
***Introduction to Spatial Hydrologic
Modeling in the US Army Corps of
Engineers:***

***GSSHA Using the Watershed Modeling
System***

Tutorials – Volume 2

GSSHA Modeling Basics

WMS 8.4 Compatible – Volume 2

Copyright © 2010 Aquaveo, LLC

All Rights Reserved

Unauthorized duplication of the *WMS* software or documentation is strictly prohibited.

AQUAVEO LLC MAKES NO WARRANTIES EITHER EXPRESS OR IMPLIED REGARDING THE PROGRAM *WMS* AND ITS FITNESS FOR ANY PARTICULAR PURPOSE OR THE VALIDITY OF THE INFORMATION CONTAINED IN THIS USER'S MANUAL

The software *WMS* is a product of the Aquaveo, LLC. For more information about this software and related products, contact Aquaveo at:

Aquaveo
3210 N. Canyon Road
Suite 300
Provo, Utah 84604
Tel.: (801) 691-5528
e-mail: wms@aquaveo.com
WWW: <http://www.aquaveo.com>

For technical support, contact Aquaveo's tech support number at (801) 691-5530 (Monday-Friday, 8am-5pm Mountain Time)

TABLE OF CONTENTS

1	GSSHA INITIAL OVERLAND FLOW MODEL SETUP	1-1
1.1	IMPORTING DEM DATA.....	1-1
1.2	COMPUTING FLOW DIRECTIONS AND FLOW ACCUMULATIONS.....	1-2
1.3	DELINEATING THE WATERSHED.....	1-3
1.4	2D GRID GENERATION	1-4
1.5	JOB CONTROL SETUP	1-5
1.6	UNIFORM INDEX MAP SETUP.....	1-6
1.7	ROUGHNESS TABLE SETUP	1-8
1.8	DEFINING UNIFORM PRECIPITATION.....	1-9
1.9	SAVE THE GSSHA MODEL.....	1-10
1.10	RUNNING THE MODEL	1-11
1.11	VISUALIZING OVERLAND FLOW RESULTS.....	1-12
2	CORRECTING OVERLAND FLOW PROBLEMS	2-1
2.1	WORKING WITH DIGITAL DAMS.....	2-1
2.2	MANUALLY ADJUSTING CELL ELEVATIONS.....	2-4
2.3	USING CLEANDAM TO FIX DIGITAL DAMS.....	2-5
2.4	SAVE AND RUN THE MODEL.....	2-6
2.5	VISUALIZING OVERLAND FLOW RESULTS.....	2-6
3	INFILTRATION	3-1
3.1	INDEX MAP SETUP	3-1
3.2	CREATE SOIL TYPE COVERAGE.....	3-2
3.3	IMPORT THE SOIL DATA	3-2
3.4	CREATING AN INDEX MAP FROM SOIL DATA.....	3-3
3.5	CREATING A MAPPING TABLE FROM THE INDEX MAP.....	3-3
3.6	RUNNING THE MODEL AND VISUALIZATION.....	3-5
3.7	DEFINING VARIABLE SURFACE ROUGHNESS.....	3-6
3.8	USING LAND USE DATA.....	3-7
3.9	CREATING AN INDEX MAP FROM LAND USE DATA	3-7
3.10	CREATING A MAPPING TABLE FROM INDEX MAP	3-8
3.11	VISUALIZATION	3-9
3.12	CREATING A COMBINED INDEX MAP.....	3-10
4	STREAMFLOW	4-1
4.1	ADDING STREAMS TO A GSSHA SIMULATION.....	4-1
4.2	SMOOTHING THE CHANNEL THALWEG ELEVATIONS.....	4-3
4.3	CHANGING OUTPUT CONTROL OPTIONS.....	4-5
4.4	RUNNING THE MODEL AND VIEWING THE OUTFLOW HYDROGRAPH	4-6
4.5	VISUALIZING STREAM FLOW DATA.....	4-6
4.6	CHANGING CONTOUR PROPERTIES.....	4-7
5	DEVELOPING GSSHA MODEL USING HYDROLOGIC MODELING WIZARD IN WMS	5-1

5.1	STARTING THE HYDROLOGIC MODELING WIZARD	5-1
5.2	DEFINING THE PROJECT BOUNDARY AND COORDINATES	5-2
5.3	IMPORTING THE DEM DATA	5-3
5.4	COMPUTING FLOW DIRECTION AND FLOW ACCUMULATION.....	5-3
5.5	DELINEATING THE WATERSHED	5-4
5.6	INITIATING GSSHA	5-5
5.7	DEFINING STREAM CHANNELS.....	5-5
5.8	CREATING A GRID	5-8
5.9	DEFINING JOB CONTROL.....	5-9
5.10	DEFINE LAND USE AND SOIL DATA.....	5-9
5.11	CREATING INDEX MAPS AND DEFINING MODEL PARAMETERS	5-11
5.12	DEFINING MODEL PARAMETERS.....	5-12
5.13	EDITING PARAMETERS.....	5-13
5.14	DEFINING A METEOROLOGICAL MODEL.....	5-14
5.15	CLEANING AND CHECKING THE MODEL	5-14
5.16	SAVE THE MODEL.....	5-14
5.17	RUN THE MODEL	5-14
6	POST-PROCESSING AND VISUALIZATION OF GSSHA MODEL RESULTS	6-1
6.1	OPEN AN EXISTING PROJECT	6-1
6.2	READ SOLUTION	6-2
6.3	VISUALIZATION	6-2
6.4	COMPARING THE SIMULATION RESULTS WITH THE OBSERVED DATA	6-10

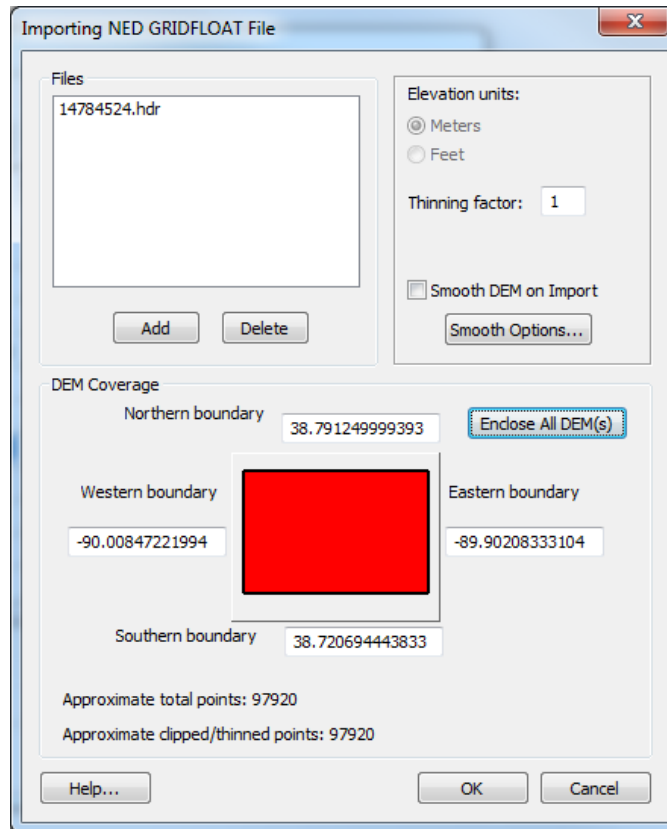
GSSHA Initial Overland Flow Model Setup

The first step in creating a GSSHA model using WMS is to delineate a watershed and obtain a watershed boundary polygon from a digital elevation model. In this tutorial you will use a DEM to create the watershed boundary polygon and the streams using the same delineation process you learned in previous tutorials. WMS interpolates cell elevations from the DEM when building a 2D grid. WMS uses the watershed boundary polygon to select whether 2D grid cells are active (inside the basin) or inactive (outside the basin). When building a GSSHA model from within WMS, the basin should not be subdivided into sub-basins as you would subdivide a lumped parameter model. GSSHA models require that you have a single watershed boundary polygon.

1.1 Importing DEM Data

You learned how to download a DEM using web services in the previous workshops. In this workshop, you will open a DEM which has already been downloaded.


1. Open a new instance of WMS.
2. Select ***File / Open***, browse and open the file ***C:\Training\Raw Data\JudysBranch\DEM\Judys_branch.hdr***.
3. The ***Importing NED GridFloat File*** dialog will open. Click OK.




4. Select *Yes* when prompted for coordinate conversion.
5. Set the horizontal *Object Projection* to *Geographic, NAD83, ARC DEGREES* and the vertical *Object Projection* to *NAVD 88(US), Meters* if different.
6. Select *Specify* option for the *Project projection*. Then select *Global projection* and click the *Set Projection* button.
7. Select *UTM, NAD83, METERS, Zone 15*, and click *OK*.
8. Make sure the *Vertical Projection* is set to *NAVD 88(US), Meters*.

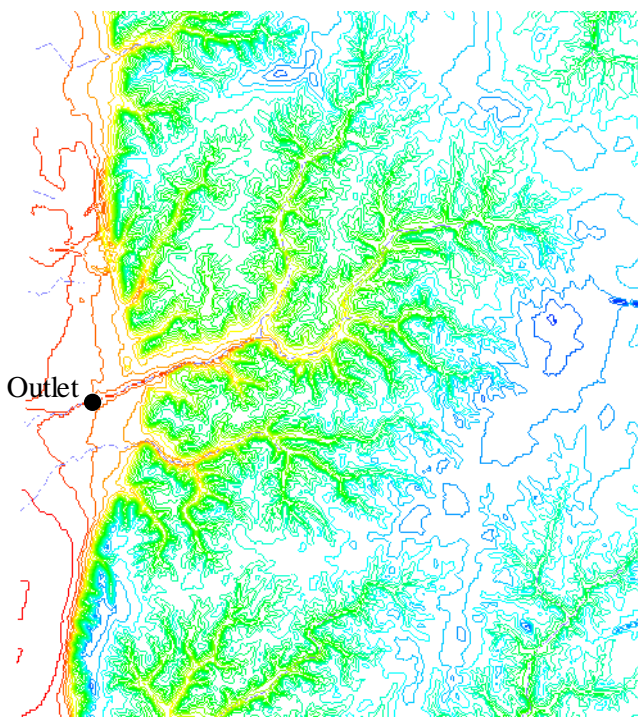
You should now be able to see the DEM contours plotted on WMS main window.

1.2 Computing Flow Directions and Flow Accumulations

1. In the *Drainage Module* , select *DEM / Compute TOPAZ Flow Data* and Click *OK* twice to accept the default TOPAZ input.

TOPAZ computes flow direction and accumulation grids. From these grids, you can infer the stream locations based on the DEM data.

2. Click on *Close* after the computations are complete.
3. You need to create an outlet point to delineate a watershed. Select the *Create Outlet Point* button . Place the outlet at approximately the location shown in the figure below.

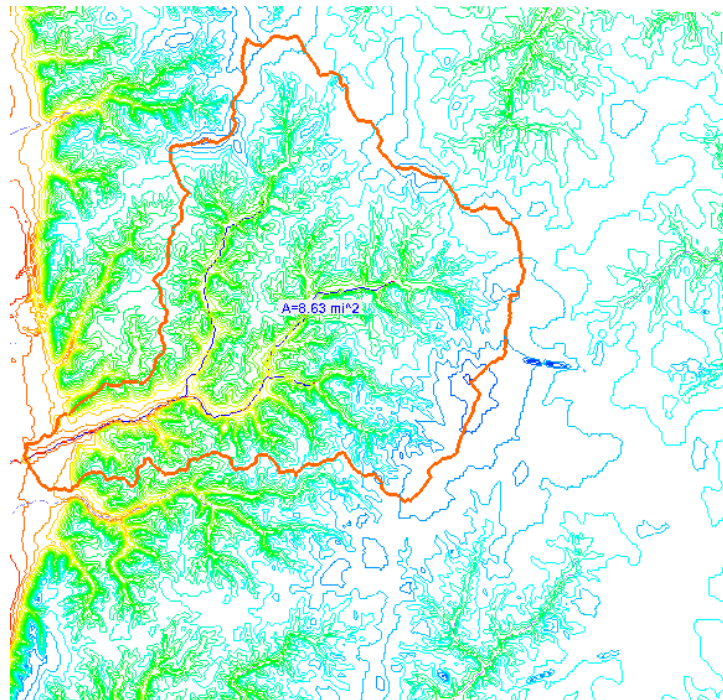



1.3 Delineating the Watershed

1. Select *DEM / Delineate Basins Wizard*.
2. Make sure that the Stream Threshold value is 1.0 sq miles. Click the *Delineate Watershed* button.

The delineate basin wizard automates the steps in the previous tutorial of converting the DEM streams to arcs, defining the basin from the DEM, converting the basin boundary to a polygon, and then computing the basin data. In a future tutorial, you will learn how to run the hydrologic modeling wizard as a guide to GSSHA model set up.



3. Click *Close* to close the *Delineate Watershed* dialog after the delineation process completes.
4. Your watershed should look somewhat like this:



5. Save your WMS project by selecting *File / Save*.  Save it as *C:\Training\Personal\BasicGSSHA\JudysBase.wms*

1.4 2D grid generation

A basic GSSHA model can be generated from a delineated basin boundary in a drainage coverage and a DEM. When delineating your watershed, it is helpful to use a value for the flow accumulation threshold when generating stream arcs from flow accumulation cells so the characteristics of your streams are correctly represented in your GSSHA model. In the following steps, you will tell WMS to use the boundary polygon and the DEM to create a 2D grid that conforms to the watershed boundary and has elevation data values that are interpolated from the DEM. For more information on selecting appropriate cell sizes to use when generating a 2D grid, see the GSSHA Primer. (http://gsshawiki.com/index.php?title=GSSHA_Software_Primer)

1. Switch to *Map Module* .
2. Click on *Select Feature Polygon Tool*  and right click anywhere within the watershed polygon. Select *Create Grid*. Select *Yes* to create a grid for GSSHA.
3. Make sure that *Base Cell Size* option is checked on and enter **90** (meters in this case) as the cell size and click *OK*.

- Click OK to interpolate grid cell elevation from the DEM, and select *Yes* when prompted to delete the DEM data.

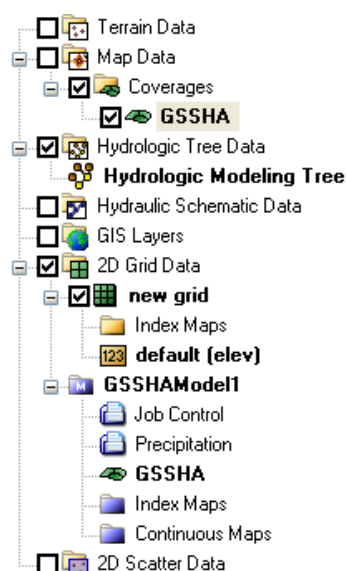
You can now see the 2D grid representation of the watershed. Also notice that under *Coverages* in the data tree, the *Drainage* coverage has been changed to a *GSSHA* coverage.

- To save your model as a GSSHA project, go to the *2D Grid module*




and select *GSSHA / Save Project File*. Save your GSSHA project file to *C:\Training\Personal\BasicGSSHA\GSSHA\basic_ov.prj*.

Notice in the project explorer that a 2D Grid called *new grid* is visible in the tree and a GSSHA model (*GSSHAModel1*) has been initialized.

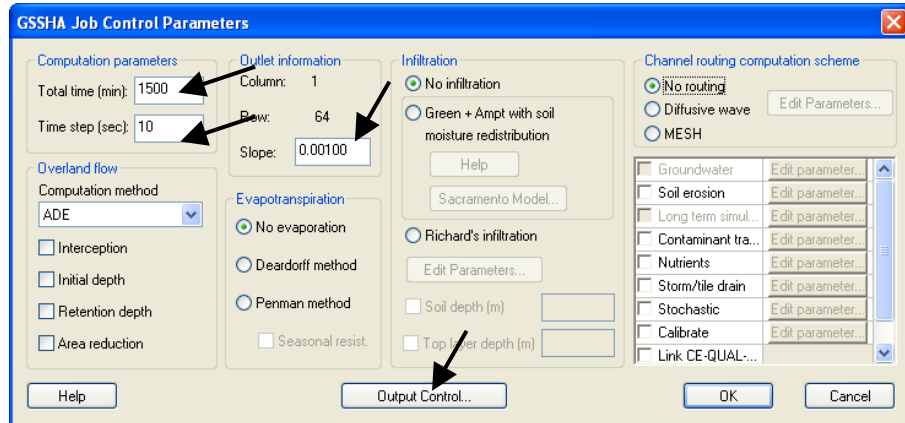


1.5 Job Control Setup

When you created a GSSHA grid, the GSSHA Job Control parameters were initialized using some default values. To get the overland flow running you will need to define some realistic values.

- In the 2D Grid module  select *GSSHA / Job Control*
- Make sure the outlet slope is set to 0.001. This ensures water will run off the grid.

- Enter a *time step* of 10 (seconds) and a *total run time* of 1500 (minutes). Make sure the values are same as those shown in the following figure.



- Click on *Output Control* and choose your *Output Units* to be **English** in the lower part of the Output Control dialog.

All computations in GSSHA are done in metric, but changing your hydrograph output to English allows you to write the outflow hydrograph in cfs (All other output will still be in metric units).

- Enter 15 minutes for the *Hydrograph write frequency*. This is the second of the two listed write frequencies. The first is for the grid output and it should remain at 30 minutes. Click *OK*.
- Select *OK*

1.6 Uniform Index Map Setup

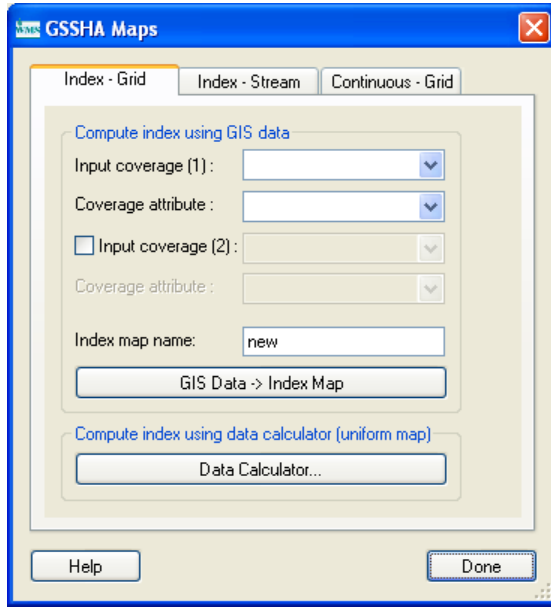
Once the Job Control parameters are set to some realistic values, there are two main parameter sets to set up to finish building an overland flow model. First, the overland flow roughness coefficients for the grid cells need to be set and second, the precipitation data needs to be specified.

To set up overland flow roughness coefficients, an index map must be set up that describes the spatial variation of the roughness and the roughness values themselves must be assigned in a mapping table. In this example, you will create a spatially uniform set of roughness values so you can examine and correct any problems with overland flow before defining spatially varied roughness and infiltration.

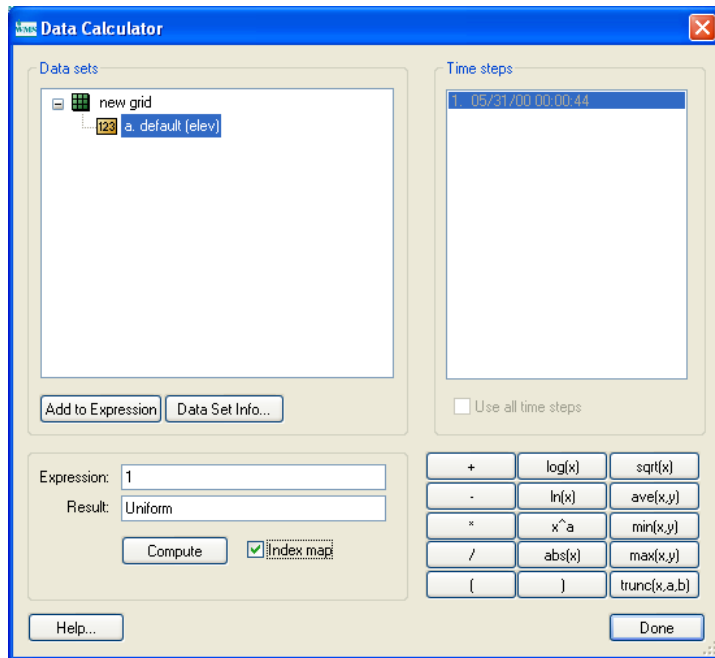
Notice that an *Index Maps* Folder was created under the GSSHA model in the project explorer. A grid may have several index maps as was demonstrated in

a previous tutorial. In this tutorial, you will add a uniform index map to this folder.

1. Select **GSSHA / Maps...** This will bring up the GSSHA Maps dialog.



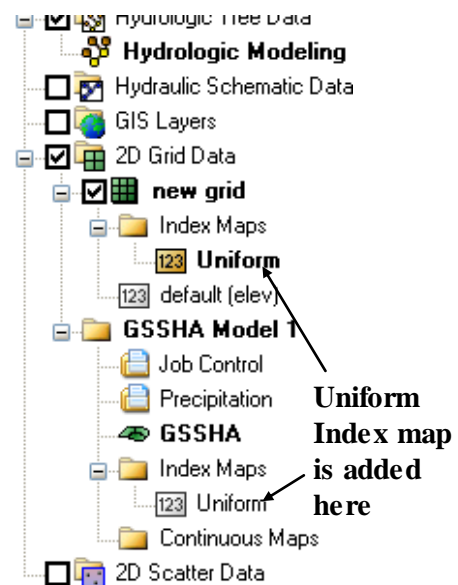
2. Select the *Data Calculator* button. This will bring up the Data Calculator dialog.



3. In the *Expression* box, type 1.
4. In the *Result* box, type “Uniform”
5. Check the *Index map* option
6. Select *Compute*.
7. Select **Done**. This takes you back to the Index Map dialog.
8. Select **Done** again to exit the *GSSHA Maps* dialog.

Notice that the *Uniform* index map is assigned to two different places in the project explorer: one in the *Index Maps* folder under the 2D grid and another under the GSSHA model (*GSSHAModell*).

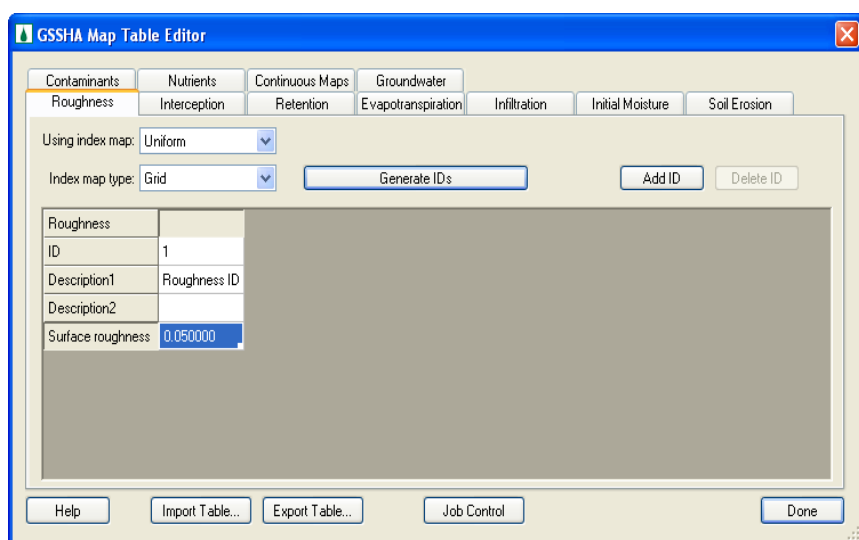
The actual index map is stored in the *Index Maps* folder under the 2D grid. When you create an index map, it is assigned to the active GSSHA model. You can *Assign and Remove* the index map(s) from a GSSHA model. If there are multiple GSSHA models, you can use the same index map for several GSSHA models. Similarly, if you have several index maps, you can use them in one or more GSSHA models (or you can remove an index map from all GSSHA models).



1.7 Roughness Table Setup

Notice that when you made the index map you assigned a value of 1 to the whole map (every grid cell). The value of 1 is an index number that you will relate to a roughness coefficient. This is done through the mapping table.

1. Select *GSSHA / Map Tables*.
2. Select the *Roughness tab* (it should be selected by default).
3. In the *Using Index Map* combo box select “Uniform”.
4. Select **Generate IDs** button.
5. In the Surface Roughness edit field, enter a value of 0.05.



6. Select *Done*

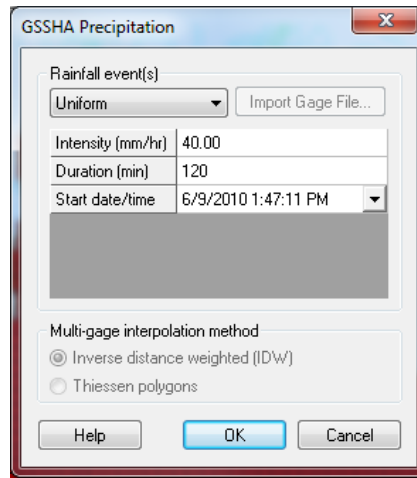
The tabs on the Mapping Table dialog list some of the mapping tables that can be set up for different watershed processes that can be modeled in GSSHA. We will set up other processes in later tutorials. The two steps for assigning values for each cell are first to assign index map ID's to your grid cells using an index map (In the case of roughness, you used a uniform index map) and then to assign roughness values to each ID used in the index map in the *GSSHA Map Table Editor*. For this simple model with uniform roughness, the use of an index map may seem like extra work since you can select all the cells in your model and assign the same roughness value to each cell. However, when you have several land use or soil type characteristics representing variations in roughness values, infiltration, and other spatial properties, it is essential to manage your GSSHA model data using index maps to represent the spatial variation and then to assign specific values to your index map IDs in the mapping tables.

1.8 Defining Uniform Precipitation

In addition to roughness, precipitation must be defined to evaluate a basic overland flow model. GSSHA has several options for defining precipitation. You will explore these options in a later workshop, but you will begin by setting up a simple uniform precipitation event for a short duration. The purpose of this tutorial is to get enough water on the grid to examine and correct any problems with overland flow.

1. Select *GSSHA / Precipitation ...*
2. Select the Uniform Rainfall in the Rainfall event(s) combo box.

3. Enter the rainfall intensity of 40 (mm\hr).
4. Enter the rainfall duration of 120 (minutes).
5. If you are simulating an actual event and know the date and time of the storm occurrence, you can enter them, but for this synthetic, uniform storm just use the defaults.
6. Select *OK*.



1.9 Save the GSSHA Model

It is advisable to create a new folder each time a significant revision is made and save the project in the new folder.

1. Select *GSSHA / Save Project File...*
2. Save the GSSHA project as *C:\Training\Personal\BasicGSSHA\GSSHA\basic_ov.prj*.

Typically, most of the files in GSSHA project share a similar base file name and only differ in the file extension. The exceptions to this rule are index map files, which all have the same extension and different base file names. The file names and extensions for the various GSSHA files may be any name desired; the defaults given in WMS are merely convention, but they do aid in quickly identifying files when you are looking through them. The following table lists a few of the conventional extensions used by WMS.

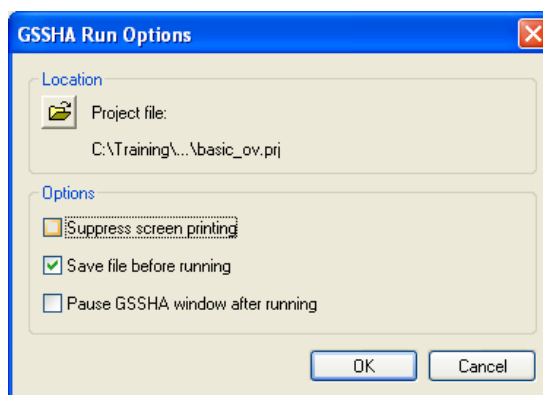
<i>Extension</i>	<i>Description</i>
prj	Project file
ele	Elevation file
msk	Watershed mask

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)

3. Open Windows Explorer and go to the folder *C:\Training\Personal\BasicGSSHA\GSSHA*.
4. Open *basic_ov.prj* using note pad (*Right click* on the file, select *Open With* and select *Notepad* to open)
5. This is the GSSHA project file. This file tells GSSHA which watershed processes to run and lists the input and output files used in the GSSHA project. Notice that there are files listed in this file with similar names but different extensions (as discussed in the above table).
6. Close *Notepad* and *Windows Explorer* and return to *WMS*.

1.10 Running the Model

1. Select *GSSHA / Run GSSHA*. Turn the *Suppress screen printing* option off.

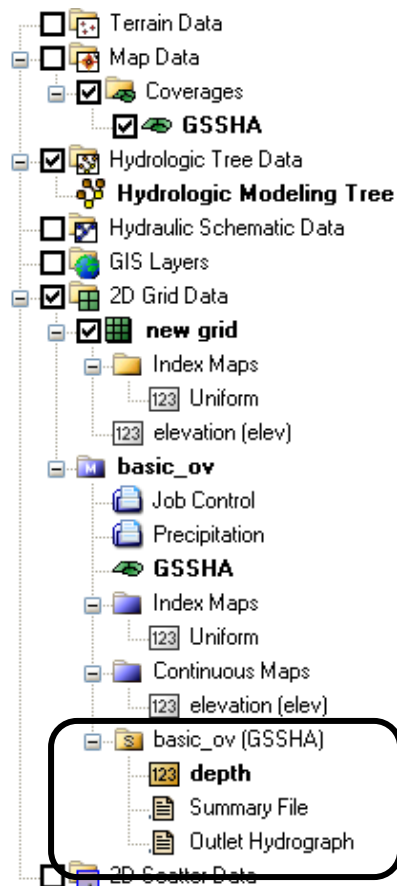


2. Select *OK*.

You will see the time steps being computed and discharge at each time step in the model wrapper. Click “*Close*” after the computation is complete, which will take some time depending on your computer speed.

1.11 Visualizing Overland Flow Results




Since you have just run your first simulation, it would be nice for you to see what happened! Notice that after you closed the Model Wrapper, WMS automatically read in some files. WMS stores the results of a run together as a solution set. There can be many solution sets in the project explorer, but they must be for the same 2D grid. These solution sets are useful because you can modify job control parameters, index maps, boundary conditions, or mapping tables and you can compare results between model runs.

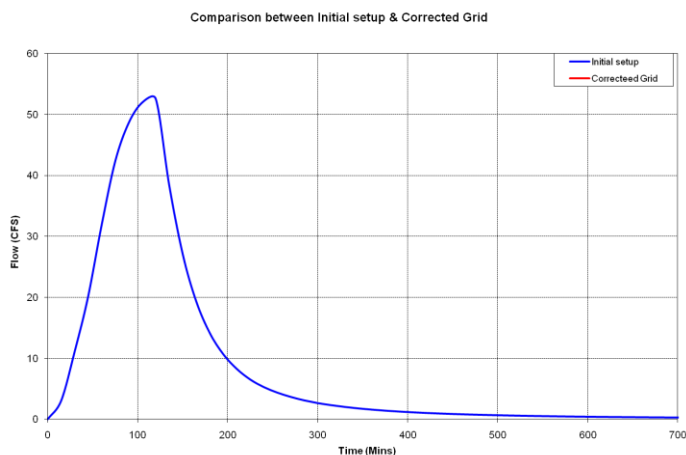


You can see in the above figure that your model is named *basic_ov* and that one of the folders within the model has a letter 'S', representing 'Solution'. The *Depth* dataset under the solution folder represents overland flow depth data. The *Summary file* is a text document that has detailed information about your model inputs, results, mass balances, and errors (if any). The *Outlet Hydrograph* is used to access a plot of the outlet hydrograph.

1.11.1 Visualizing the Outlet Hydrograph

As soon as the results are read you can see a small hydrograph icon at the outlet of your watershed. This icon is used access the plot of the outflow hydrograph.

1. In the 2-D grid module , click on the “Select Hydrograph” tool  and double click on the small hydrograph icon  near the outlet.
2. This will open the hydrograph in a plot window.
3. Right click on the hydrograph plot and choose *View Values*. This will display the hydrograph ordinates and corresponding times. You can copy and paste these data to a spreadsheet program.
4. Select the data values under *Flow (cfs)* column, right click and select *Copy*.
5. Open the spreadsheet *C:\Training\InitialGSSHAComparison.xls* and paste the hydrograph ordinates in the *Data* tab, in the *Initial Setup & Corrected Grid* section, under the column *Initial Setup*.
6. When pasting data in the spreadsheet, paste your data in the white areas only. Notice some additional tabs in this spreadsheet that can be used to view the results. Select the *Initial_Corrected* tab to view a plot of your hydrograph.
7. Return to WMS and *Close* the *View Values* and *Hydrograph plot* windows after you are done. Your hydrograph will look something like this:



1.11.2 Examining the Summary File


1. Double-click on Summary File under the solution folder.
2. If WMS asks for your editor, select Notepad and click *OK*.
3. Look through the summary file, in particular note the following:
 - Total precipitation volume
 - Total discharge
 - Volume remaining on the surface
 - Mass conservation error etc
4. Notice from these numbers that most of the water remained on the watershed grid in this simulation instead of running off the grid.
5. When you are done you can close the summary file.

There could be a couple of reasons for the water remaining on the grid instead of being converted to runoff:

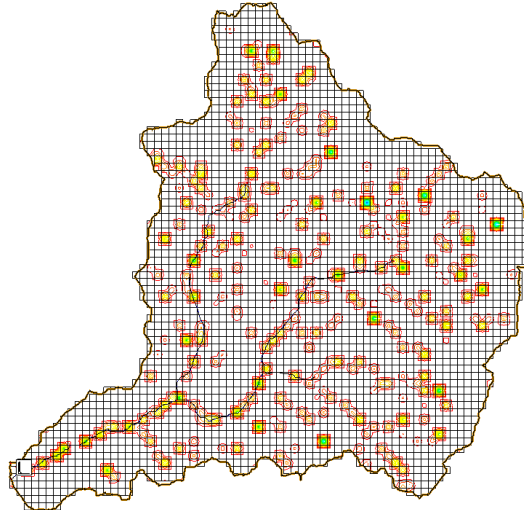
- The simulation did not run long enough,
- There are problems with the elevation grid that result in water ponding on the grid, or
- Both of the above reasons

Let's look at the water depths to examine the surface runoff behavior.

1.11.3 Visualizing Depth Contours

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. Make sure the *Contour Method* drop-down box is set to *Normal Linear*.

7. Select the *Legend...* button.
8. Turn on the legend Display.
9. Click *OK*.
10. Click *OK*.
11. In the properties window (normally to the right side of the WMS screen), a set of time steps appear.
12. In the Properties window on the right side of the WMS application, click on first time step and use the down arrow key (on the keyboard) to toggle through the time steps. You can see the depth contours varying in color as the time steps change.



It is fairly easy to see that the water has ponded on the grid (see the circular spots on your grid and in the above figure). These puddles are the result of digital dams. Digital dams are local depressions that exist as a result of a lack of a grid's elevation resolution. Your lack of runoff is not due to a short simulation time but rather due to the presence of digital dams. Each grid cell in your model is 90m x 90m or 8100 meters squared, so a large amount of water can be retained by each "digital dam" cell and any cells flowing to this cell.

Since not a lot of water runs off the watershed, but ponds in digital dams, this situation should be fixed to correct the model. Often, when there are many digital dams, your simulation will not run to completion. The problem is usually due to digital dams. These digital dams are artificial depressions in the 2D Grid caused by a lack of resolution in both the original DEM and the derived grid. Fixing digital dams is the subject of the next tutorial.

Correcting Overland Flow Problems

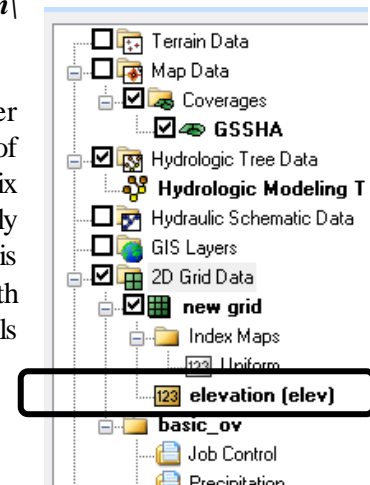
In the previous tutorial, you saw that the presence of digital dams creates problems with surface runoff in your GSSHA model. You will learn how digital dams can be removed from your model.

2.1 Working with Digital Dams

In this tutorial you will continue working with the same model. If you have created the project but have already closed the project, open *C:\Training\Personal\BasicGSSHA\GSSHA\ basic_ov.prj*. If you do not have a project saved, a backup can be opened from *C:\Training\BasicGSSHA\ GSSHA\basic_ov.prj*.

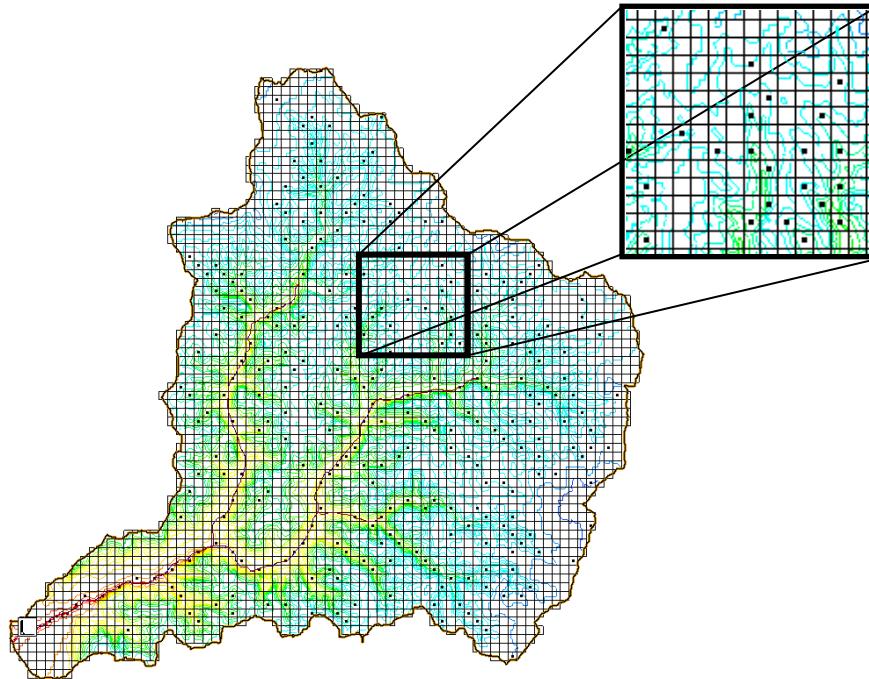
You should save this project with a different file name so the original project is unchanged. In the 2D-Grid module, select *GSSHA / Save Project file*. Save it as *C:\Training\Personal\DigitalDam\Clean.prj*.

The problem with digital dams is that the water ponds in artificial depressions that result because of a lack of resolution. There are two methods to fix the digital dams. The first method is to manually adjust the cell elevations and the second method is to use the *cleandam* tool to automatically smooth the cell elevations. To quickly identify which cells have digital dams:



2-2 GSSHA Modeling Using WMS – Volume 2

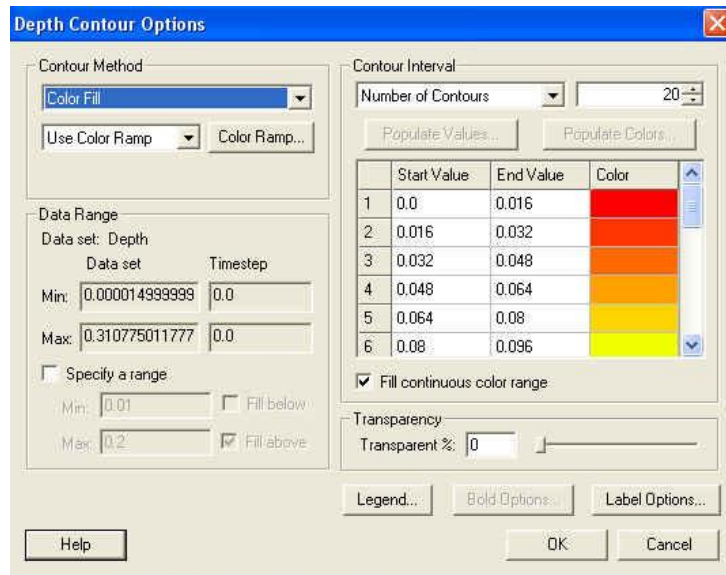
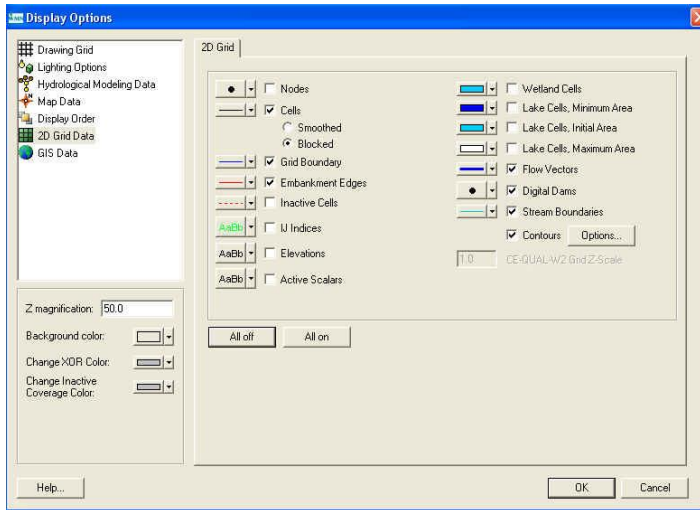
1. Click on the *elevation (elev)* dataset in the project explorer under *2D Grid Data* to select it.
2. Select **Display / Display Options**.
3. In the *2D Grid* tab, turn on *Digital Dams*.
4. Select *OK*. The Grid will look something like the following figure.



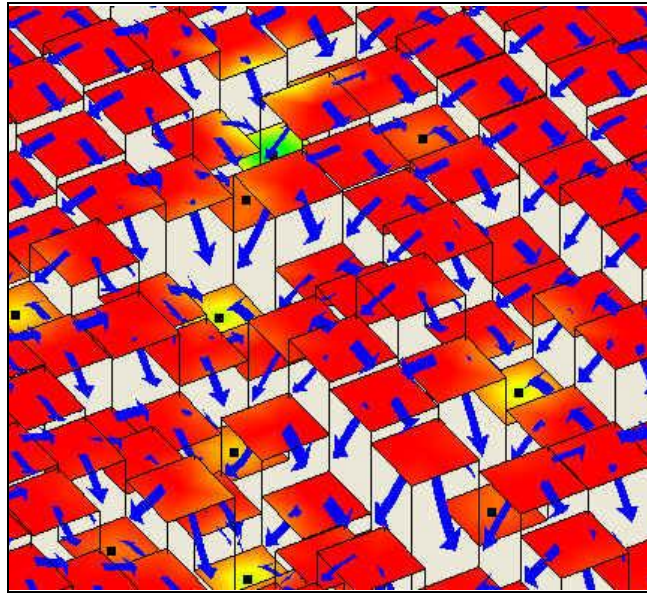
To better visualize why these cells are digital dam cells:

1. Select **Display / Display Options**.
2. Select the *2D Grid Data* item from the list. Under the *Cells* toggle box, select the *Blocked* cells option.
3. If not already on, turn on the *2D Grid Contours*.
4. Click on the *Contour Options...* button.
5. Under the *Contour Method*, Choose *Color Fill*.
6. Click *OK*.

Correcting Overland Flow Problems 2-3



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



The black dots denote the digital dam cells. The arrows indicate the preferred GSSHA overland flow paths (remember that GSSHA only flows in one of four directions (left, right, up, down from each cell). The cells with all four sides marked as pointing inward are flagged as digital dam cells. The blocked cells display option further helps to illustrate this. You can use the *Display / View / Z Magnification* option to enhance the z scale (try adjusting the Z magnification to a value of 10.0). Notice that digital dam locations are at low points in the grid.


2.2 Manually Adjusting Cell Elevations

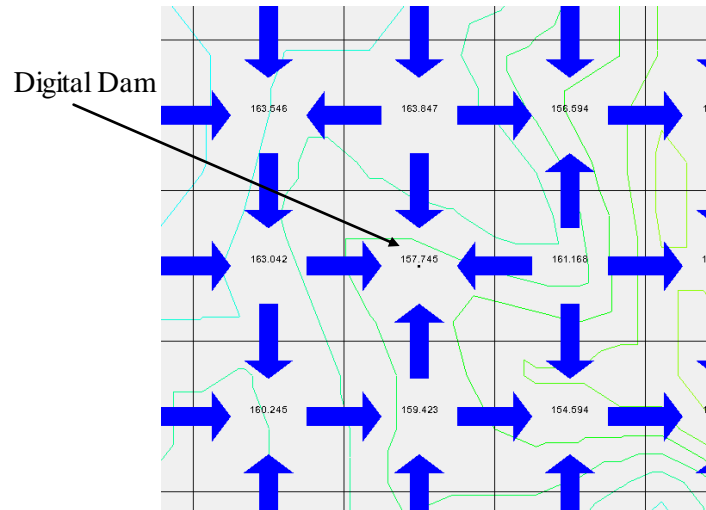
You could adjust cell elevations manually to make the water flow in an appropriate direction. However, the number of digital dams in this model is typical and manual adjustment would be tedious. You will edit one or two of these digital dams by hand and then use some automated tools to speed up the process of removing the low points.


Occasionally, only the digital dam cells have to be adjusted, but usually one or more neighboring cells must also be adjusted to remove digital dams.

To adjust the elevation of a cell, click on the cell. In the WMS properties window, there is an edit field with the cell elevation. Let's try a couple.

1. In the display options, change the following so that you can see the elevation of surrounding cells and flow direction in each cell.
 - turn on the display of *Elevations*
 - turn on *Flow Vectors*
 - Change the DEM contour method to *normal linear*

2. Select the *Plan View* button  and
3. Select the zoom tool and zoom in to any one of the digital dams so that the cell with digital dam and surrounding cells are clearly visible.



4. Switch to the *2d Grid* Module and select a cell with a digital dam and increase its elevation in such a way that flow occurs from this cell to surrounding cells. It might take few trials. You might end up removing the digital dam from this cell but create a new one up or downstream of it.
5. Once the digital dams are fixed by changing the cell elevations, you will need to right click the *elevation* dataset in the *Project Explorer Window* (under the *new grid 2D grid*) and select the *Set as Elevation* option. Doing this will permanently update the elevation data.
6. After you have manually fixed a few digital dams, go to the display options and turn off the display of *Elevations* and *Flow Vectors*. Then select the *Frame* tool  to zoom to full extent.

2.3 Using Cleandam to Fix Digital Dams

Manually adjusting cell elevations works fine for a small area with a few digital dams or if there are one or two particularly troublesome digital dam cells in a larger watershed model. It quickly becomes tedious, however, when there are hundreds 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.

To run *Cleandam*:

1. Select **GSSHA / Clean Digital Dams**
2. You should now see cleandam running in the model wrapper. When it is done select *Close*.
3. If you still see any digital dams present, you could manually adjust them.




2.4 Save and Run the Model

Once done cleaning the digital dams, save the project as **C:\Training\Personal\DigitalDam\clean.prj**.

1. Select **GSSHA / Run GSSHA**. Turn off the *Suppress screen Printing* option.
2. Select *OK*.
3. You can see the time steps being computed and discharge at each time step in the model wrapper. Click “*Close*” after the computation is complete.

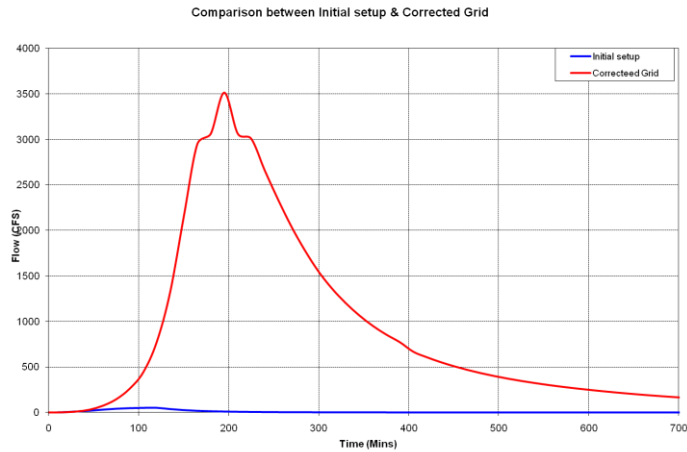
2.5 Visualizing Overland Flow Results

2.5.1 Visualizing Hydrographs

In the 2-D grid module , click on the “*Select Hydrograph*” tool  and double click on the hydrograph icon  near the watershed outlet. Since digital dams have been removed, your model should have produced more runoff than the previous model.

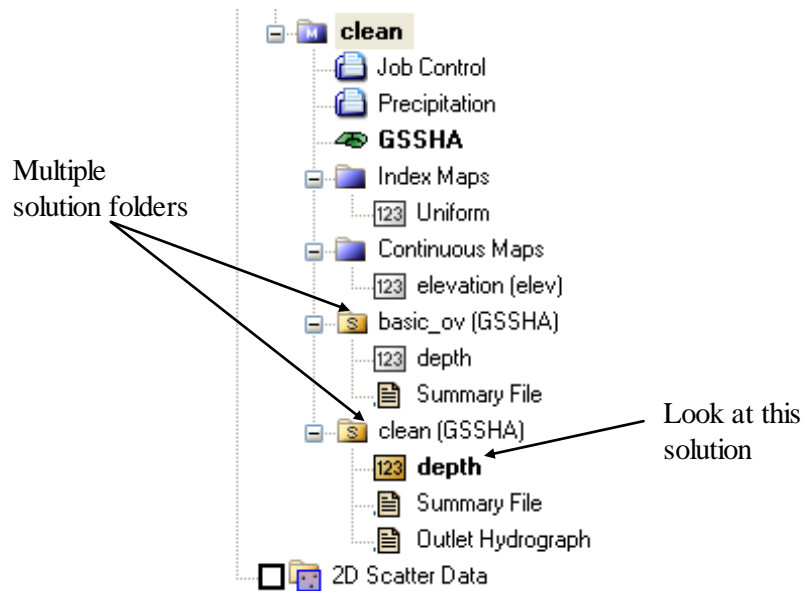
1. Right click on the hydrograph plot and choose *View Values* which will display the hydrograph ordinates and corresponding times.
2. Select the data under *Flow* column, right click and then select *Copy*.
3. Open the spreadsheet **C:\Training\InitialGSSHAComparison.xls** and paste the hydrograph ordinates under the column *Corrected Grid*.
4. Paste your data in the white areas of the spreadsheet only and compare the plots before and after correcting the digital dams by selecting the *Initial_Corrected* tab in the spreadsheet.

- See the following plot which compares the hydrograph before and after fixing the digital dams. Is your plot similar to this (at least of the same magnitude)?
- In WMS, close the *Hydrograph* plot window after you are done.



2.5.2 Visualizing Summary File

- Under the GSSHA model folder, you can see one or more solutions. If you have been working with the same model from the previous exercise, you might have two or more solution folders. Go to the solution folder with the name *clean (GSSHA)*. See the following figure:




- Open the summary file and:

- Compare the amount of water remaining on the surface
- Volume of discharge

You should have noticed that the volume of water remaining on the surface has significantly reduced as compared to the previous simulation where the digital dams were not fixed. In the previous simulation, nearly all of the rainfall remained on the surface and there was very little discharge. In this simulation, much of the water was converted to discharge. There is still some water remaining in the overland cells because this model does not have infiltration or streams turned on.

3. When you are done you can close the summary file.

2.5.3 Visualizing Depth Contours


1. In the 2D Grid module , select *Display / Display Options*.
2. Turn on the *2D Grid Contours*. Select *OK*.
3. In the data tree, right-click on *Depth* under the solution folder.
4. Select *Contour Options* from the popup menu.
5. Under the *Contour Method*, select *Color Fill*. Select *OK* in the *Contour Options* dialog to save your changes.
6. Now in the project explorer, click on *Depth* to select it. In the properties window (to the right side), a set of time steps appear. Click around on a few time steps.
7. Look at the depth contours toward the end of the simulation to see if water is ponding in some part of the watershed. If it is, then you can adjust the elevations manually and try re-running the simulation as described above.

What differences do you see in the results before and after fixing the digital dams?

Infiltration

Infiltration is a key process in a rainfall runoff model and to this point you have not simulated it since you were learning other concepts and you should first make sure the overland flow process runs correctly. During this tutorial, you will set up the inputs needed for the *Green & Ampt with Soil Moisture Redistribution* model.

Keep working with the model you have been developing. If you are continuing from the last tutorial then you can skip the first two steps which open your model (or the backup saved in case you have problems) and go to step 3 below.

1. In the 2D Grid Module  select *GSSHA / Open Project File*.
2. Browse and open the file *C:\Training\Personal\DigitalDam\Clean.prj*. If this file does not exist or is incorrect, you may open *C:\Training\DigitalDam\Clean.prj*.
3. Save this project with a different name so that the original project remains unchanged. Select *GSSHA / Save project File* and save the project as *C:\Training\Personal\ Infiltration\Infiltration.prj*.

3.1 Index Map Setup

Describing the spatial variability of almost all parameters is done by setting up an index map and then assigning parameter values to a mapping table. An index

map is a grid of ID numbers. Parameter values can be assigned to each index map ID number in the project mapping tables. Index maps are generic and may apply to any number of mapping 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.

In this tutorial, you will use a soil type shape file from the NRCS SSURGO soil database to create a soil type index map. You will learn to perform necessary join operations to derive infiltration parameters from SSURGO soil data.

3.2 Create Soil Type Coverage



1. Right-click on the *Coverages* folder in the project explorer and select *New Coverage....*
2. Change the *Coverage Type* to *Soil Type*.
3. Select *OK*.

3.3 Import the Soil Data

1. Right-click on *GIS Layers* in the project explorer.
2. Select *Add Shape file Data*. Browse and open *C:\Training\Raw Data\JudysBranch\SSURGOsoil\Raw\Spatial\soilmu_a_il119.shp*.
3. Right click on *soilmu_a_il119.shp* in the project explorer and select *Open Attribute Table*. In the attribute table, notice that there are no soil parameters other than *MUSYM* and *MUKEY* assigned to the soil polygons. Select *OK* to close the *Attributes* dialog.
4. Right click on *soilmu_a_il119.shp* and select *Join NRCS Data*.
5. Toggle the *Fill blank values* and *Compute Hydraulic...* options on.
6. Leave the fields to fill (i.e. B and Silt loam) as is displayed and click *OK*. After the join is completed, you may check the attributes table by right-clicking on *soilmu_a_il119.shp* and selecting *Open Attribute Table*; notice that the attributes are now joined to the shape file.


Now we can convert the shape file to a coverage

7. Right-click on the *GSSHA* coverage and select *Zoom To Layer* to set the current view so you are zoomed to your GSSHA model.

8. Click on the *Soil Type* coverage in the *Coverages* list to make it the active coverage.
9. Select the *soilmu_a_il119.shp* GIS layer by clicking on it. This will change the active module to the *GIS module* .
10. Click the *Select shapes tool*  and drag a rectangle around (and a little outside) the watershed to select the soil polygons that overlay the watershed.
11. In the GIS Module, select *Mapping / Shapes -> Feature Objects*.
12. Select *Next*. Notice that the fields should be mapped properly. Click *Next* again.
13. Select *Finish* and wait for WMS to convert the selected shapes to feature objects in the map module.
14. You may now delete the shape file under GIS Layers (**Right click / Delete**) since you are done with it.
15. Toggle off the display of the Soil Type coverage so the WMS display will update faster.

3.4 Creating an Index Map from Soil Data


Now you can generate an index map out of the soil type coverage.

1. Switch to the *2D Grid Module* .
2. Select *GSSHA / Maps...*
3. For the *Input coverage (1)*, use the *Soil Type* coverage. Do not use a second GIS data source.
4. Make sure the Coverage Attribute is set to *Texture*.
5. Change the Index map name to *SoilType*
6. Click on the *Coverages -> Index Map* button. It might take a few moments to build the Index map, but when it does you will see the grid display updated and the *SoilType* index map listed under the Index Maps folder in the project explorer.
7. Select *Done*

3.5 Creating a Mapping Table from the Index Map

Now you can define IDs based on your *SoilType* map in your infiltration mapping table and assign infiltration parameters to each ID.

3-4 GSSHA Modeling Using WMS – Volume 2

1. In the 2D Grid Module  select *GSSHA / Map Tables*.
2. Select the *Infiltration* tab.
3. Select *Yes* to open the *Job Control* window so you can turn on the simulation of infiltration for your GSSHA model.
4. Under the *Infiltration* heading, select the *Green + Ampt with soil moisture redistribution* option.
5. Select OK.
6. In the *Using Index Map* drop down box, choose *Soil Type*.
7. Click on *Generate IDs* button

The Generate IDs button should have created some IDs, among which IDs 15, 17 and 18 may exist. You can see the soil texture name in the *Description1* field below each ID. You will now define infiltration properties for each of these soil textures.

8. Using the following table, enter the values for each soils type

Parameters	Clay loam (Id 15)	Silt loam (Id 17)	Silty clay loam (Id 18)
Hydraulic Conductivity (cm\hr)	0.2	0.68	0.2
Capillary Head (cm)	20.88	16.68	27.3
Porosity (m ³ \m ³)	0.464	0.501	0.471
Pore distribution index (cm\cm)	0.242	0.234	0.177
Residual Saturation (m ³ \m ³)	0.075	0.015	0.04
Field Capacity (m ³ \m ³)	0.318	0.33	0.366
Wilting Point (m ³ \m ³)	0.1	0.1	0.1

9. Switch to the *Initial Moisture tab*.
10. In the *Using index map* drop down box, select the *Soil Type* index map.
11. Click on the *Generate IDs* button.

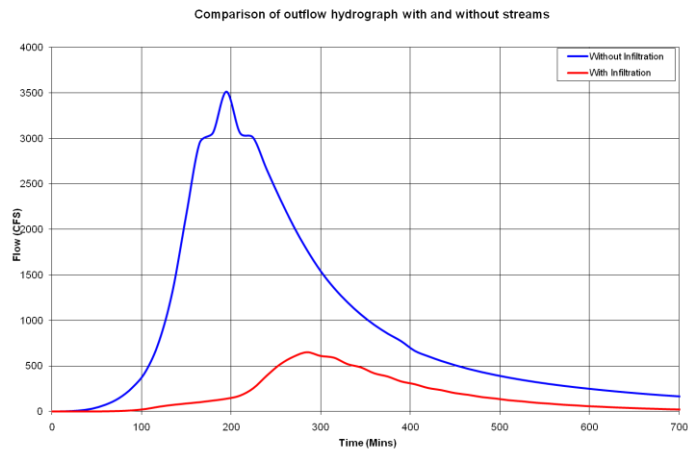
12. Enter *Initial Moisture* values of 0.15, 0.1 and 0.2 for Clay loam, Silt loam and Silty clay loam respectively. The initial moisture value must always be less than porosity.
13. Click *Done*.
14. Save your project as *C:\Training\Personal\Infiltration\Infiltration.prj*.
15. Run the model using the *Run GSSHA...* command in the *GSSHA* menu.

3.6 Running the model and Visualization

You can see the volume infiltrated or volume of discharge and other summary values by opening the summary file.

1. In the data tree, under the *Infiltration* solution, double-click on the summary file to open the file. If your *Clean* solution is still open, double-click on this solution's summary file also.
2. Find the following values in each summary file and compare the values with each other:
 - Volume of infiltrated water
 - Volume of discharge
 - Volume ground water recharge
 - Volume remaining on surface
 - Mass conservation error
3. Double-click on the *Outlet Hydrograph* to view it.
4. To export the hydrograph ordinates to an Excel spreadsheet, right click on the hydrograph plot and choose *View Values*.
5. Select all the *Flow* data values, right click and select *Copy*.
6. Open the spreadsheet *C:\Training\InitialGSSHAComparison.xls* and paste the hydrograph ordinates under the column *With Infiltration-Flow (CFS)*. Paste only the data values (no text) and paste your data in the white areas only.

7. Visualize the difference in outflow hydrograph in the two models. Your plot may be similar to this:



You should have noticed that the peak flow and the runoff volume have significantly decreased. Some of the rainfall should have infiltrated into the soil with the infiltration option turned on.


8. In WMS, close the Hydrograph plot window after you are done comparing the infiltration plot with the plot from the model without infiltration.

3.7 Defining variable surface roughness



So far in your model you have set up uniform watershed roughness, uniform precipitation, and variable infiltration parameters. You will now setup spatially varied roughness coefficients. Setting up non-uniform precipitation will be covered in a later workshop.

You will be using a land use GIS file to create an index map that will be used with the roughness mapping table. Using land use data to create an index map for roughness makes your model more closely represent real-world conditions since your model can use both a land use map to describe watershed roughness and a soil type map to describe watershed infiltration. Before the model is complete, you will consider how land use affects infiltration and adjust the infiltration parameters based on a combined land use/soil index map.

Continue working on the model in the previous section. If you have already closed the model then open your saved model or the backup model and save your model as a new project.


1. In the 2D Grid module , select **GSSHA / Open Project File**. If it is not already open, browse to and open the file **C:\Training\Personal\Infiltration\Infiltration.prj**. If you do not have this project file saved you may open **C:\Training\Infiltration\Infiltration.prj**
2. Save this project with a new name using GSSHA | Save Project File so that the original project remains unchanged. Save the new project as **C:\Training\Personal\Roughness\Roughness.prj**.

3.8 Using Land Use Data

1. Right-click on Coverages folder in the project explorer and select *New Coverage*.
2. Change the Coverage Type to *Land Use*. Click *OK*. You should have three coverages: GSSHA, Soil Type and Land Use.
3. Right click on the *GIS Layers* folder in the project explorer and select *Add shapefile data...*
4. Browse to the folder **C:\Training\Raw Data\JudysBranch\Landuse** and open the files **Belleville.shp and StLouis.shp**.
5. Right-click on the *GSSHA* coverage and select *Zoom To Layer* to set the current view so you are zoomed to your GSSHA model.
6. Click on the *Land Use* coverage in the *Coverages* list to make it the active coverage.
7. Select the *Belleville.shp* GIS layer by clicking on it. This will change the active module to the **GIS module** .
8. Click the *Select shapes tool*  and drag a rectangle around (and a little outside) the watershed to select the soil polygons that overlay the watershed.
9. In the GIS Module, select **Mapping / Shapes -> Feature Objects**.
10. Select *Next*. Make sure *LUCODE* is mapped to *Land Use*. Click *Next* to accept the attribute mappings.
11. Select *Finish* and wait for WMS to convert the selected shapes to feature objects in the map module.
12. You may now delete both shape files under *GIS Layers (Right click / Delete)* since you are done with them.
13. Toggle off the display of the Land Use coverage so the WMS display will update faster. If your Soil Type coverage is displayed, you should also toggle off the display of this coverage.
14. Select the *GSSHA* coverage to make this the active coverage.

3.9 Creating an Index Map from Land Use Data

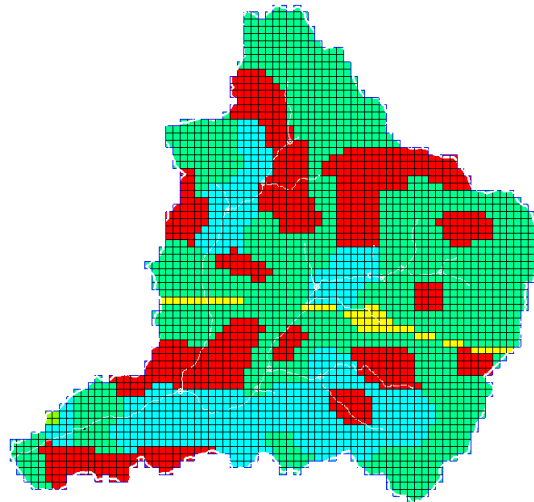
Now you can make an index map out of the land use coverage:

1. Switch to the 2D Grid Module .
2. Select **GSSHA / Maps...**
3. In the *Index – Grid* tab, for the *Input coverage (I)* use the *Land Use* coverage. Do not use a second GIS data source.
4. Set coverage attribute to *Id*.
5. Change the Index map name to *LandUse*.
6. Click on the **Coverages -> Index Map** button.
7. Select *Done*.

Under the 2D Grid Data folder in the project explorer you will notice that there are now three index maps listed: *Uniform*, *SoilType* and *LandUse*.

3.10 Creating a Mapping Table from Index Map

When you select the land use index map, the display should show grid cells colored in several colors representing the different land use IDs of the polygons that were mapped to the grid cells. Next, you 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 / Map Tables...**
2. Select the *Roughness* tab if it is not already selected.
3. Under '*Using index map*' choose the "Land Use" index map.
4. Click on "*Generate IDs*". Click *Yes* when asked to delete the selected process' existing IDs.
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.) If one of the roughness values does not show up

in your mapping table, just define roughness parameters for the values that show up in the mapping table.

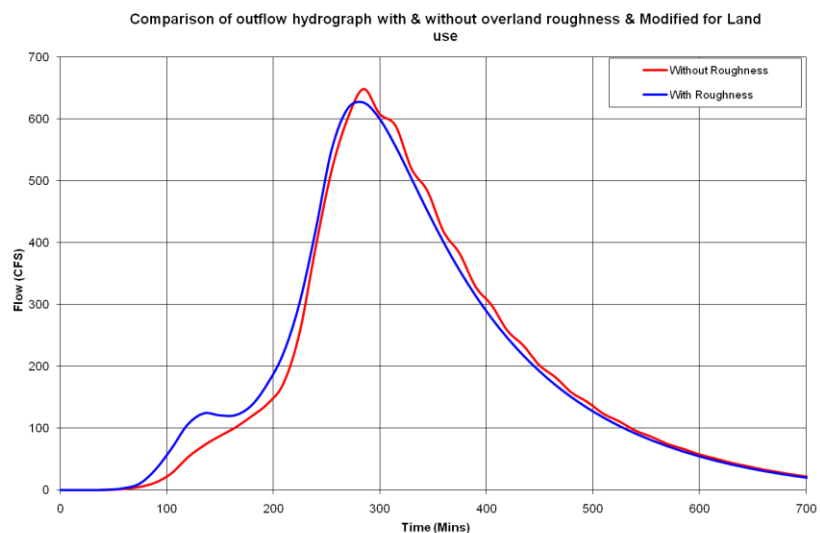
ID	Description	Roughness
11	Land ID #11, Residential	0.011
12	Land ID #12, Commercial Services	0.012
14	Land ID #14, Highway	0.011
16	Land ID #16, Mixed Urban or Built-Up Land	0.011
21	Land ID #21, Cropland and Pasture	0.035
23	Land ID #23, Confined Feeding Operations	0.035
41	Land ID #41, Deciduous Forest Land	0.100

6. Once you are done entering the roughness values, select *Done*.
7. Save your project as *C:\Training\Personal\Roughness\roughness.prj*.
8. Run GSSHA.

3.11 Visualization

After GSSHA successfully runs, open the outflow hydrograph.

1. Double-click on the outlet hydrograph to bring up the hydrograph plot.
2. Right click on the hydrograph plot and choose *View Values...*
3. Select the values under the *Flow* column, Right Click and select *Copy*.
4. Open the spreadsheet *C:\Training\InitialGSSHAComparison.xls* and paste the hydrograph ordinates under the column *With Roughness*. Paste your data on the white areas only.
5. Visualize the difference in the outflow hydrograph (under the *W_WO_Roughness* tab) in the two models.
6. Close the *View Values* window and the *Hydrograph* plot window in WMS after you are done.

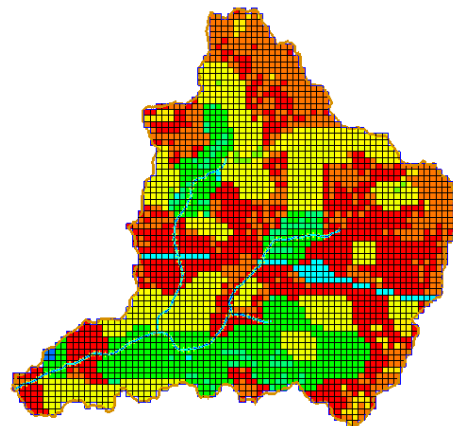


3.12 Creating a Combined Index Map

You will now adjust the model to take into account the effects of land use on the infiltration parameters. The values you have been using from Rawls and Brakensiek are for “bare earth” and do not account for the effects of land use. For instance, cropland with sandy soil and the concrete parking lot built on sandy soil will obviously have different “effective” infiltration rates. In this section, you will see how you can determine the effects of land use on infiltration in a GSSHA model.

You will continue with the model you have been working on. The first thing you will start with is to create a combined index map which will use both the soil and the land use coverages.

1. Select **GSSHA | Maps...**
2. In the *GSSHA Maps* dialog, select the following:
Input Coverage (1): Soil Type
Coverage attribute: Texture
Check on *Input coverage (2)* and select: Land Use
Coverage attribute: Id
Index map name: Combined
3. Click on the **Coverages -> Index Map** button. WMS will compute a new index map which specifies a unique ID for each combination of land use and soil type.
4. Click *Done* once the calculation is complete to close the *GSSHA Maps* dialog.
5. Now you can see an index map which should look something like this:



6. Next you will define the infiltration parameters for this combined index map in the infiltration mapping table. Select **GSSHA | Map Tables...**
7. In the *GSSHA Map Table Editor*, switch to the Infiltration tab. For the *Using Index map* field select *Combined* and click on the *Generate IDs* button. Click *Yes* to delete the existing IDs.

- Several fields are added in the table. These fields show all possible combinations of land use and soil type present in the watershed.

The infiltration parameters listed in the Rawls and Brakensiek table represent the bare earth soil infiltration parameters. However, you have generated a combined index map that combines the effects of land use and soil cover. The mapping table for this index map should account for the change in infiltration parameters caused by land use changes in a watershed.

- In the infiltration tab, click on the *Import Table...* button in the lower part of the editor.

- Browse and open *C:\Training\gssha.cmt*.

Now you can see the values filled in for all the fields. These are the values from standard Rawls and Brakensiek table. You will need to modify the values as follows:

- The values for *Roughness* and *Initial moisture* are correct, you do not need to edit those.
- Switch to *Infiltration* tab and replace the values in the upper two rows (i.e. for Hydraulic conductivity and Capillary head) with the values in the following table.

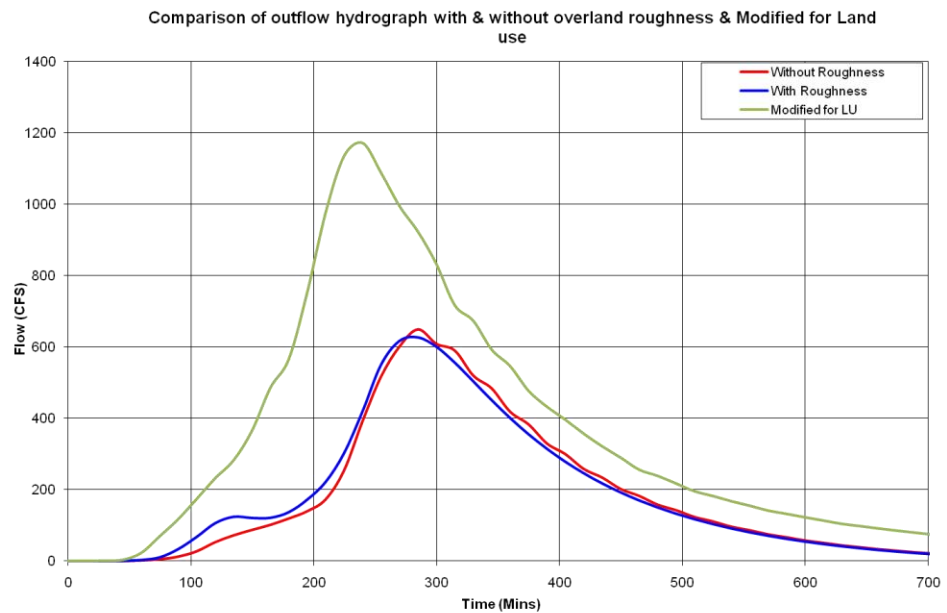
Note: Only the **Bold** faced numbers are changed from the Rawls and Brakensiek values. The rest of the parameters are the same as the values from standard GSSHA.cmt (Rawls and Brakensiek) table.

1	2	3	4	5	6	7	8	9	10	11	12
Silt loam	Silt loam	Silty clay loam	Silt loam	Silty clay loam	Silty clay loam	Silt loam	Silty clay loam	Clay loam	Silt loam	Clay loam	Silty clay loam
<i>Land ID #21</i>	<i>Land ID #11</i>	<i>Land ID #21</i>	<i>Land ID #41</i>	<i>Land ID #11</i>	<i>Land ID #41</i>	<i>Land ID #14</i>	<i>Land ID #14</i>	<i>Land ID #16</i>	<i>Land ID #16</i>	<i>Land ID #21</i>	
0.5	0.1	0.15	0.6	0.05	0.15	0.1	0.05	0.05	0.1	0.1	0.1
17.68	16.68	27.3	16.68	30.1	27.3	16.68	30.1	30.1	16.68	23.25	29.4
0.501	0.501	0.471	0.501	0.471	0.471	0.501	0.471	0.464	0.501	0.464	0.471
0.234	0.234	0.177	0.234	0.177	0.177	0.234	0.177	0.242	0.234	0.242	0.177
0.015	0.015	0.04	0.015	0.04	0.04	0.015	0.04	0.075	0.015	0.075	0.04
0.33	0.33	0.366	0.33	0.366	0.366	0.33	0.366	0.318	0.33	0.318	0.366
0.1	0.1	0.1	0.1	0.1	0.1	0.1	0.1	0.1	0.1	0.1	0.1

- Once you have filled in the values, click *Done*.
- Save the GSSHA project as *C:\Training\Personal\Infiltration\UpdatedInfil.prj*
- Run GSSHA.
- Open the outflow hydrograph.
- Right click on the hydrograph plot and choose *View Values*. Select the hydrograph ordinates, right click, and select *copy*.

3-12 GSSHA Modeling Using WMS – Volume 2


18. Open the spreadsheet *C:\Training\InitialGSSHAComparison.xls* and paste the hydrograph ordinates under the column *Modified for roughness*. Paste your data on the white areas only.
19. Compare the results you get here with the results from your previous simulation where you did not modify the parameters based on land use.
20. Close the *View Values* window and the *Hydrograph* plot window in WMS after you are done.



Stream Flow

4.1 Adding Streams to a GSSHA Simulation

While there are streams in the GSSHA coverage, channel simulation has not been turned on and channel properties are undefined. You define channel properties on the stream arcs in WMS. You will continue to work with the same project. Open your project or the backup copy if needed and then save your stream project to a new file to get started.

1. In the 2D Grid module  select ***GSSHA / Open Project File***.
2. If you saved your project in the last workshop, browse to and open the file ***C:\Training\Personal\Infiltration\UpdatedInfil.prj***. If the project was not saved, you may open this file: ***C:\Training\Infiltration\UpdatedInfil.prj***.
3. Save this project file with a different name so the original project remains unchanged. Select ***GSSHA / Save project File*** and save as ***C:\Training\Personal\Streams\stream.prj***.



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:

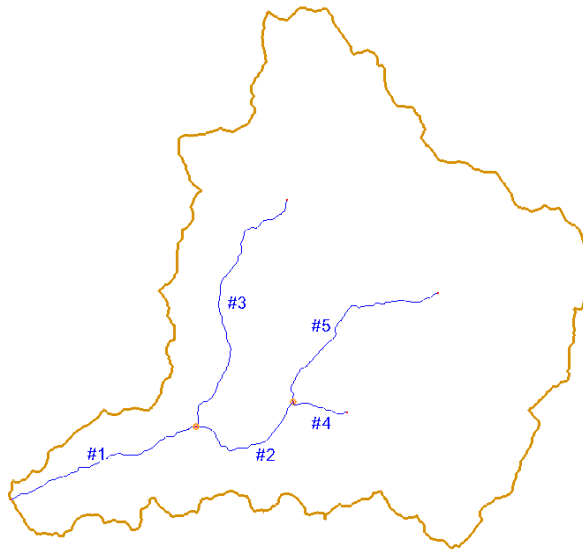
- Set the stream arcs to a GSSHA stream arc type
- Define the channel cross section geometry
- Define the channel thalwegs
- Turn on the channel routing job control item

4-2 GSSHA Modeling Using WMS – Volume 2

- Turn on additional output options to visualize the GSSHA stream Output

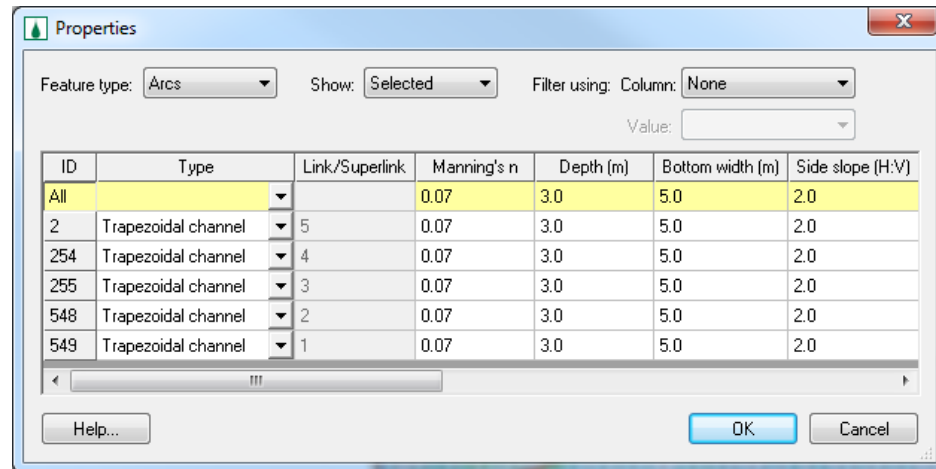
When you originally delineated this watershed, stream arcs in the GSSHA coverage were created. These stream arcs, however, are currently assigned a generic stream attribute. Further, the stream density (node spacing) is defined according to the threshold limit used when originally delineating the watershed. You can manually edit the streams after the watershed has been delineated, but you should determine the appropriate stream density when you are first delineating the watershed. You will see the importance of defining your stream point density in the hydrologic modeling wizard tutorial.

1. Select the **Map Module** .
2. Select 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. In the Properties dialog, select the “*Trapezoidal Channel*” type for the first row in the table (colored yellow) which allows you to define channel geometry for all selected stream arc(s). See the following figure.
6. Enter the remaining channel properties in the same first row of the Properties dialog according to the following table.

Parameter	Value
Manning's N	0.07
Channel Depth	3
Bottom Width	5
Side Slope	2



7. Select *OK*

So far all the stream arcs in the basin are defined as trapezoidal channels and have identical properties. There are two types of channels that GSSHA recognizes: trapezoidal and break-point (derived from actual cross section geometry). It would be nice to add some variation to our streams to represent the streams narrowing upstream. To do this you will increase the dimensions of the most downstream arc.

8. Click on the *Select Arc* tool  (be sure not to select the *Create Arc* tool which looks similar but does not have the arrow).
9. Select the most downstream arc (#1 in the figure above) and select ***Feature Objects/Attributes***
10. Enter the following attributes:

Manning's n	Depth	Bottom width	Side slope
0.075	3.5	7	2


11. Click *OK* to close the Properties window.

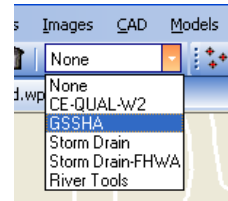
The stream arcs now have defined geometries and are ready to be smoothed.

4.2 Smoothing the Channel Thalweg Elevations


Now that the channel geometry for each stream arc has been defined, it is important to ensure that each arc is flowing downhill. Due to the coarse resolution of the DEM data, stream arcs will often have segments that flow uphill, which causes digital dams in the stream network. To avoid this problem, you should look at the stream profile and modify the stream node elevations until each arc segment goes downhill in the upstream-downstream direction. First, however, you should redistribute the vertices on the arcs to a more manageable spacing since by default stream nodes were created at the resolution of the original DEM, which is approximately 10 meters.

1. Select the *Map Module* .

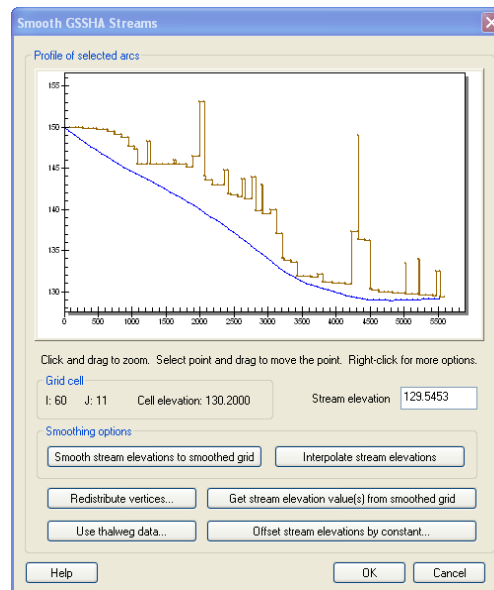
2. If the Model combo box on the X,Y,Z edit bar is not set to GSSHA, change it to GSSHA now.
3. Click on the *Select Feature Arc Branch* tool .
4. Click on the stream segment labeled “#1” from the previous figure.
5. Select *Feature Objects / Redistribute*.
6. Type 90 in the box next to *Spacing*, check the option to *Use Cubic Spline* and click *OK*.



The node spacing on the arcs is now about similar to the grid size. You should now smooth the thalweg elevations by selecting continuous channel segments until all channel segments have been smoothed.

7. Select the *Select Arc* tool .
8. Select arc # 1 as shown on the figure above, and while holding down the shift key, select stream arc “#3” so this continuous channel segment is selected with no branches.
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 naturally. We will mitigate this problem by making slight changes to the node elevations along the segment.



10. Select the “*Interpolate stream elevations*” buttons as many times as needed (probably 10-15 times) to generate a smooth stream segment with no uphill flow, then click *OK*.
11. If uphill flow cannot be eliminated in this manner, you should edit individual points by 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 do not create adverse slopes. Moving nodes manually and selecting the *Interpolate stream elevations* button is an effective way of eliminating any adverse slopes in your stream segments.

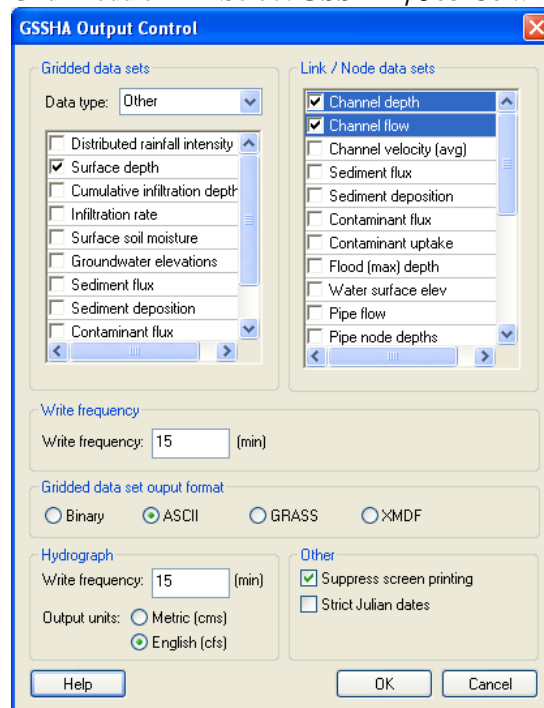
12. You may drag a box to zoom into any area where you want more detail. Other options for modifying the stream plot display can be accessed by a right click menu in the stream plot window.
13. After you have finished smoothing the streams and removing any adverse slopes, click *OK* to close the *Smooth GSSHA Streams* dialog.

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 13 until adverse slopes have been removed from all stream segments in the basin. After removing adverse slopes, your streams are now ready for use in the GSSHA model.

Before saving the model, the stream routing option must be turned on in the Job Control dialog for GSSHA to run with the 1D stream flow option.

4.3 Changing Output Control Options

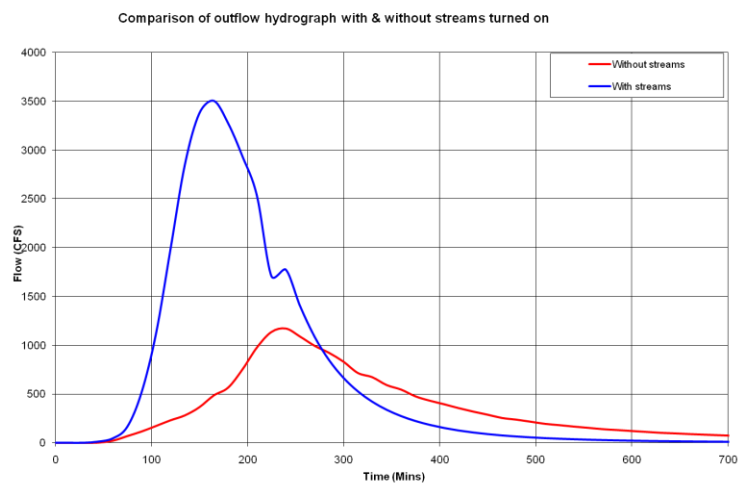
1. In the 2D Grid Module  Select *GSSHA / Job Control...*



2. Under *Channel routing computation scheme* select *Diffusive Wave*.
3. Click on *Output Control* and toggle on *Channel Depth* and *Channel Flow* and select *OK*.
4. Click *OK*.
5. Save the model as *C:\Training\Personal\Streams\stream.prj*

4.4 Running the model and viewing the outflow hydrograph

1. Select *GSSHA / Run GSSHA*. Select *OK* to run the model.
2. After the simulation has finished running, open the *Summary* file. View and compare the total amount of discharge, Final volume on surface, final volume in channels, infiltration, and other parameters. If the channel profile is not smoothed correctly, you will see a large amount of water trapped in the channel. Close the summary file and return to WMS.
3. Open the outlet hydrograph.
4. Right click on the hydrograph plot and select *View Values*. Then select and copy the flow values.
5. Open the spreadsheet *C:\Training\InitialGSSHAComparison.xls* and paste the hydrograph ordinates under the column *With Streams* (under the *Data* tab). Paste your data in the white areas only.
6. Visualize the difference in outflow hydrographs between the two models in the *W_WO_Streams* tab.
7. Close the *View Values* window and the *Hydrograph* plot window in WMS after you are done.



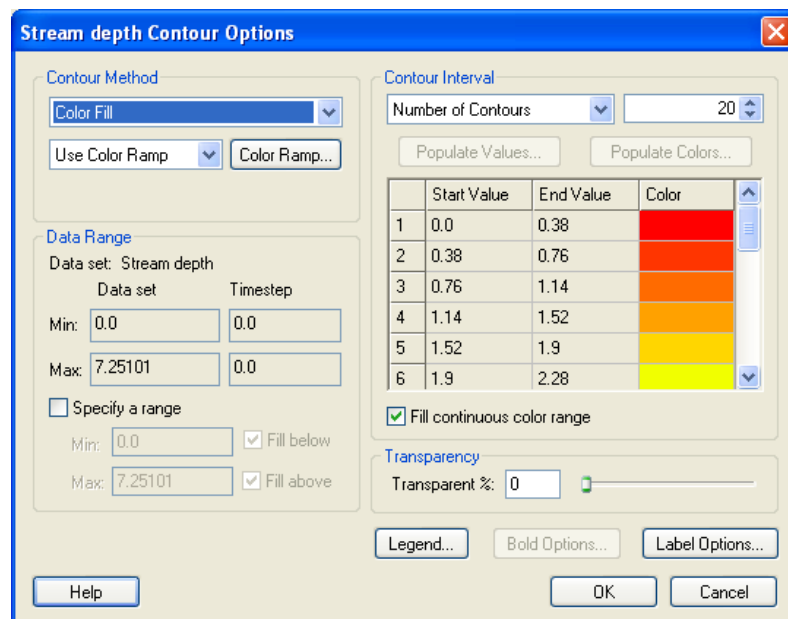
4.5 Visualizing Stream Flow Data

While the streams are connected to the overland flow plane, they represent a separate but coupled model. The depth and flow outputs from this stream hydraulic model are in a format called the link/node data set format. These files hold a data value for every node (arc segment) of every link at the same time step as that for the gridded output data. Two of the most common files in this format are the channel depth file (*.cdp) and the channel discharge file (*.cdq.).

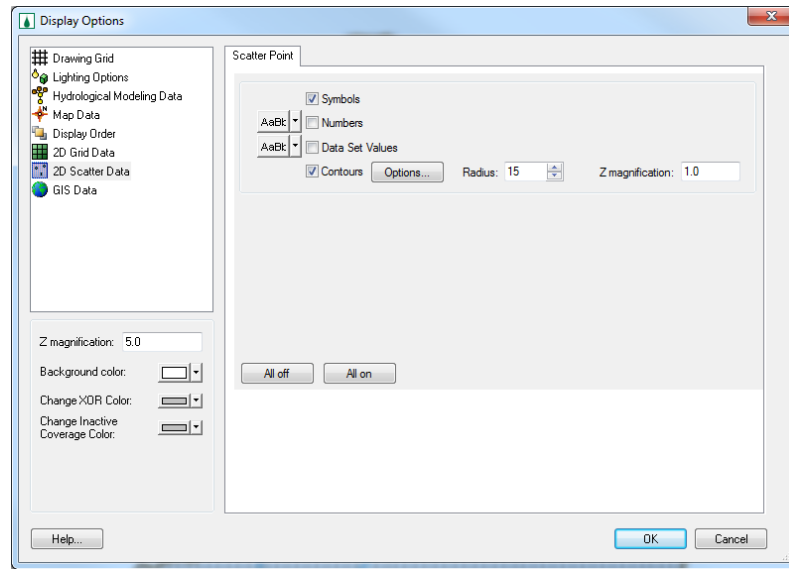
These stream data sets can be visualized in WMS. You can create depth contours, flood histograms, or a movie of how these values vary with time. In this tutorial, you will see some of these WMS visualization tools.


4.6 Changing Contour Properties

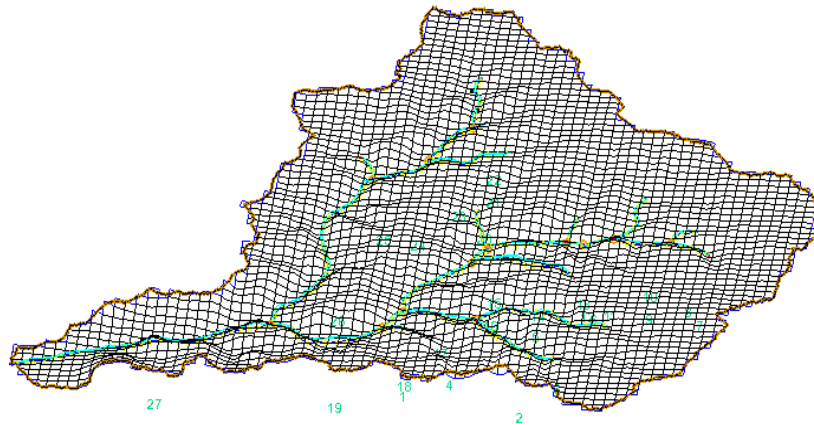
1. In the project explorer (near the bottom, under *2D Scatter Data*) click on the *Stream depth* dataset to select it.
2. Select **Display / Display Options**. In the 2D Scatter Data window, toggle off the display of symbols and toggle on the display of *Contours* and click on *Options*.
3. Change the Contour Method to *Color fill* and click *OK*.



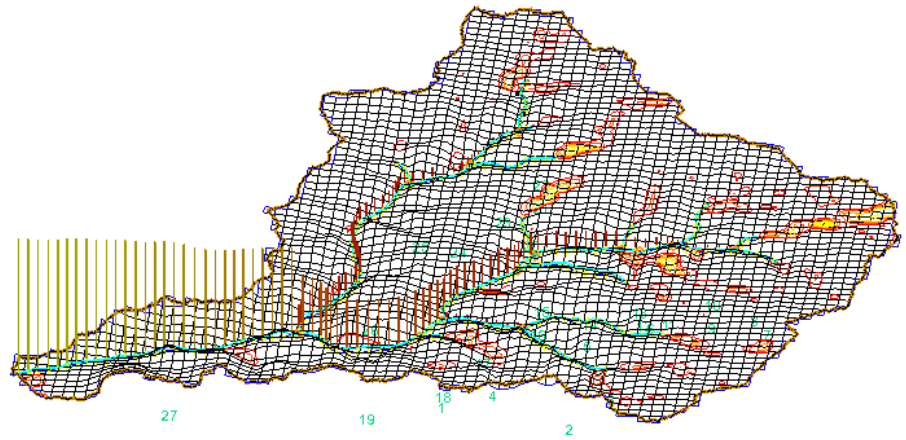
4. Change the *Radius* field to 15 and *Z-magnification* for the contours to 1.
5. On the left side of the dialog, set the Z-magnification to 5 (This sets the Z-magnification for all your data while the Z magnification under the scatter point options only sets the Z-magnification for scatter point contours).
6. Select *2D Grid Data* option in the list. Make sure that *Cells* option is set to *Smoothed*.
7. Click *OK*.



8. Click and select the “Rotate” tool . Then rotate the watershed by using the left mouse click so that the display is as shown in the following figure (Use the middle button on the mouse if you need to pan the display while the rotate tool is selected).



9. Now click on *Stream Flow* in the project explorer to select and toggle through different time steps in the *Properties* window at the right side of the WMS main window. As you cycle through the time steps, you can see cylinder diagrams representing the discharge flood wave along the streams. You can also see the depth contours along the 2D overland flow plane and view the effects of overland depth on stream flow as water enters the streams. Viewing the 2D overland flow and the 1D stream flow in this way should give you a clearer idea of the coupling of overland and stream flow in GSSHA.




Developing GSSHA Model Using Hydrologic Modeling Wizard in WMS

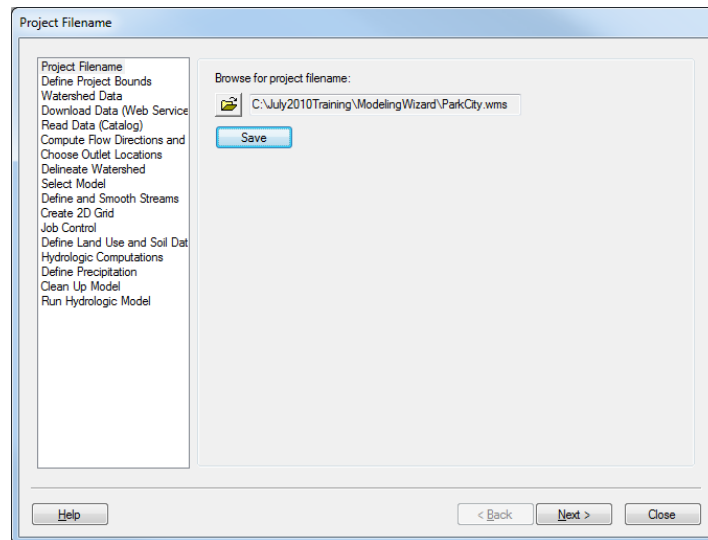
Because you may not use the modeling tools on a day to day basis it can be difficult to remember the step-by-step processes you must go through to develop a hydrologic model using WMS. The Primer on the GSSHA wiki is an excellent reference to help you remember the process, but the WMS interface also includes a hydrologic modeling wizard to help guide you through the basic model setup process. The wizard also streamlines and improves some of the processes you went through in the previous tutorial. Two processes that are streamlined by the wizard are the smoothing of stream channels prior to building the 2D grid and the creation of index maps from commonly available GIS shapefile data.


In this tutorial, you will create a GSSHA model with the Hydrologic Modeling Wizard using data in the area of Park City, UT. This model will use the same basic simulation options (2D overland flow, infiltration, and 1D stream flow) you have just set up in the previous tutorials and will serve as a review of what you have learned so far. The raw data for this Park City model can be found at *C:\Training\Raw Data\ParkCity*.

5.1 Starting the Hydrologic Modeling Wizard

1. Open a new instance of WMS. If you have started WMS previously, close it and restart WMS.

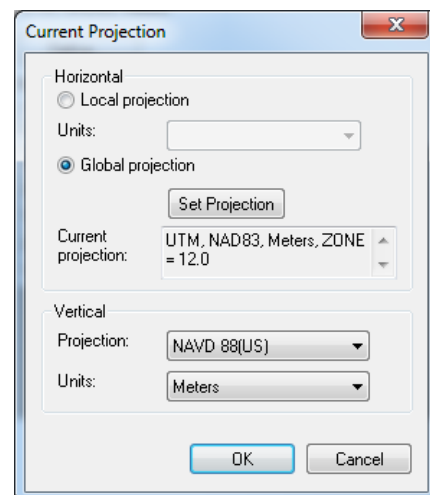
- Click on the “**Hydrologic Modeling Wizard**” button  on the bottom toolbar of the WMS interface to start the Hydrologic Modeling Wizard. This wizard will guide you through the basic set up of a GSSHA model (or any other hydrologic model supported by WMS). You can start at the beginning of the wizard or jump in and out at any step as needed. All of the WMS menus\tools work while the wizard is open, though you may have to move the wizard window aside to see the WMS windows.




- Click on the *Browse* button  to specify the project name and location to save. Enter **C:\Training\Personal\ModelingWizard\Parkcity.wms** as the project name and click *Save*.
- Click on the *Save* button in the wizard.

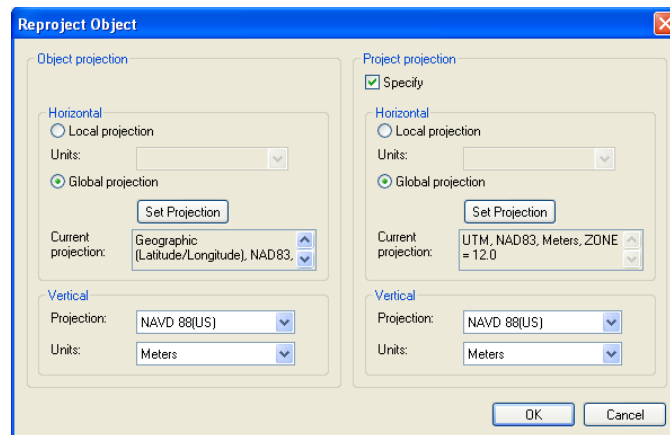
5.2 Defining the Project Boundary and Coordinates

- In the modeling wizard, click *Next* to go to the *Define Project Bounds* step of the wizard. Here you need to define your project coordinates and project bounds.
- Click on the *Define* button under *Project coordinate system*.
- Set the Projection and Datum to be *UTM NAD 83, Zone 12* and set the horizontal and vertical units to *meters*. Set the vertical projection to *NAVD 88 (US)* and click *OK*.



5.3 Importing the DEM Data

1. Click *Next* on the wizard to import watershed data. The data can be downloaded from the web, from catalog files with listings of local data and their boundaries, or data can be imported from any location on your computer or local network. Here you will import the data from a file on your computer.
2. Next to the *Open file(s)* text in the wizard, click on the browse button . Open the file *C:\Training\Raw Data\ParkCity\DEM\ned_35172081.hdr*.
3. Click *OK* to verify the data you have selected and Click *Yes* if prompted to change the coordinate system. *Skip steps 4 and 5 if you are not prompted to change the coordinate system.*
4. In the coordinate transformation dialog (See below), make sure that the “Convert From” coordinate is *Geographic NAD 83(US)* and that vertical units are set to meters (use *NAVD 88 (US)* for the vertical system). Then, if needed, check on the *Edit Project coordinate system* under the “New projection” option.



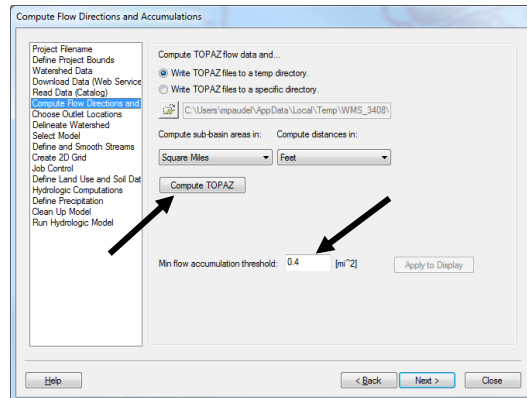
5. Set the *New Projection* Horizontal system to *UTM NAD 83 (US)*. Make sure that the UTM zone is 12 and the vertical system is *NAVD 88 (US)* in the new projection. The horizontal units for both the current and the new projections should be meters. Click *OK* and WMS will convert the coordinates of the DEM from Geographic NAD 83 to UTM NAD 83. After WMS converts the coordinates, you should be able to see the DEM contours behind the modeling wizard in the WMS main window.


5.4 Computing Flow Direction and Flow Accumulation

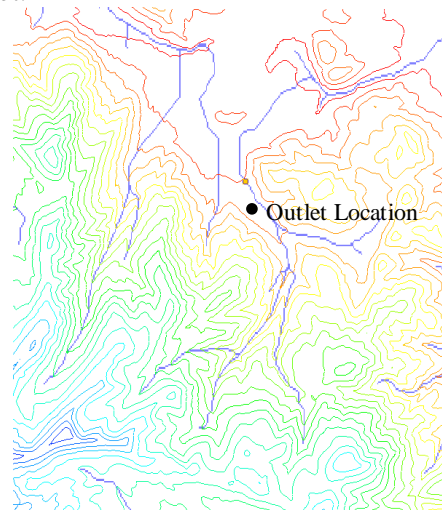
1. In the Modeling Wizard, click *Next* and *Next* again. You should be in the *Compute Flow Direction and Accumulation* part of the wizard.

5-4 GSSHA Modeling Using WMS – Volume 2

2. Set the *Min flow accumulation threshold* to 0.2 sq mile.

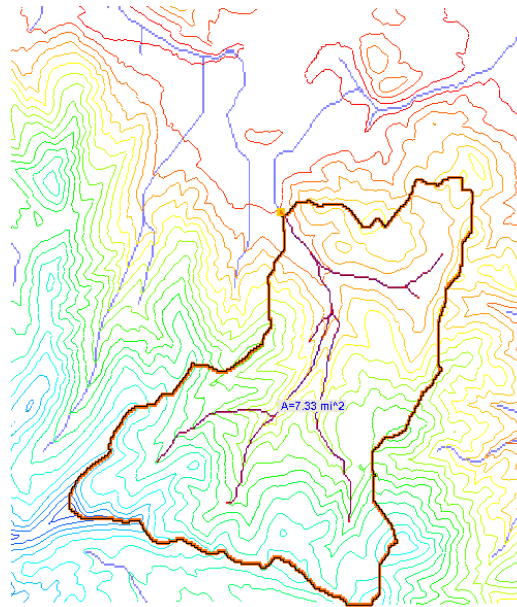



3. Click on the “*Compute Topaz*” button which will compute the flow directions and accumulations and infer the streams based on DEM data. Click on “*Close*” after computations are complete. If you move the modeling wizard dialog to the side of the WMS window, you can see blue lines added to the display that represent areas of high flow accumulation.
4. Click Next on the modeling wizard and Select *Create Outlet Point* . When you move the mouse pointer out of the wizard dialog, it will now change to a small cross hair. Locate the point where you want the outlet for the watershed to be, as shown in the following figure and click near a flow accumulation cell at this point to create an outlet.



5.5 Delineating the Watershed

1. Click *Next* on the modeling wizard and click on the *Delineate Watershed* button.
2. Your watershed should look like the following figure:



3. Save your WMS project by selecting **File / Save**.  Replace the existing file if prompted that it already exists.

5.6 Initiating GSSHA

Click *Next* on the wizard dialog and select the desired model to be *GSSHA* in the drop down box and click *Initialize Model Data*.

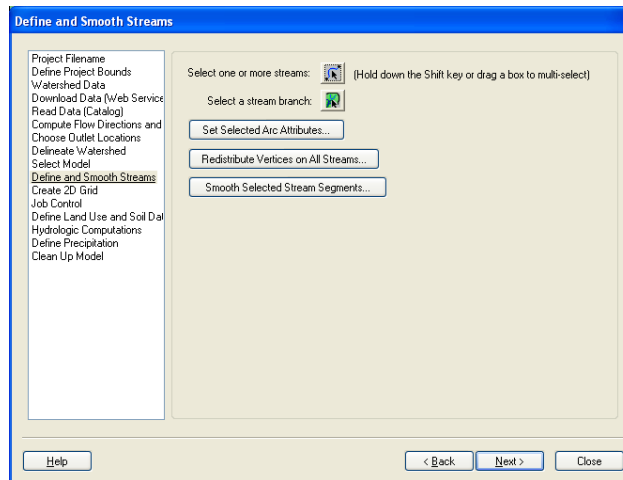
This generates the basic parameters necessary for GSSHA. Notice that the *Drainage* coverage in the project explorer has been converted to a *GSSHA* coverage and a default GSSHA model (GSSHAModel1) has been created under the 2D Grid Data folder.


5.7 Defining Stream Channels

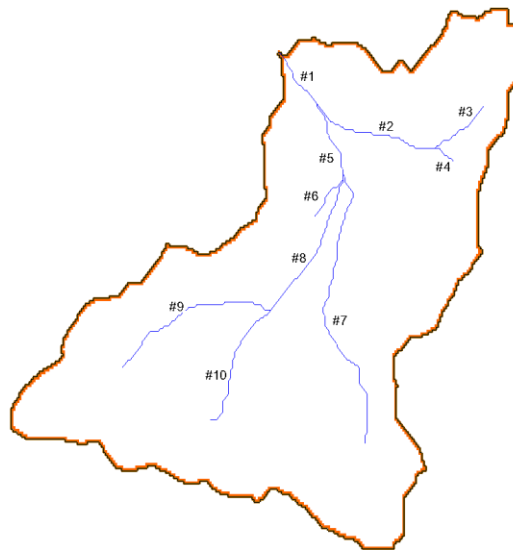
All the streams defined so far in GSSHA are generic and they do not have any defined shape or geometry. Also, because of a lack of resolution in the DEM data, the stream bed may not go from higher to lower elevations in the downstream direction which might create problems during simulation. In this section, you will define the channel cross section and roughness properties and you will also check and smooth the channel bottom.

5.7.1 Defining stream cross sections

1. In the wizard dialog, click *Next*. This will display the *Define and Smooth Streams* step as shown in the following figure:

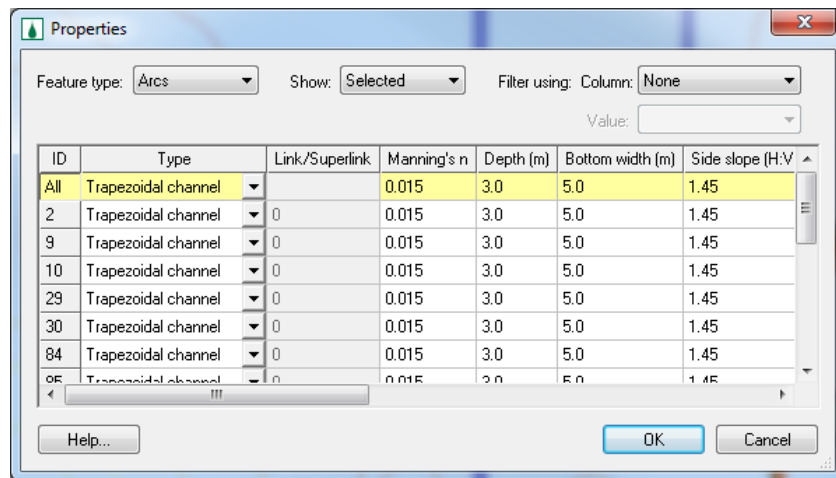


2. In the wizard, click on the *Select feature line branch tool*  and click on stream segment #1 (most downstream stream segment that is connected to the watershed outlet, you might need to zoom to the outlet area) in the following figure.
3. Select the *Set Selected ArcAttributes* button.



4. This will open up the *Channel Properties* dialog. Select the *Trapezoidal channel* option for all the arcs (from the *All* row). Enter the following values for all the streams:

Manning's n	0.015
Channel depth (m)	3
Bottom Width (m)	5
Side slope (H:V)	1.45




5. Click *OK*. Here we are assuming that all the stream segments have the same cross section. However, as demonstrated previously, you could define different stream properties for each arc.

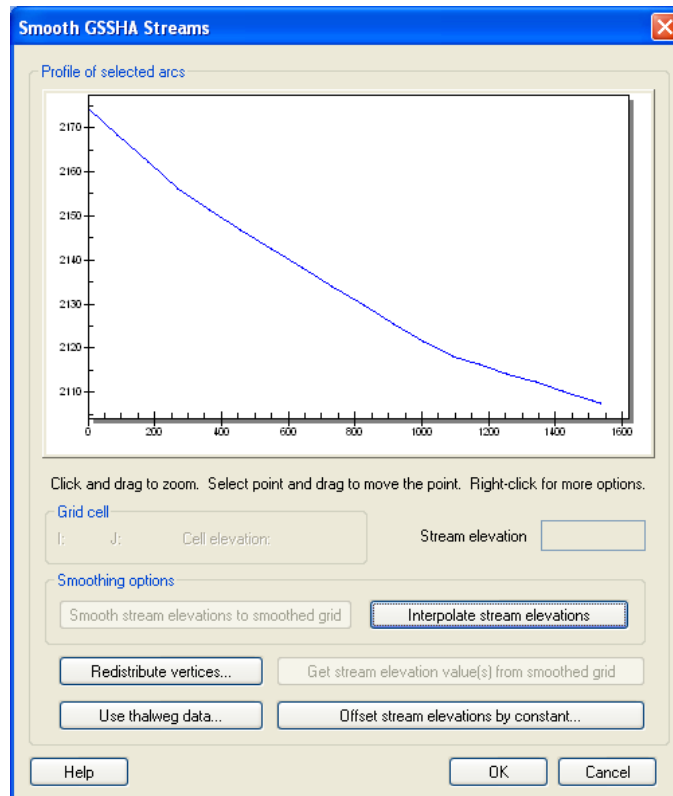
5.7.2 Smoothing the stream thalwegs

1. With the entire stream network still selected, click on the *Redistribute Vertices on All Streams* button in the wizard.
2. Enter 90 for the Spacing and turn on the option to *Use Cubic Spline*.
3. Click *OK*.

The node spacing on the arcs is now something greater than the DEM resolution and approximately the resolution of the computational grid we will generate later. We will now proceed to smooth the thalweg elevations by selecting continuous stream segments in the channel until all stream segments have been smoothed.

4. In the wizard, click on the *Select Arc* tool  and select a line of streams from the bottom to the top of the stream network without selecting any branching streams. Start at the most downstream segment and select (Hold Shift Key down and click) all arcs to the left (or right) until you reach the top. Make sure that there is no branch in your selection and the selection of streams is continuous.
5. With these streams selected, select *Smooth Selected Stream Segments* button on the wizard dialog.

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 present in a model where they do not exist naturally. You can solve this adverse slope problem by making slight changes to the node elevations along each segment.



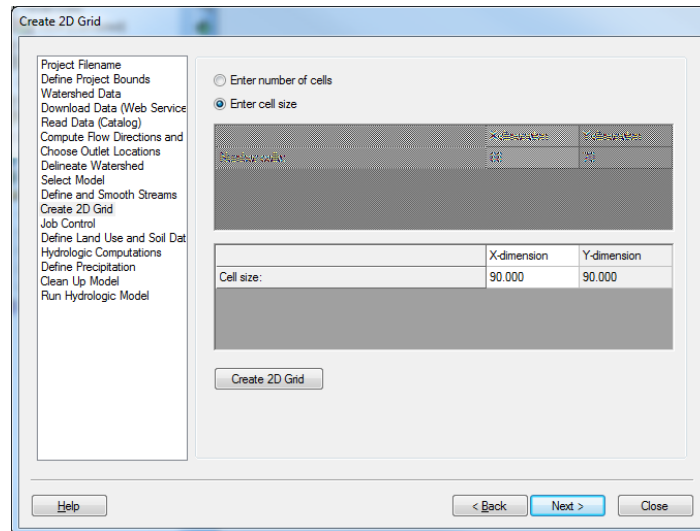
6. Select the *Interpolate Stream Elevations* button as many times as needed to generate a smooth stream segment with no uphill flow, then click *OK*.
7. If uphill flow cannot be eliminated in this manner, you can edit individual points by selecting the node and then dragging the point to a new position or editing the value in the box next to *Stream elevation*. Be especially careful to make sure the slope next to the outlet is not adverse.

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 4 through 7 until no stream arcs in the basin have adverse slopes. Your streams are now ready for use in the GSSHA model.

5.8 Creating a Grid

Once the channel is smoothed, we will create the 2D grid for GSSHA. Click *Next* on the wizard dialog.

1. Make sure that the *Enter Cell size* option is selected and enter a cells size of **90** (meters) in the X-dimension field. The same cell size is used for the Y-dimension field because GSSHA grid cells must be square.




2. Click on the *Create 2D Grid* button to create the 2D grid for the watershed.
3. Click *OK* to interpolate the grid elevations from the DEM and Click *Yes* to delete the existing background DEM.



5.9 Defining Job Control

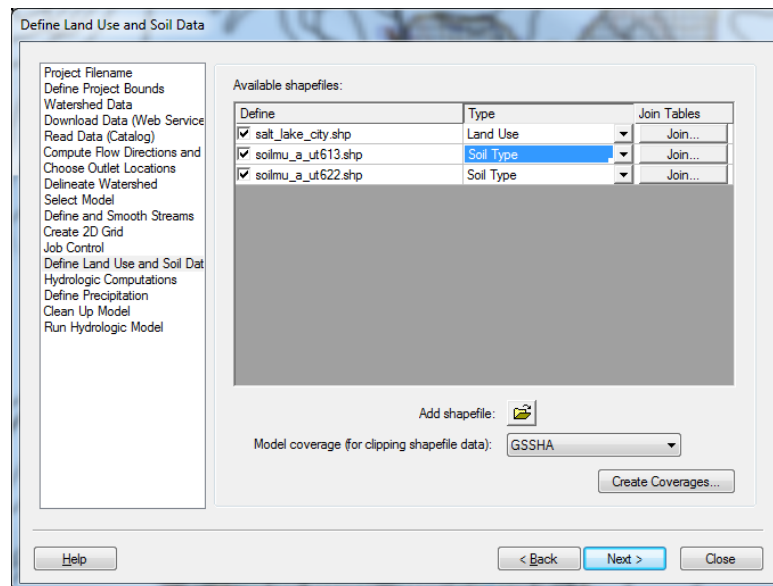
1. Click *Next* in the wizard dialog.
2. Enter start and end dates of (07\01\2010, 12:00:00PM to 07\02\2010, 12:00:00PM) with a computational time interval of 10 sec. for the model simulation. Click on *Set Job Control Data*.
3. Click *Next*.

5.10 Define Land use and Soil Data

GSSHA models can be parameterized using readily available GIS shapefiles for land use and soil information. In this section, you will import these shapefile layers and map them to WMS coverages.

1. Click on the *Add shapefile* button  in the lower section of the *Define Land Use and Soil Data* window. Browse to and open the file *C:\Training\Raw Data\ParkCity\Luse\salt_lake_city.shp*. Notice that the land use shape file is added to the “GIS Layers” in the WMS Project Explorer window.

2. Click on the *Add Shapefile* button again  and open the file *C:\Training\Raw Data\ParkCity\SSURGO_Soil\soil_ut613\spatial\soilmu_a_ut613.shp*. Notice that this soil shapefile does not overlay your watershed completely. Because the shapefile does not overlay the entire watershed, you need to open an adjacent soil dataset so you have soil data that covers your entire watershed.
3. Click on the *Add Shapefile* button again  and open the file *C:\Training\Raw Data\ParkCity\SSURGO_Soil\soil_ut622\spatial\soilmu_a_ut622.shp*. This will cover the entire watershed.
4. Since these are the raw SSURGO data, you need to join the attributes to them. The NRCS SSURGO attributes are joined using a menu command available in the project explorer. In the project explorer, right click each of the SSURGO shapefiles (*soilmu_a_ut613.shp* and *soilmu_a_ut622.shp*) and select *Join NRCS data*. Check both toggles on and click *OK*.
5. In the Wizard, make sure that the *Type* field is correctly selected for the corresponding shapefiles. The *Type* field should be set to *Land Use* for *salt_lake_city.shp* and to *Soil Type* for the other shapefiles.



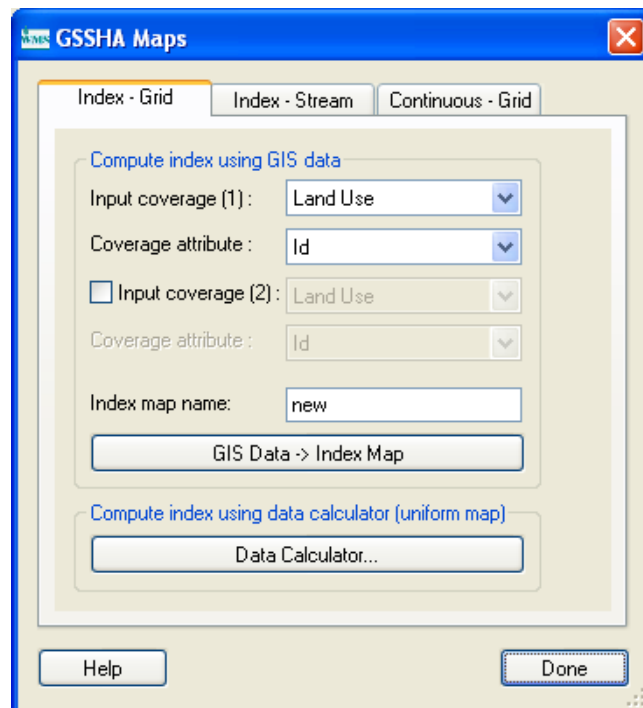
6. In the wizard, click on *Create coverages* to create land use and soil coverages and map the corresponding shape file polygons to their respective coverages. Because the shapefiles use standard attribute names the mapping should be defaulted correctly, so accept the defaults in the *GIS to Feature Objects Wizard* by clicking *Next* and *Finish* when prompted.
7. After both the coverages are created, toggle off the display of your soil shapefiles (all the *.shp* files under the *GIS Layers* folder) and toggle off the *Land Use* and *Soil Type* coverage display under the

Coverages folder in the *WMS Project Explorer*. Doing this cleans up your display and makes the model display faster.

8. Click *Next* on the modeling wizard.

5.11 Creating Index maps and defining Model Parameters

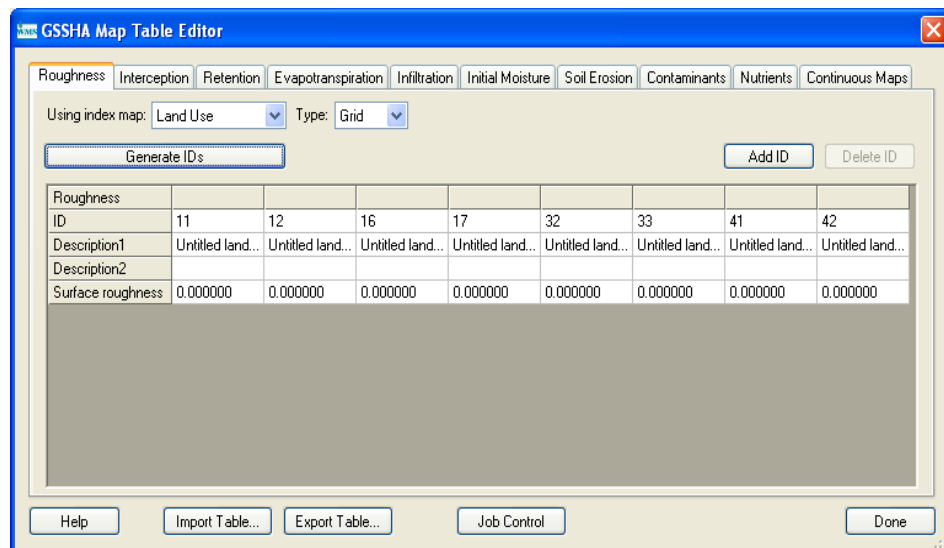
1. Click on the *Compute Index Mapping Tables* button which will bring up GSSHA Maps dialog.



2. Select *Land Use* as the Input Coverage and *ID* as the Coverage Attribute. Enter *Land Use* as the Index Map Name. Then click on the ***Coverages -> Index Map*** button.
3. Select *Soil type* as the Input Coverage and *Texture* as the Coverage Attribute. Enter *Soil Type* as the Index Map Name. Then click on the ***Coverages -> Index Map*** button.
4. Toggle on the Input coverage (2) option so that we can create a combined index map using both the land use and soil type coverages. Select *Land Use* as the Input Coverage (1) and *ID* as the Coverage Attribute. Select *Soil Type* as the Input Coverage (2) and *Texture* as the Coverage Attribute. Enter *Combined* as the Index Map Name. Then click on the ***Coverages -> Index Map*** button.
5. Click *Done*. Notice that three index maps have been added to your GSSHA model (*Land Use*, *Soil Type*, and *Combined*).
6. The *GSSHA Map Table Editor* dialog will pop up. You need to define the index mapping parameters.

5.12 Defining Model Parameters

1. In the GSSHA Map Table Editor dialog, switch to the Roughness tab (it is selected by default).
2. In the *Using index map* field, select *Land Use* and click on the *Generate IDs* button. You can see the different fields populated based on the land use data.



3. Similarly switch to the *Infiltration* tab and click *Yes* to turn the infiltration option ON. In the *Job Control* Dialog, select the *Green + Ampt with soil moisture redistribution* option and click OK.
4. In the *Using Index map* field, select *Combined* and click the *Generate IDs* button. This will generate a unique ID for all combinations of land use and soil type data within the watershed.
5. Switch to the *Initial Moisture* tab, select *Soil Type* in the *Using Index Map* field and select the *Generate IDs* button.
6. Now select the *Import table* button at the bottom left of the *GSSHA Map Table Editor Dialog*.
7. Browse and open the file *C:\Training\gssha.cmt*.

NOTE: This table has typical values for the watershed parameters based on standard soil types and land use classifications. Importing this table will populate values in the mapping tables which we will verify later. One important thing to remember is that the values from the table are not absolute and the modeler needs to take ownership of them and adjust as necessary for the specific conditions of a given area. Generally these values are best estimated either by field measurement or by the calibration of the model. Values obtained from this table should be used only as initial parameter estimates.

8. Now if you switch back to the *Roughness* tab, some preliminary values for roughness are already mapped based on the land use data. Make

sure that each field has a non zero value. If one does then you must enter a suitable value.

9. Similarly, switch to the *Infiltration* tab. The parameter values listed in this mapping table are based on soil texture only. However, the *Combined* index map is a combination of both land use and soil type information. Because of this, you should adjust the parameters to reflect the impact of different types of land use on the soil-based mapping table parameters. For example, “sand” in an agricultural area and in a residential area have different properties. Similarly, “Loam” in a forested area and in an Industrial area have different properties.
10. The land use IDs are the same as the IDs used in the USGS Land Use Classification Codes. You can go to:
http://emrl.byu.edu/gsda/data_tips/tip_landuse_table.html
to find a table that relates land use classification code IDs to land use descriptions.
11. For the columns with land use IDs of 11, 12, 16, and 17, reduce the values of Hydraulic conductivity to half of the values currently in the table. These land uses are impervious relative to bare soil, causing the infiltration rate to decrease. In a modeling situation, you would use your judgment to approximate the effect of land use on hydraulic conductivity and other infiltration values.
12. In the *Initial Moisture* tab, enter the values 0.1, 0.15, 0.2 for Loam, Sandy clay Loam and Clay Loam respectively. Make sure that the values you enter here are smaller than the values for porosity in the *Infiltration* tab.
13. Once you are done defining the parameters, click “Done”.

5.13 Editing Parameters

1. Click on the *Edit Parameters* button on the modeling wizard. This will open the *GSSHA Job Control Parameters* dialog.
2. Select the *Diffusive Wave* option for the “Channel routing computation scheme” in the top right side of job control dialog.
3. Select the *Output Control* button. Toggle on the *Cumulative infiltration depth* and *Infiltration rate* gridded data sets and the *Channel depth* and *Channel flow* link/node data sets.
4. Enter a *write frequency* of 15 minutes for both the overland model (this value is in the *Write frequency* box) and the hydrograph and change the hydrograph units to be *English* (*Metric* is selected by default).
5. Click *OK* to close the *GSSHA Output Control* dialog.
6. Click *OK* to close the *GSSHA Job Control Parameters* dialog.
7. Click *Next* on the modeling wizard to define the precipitation data.

5.14 Defining a Meteorological Model

1. In the wizard, click on the *Define Precipitation...* button. In the *GSSHA Precipitation* dialog, change the *Rainfall event(s)* to *Hyetograph*. Then click on the *Define Distribution* button.
2. In the XY Series Editor window that opens, select “*Type I-24 hour*” under the *Selected Curve* option. This is the dimensionless temporal distribution of an SCS type I-24 hour storm. Click *OK*.
3. Enter an Average Depth of 63.5 mm and make sure the start date\time is set to 07\01\2010 12:00:00 PM. Click *OK*.
4. Click *Next* on the wizard.

5.15 Cleaning and Checking the Model

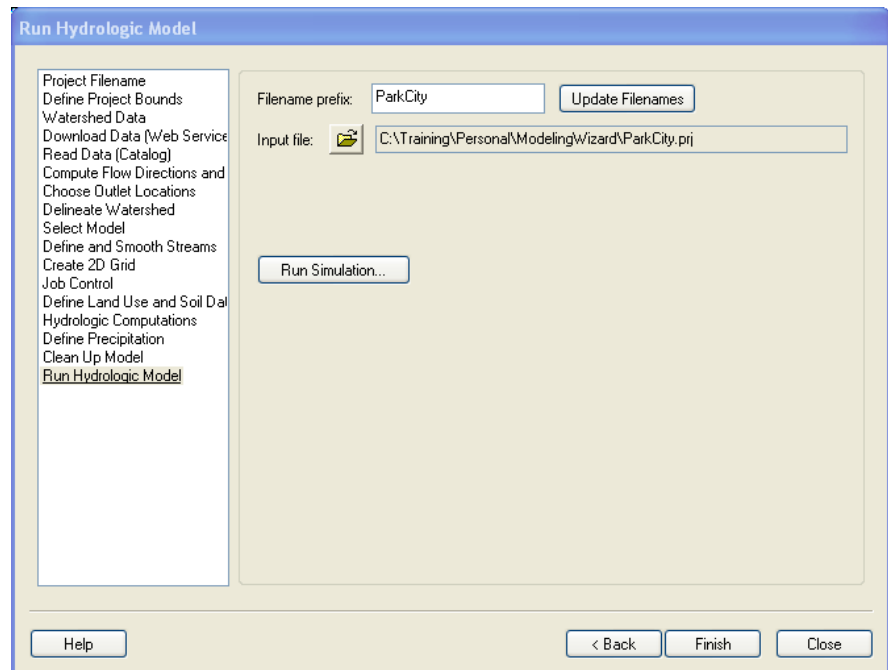
1. When you reach this step in the hydrologic modeling wizard, you are done defining most of the parameters and components required for the model. Click on the *Clean Up Model* button. This button does several things to clean up the display of the model. This button also prepares your model for running by running the Clean Dam routine to improve the overland flow and bringing up the model checker to show any input errors. In the model checker, clicking on an error takes you to a GSSHA dialog where you can make suitable corrections to the input data.
2. Click *Done* to close the *GSSHA Model Check* window. Click *Close* to close the *Model Wrapper* for the “Cleandam” program

5.16 Save the Model

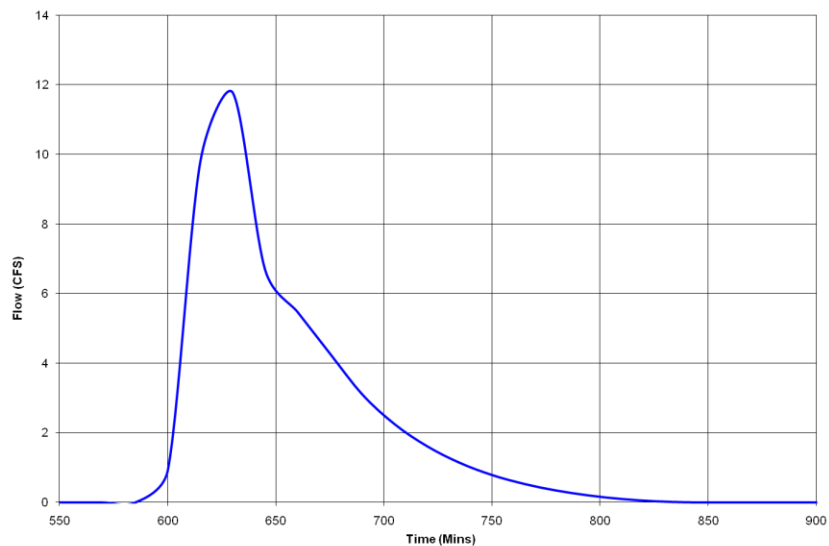
1. Click on the *Save* button to save the GSSHA project file. The GSSHA project file is in a different format than the WMS project file, so even though you have saved the WMS project, this step is necessary to save the GSSHA project file.
2. Save your project as ***C:\Training\Personal\Modeling Wizard\Parkcity.prj***.

5.17 Run the Model

1. Click *Next* in the modeling wizard. This will bring you to the *Run Hydrologic Model* wizard step.
2. Make sure the path you defined to save the project is correct (***C:\Training\Personal\Modeling Wizard\Parkcity.prj***)
3. Click the *Run Simulation* button, turn off the option to *Suppress screen printing*, and click *OK* to run GSSHA.



4. As GSSHA runs, you can see the time steps being computed and discharge at each time step in the model wrapper. Click “Close” after the simulation is complete to exit the model wrapper and read the solution.
5. After the solution is read, look at the outlet hydrograph and depth dataset contours.



In this tutorial you learned how WMS automates GSSHA model generation. This tutorial repeated all the steps in the previous several workshops from the hydrologic modeling wizard. The hydrologic modeling wizard provides a standard framework for developing a basic GSSHA model. Once you setup a basic GSSHA model, the model will be ready for modeling additional hydrologic processes.

Post-Processing and Visualization of GSSHA Model Results

In this tutorial, you will explore different post-processing options. You have already been using some of these post-processing tools in the previous tutorials. This tutorial explains and leads you through exploring some of the post processing tools in WMS.

6.1 Open an Existing Project

1. Start WMS. If you are continuing from a previous tutorial close and restart WMS.
2. In the *2D Grid Module*, select ***GSSHA / Open Project File***. Browse to and open the file ***C:\Training\Visualization\visualization.prj***.

This Judy's Branch project models distributed roughness, infiltration, and distributed precipitation. Notice several precipitation gages covering the watershed area.

3. Because of the land use and soil type coverages as well as the precipitation gages, the display looks cluttered and the screen refreshes slowly. Turn off the display of all items under *Coverages* in the *WMS Project Explorer* window. Then turn the display of the *GSSHA* coverage back on. Select the *Frame* button to zoom into the *GSSHA* coverage after the *GSSHA* coverage is turned on. See the following figure:



6.2 Read Solution

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 project explorer but they must be for the same grid and streams. For example, you could vary roughness parameters in a single GSSHA model and have solution sets for each set of roughness parameters and then you can compare results for each set of input parameters.




Since this project has already been run, you do not have to run it in order to view the results. However, the results are not read when you read the project file, so you need to tell WMS to read the solution files.

1. In the *2D grid* module, select ***GSSHA / Read Solution..***
2. A dialog box pops up here that allows you to select a simulation (though the current project is there by default). Select *OK* in this dialog to start reading the solution.
3. Notice that solutions are added to the project explorer under the *2D Grid Data* and the *2D Scatter Data* folders.

6.3 Visualization


6.3.1 Visualizing the Outlet Hydrograph

As soon as the result is read, you can see a small hydrograph icon at the outlet of your watershed. Clicking on this icon shows the outflow hydrograph in a plot window.

1. In the 2-D grid module , click on the “**Select Hydrograph**” tool  and double click on the small hydrograph icon  near the outlet.
2. Clicking on the icon opens the hydrograph in a plot window.
3. There are several options to control the display of the hydrograph plot, including options to view and export the plotted values by right clicking on the hydrograph plot and selecting the appropriate menu item.
4. The simplest way to copy the plot’s data to a spreadsheet (as you have seen in previous tutorials) is to choose the *View Values* option and then *copy\paste* selected values to a spreadsheet.


5. Close the Hydrograph plot window when you are done.

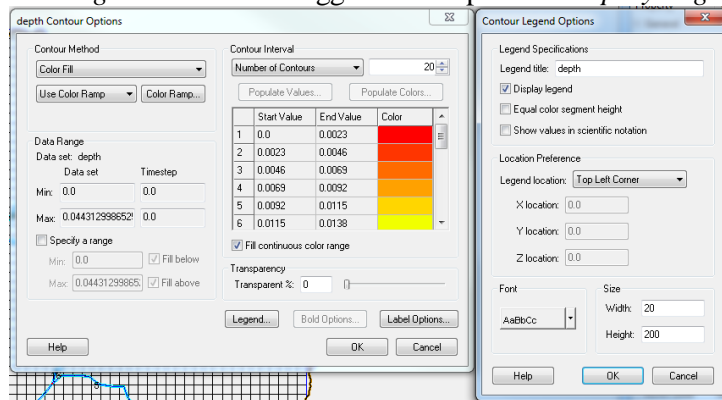
6.3.2 Reading the Summary File

Notice that there is a folder named *visualization* under the *Visualization* project folder. The solution folder has a symbol 's' for solution  visualization (GSSHA) . Notice the summary file under this project's solution.

1. Double-click on the *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 the mass balance error and the volume remaining on the surface to know that GSSHA is simulating the processes correctly.
4. When you are done, close the summary file

6.3.3 Visualizing Depth Contours

1. In the *Project Explorer*, right-click on *Depth* under the *visualization* folder.
2. Select *Contour Options* (this menu item can also be found under the *Display* menu or by using the Contour Options macro )
3. Under the Contour Method select the option to *Color fill* from the drop down box.
4. Click the *Legend* button and toggle on the option to *Display Legend*.






5. Click *OK* on the *Contour Legend Options* dialog.
6. Click *OK* on the *Contour Options* dialog.
7. In the *Properties* window (to the right side of the WMS window), a set of time steps appear. If the time steps are not showing, either the *Depth* data set is not selected in the project explorer or something else has been selected since selecting the Depth dataset. If this happens, click somewhere outside the watershed boundary and select the *Depth* data set with your left mouse button.
8. Click on first time step and use the down arrow key (on the keyboard) to cycle through the time steps. About half way through the time steps, notice that the depth contours vary in color at different time steps. You

do not have to go through the time steps consecutively; you can select any time step to make it active. This model is actually not that interesting for surface depth because it “drains” very well so it takes a few time steps before you see any changes and even then the changes are modest.

9. Notice that the legend is also displayed.

You can also adjust the lighting (*Display / Display Options / Lighting Options*) and the vertical exaggeration (*Display / View / Z Magnification*).

Try panning , zooming , and rotating .

6.3.4 Visualizing Stream Flow Results

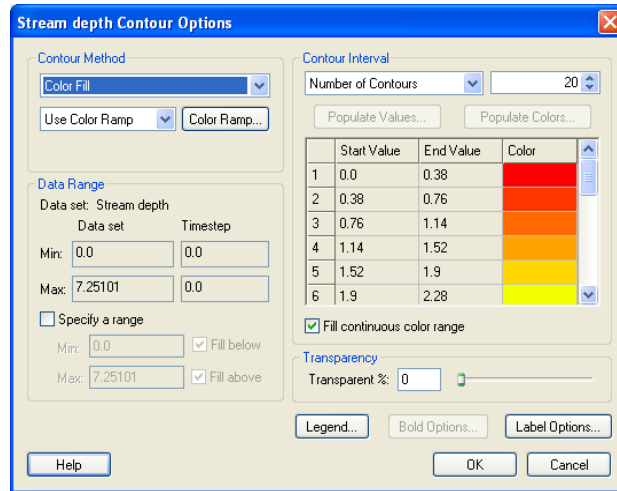
While the streams are connected to the overland flow plane, they represent a separate but coupled model. The depth and flow outputs from this stream hydraulic model are in a format called the link/node data set format. These files hold a data value for every node (arc segment) of every link at the same time step as that for the gridded output data. Two of the most common files in this format are the channel depth file (*.cdp) and the channel discharge file (*.cdq).

These stream data sets can be visualized in WMS. You can create depth contours, flood histograms, or a movie of how these values vary with time. In this tutorial, you will see some of these WMS visualization tools.

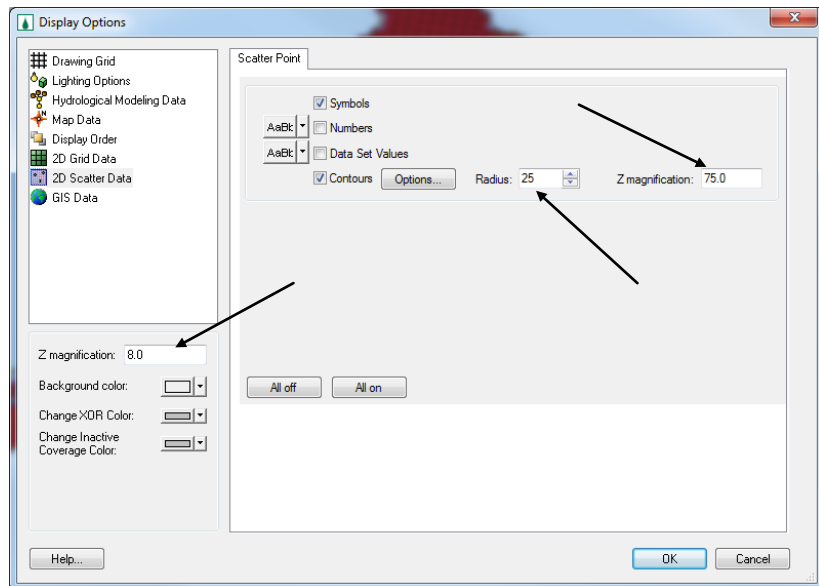
In the *2D Scatter Data* folder in the project explorer, you’ll see two data sets, *Stream Depth*, and *Stream Flow*. The *Stream Depth* and *Stream Flow* data sets are link/node data sets. In order to visualize stream depth and/or stream flow, follow these steps:


6.3.4.1 Visualizing Depth Contours

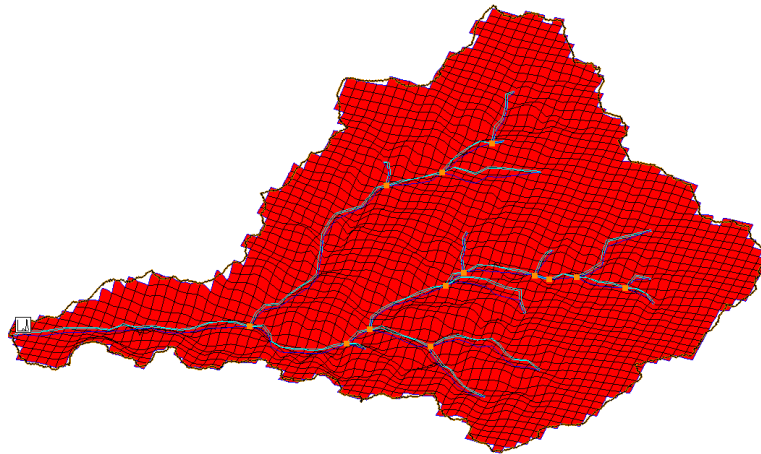
1. In the project explorer click on *Stream Depth* to select it.
2. Select *Display / Display Options*. In the *2D Scatter Data* window, toggle off the display of *Symbols* and toggle on the display of *Contours* and click on *Options*.
3. Change the contour Method to *Color fill* and click *OK*



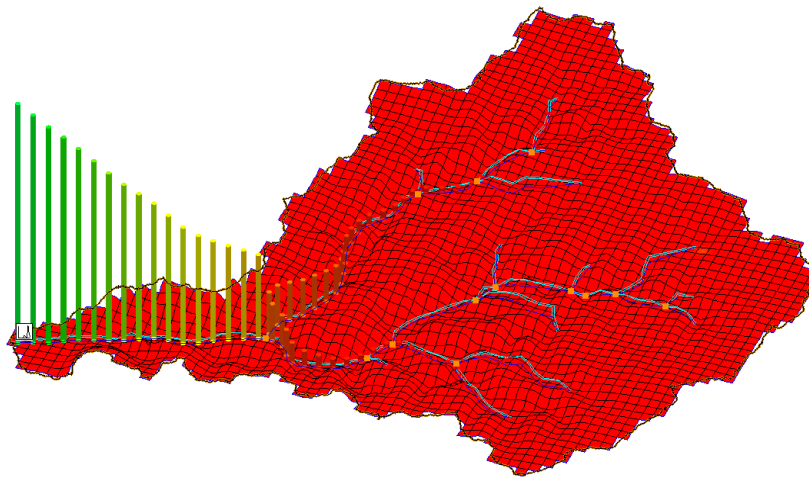
- In the *Display Options* dialog, change the *Radius* to 25 and *Z-magnification* to 75.



- On the left side of the dialog, set the *Z-magnification* to 8 and Click *OK*
- Select the “*Rotate*” tool . Rotate the watershed by clicking on the left mouse button and dragging in the window until it looks similar to the following figure:



7. Now click on the *Stream Flow* in the Project Explorer to select it and cycle through the different time steps (shown on the properties window at the right side of WMS main window). Notice the bar diagrams representing the flood wave being generated and traveling along the streams to the outlet.




Notice the overland flow depth contour changing with the flood wave bar diagrams. This gives a clearer idea of the link between overland flow and stream flow.

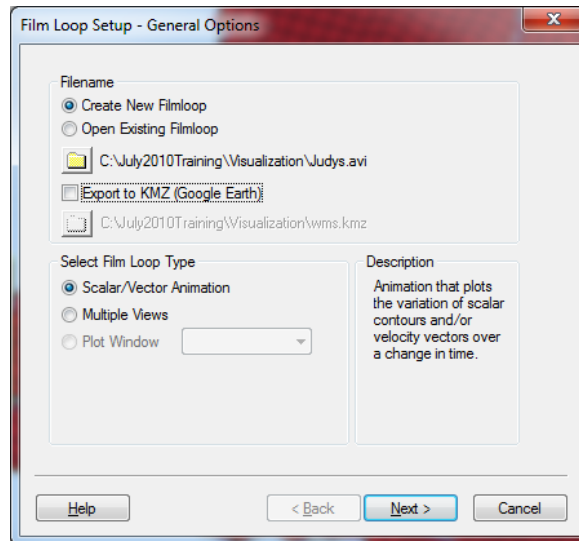
6.3.5 Creating an Animation Film Loop

These processes can be animated using the film loop command.

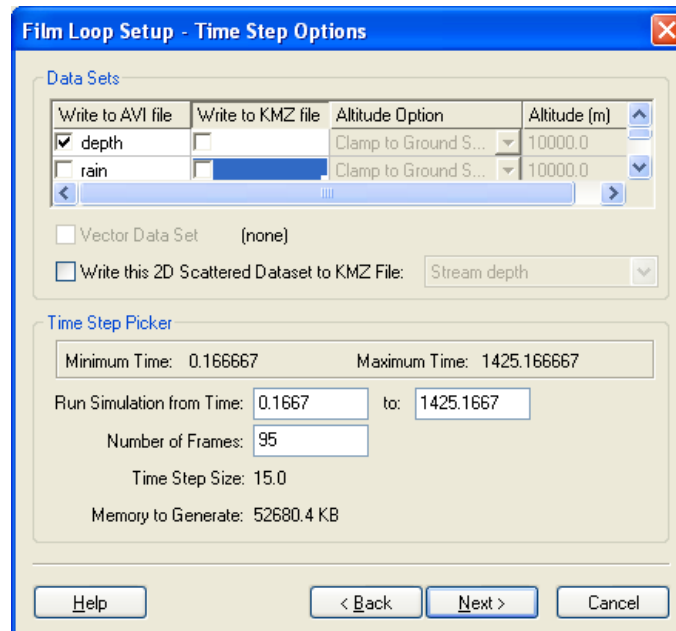
1. In the 2D-Grid Module , select *Data / Film Loop...*

Post-Processing and Visualization of GSSHA Model Results 6-7

2. Make sure to select “*Create New Filmloop*” and choose the location to save your AVI movie file by clicking on the *Browse* button . Save your movie as *C:\Training\Personal\Visualization\Judys.avi*
3. We will NOT export Google Earth animation at this moment. Make sure the *Export to KMZ* option is checked OFF.



4. Select the *Scalar\Vector Animation* option and click *Next*.
5. Under *Write to AVI file*, check *depth* data set to select it. Turn off all the boxes under *Write to KMZ file*.
6. Turn off the *Write this 2D Scattered Dataset to KMZ File* option.






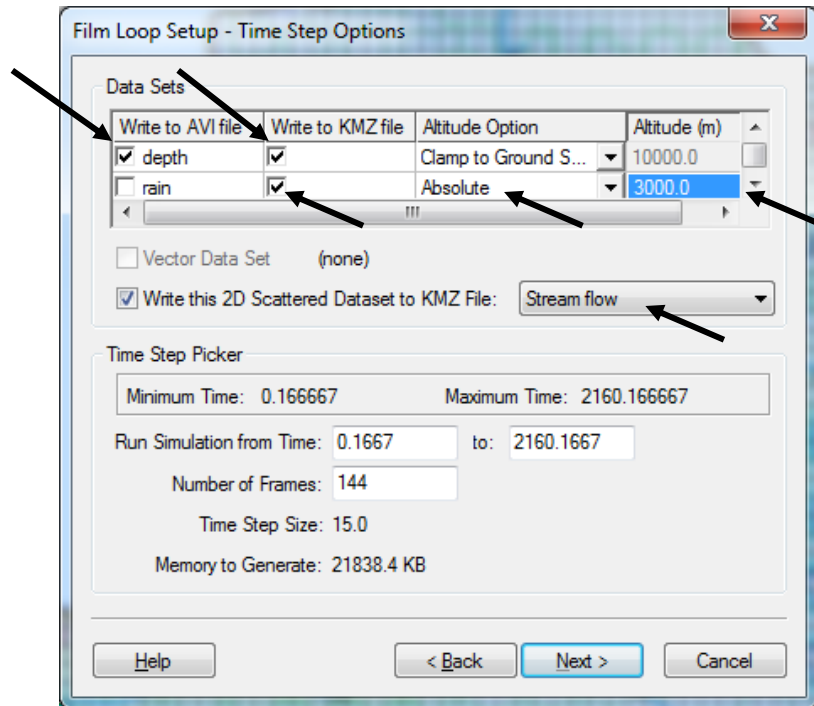
7. Click *Next*.

8. Do not change any clock options. Click “*Finish*”. WMS takes some time to create the movie. The movie begins playing once the entire movie is saved. In the movie, you can see the overland flow depth contours and stream flood waves animated simultaneously. The AVI file is saved and can be put inside a presentation or played separately as needed. Close the animation player when you are done viewing it.

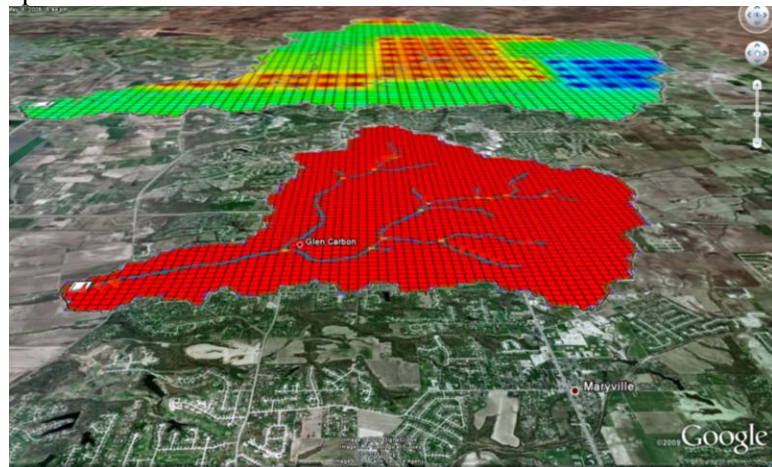
6.3.6 Creating a Google Earth Animation

You can export an animation to Google Earth and display your animation in its real-world location.

1. Change the display to plan view by clicking *Plan View* . You cannot export a movie to Google Earth in an oblique view.
2. In the 2D-Grid Module , select *Data / Film Loop...*
3. Choose the location to save your AVI movie file by clicking on the *Browse* button . Save your movie as *C:\Training\Personal\Visualization\Judys.avi*
4. Select the option *Export to KMZ (Google Earth)*. Define the location to save the KMZ file as *C:\Training\Personal\Visualization\Judys.kmz*
5. Toggle the on “*Scalar\Vector Animation*” option and click Next.
6. Under *Write to AVI file*, turn on the *depth* data set to select it.
7. Turn on the toggle boxes under *Write to KMZ file* corresponding to both the *Depth* and *Rain* datasets.
8. For the *Rain* dataset, select *Absolute* under *Altitude Option* and enter 3000 meters for *Altitude*.
9. Check on the option *Write this 2D Scattered Dataset to KMZ File* and select *Stream flow*.



10. Click *Next*.
11. Do not change any clock options. Click "*Finish*". WMS will take some time to (3-5 minutes) to create the movie. After the movie is completed, WMS opens the Google Earth animation.
12. Once in Google Earth, tilt the Google Earth view to see the animated Stream Flow lines. Play around with some of the Google Earth display options, such as changing the transparency of the data and changing the speed of the animation.





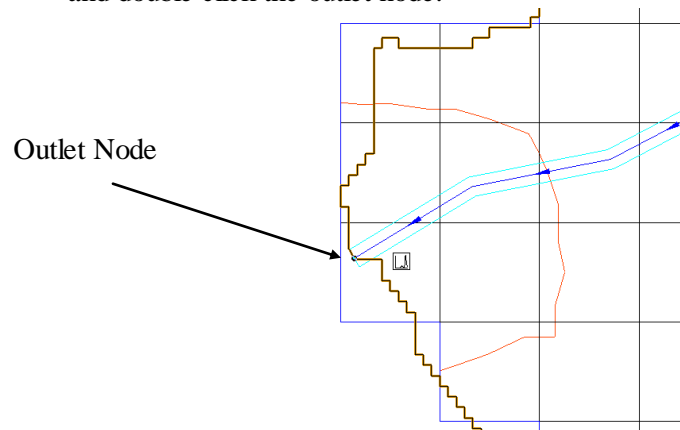
The upper layer represents the distributed rainfall input and the lower layer represents overland runoff.

6.4 Comparing the simulation results with the Observed data

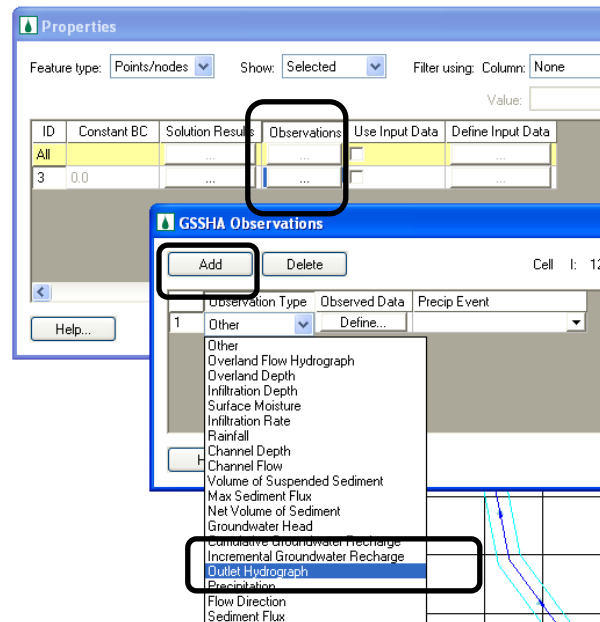
If you have some measured data at the watershed outlet or at any stream node upstream in the watershed, you can enter the measured data and compare the simulation result with that data.

We will use an arbitrary observed flow hydrograph at the outlet and compare it with simulation results.

1. Zoom in around the watershed outlet.
2. In the map module , select the *Select Feature Point\Node* tool  and double click the outlet node.



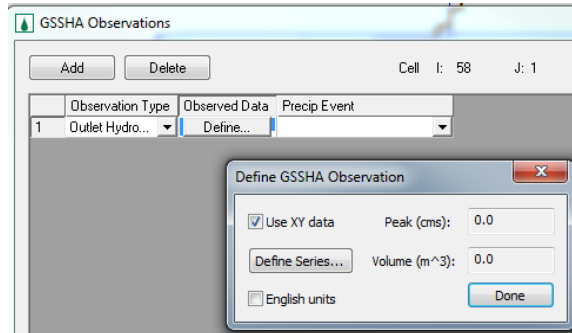
3. In the *Properties* dialog that opens, scroll all the way to the right and under *Observations* select the browse button.



4. In *GSSHA Observations* dialog, click *Add* and select *Outlet Hydrograph* under *Observation Type*.

Post-Processing and Visualization of GSSHA Model Results 6-11

- Click the *Define* button.
- Check *Use XY data* option and click the *Define Series* button.



- The observed flow data can be found at **C:\Training\Raw Data\JudysBranch\Judys_ObsFlow.txt**.
- In Notepad, open the **Judys_ObsFlow.txt** file and copy and paste the data into the XY Series editor in WMS. You should see the hydrograph plotted on the XY-series editor.
- Click OK, click *Done*, and click OK. This will close all but the properties dialog.
- Click the Browse button under *Solution Results* (just to the left of *Observations*).
- You can now compare the observed flow data with the simulation results in the *GSSHA Solution Analysis* dialog by clicking on the *Observed (Outlet Hydrograph)* and the *visualization (GSSHA) Stream flow (Channel Flow)* check boxes.

