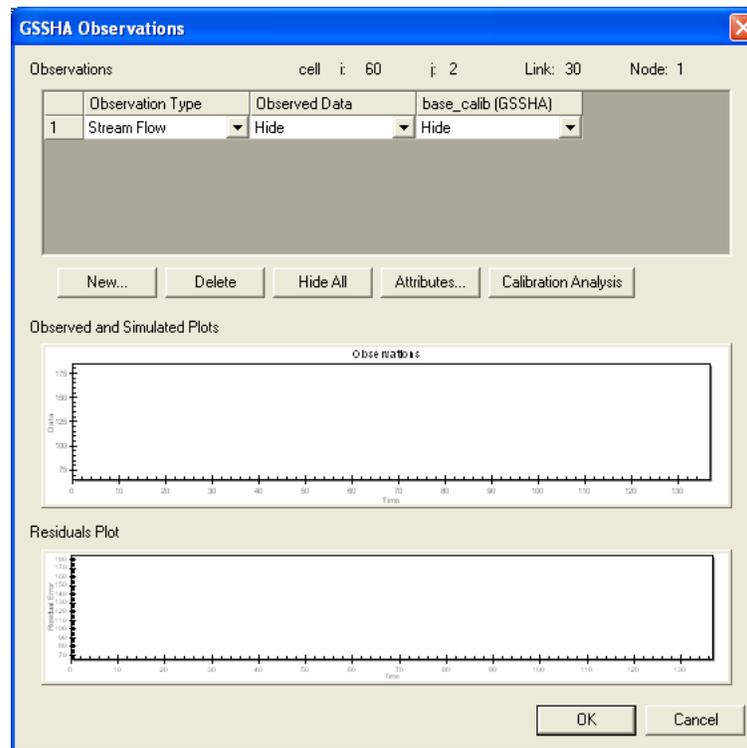
This is the project we shall use as the starter project. Let's look at the data.

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.



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.

1. 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:

1. 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.