
***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
75 South 200 East, Suite 201
Provo, Utah 84606
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 THE DEM DATA	1-1
1.2	COMPUTING FLOW DIRECTION AND FLOW ACCUMULATION	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	1-11
1.12	OPENING SUMMARY FILE	1-15
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	2-5
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
4	USING THE MAPPING TABLE AND GIS DATA TO DEFINE VARIABLE SURFACE ROUGHNESS.....	4-1
4.1	USING LAND USE DATA	4-2
4.2	CREATING AN INDEX MAP FROM LAND USE DATA	4-2
4.3	CREATING A MAPPING TABLE FROM INDEX MAP	4-3
4.4	VISUALIZATION	4-4
4.5	CREATING A COMBINED INDEX MAP	4-4
5	STREAM FLOW	5-1
5.1	ADDING STREAMS TO A GSSHA SIMULATION	5-1
5.2	SMOOTHING THE CHANNEL THALWEG ELEVATIONS	5-3
5.3	CHANGING OUTPUT CONTROL OPTIONS	5-5
5.4	RUNNING THE MODEL AND VIEWING THE OUTFLOW HYDROGRAPH	5-6
5.5	VISUALIZING STREAM FLOW DATA	5-7
5.6	CHANGING CONTOUR PROPERTIES	5-7

6	DEVELOPING GSSHA MODEL USING HYDROLOGIC MODELING WIZARD IN WMS	6-1
6.1	INITIATING THE WIZARD.....	6-1
6.2	DEFINING THE PROJECT BOUNDARY AND COORDINATES.....	6-2
6.3	IMPORTING THE DEM DATA.....	6-3
6.4	COMPUTING FLOW DIRECTION AND FLOW ACCUMULATION.....	6-4
6.5	DELINEATING THE WATERSHED	6-5
6.6	INITIATING GSSHA	6-5
6.7	DEFINING STREAM CHANNELS.....	6-5
6.8	CREATING A GRID.....	6-8
6.9	DEFINING JOB CONTROL.....	6-9
6.10	DEFINE LAND USE AND SOIL DATA.....	6-9
6.11	CREATING INDEX MAPS AND DEFINING MODEL PARAMETERS	6-11
6.12	DEFINING MODEL PARAMETERS	6-12
6.13	EDITING PARAMETERS	6-13
6.14	DEFINING A METEOROLOGICAL MODEL	6-14
6.15	CLEANING AND CHECKING THE MODEL.....	6-14
6.16	SAVE THE MODEL	6-14
6.17	RUN THE MODEL	6-15
7	POST-PROCESSING AND VISUALIZATION OF GSSHA MODEL RESULTS	7-1
7.1	OPEN AN EXISTING PROJECT.....	7-1
7.2	READ SOLUTION	7-2
7.3	VISUALIZATION	7-2
7.4	COMPARING THE SIMULATION RESULTS WITH THE OBSERVED DATA	7-9

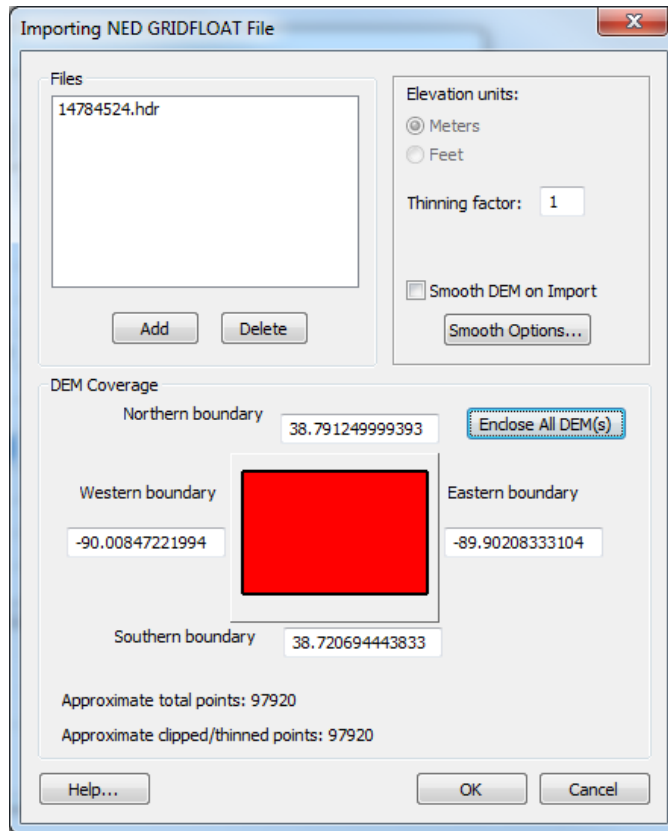
GSSHA Initial Overland Flow Model Setup

To start a GSSHA model, you must have a digital elevation model and a boundary polygon for the watershed. For this tutorial we will use a DEM as the elevation model and also create the boundary polygon and the streams using the same delineation process in WMS we learned in the previous tutorial. WMS will then use the DEM to interpolate cell elevations and the boundary polygon to select whether or not a cell is active (inside the basin) or inactive (outside the basin). Unlike lumped parameter watershed models, the basin should not be subdivided into sub-basins. There should only be a single basin boundary.

1.1 Importing the DEM Data

We learned how to download DEM using web services in the previous workshops. Here we will open a DEM which has already been downloaded.


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




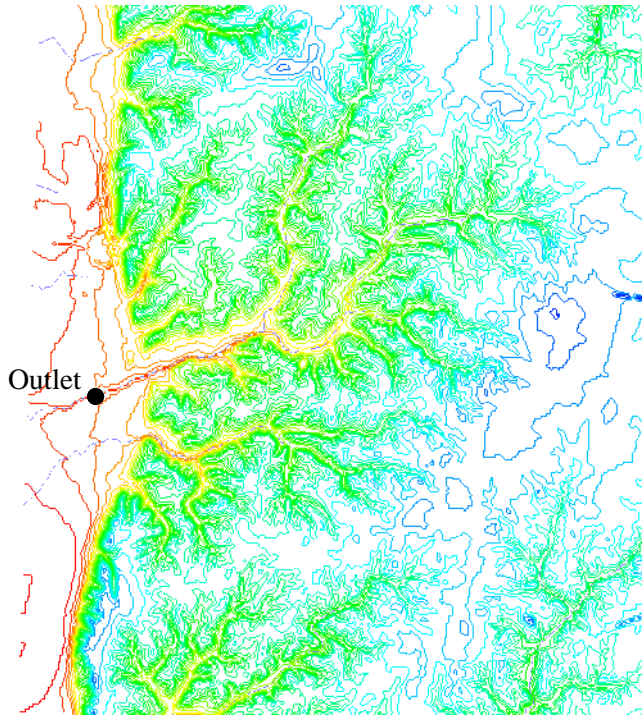
4. Select *Yes* when prompted for coordinate conversion.
5. Set the *Object Projection* to *Geographic NAD83* if different.
6. Select *Specify* option for *Project projection*. Then select *Global projection* and click *Set Projection* button.
7. Select *UTM NAD83 Zone 15* and click *OK*.
8. Make sure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

Now you can see the DEM contours plotted on WMS main window.

1.2 Computing Flow Direction and Flow Accumulation

1. In the *Drainage Module* , select *DEM | Compute TOPAZ Flow Data* and Click *OK* twice to accept the default processing. TOPAZ will compute the flow direction and accumulation and infer the streams 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 . Locate the outlet in approximately the location shown in the figure below.

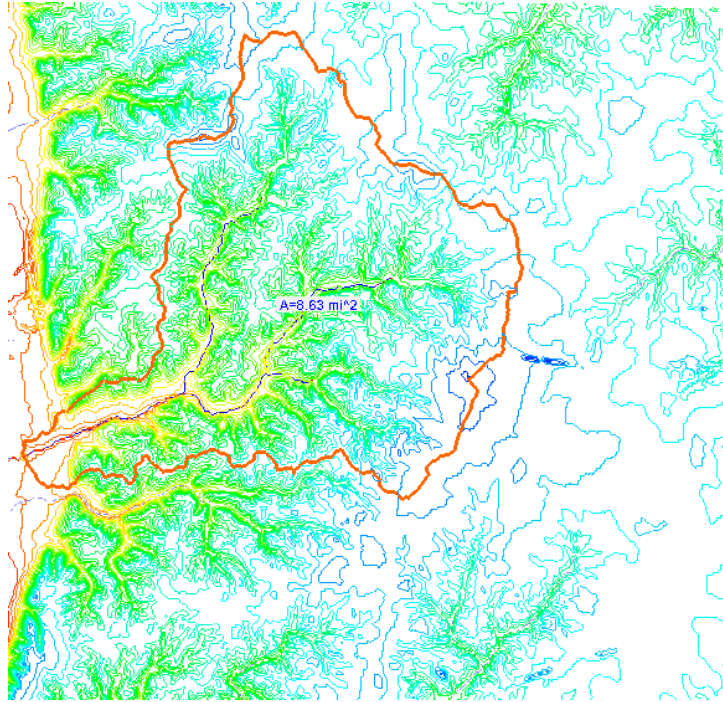



1.3 Delineating the Watershed

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 completes the process you did manually in the previous tutorial of converting the DEM streams to arcs, defining the basin from the DEM and converting the boundary to a polygon, and then computing the basin data all at the same time. Later you will learn how to run the complete 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. Click on “*Frame*” button . Your watershed should look somewhat like this (**Error! Reference source not found.**).




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

1.4 2D grid generation

The basic GSSHA model begins with the basin boundary in the Drainage coverage and a DEM. It is helpful to have the streams developed at an appropriate density for your model as well. In this step you are essentially telling WMS to take the boundary polygon and the DEM and create a 2D grid that fits the boundary and has an elevation data set that is interpolated from the DEM. For more information on selecting appropriate cell sizes see the GSSHA Primer. (http://gsshawiki.com/index.php?title=GSSHA_Software_Primer)

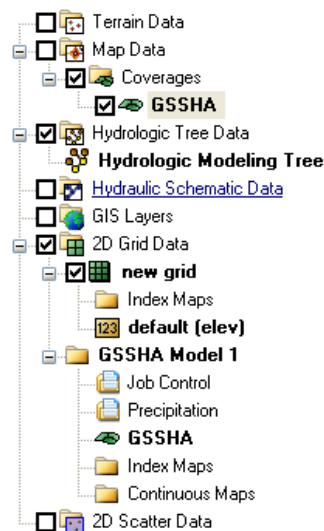
1. Switch to *Map Module* .
2. Click on *Select Feature Polygon Tool*  and right click anywhere within the watershed polygon. Then select *Create Grid*. Select *Yes*.

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*.
4. Click *OK* to interpolate grid cell elevation from the DEM, and select *Yes* when prompted if you want to delete the DEM data.
5. Click on “*Frame*” button .

You can now see grid cells over the watershed. You should have also noticed that under *Coverages* in the data tree, the *Drainage* coverage has been now changed to *GSSHA*.


6. Save your WMS project, *File / Save*.

You’ll notice in the project explorer that a 2D Grid called new grid is also visible in the tree now and a GSSHA model (GSSHA Model 1) has been initialized.

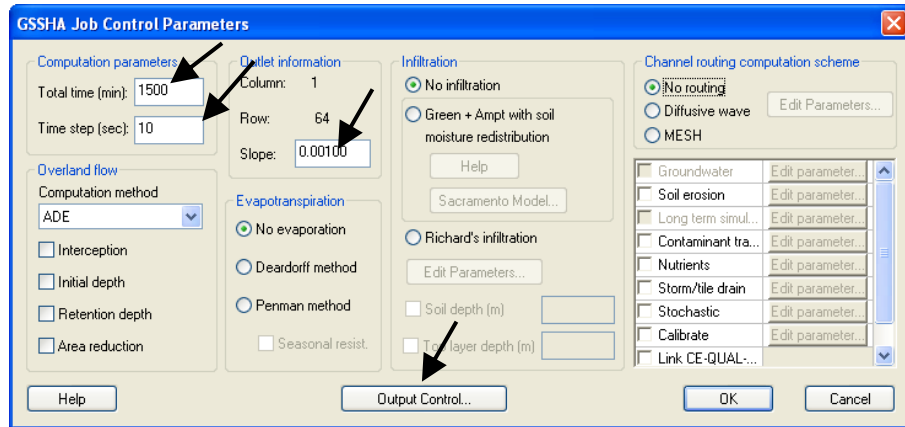


1.5 Job Control Setup

In the previous step (when you created GSSHA grids), the GSSHA Job Control parameters were initialized using some default values. To get the overland flow up and running you will need to define some realistic values.

1. In the 2D Grid module  select *GSSHA / Job Control*
2. The default outlet slope should be 0.001, but if it is not then change it now. This ensures water will run off the grid.

3. 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.



4. 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 here you have an option to write the outflow hydrograph in cfs (however grid results will still be in metric units).

5. Enter 15 minutes as the hydrograph write frequency. This is the second of the two listed write frequencies. The first is for gridded output and it should remain at 30 minutes. Click *OK*.
6. Select *OK*

1.6 Uniform Index Map Setup

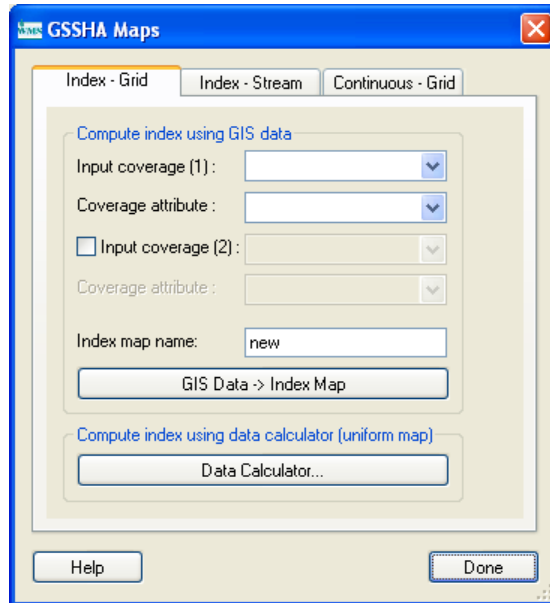
Once the Job Control parameters are set to some realistic values, there are two main areas to set up for overland flow. First, the overland flow roughness coefficients for the grid cells need to be set and secondly, the precipitation data needs to be specified.

There are two parts to setting up the overland flow roughness coefficients; first an index map must be set up that describes the spatial variation of the roughness and next, the roughness values themselves must be set. In this case we will create a spatially uniform set of roughness values so that we can examine and correct any problems with overland flow before moving on to define spatially varying roughness and infiltration.

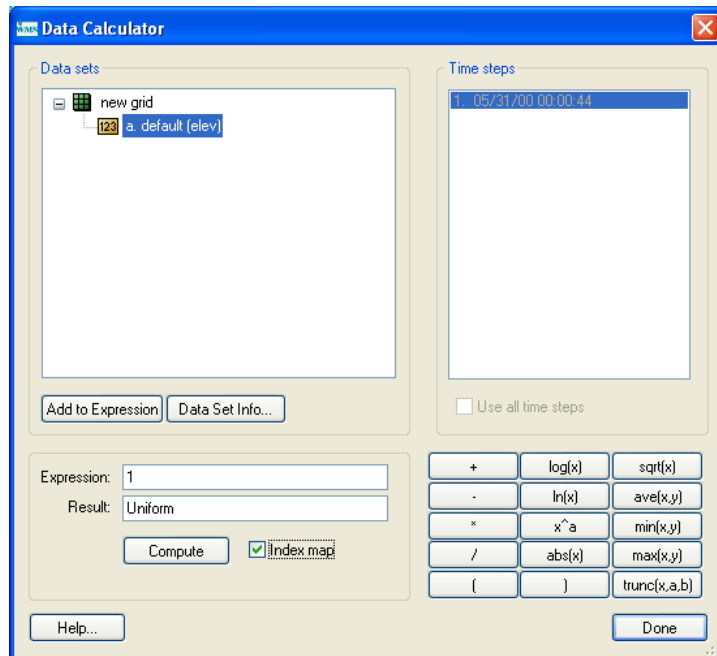
You may have noticed that an Index Maps Folder was created under the new grid object in the project explorer. A grid may have several index maps as was

demonstrated in a previous tutorial and we will now add a uniform index map to this folder.

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



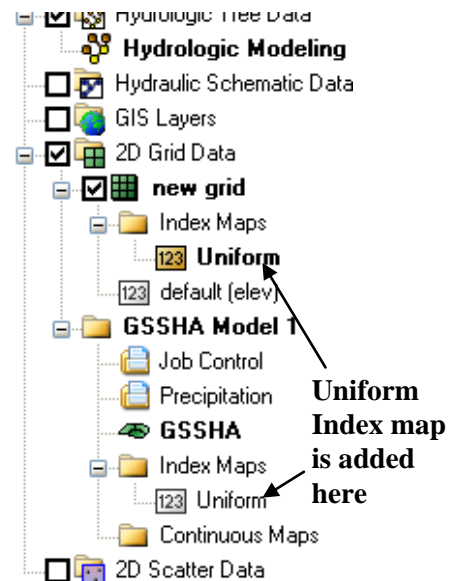
2. Select the *Data Calculator* button. This will bring up the Data Calculator dialog.



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

You should have noticed that the *Uniform* index map is populated in two different places, one in *Index Maps* folder under *new grid* and another under GSSHA model.

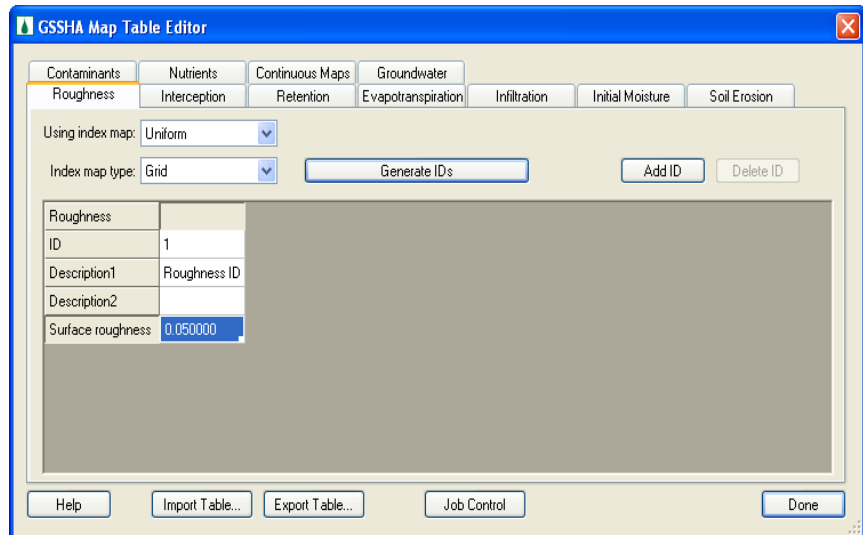
The index maps are in fact stored in *Index Maps* folder under *new grid* and are assigned to the GSSHA model(s). You can *Assign and Remove* the index map(s) from the GSSHA models. If there are multiple GSSHA models, you can use the same index maps in different GSSHA models. Similarly, if you have several index maps, you can use these to single or multiple GSSHA models.



1.7 Roughness Table Setup

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

1. Select **GSSHA / 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*

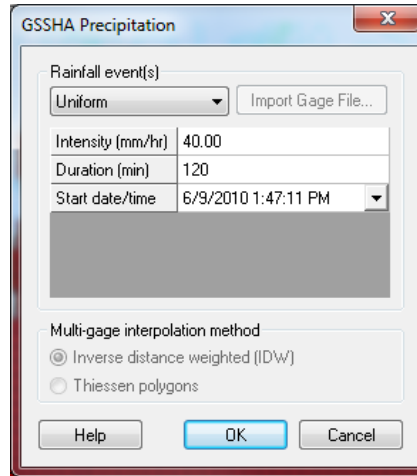
The tabs on the Mapping Table dialog list some of the mapping tables that can be set up. We will set up other processes in later tutorials. Through these two steps we have set up the roughness value (by assigning it the uniform index map) as well as an assigned roughness value to the lone ID in the map. Of course for this simple model with uniform roughness the use of an index map seems like extra work, but when we have several different land uses (roughnesses) to map as well as different infiltration and other spatial properties it is essential to managing the data needed for a GSSHA model.

1.8 Defining Uniform Precipitation

Besides the roughness, the precipitation must be defined in order to evaluate the basic overland flow model. GSSHA has several options for defining precipitation that will be explored later, but for now we will set up a simple uniform precipitation event for a short duration. For now we just need to get some amount of water on the grid in order 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 it.

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

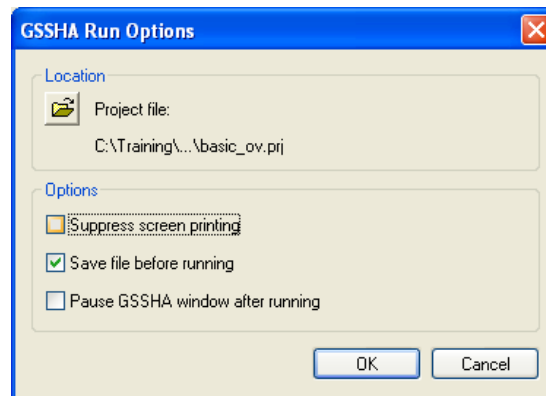
Typically, most of the files in GSSHA project share a similar base file name and only differ in extension. The exceptions to the rule are the index maps, which all have the same extension and different base file names. The file names and extensions may be any name desired; the defaults given in WMS are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

<i>Extension</i>	<i>Description</i>
prj	Project file
ele	Elevation file
msh	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. In Windows explorer go to the folder *C:\Training\Personal\BasicGSSHA\GSSHA*.
4. Open *basic_ov.prj* using note pad (Right click, Openwith and select notepad to open)
5. This is the GSSHA project file which tells GSSHA what to do and which files to refer to. You should have noticed there are several other files with similar names but different extensions (as discussed in the above table).

1.10 Running the Model

1. Select *GSSHA / Run GSSHA*. For now make sure the Suppress screen Printing option is off.

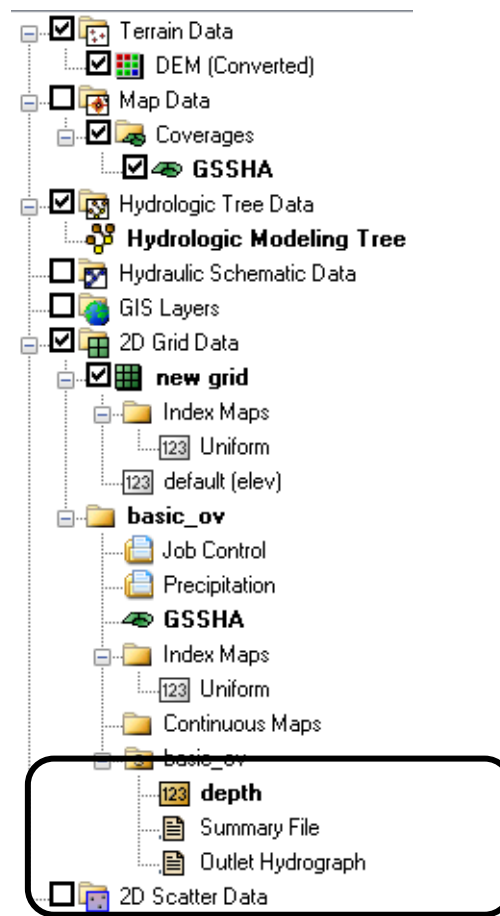


2. Select *OK*.

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, which depending on your computer should take about a minute.

1.11 Visualizing Overland Flow Results




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



You can see in the above figure that the project explorer has renamed your model to *basic_ov* and that one of the folders within the model has a letter 'S', representing 'Solution'. The *Depth* data under the solution folder represents overland flow depth data. Similarly, the *Summary file* is a text document that has detailed information about model inputs, results, mass balances and errors if any. The *Outlet Hydrograph* is the time series data used primarily for plotting, which has the ordinates 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 to plot 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 for the simulation run made.

3. Right click on the hydrograph plot and choose *View Values* which will display the hydrograph ordinates and corresponding times. You can copy and paste these data to Excel spreadsheet.
4. Select the data under *Flow* column, right click and select *Copy*.
5. Open the spreadsheet *C:\Training\InitialGSSHAComparison.xls* and paste the hydrograph ordinates under the column *Initial Setup*.
6. Paste your data on the white areas only. You will also notice some additional tabs in this spreadsheet that can be used to view the results. Select the *Initial_Corrected* tab to view your hydrograph as an Excel Plot.
7. Close the *View Values* and *Hydrograph plot* windows after you are done.

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 mass balance and the volume of discharge and volume remaining on the surface.
4. When you are done you can close the summary file.


From the summary file we learned that most of the water remained on the grid instead of running off. There could be a couple of reasons for this.

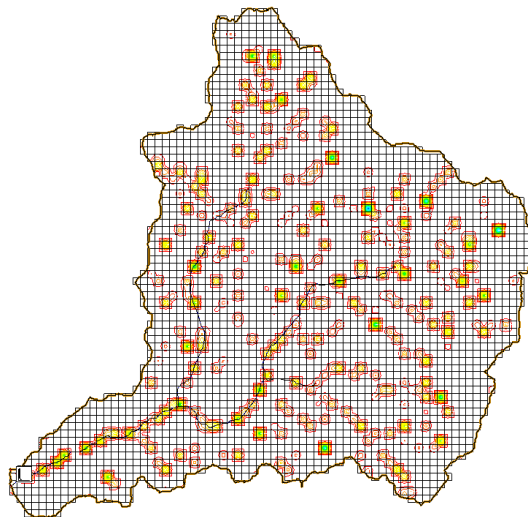
- Maybe the simulation did not run long enough, or
- There are problems with the elevation grid that result in large ponding, or
- Both.

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. Under Contour Method toggle to Color fill.
7. Select the *Legend...* button.
8. Turn on the legend Display.
9. Click *OK*.
10. Click *OK*.
11. Now in the project explorer, right click ***Depth dataset*** and select *Contour Options* or you may select the data and click on *Contour Option macro*  to open *Contour Options* dialog. Make sure that *Contour Method* is set to *Normal Linear*.
12. On the properties window (to the right side), a set of time steps appear.
13. 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 contour varying in color at different time steps.




It is fairly easy to see that the water has ponded in many little puddles on the grid (circular spots in the above figure). These puddles are the result of digital dams, or local depressions that exist as a result of a lack of resolution. Thus our lack of runoff is not due to a shorter simulation time but rather due to the presence of digital dams. Remember that each grid cell is 90m x 90m or 8100 meters squared, so a large amount of water can be detained in each such cell.

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

1.12 Opening Summary File

You should skip this step for now as our project solution is already open, but for future reference this is how you open an existing GSSHA project:

1. In the 2-D grid module , select **GSSHA / Open Project File**
2. Browse to the **C:\Training\BasicGSSHA\GSSHA\basic_ov.prj** file.
3. Select *Open*.

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

1. Select **GSSHA / Read Solution...**
2. Make sure that the simulation points to the correct simulation (**basic_ov.prj** in this case.)

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

3. Select *OK*.
4. Expand the 'new grid' 2D grid folder in the data tree
5. Expand the 'basic_ov (GSSHA)' folder.

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

Correcting Overland Flow Problems

We saw in the previous model that the presence of digital dams creates problems with surface runoff in our GSSHA model. We will learn how such digital dams can be removed from the model.

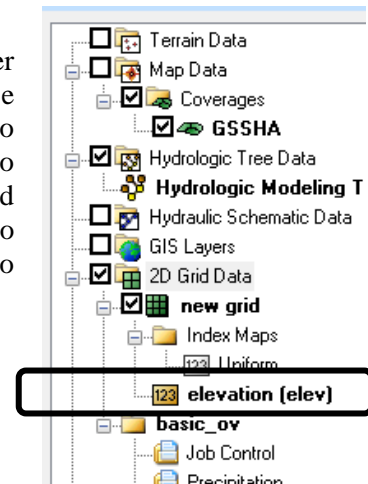
2.1 Working with Digital Dams

In this tutorial you will continue working with the same model. If you have already closed the project or otherwise had problems generating an accurate solution then open *C:\Training\Personal\BasicGSSHA\GSSHA\ basic_ov.prj*. If you do not have such project saved a backup can be opened from *C:\Training\ BasicGSSHA\GSSHA\basic_ov.prj*.

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

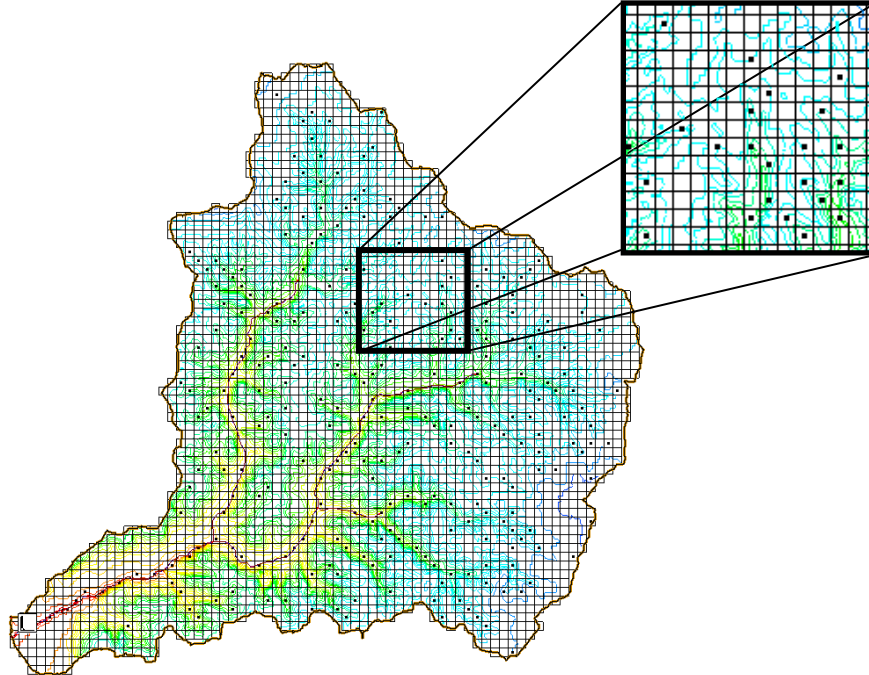
The problem with digital dams is that the water ponds in artificial depressions that result because of a lack of resolution. There are two methods to fix the digital dams. The first method is to manually adjust the cell elevations and the second method is to use the *cleandam* tool to automatically smooth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

1. Click on *elevation (elev)* data in the project explorer under *2D Grid Data* to select it.
2. Select *Display / Display Options*.



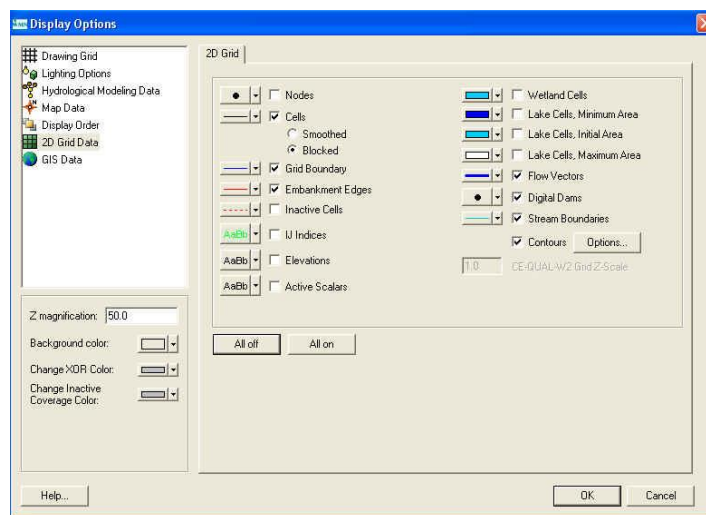
2-2 GSSHA Modeling Using WMS – Volume 2

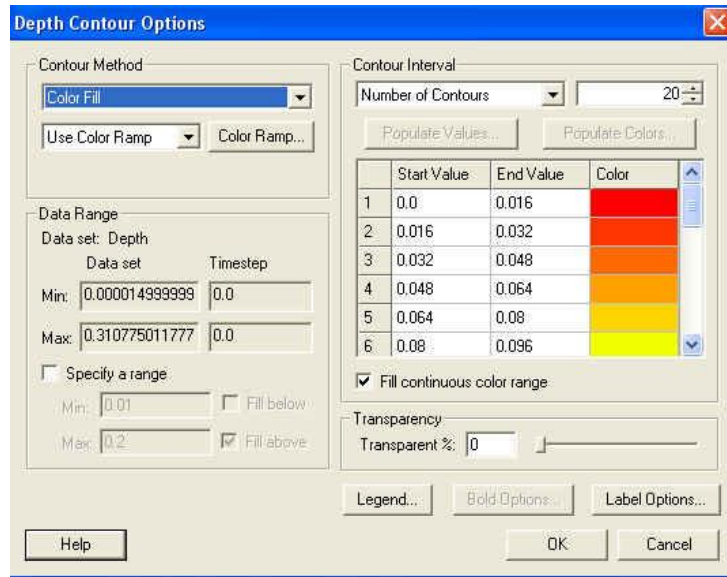
3. In the *2D Grid* tab, turn on *Digital Dams*.
4. Select *OK*. The Grid will look something like the following figure.






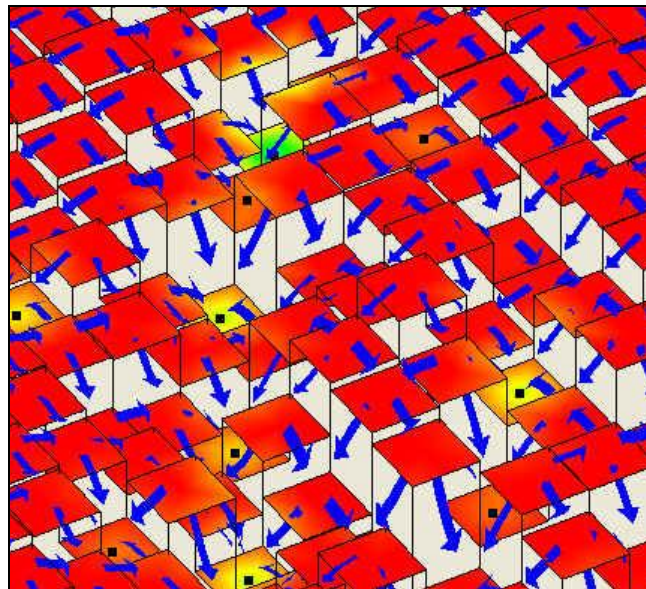
To better visualize why these cells are digital dam cells:

1. Select *Display / Display Options*.
2. Under the *2D Grid* tab, select the blocked cells toggle.
3. Turn on the *2D Grid Contours*.
4. Click on the *2D Grid Contours Options* button.
5. Choose *Color Fill Contours*.
6. Click *OK*





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.

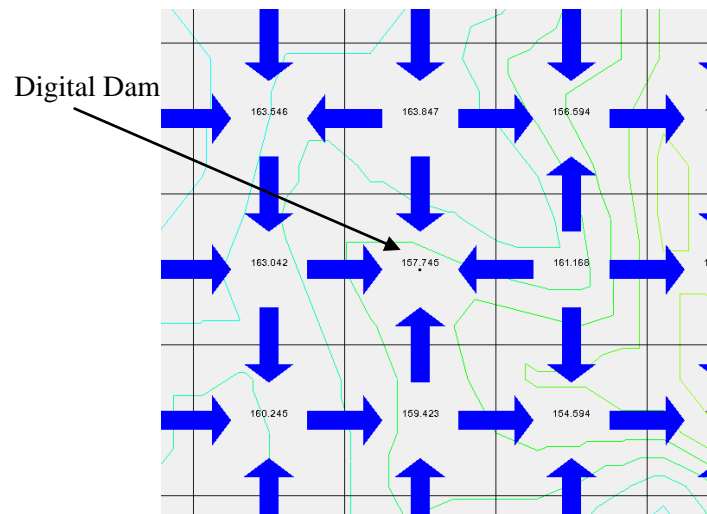
2.2 Manually Adjusting Cell Elevations

It is apparent that you could iteratively 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 by hands and then use some automated tools to speed up the process.

Occasionally only the digital dam cell has to be adjusted but usually one or more neighboring cells are the source of error.


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

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



- Switch to *2d Grid* Module and select the cell with digital dam and increase its elevation in such a way that flow occurs from to this cells to from 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.
- Once the digital dam is fixed by changing the cell elevations, you will need to right click the *elevation* data in the *Project Explorer Window*

and select the *Set as Elevation* option. Doing this will permanently update the elevation data.

6. Turn off the display of *Elevations* and *Flow Vectors*. Then select *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 see any digital dam still present, you might want to 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

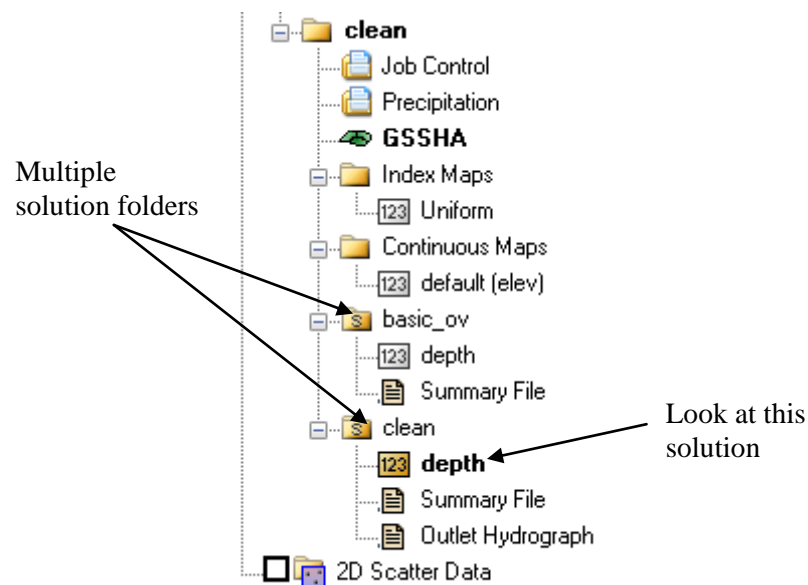
2.5.1 Visualizing Hydrograph

In the 2-D grid module , click on the “*Select Hydrograph*” tool  and double click on the little hydrograph icon  near the watershed outlet. Your model should have produced a lot more runoff this time.

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 spreadsheet *C:\Training\InitialGSSHAComparison.xls* and paste the hydrograph ordinates under the column *Corrected Grid*.
4. Paste your data on the white areas only.
5. Close the Hydrograph plot window after you are done.


2.5.2 Visualizing Summary File

1. Under the ‘clean’ folder, you can see solutions. If you have been working with the same model from previous exercise, you might have two or more solution folders. Check the solution folder that has name *clean*. See the following figure:



2. Open the summary file and compare see the amount of water remaining on the surface. The volume should have reduced by a lot as compared to the previous run.
3. When you are done you can close the summary file. You should have seen the effects of digital dam on the overland flow model.

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*.
5. Under *Contour Method* toggle to *Color fill between contours*.
6. Now in the project explorer, click on *Depth* to select it. On the properties window (to the right side), a set of time steps appear. Click around on a few. It would be helpful if we knew what the colors represented.
7. Look at the depth contour if you still think that water is ponding in some part of the watershed. If it is, then you can adjust the elevations manually 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 we have not simulated it since we first wanted to insure the overland flow process was functioning correctly. During this tutorial you will set up the inputs needed for the Green & Ampt with Soil Moisture Redistribution model.

Let us keep working with the model we 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 file ***C:\Training\Personal\DigitalDam\Clean.prj*** and if this file is not saved you may open ***C:\Training\DigitalDam\Clean.prj.***
3. Save this project with different name so that the original project remains unchanged. Select ***GSSHA / Save project File*** and save as ***C:\Training\Personal\ Infiltration\Infiltration.prj.***

3.1 Index Map Setup

Describing the spatial variability of almost all parameters is done through setting up an index map and a mapping table. The index map is a grid of ID numbers. The ID numbers refer to numbers in the mapping table. Index maps are generic and may apply to any number of tables. Each table lists the name of

the index map associated with the table and all the IDs that the index map (should or could) have, along with parameters for the IDs.

We will use a soil type shape file from the SSURGO database of the area to create a soil type index map. You will need to perform necessary join operations.



3.2 Create Soil Type Coverage

1. Right-click on Coverages 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

4. Right-click on *GIS Layers* in the project explorer.
5. Select *Add Shape file Data*. Browse and open *C:\Training\RawData\JudysBranch\SSURGOsoil\Raw\Spatial\soilmu_a_il119.shp*
6. Right click *soilmu_a_il119.shp* on the project explorer and select *Open Attribute Table*. See the attribute table. You should have noticed that there are no attributes assigned to these soil polygons.
7. Right click on *soilmu_a_il119.shp* and select *Join NRCS Data*.
8. Toggle *Fill blank values* and *Compute Hydraulic* options on.
9. Leave the fields to fill (i.e. B and Silt loam) as is displayed and click *OK*. You may check the attributes table if you want, the attributes are not joined to the shape file.


Now we can convert the shape file to a coverage

10. Make sure the Soil Type coverage is the current active coverage by clicking on it in the Coverage list.
11. Select the *soilmu_a_il119.shp* GIS layer by clicking on it. This will change the active module to the *GIS module* .
12. Click *Select shapes tool*  and drag a rectangle around the watershed to select the soil polygons that overlay the watershed. If you selected few rectangles outside the watershed boundary, that is okay because WMS will clip the soil polygons using watershed boundary.
13. In the GIS Module select, *Mapping / Shapes -> Feature Objects*.

14. Select *Next*. Make sure that fields are mapped properly. Click *Next* again.
15. Select *Finish*.
16. Wait for WMS to convert the shape file into the coverage.
17. You may delete the shape file under GIS Layers (**Right click / Delete**)
18. Toggle off the display of soil type coverage so that the display will be faster.


3.4 Creating an Index Map from Soil Data

Now we can make 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. Change the Coverage Attribute to “*Texture*”.
5. Change the Index map name to “*SoilType*”
6. Click on **Coverages -> Index Map** button, it might take few moments to finish creating 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 we can set up the mapping tables for infiltration.

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 area select the *Green + Ampt with soil moisture redistribution* option.
5. Select OK.

3-4 GSSHA Modeling Using WMS – Volume 2

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 three IDs, 15, 17 and 18. You can see the soil texture name in the Description1 field right below each ID. For our model we have four different soil classifications for which we need to define properties.

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)	14.8	14.1	21.2

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 “*Generate IDs*” button.
12. Enter a values of 0.15, 0.1 and 0.2 for Clay loam, Silt loam and Silty clay loam respectively in edit boxes for the initial moisture. Make sure that the initial moisture is always less than porosity.
13. Click *Done*.
14. Save your project as *C:\Training\Personal\Infiltration\Infiltration.prj*.
15. Run the model.

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 solution that was just read in, double-click on the summary file.
2. Check the mass balance errors, check the volumes of rainfall, discharge, infiltration and others.
3. Open and have a look at the outflow hydrograph
 1. Right click on the hydrograph plot and choose *View Values* to export the hydrograph ordinates to excel spreadsheet.
 2. Select the data under *Flow*, right click and select *Copy*.
 3. Open spreadsheet *C:\Training\InitialGSSHAComparison.xls* and paste the hydrograph ordinates under the column *With Infiltration*. Paste your data on the white areas only.
 4. Visualize the difference in outflow hydrograph in the two models.
 5. Close the Hydrograph plot window in WMS after you are done.

Using the Mapping Table and GIS Data to define variable surface roughness


So far in our model we have set up uniform roughness, uniform precipitation and variable infiltration conditions. We shall now look at setting up spatially varied roughness coefficients. Setting up non-uniform precipitation will be covered later.

We will be using a land use GIS file to create the IDs for the index map to be used with the roughness table. It makes our model more comprehensive as it considers both variable roughness based on land use and variable infiltration based on soil type data. Before completing the model we will consider how land use affects infiltration and adjust the parameters based on a combined index map.

Let us continue working on the previous model. If you have already closed the model used in the previous workshop then open your saved or the backup version and either way save your model as a new project.


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

4.1 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*. Now you will 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 and open the files *Belleville.shp* and *StLouis.shp* located in the folder *C:\Training\RawData\JudysBranch\Landuse*
5. Make sure the Land Use coverage is the current active coverage by clicking on it in the Coverage list.
6. Select both the shapefiles under *GIS Layers folder* (press Ctrl key in the keyboard and click on the shapefiles). Notice that this changes the active module to the GIS module.
7. Click on *Select Shapes Tool*  and drag a rectangle around the watershed boundary which will select the polygons overlaying the watershed.
8. Select *Mapping / Shapes -> Feature Objects*.
9. Click *Next*.
10. Make sure *LUCODE* is mapped to *Land Use*. Click *Next* to accept the attribute mappings.
11. Click *Finish*.
12. You may now delete both GIS shape files.
13. Select the GSSHA coverage under the Coverages folder in the project explorer. You may choose to turn off the Soil Type and land use coverages at this point.

4.2 Creating an Index Map from Land Use Data

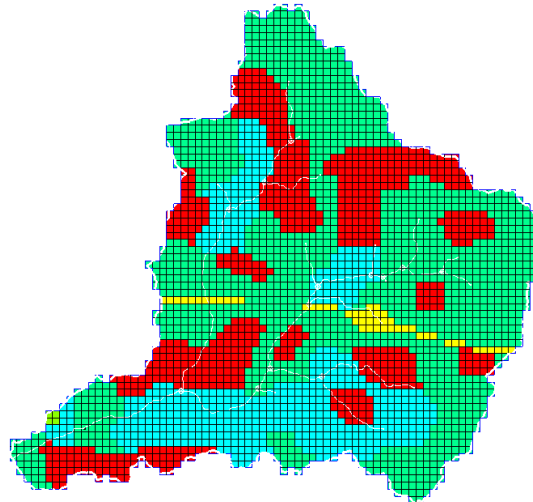
Now we can make an index map out of the land use coverage.

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

4.3 Creating a Mapping Table from Index Map

When you clicked the land use index map, the grid should now have cells colored in several colors representing the different IDs of the polygons that were mapped to the grid cells. Next we will assign the Land Use index map to the roughness table and set up roughness values for each of the IDs in the Land Use index map.



1. Select *GSSHA / 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.)

<i>ID</i>	<i>Description</i>	<i>Roughness</i>
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
41	Land ID #41, Deciduous Forest Land	0.100

6. You have defined the roughness values based on different land uses.
7. Once you are done, select *Done*.
8. Save your project as *C:\Training\Personal\Roughness\roughness.prj*.
9. Run GSSHA.

4.4 Visualization

After it successfully runs, open the outflow hydrograph.

1. Right click on the hydrograph plot and choose *View Values ...*
2. Select the data under *Flow* column, Right Click and select *Copy*.
3. Select Data toggle and click on Export.
4. Open 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 outflow hydrograph in the two models.
6. Close the Hydrograph plot window in WMS after you are done.

4.5 Creating a Combined Index Map

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

We will continue with the model you have been working on so far and the first thing we will start with is creating 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

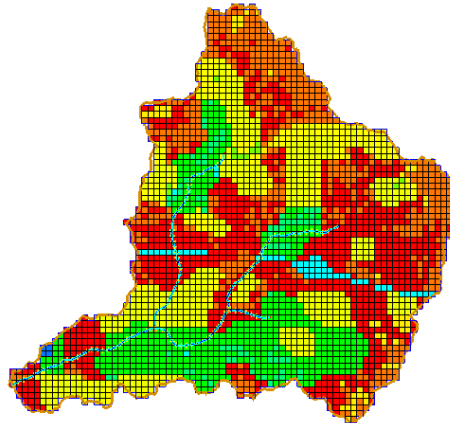
Check on Input coverage (2) and select: Land use

Coverage attribute: Id

Index map name: Combined

Using the Mapping Table and GIS Data to define variable surface roughness 4-5

3. Click on the *Coverages -> Index Map* button. WMS will take some time to compute the new index map which creates a unique ID for each unique combination of land use and soil.
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 we will define the infiltration parameters in the infiltration mapping table. Select *GSSHA | Map Tables...*
7. In the *GSSHA Map Table Editor* dialog, switch to the Infiltration tab. For *Using Index map* field select *Combined* and click on the *Generate IDs* button. Click *Yes* to delete existing IDs.
8. Now you can see several fields added in the table which has all possible combinations of land use and soil type present in the watershed. A total of 18 possibilities exist since there were 6 land use types and 3 soils. There are only 12 combinations in this case, because four other combinations do not exist.

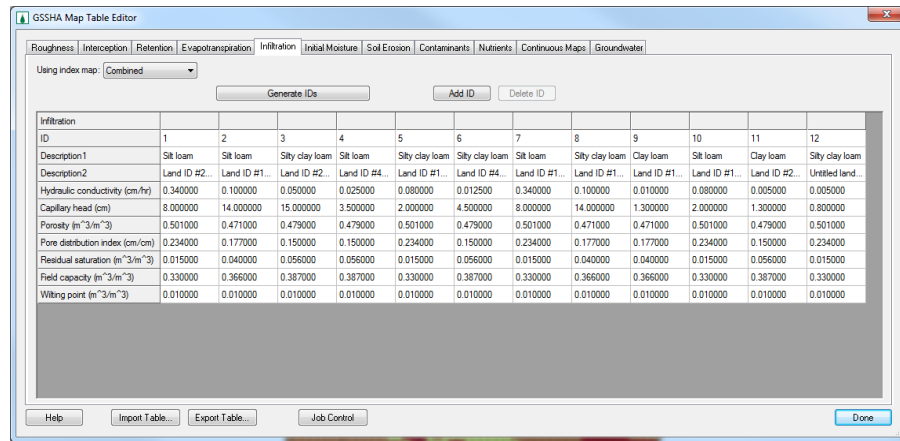
The infiltration parameters listed in Rawls and Brakensiek table represent the bare earth soil infiltration parameters. But, we have combined index map that shows a combination of both land use and soil cover and we can consider for the change in infiltration parameters introduced by overlaying land cover.

You can import the standard GSSHA .cmt file and change the infiltration parameters in the mapping table but for this exercise an updated .cmt file has already been created (*C:\Training\Infiltration\UpdatedInfilParams.cmt*)

9. In the infiltration tab, click on the *Import Table...* button at the lower left part of the editor.
10. Browse and open *C:\Training\Infiltration\UpdatedInfilParams.cmt*.

4-6 GSSHA Modeling Using WMS – Volume 2

Now you can see the values filled in for all the fields.



Use the following table to compare if the values are mapped in properly in the *Infiltration* tab.

Combination	Hydraulic Conductivity	Capillary head	Porosity	Pore distrib index	Residual saturation	Field capacity	Wilting Point
Silt loam Pasture (#21)	0.5	16.68	0.501	0.234	0.015	0.33	0.1
Silt loam Residential (#11)	0.1	16.68	0.501	0.234	0.015	0.33	0.1
Silty clay loam Pasture (#21)	0.15	27.3	0.471	0.177	0.04	0.366	0.1
Silt loam Forest (#41)	0.6	16.68	0.501	0.234	0.015	0.33	0.1
Silty clay loam Residential (#11)	0.05	27.3	0.471	0.177	0.04	0.366	0.1
Silty clay loam Forest (#41)	0.15	27.3	0.471	0.177	0.04	0.366	0.1
Silt loam Industrial (#14)	0.1	16.68	0.501	0.234	0.015	0.33	0.1
Silty clay loam Industrial (#14)	0.05	27.3	0.471	0.177	0.04	0.366	0.1
Clay loam Transportation (#16)	0.05	20.88	0.464	0.242	0.075	0.318	0.1
Silt loam Transportation (#16)	0.1	16.68	0.501	0.234	0.015	0.33	0.1
Clay loam Pasture (#21)	0.1	20.88	0.464	0.242	0.075	0.318	0.1
Silty clay loam Untitled land use	0.1	27.3	0.471	0.177	0.04	0.366	0.1

Using the Mapping Table and GIS Data to define variable surface roughness 4-7


ote: Only the **Bold** faced numbers are changed, the rest of the parameters are the same as the vales from standard GSSHA .cmt table.

1. Once you have filled in the values, click *Done*.
2. Save the GSSHA project as *C:\Training\Personal\Infiltration\UpdatedInfil.prj*
3. Run GSSHA.
4. Open the outflow hydrograph.
5. Right click on the hydrograph plot and choose View Values. Select the hydrograph ordinates, right click and select copy.
6. Open spreadsheet *C:\Training\InitialGSSHAComparison.xls* and paste the hydrograph ordinates under the column *Modified for Land use*. Paste your data on the white areas only.
7. Compare the results you get here with the results from your previous simulation where you did not modify the parameters based on land use.
8. Close the Hydrograph plot window in WMS after you are done.

Stream Flow

5.1 Adding Streams to a GSSHA Simulation

While there are stream arcs present in the WMS display, channel simulation has not been turned on and the necessary channel properties are undefined. Channel properties are defined on the stream arcs in WMS. Let us keep working with the same project you have been working on. Open your project or the backup copy if needed and then save your stream project to get started.

1. In the 2D Grid module  select **GSSHA / Open Project File**.
2. Browse and open file **C:\Training\Personal\Infiltration\UpdatedInfil.prj** and if the project was not saved, you may open this file **C:\Training\Infiltration\UpdatedInfil.prj**
3. Save this project file with different name so that 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:

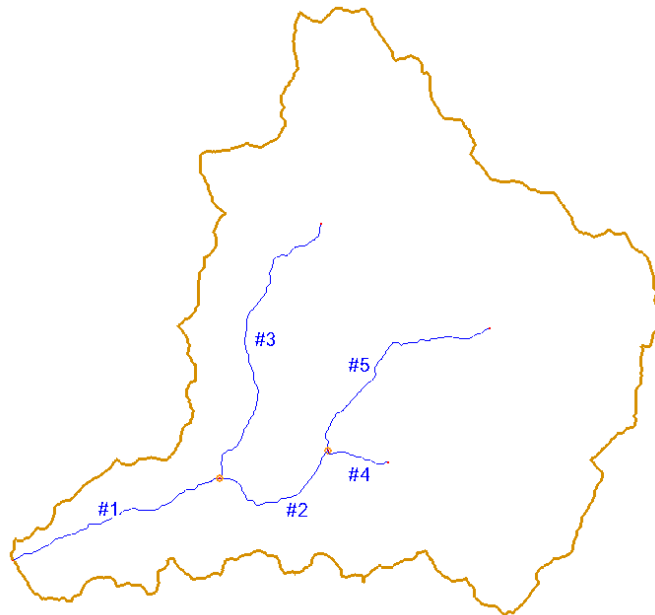
- The stream arcs must be set to a GSSHA stream arc type
- The channel thalwegs defined
- The channel cross section geometry defined.
- The channel routing job control item turned on

5-2 GSSHA Modeling Using WMS – Volume 2

- Additional Output options to visualize streams turned on.

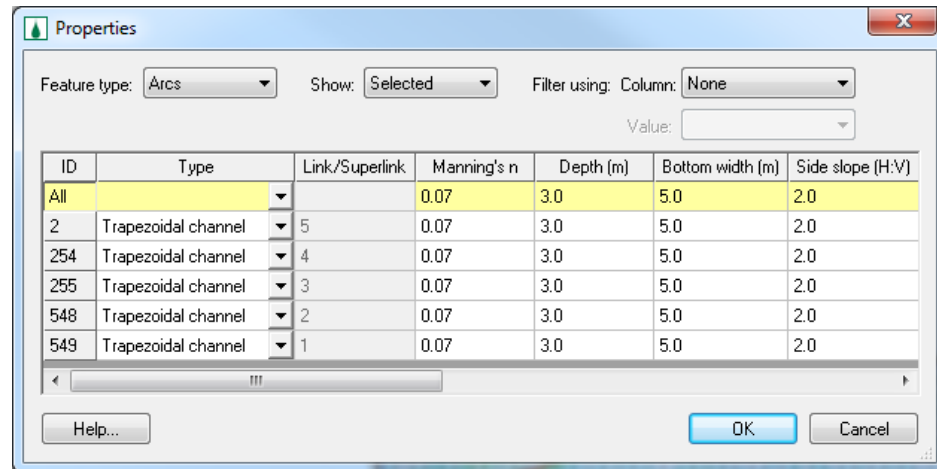
As part of the original delineation a set of stream arcs in the GSSHA coverage were created. These stream arcs, however, are currently set to be generic streams. Further, the stream density (node spacing) is defined according to the threshold limit used when delineating the basin at the beginning, and while you can manually edit the streams now it is good to be thinking about the appropriate stream density when you are first delineating the watershed which we will demonstrate more clearly in the 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 Type as “**Trapezoidal Channel**” for the first row in the table (colored yellow) which allows you to define channel geometry for all selected stream arc(s). See 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 if we could add some variation to our streams in order to represent the streams narrowing upstream. To do this we will increase the dimensions of the most downstream arc.

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

Arc Number	Channel Depth	Bottom Width	Manning's Rough	Side Slope
1	3.5	7	0.075	2



11. Click *OK*.

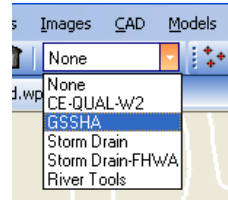
The stream arcs now have defined geometries and are ready to be smoothed.

5.2 Smoothing the Channel Thalweg Elevations


Now that the geometry for each stream arc has been defined, it is important to ensure that each arc is flowing downhill. Due to coarse resolutions in the DEM data and grid construction, stream arcs will often have segments that flow uphill, which causes digital dams and other problems. To avoid problems in the simulations, we will look at the stream profiles and modify the node elevations until each arc is flowing downhill. First, however, we need to redistribute the vertices on the arcs to a more manageable spacing because by default vertices were created approximately 10 meters apart based on the resolution of the original DEM.

5-4 GSSHA Modeling Using WMS – Volume 2

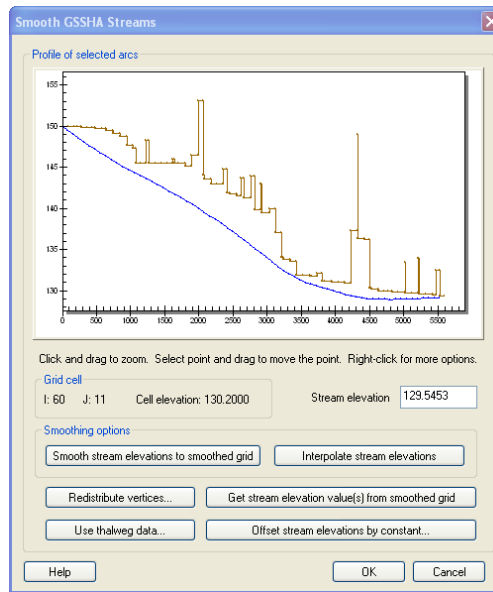
1. Select the **Map Module** .
2. If the Model combo box on the X,Y,Z edit bar is not set to GSSHA, change it to GSSHA now.
3. Click on the **Select Feature Arc Branch** tool .
4. Click on the stream segment labeled “#1” from the previous figure.
5. Select **Feature Objects / Redistribute**.
6. Type 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. We will now proceed to smooth the thalweg elevations selecting continuous branches of the channel until all 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” along the continuous branch until you have selected the most upstream segment. Make sure that there is no branching in your selection and the selection is continuous.
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.



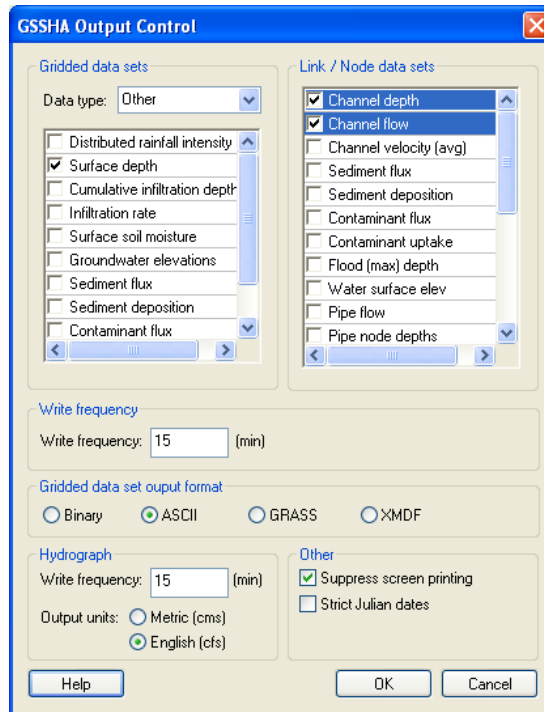
10. Select the “*Interpolate stream elevations*” buttons as many times as needed (probably 10-15 times) to generate a smooth stream segment with no uphill flow, then click *OK*.
11. If uphill flow cannot be eliminated in this manner, you can edit individual points by dragging the point to a new position 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 an adverse slope.
12. You may draw a box in the area you want to zoom. Similarly, other editing options are available on right click menu.

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

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

5.3 Changing Output Control Options

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



2. Under Channel Routing toggle on *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\ Stream\stream.prj*

5.4 Running the model and viewing the outflow hydrograph

1. Select *GSSHA / Run GSSHA*. To run the model. After the completion of simulation, open the outflow hydrograph.
2. Right click on the hydrograph plot and select View Values. Then select and copy the hydrograph ordinates.
3. Open spreadsheet *C:\Training\InitialGSSHAComparison.xls* and paste the hydrograph ordinates under the column *With Stream*. Paste your data on the white areas only.
4. Visualize the difference in outflow hydrograph in the two models.
5. Close the Hydrograph plot window in WMS after you are done.

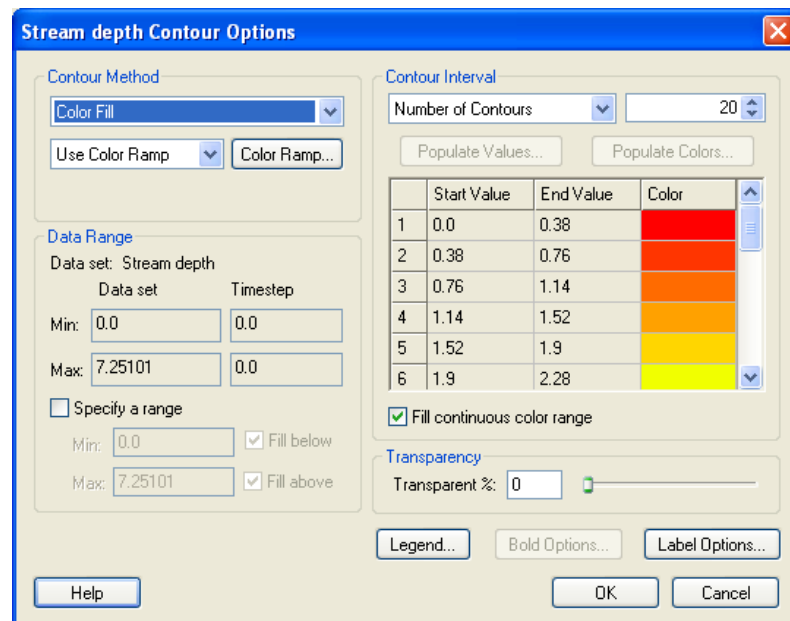
5.5 Visualizing Stream Flow Data

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

These data sets can be visualized in WMS in different interactive forms. We can create depth contours, flood histograms and/or a movie of how these values vary with time. In this tutorial we will see some of these visualization processes.

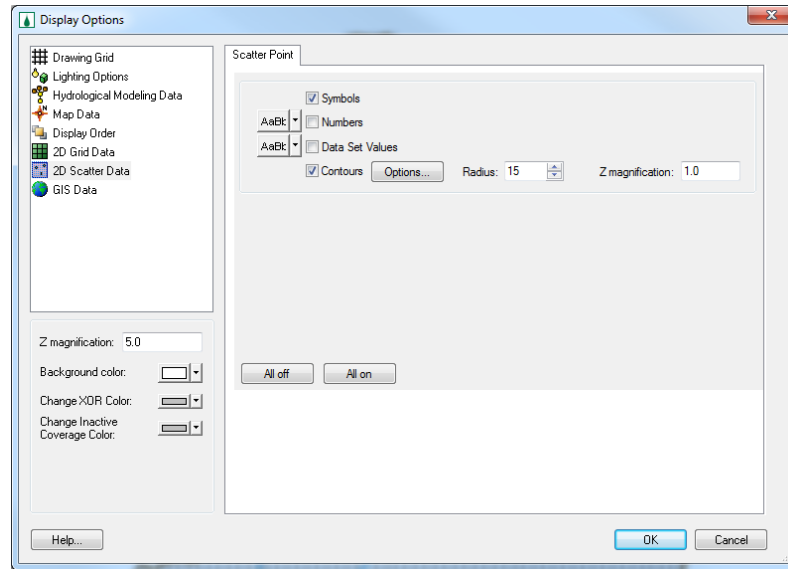
5.6 Changing Contour Properties


1. In the project explorer (near the bottom) click on *Stream Depth* to select it.
2. Select **Display / Display Options**. In the 2D Scatter Data, 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*.

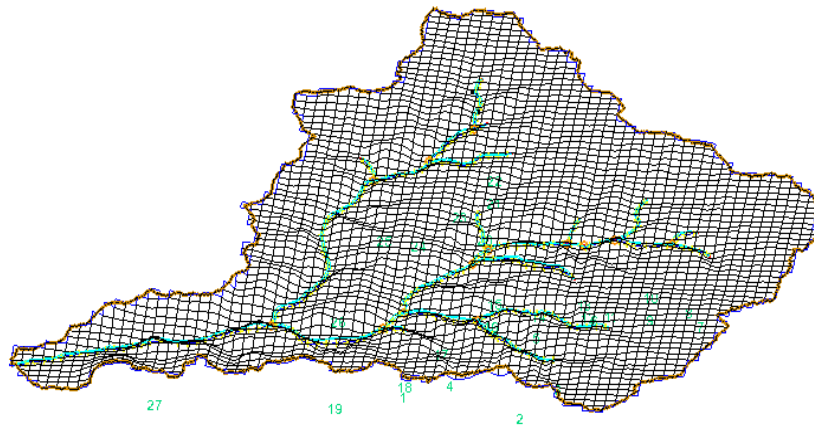


4. Change *Radius* field to 15 and *Z-magnification* for the contours to 1.
5. On the left side of the dialog, set the *Z-magnification* to 5.

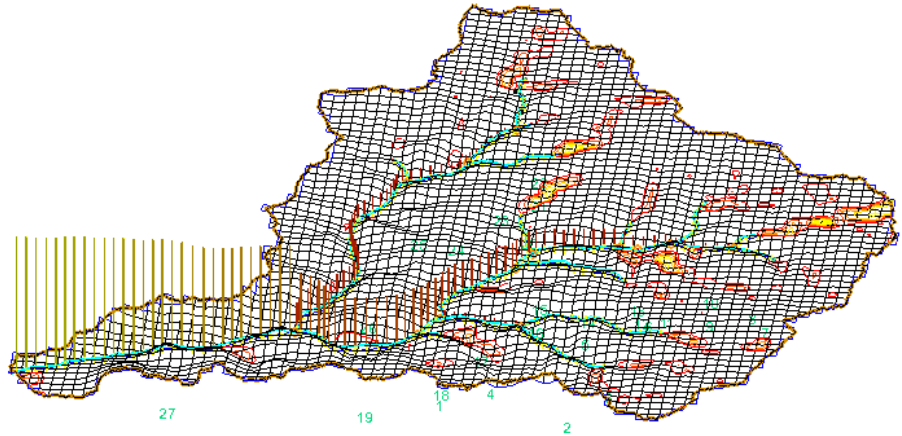
6. Select *2D Grid Data* option in the list. Make sure that *Cells* option is selected to be *Smoothed*.
7. Click *OK*.



8. Click and select “*Rotate*” tool . Then rotate the watershed by using left mouse click so that it looks as shown in the following figure. (Use middle button on the mouse to pan the display).



9. Now click on *Stream Flow* in the project explorer to select and toggle through different time steps (shown on the properties window at the right side of WMS main window). You can see the bar diagrams representing flood wave generating and getting accumulated to create peak flow at the outlet.




You can see the overland flow depth contour changing along with the flood wave bar diagrams. This gives more clear idea of the actual natural process of overland flow procedure.

Developing GSSHA Model Using Hydrologic Modeling Wizard in WMS

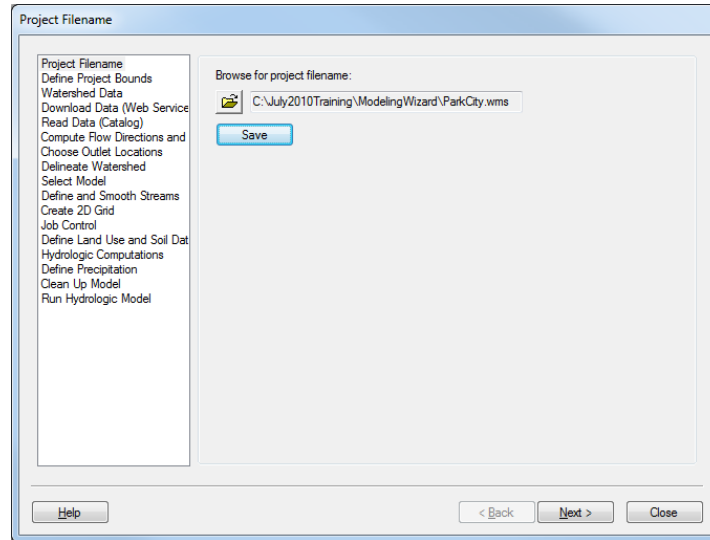
Because you may not use the modeling tools on a day to day basis it can be a bit difficult to remember the basic and important processes you must go through to develop a hydrologic model with WMS. The Primer on the GSSHA wiki is an excellent reference to help you remember the process, but the WMS interface also includes a wizard to help guide you through the basic set up process. It also streamlines and improves some of the processes we went through in the previous tutorial like smoothing of stream channels prior to grid development and creation of index maps from commonly available GIS shapefile data.


In this tutorial, we will create a GSSHA model using the Park City data and the Hydrologic Modeling wizard. It will use the same basic simulation options we have just set up one at a time in the previous tutorials and will further serve as a review of what you have learned so far. The raw data for Park City Branch can be found at *C:\Training\RawData\ParkCity*.

6.1 Initiating the Wizard

1. Open a new instance of WMS. If you have a previous version up close it and restart WMS.
2. Click on the “*Hydrologic Modeling Wizard*” button  at the bottom left bar of the interface to initiate the modeling wizard dialog. This wizard will guide you through the basic set up of a GSSHA model.

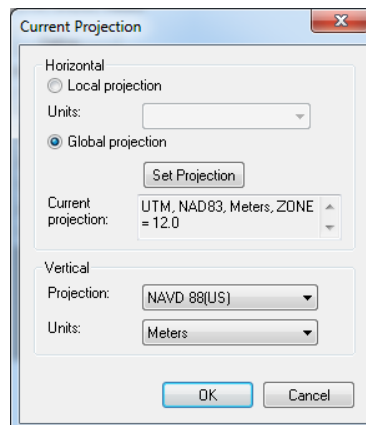
You can follow each step from the beginning or jump in and out at any step as needed. All of the WMS menus\tools work while the wizard is open, though you may have to move it aside to see the WMS windows.



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


6.2 Defining the Project Boundary and Coordinates

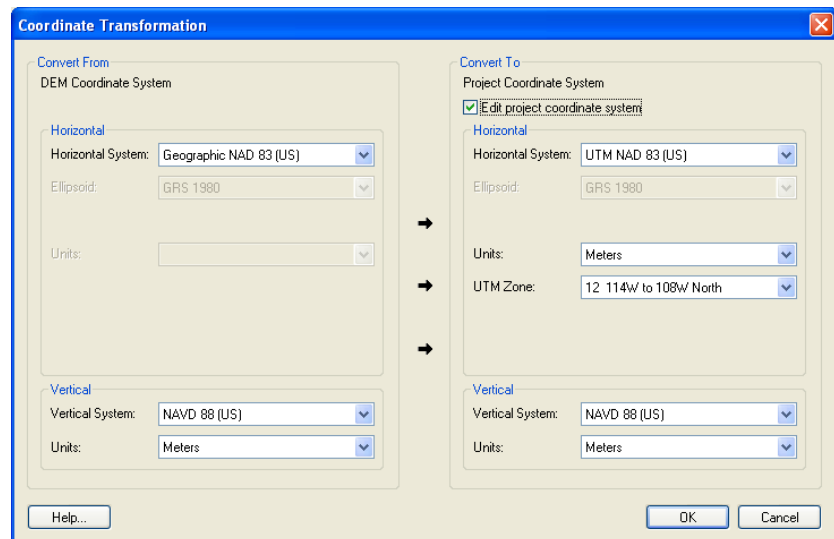
1. On the modeling wizard, click *Next* to go to the *Define Project Bounds* dialog. Here you need to define your project coordinates and project bounds.
2. Click on *Define* button under *Project coordinate system*.



3. Select the coordinate system to be UTM NAD 83 (US), units for both horizontal and vertical are meters, UTM zone is 12 and the vertical system as NAVD 88 (US) and click *OK*.

6.3 Importing the DEM Data

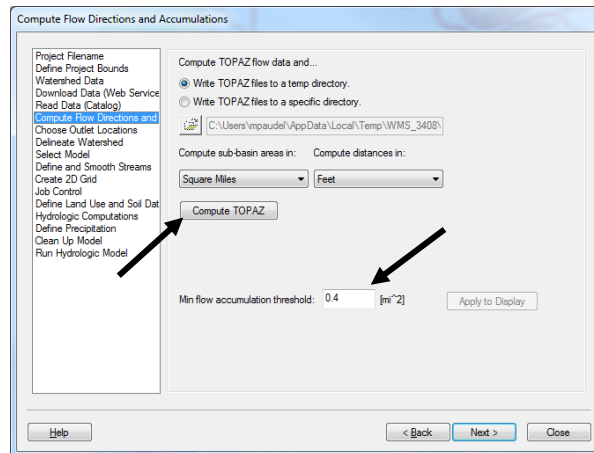
1. Click Next on the wizard to proceed to import watershed data. The data can be downloaded from web or you may import the data from the catalog files or import the file manually from any other location in the computer. Here we will import the data manually from our computer.
2. Select *Open file(s)* in the wizard dialog and click on the browse button . Open the file *C:\Training\RawData\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 4 and 5 if no coordinate transformation is necessary.
4. In the coordinate transformation dialog (**Error! Reference source not found.**), make sure that “Convert From” coordinate is *Geographic NAD 83(US)* and vertical unit is meters. Then check on the *Edit Project coordinate system* under “Convert To” option.




5. Select the *Convert To* Horizontal system as *UTM NAD 83 (US)*, make sure that the UTM zone is 12 and vertical systems are NADV 88 (US) in both areas. The horizontal units on both areas should be meters. Click *OK* and WMS will convert the coordinates of the DEM from Geographic NAD 83 to UTM NAD 83. Now you can see the DEM contours behind the modeling wizard in WMS main window.

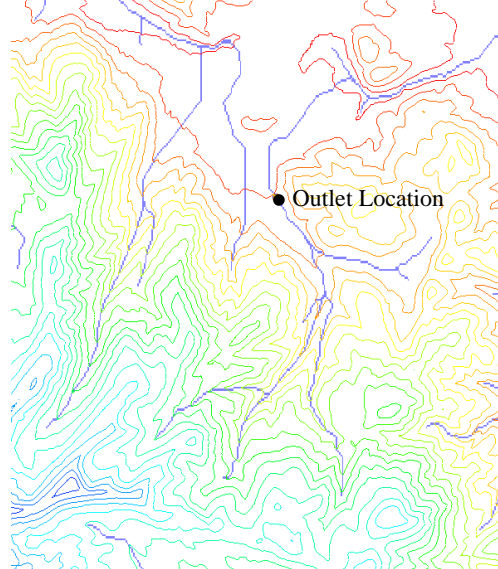
6.4 Computing Flow Direction and Flow Accumulation

1. In the Modeling Wizard, click *Next* and *Next* again. You should be in the *Compute Flow Direction and Accumulation* part of the wizard.
2. Set the *Min flow accumulation threshold* to 0.2 sq mile.



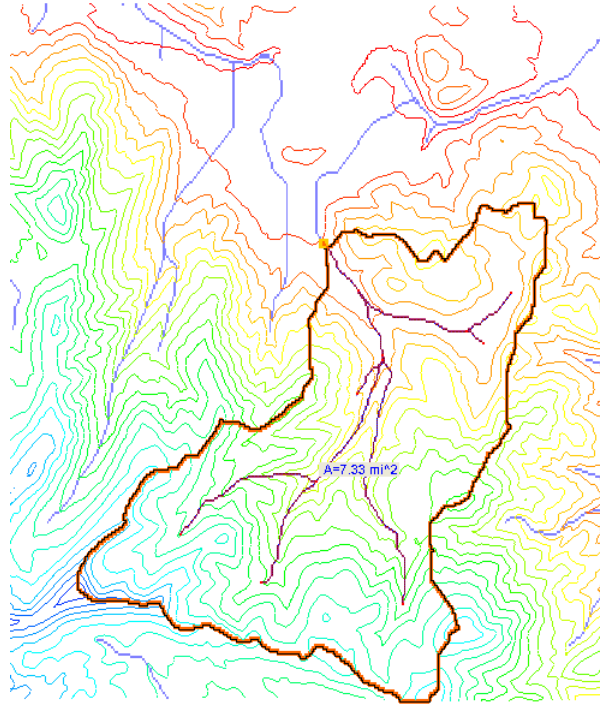
3. Click on the “*Compute Topaz*” button which will compute the flow direction and accumulations and infer the streams based on DEM data. Click on “*Close*” after computations are complete. If you move the modeling wizard dialog to the sides, you can see the blue lines added to the display representing the streams.
4. Click *Next* on the modeling wizard and Select *Create Outlet Point*


Create outlet point:  , and when you take 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. See the following figure for the location of outlet (you can use the middle scroll button of the mouse to zoom in or out).



6.5 Delineating the Watershed

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



3. Save your WMS project by selecting *File / Save*. 

6.6 Initiating GSSHA

1. 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. You should have noticed 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.

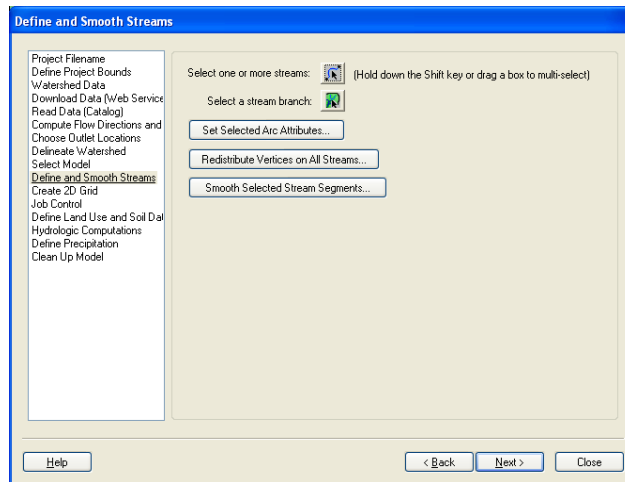
6.7 Defining Stream Channels


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

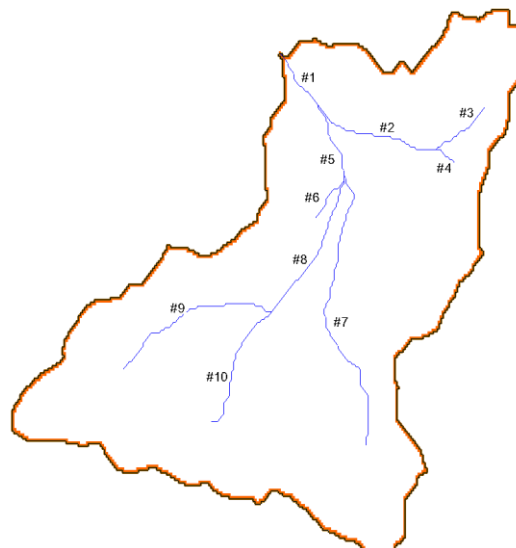
DEM data, the stream bed is often not smooth which might create problems during simulation. In this step, we will define the channel cross section and roughness properties and we will also check and smooth the channel bottom.

6.7.1 Defining stream cross sections

1. In the wizard dialog, click *Next* which will bring the *Define and smooth Streams* dialog, see following figure.

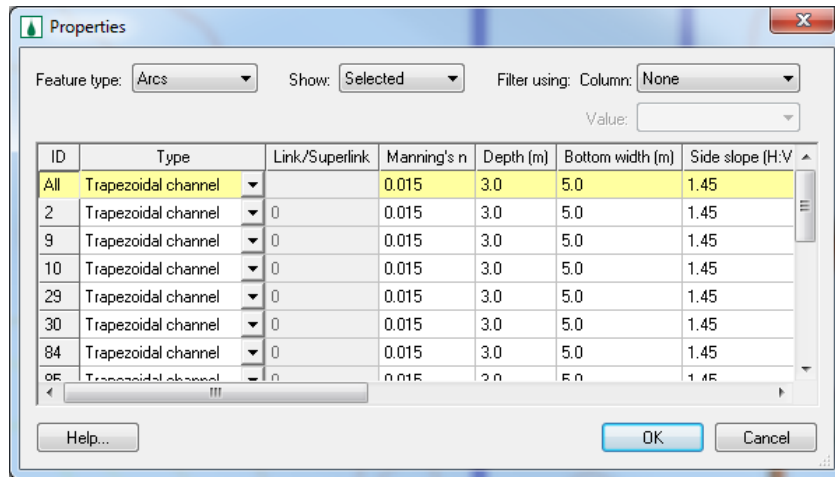


2. In the wizard, click on the *Select feature line branch tool*  and click on stream segment #1 (most downstream stream segment that is connected to the watershed outlet, you might need to zoom in that area) as shown in the following figure.
3. Select the *Set Selected ArcAttributes* button.



- This will open up the *Channel Properties* dialog. Select *Trapezoidal channel* option. Enter the following values.

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




- Click *OK*. Here we are assuming that all the stream segments have the same cross section, but as demonstrated previously each arc could represent different stream properties.

6.7.2 Smoothing the stream thalwegs

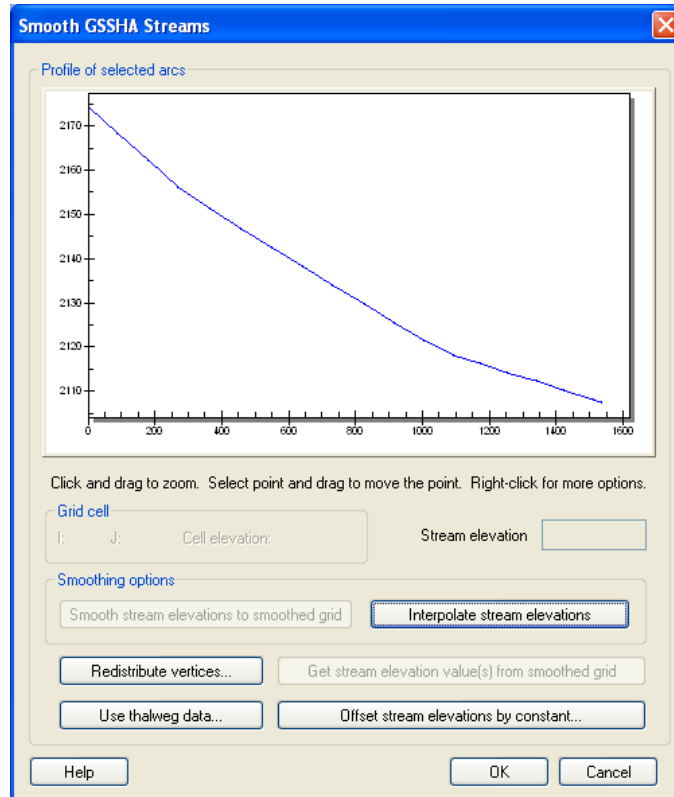
- With the entire stream network still selected, click on *Redistribute Vertices on All Streams* button in the wizard.
- Type 90 in the box next to Spacing and check on the option to *Use Cubic Spline*.
- Click *OK*.

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 selecting continuous branches of the channel until all have been smoothed.

- Click on *Select Arc* tool  and select the stream arcs. 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 is continuous.
- 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 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.



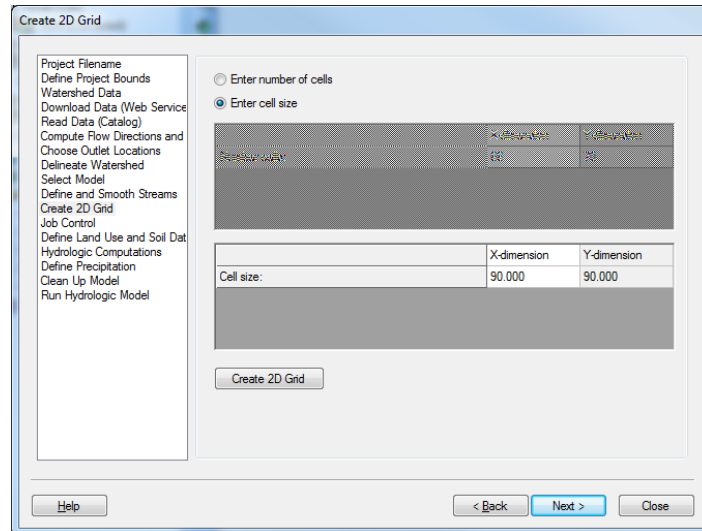
6. Select the *Interpolate Stream Elevations* button as many times as needed to generate a smooth stream segment with no uphill flow, then click *OK*.
7. If uphill flow cannot be eliminated in this manner, you can edit individual points by selecting the node and then dragging the point to a new position 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 all stream arcs in the basin are smooth. Your streams are now ready for use in the GSSHA model.

6.8 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 grid on the watershed.
3. Click *OK* to interpolate the grid elevations from the DEM and Click *Yes* to delete the existing background DEM.

6.9 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*.

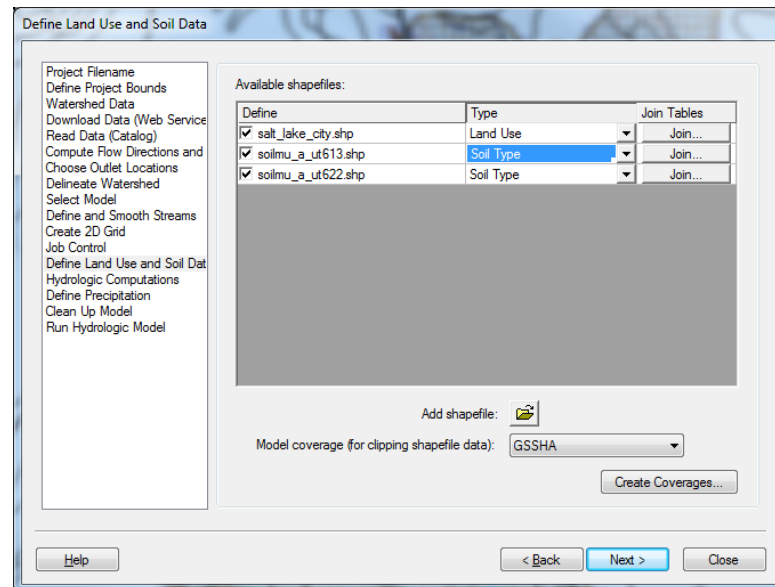
6.10 Define Land use and Soil Data

The model is parameterized using GIS shapefile data for land use and soil information for the watershed. Here we will import the shapefiles and map them to WMS coverages.

1. Click on the *Add Shapefile* button  at the lower end of the dialog. Browse and open the file *C:\Training\RawData\ParkCity\Luse*

salt_lake_city.shp. You can see the land use shape file added to the “GIS Layers” in WMS Project Explorer.

2. Click on the *Add Shapefile* button again  and open the file *C:\Training\RawData\ParkCity\SSURGO_Soil\soil_ut613\spatial\soilmu_a_ut613.shp*. You should have noticed that this soil shapefile does not overlay our watershed completely. So, we will need to open another soil data.
3. Click on the *Add Shapefile* button again  and open the file *C:\Training\RawData\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 which is done outside the wizard for now. 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 properly selected for the corresponding shapefiles, for example the *Type* field should be set to *Land Use* for *salt_lake_city.shp*.

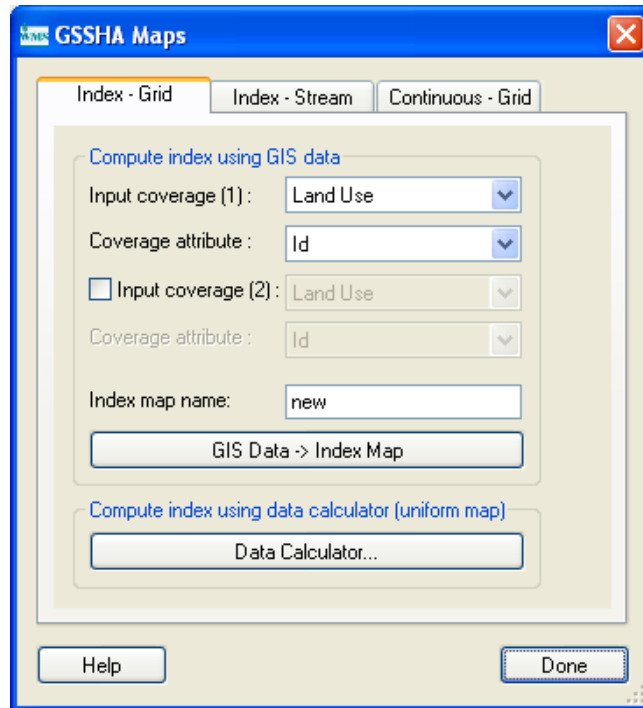


6. In the wizard, click on *Create coverages* to create land use and soil coverages and map the corresponding shape file polygons to their respective coverages. You will need to navigate through the attribute mapping dialogs by clicking *Next* and *Finish* as prompted, but because the shapefiles use standard attribute names the mapping should be defaulted correctly.

7. Once both the coverages are created, click *Next* on the modeling wizard

6.11 Creating Index maps and defining Model Parameters

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

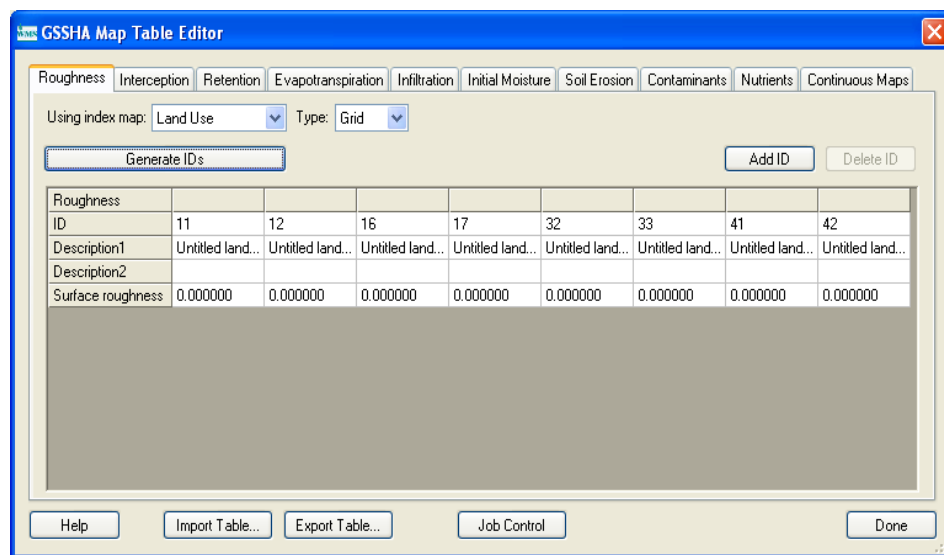


2. Select *Land Use* as the Input Coverage and *ID* as the Coverage Attribute. Enter the Index Map Name to be *Land Use*. Then click on the **Coverages -> Index Map** button.
3. Select *Soil type* as the Input Coverage and *Texture* as the Coverage Attribute. Enter the Index Map Name to be *Soil Type*. 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 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 the Index Map Name to be *Combined*. Then click on the **Coverages -> Index Map** button.
5. Click *Done*. Now, you can see three index maps on the data tree.

- The *GSSHA Map Table Editor* dialog will pop up itself. You will now need to define the index mapping parameters.

6.12 Defining Model Parameters

- In the *GSSHA Map Table Editor* dialog, switch to the *Roughness* tab (it is generally selected by default).
- 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.



- 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.
- In the *Using Index map* field, select *Combined* and click the *Generate IDs* button. This will generate a unique ID for all possible combinations of land use and soil type data within the watershed.
- Switch to the *Initial Moisture* tab, select *Soil Type* in the *Using Index Map* field and select the *Generate IDs* button.
- Now select the *Import table* button at the bottom left of the *GSSHA Map Table Editor* Dialog.
- Browse and open the file *C:\Training\gssha.cmt*.

NOTE: This table has typical values for the watershed parameters based on standard soil type and land use classification. Importing this table will populate these values in the respective fields 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 an initial estimate of the corresponding parameters.

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 values listed there for each parameter are based on soil texture only. But the index map we are using is the combination of both land use and soil type and so we should make suitable adjustment to the parameters. It is because we can see the properties of “sand” in agricultural area and in residential area will be different. Similarly, “Loam” in Forest will have different infiltration properties as compared to “Loam” in Industrial area.
10. The land use IDs are the same as the IDs used in USDA soil classification. You can go to http://emrl.byu.edu/gsa/data_tips/tip_landuse_table.html to see the land use names and related IDs).
11. For the columns with land ID 11, 12, 16 and 17 reduce the values of Hydraulic conductivity to about half of what is in the table. These land uses are relatively impervious and we are assuming that the infiltration rate will decrease because of these land uses. In most of the situations, this is the place where you need to use your judgment.
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*”.

6.13 Editing Parameters

1. Click on the “*Edit Parameters*” button on the modeling wizard. This will open the “*Job Control*” dialog. Make sure that all the fields are defined and turned on properly.
2. Select the “*Diffusive Wave*” option for the “*Channel routing computation scheme*” at the top right side of job control dialog.

3. Select “*Output Control*” button. Toggle on the *Cumulative infiltration depth, Infiltration rate, Channel Depth* and *Channel flow* options.
4. Enter the write frequency to be 15 minutes for both grid and hydrograph and change the units to be English (Metric is selected by default).
5. Click *OK* to close the *Output Control* dialog.
6. Click *OK* to close the *Job control* dialog.
7. Click Next on the modeling wizard to define the precipitation data.

6.14 Defining a Meteorological Model

1. In the wizard, click on the “*Define Precipitation...*” button. In the “*GSSHA Precipitation*” dialog, select Rainfall event(s) to be “*Hyetograph*”. Then click on the “*Define Distribution*” button.
2. In the XY Series Editor window that opens, select “*TypeI-24 hour*” under *Selected Curve* option. This is the temporal distribution of SCS typeI-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.

6.15 Cleaning and Checking the Model

1. You are done defining most of the parameters and components required for the model.
2. Click on the “*Clean Up Model*” button. This does several things to clean up the display of the model as well as finish preparations of the model. It will run the Clean Dam routine to improve the overland flow, as well as bring up the model checker and it will show known input errors if there are any. You can click on the error which will take you to the corresponding dialog to make suitable corrections.
3. Click “*Done*” to close the model checker. Click “*Close*” to close the “*Cleandam model Wrapper*”.

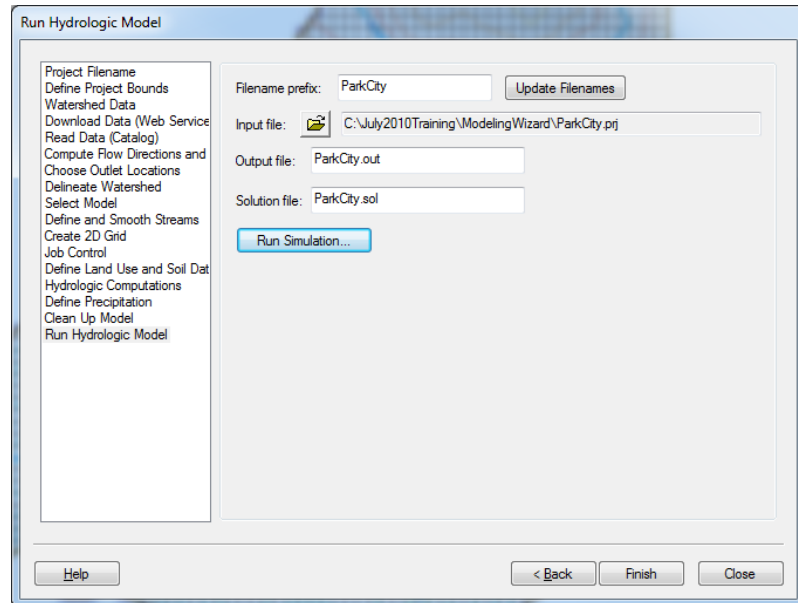
6.16 Save the Model

1. Click on “*Save*” button to save the GSSHA project file. It is different from WMS project file so even though you have been saving the WMS project, this step is necessary to save the GSSHA project file.

2. Browse and save your project as **C:\Training\Personal\Modalign Wizard\Parkcity.prj**.

6.17 Run the Model

1. Click *Next* in the modeling wizard dialog which will bring *Run Hydrologic Model* dialog.
2. Make sure the path you defined to save the project is correct (**C:\Training\Personal\Modalign Wizard\ Parkcity.prj**)
3. Then Click *Run Simulation* button to start GSSHA run.



4. 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.
5. Once done running, look at the outlet hydrograph and depth contours.

In this tutorial you learned how WMS automates GSSHA model creation process. This workshop repeats all the steps that you did in previous several workshops but the wizard guided you through the process. This is a standard way of creating initial GSSHA models. Once, you get your model at this point, it will then be ready for performing further hydrologic analysis.

Post-Processing and Visualization of GSSHA Model Results

In this section you will explore different post-processing and result visualization options. You have already been using some of these tools in the previous tutorials. We will use a model of Judy's Branch that has already been executed and explore some of the post processing tools in WMS.

7.1 Open an Existing Project

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

This will open a Judy's branch project which uses distributed roughness, infiltration and also distributed precipitation. You will notice several precipitation gages over the watershed.

Because of the land use and soil polygons as well as precipitation gages, the display looks cluttered and the screen refreshes slowly. Check off the display of all items under *Coverages* in the data tree. Then turn the display of GSSHA back on. See the following figure:



7.2 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 have solution sets for different roughness parameters or different time steps and then you can compare results across the data sets.




Since this project is already run, you do not have to run it in order to view the results. But, the results are not read in by default, you will have 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) and you have to answer OK to start reading the solution.
3. and select OK. WMS might take a while to finish reading the solution.
4. You can see solution datasets added to the project explorer under *2D Grid Data* and *2D Scatter Data*.


7.3 Visualization

7.3.1 Visualizing the outlet Hydrograph

As soon as the result is read, you can see a little hydrograph icon at the outlet of your watershed. This icon is used to see the outflow hydrograph.


1. In the 2-D grid module , click on the “**Select Hydrograph**” tool  and double click on the little hydrograph icon  near the outlet.
2. This will open the hydrograph in a plot window for the simulation run made.
3. There are several options to control the hydrograph window, including exporting by right clicking on the hydrograph plot.
4. Probably the simplest way to move the data to a spreadsheet as you have seen in previous tutorials is to choose the *View Values* option and then *copy\paste* those values to a spreadsheet.
5. *Close* the Hydrograph plot window when you are done.

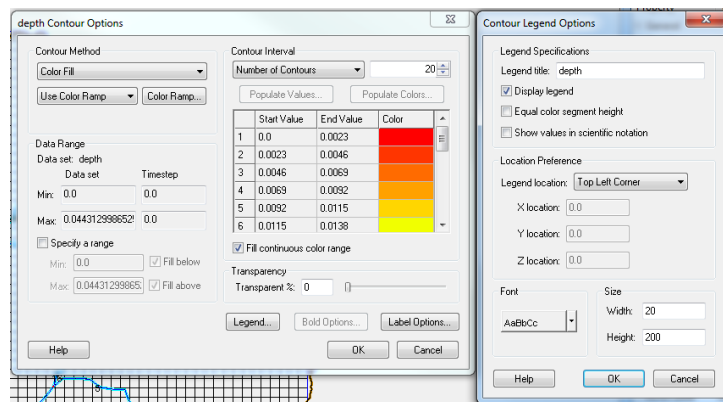
7.3.2 Reading the Summary File

Notice that there is a folder named *Solution* under *Visualization* project folder. The solution folder has a symbol ‘s’ for solution  *Solution*. Now let’s look at the summary file.

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 and the volume remaining on the surface to know that GSSHA is simulating the processes correctly.
4. When you are done you can *close* the summary file

7.3.3 Visualizing Depth Contours




1. In the data tree, right-click on *Depth* under the *solution* folder.
2. Select *Contour Options* (it can be found under the Display menu or 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*.
6. Click *OK*
7. In the properties window (to the right side), a set of time steps appear. If the time steps are not showing, it might be either *Depth* data set is not selected on the project explorer or you have selected a single grid cell. If this happens, click somewhere outside the watershed boundary and select *Depth* data set.
8. Click on first time step and use the down arrow key (on the keyboard) to cycle through the time steps. You can see the depth contour varying 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. You can see the legend 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 .

7.3.4 Visualizing Stream Flow Results

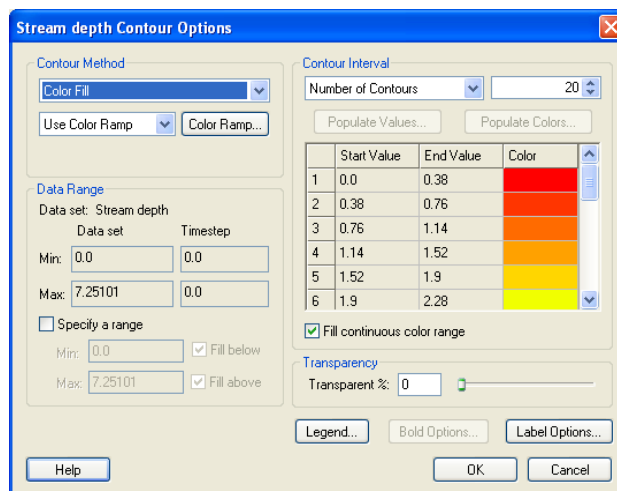
While the streams are connected to the overland flow plane they are a separate coupled model, each having its own input files and data formats. The channel output data is in a link\node data set format. These files hold a data value for every link and node at the same time step as that for the gridded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

These data sets can be visualized in WMS in different interactive forms. We can create depth contours, flood histograms and/or animations of how these values vary with time. In this tutorial we will see some of these visualization processes.

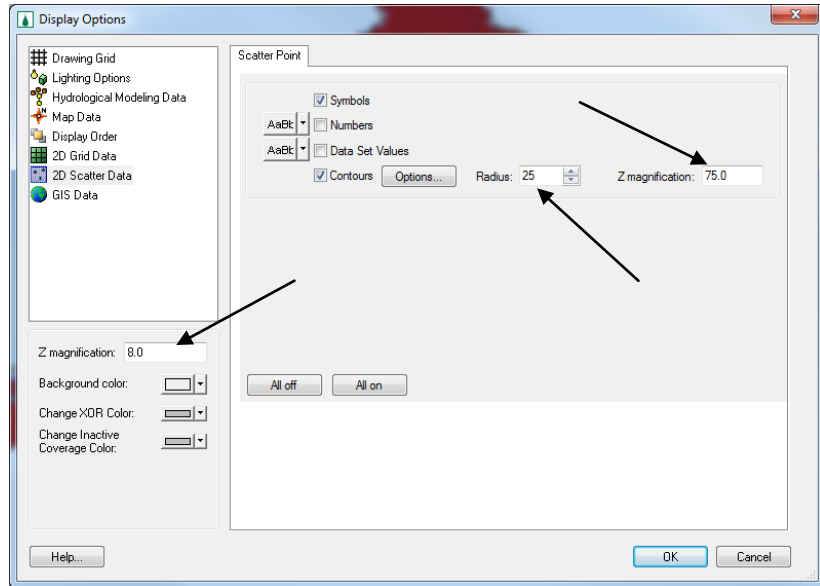
In the *2D Scatter Data* folder in the project explorer, you'll see two data sets, *Stream Depth*, and *Stream Flow*. The Stream Depth and Stream Flow data sets are link\node data sets. In order to visualize stream depth and/or stream flow, we need to follow these steps.


7.3.4.1 Visualizing Depth Contours

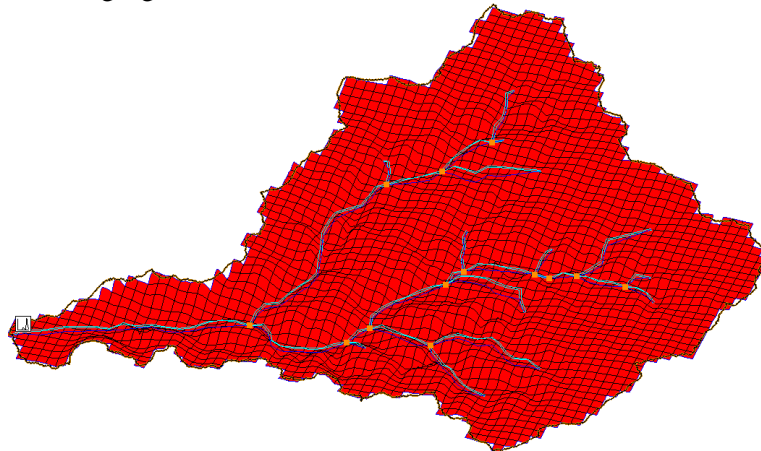
1. In the project explorer click on *Stream Flow* to select it.
2. Select *Display / Display Options*. In the 2D Scatter Data, toggle off the display of symbols and toggle on the display of Contours and click on *Options*.
3. Change the contour Method to *Color fill* and click *OK*



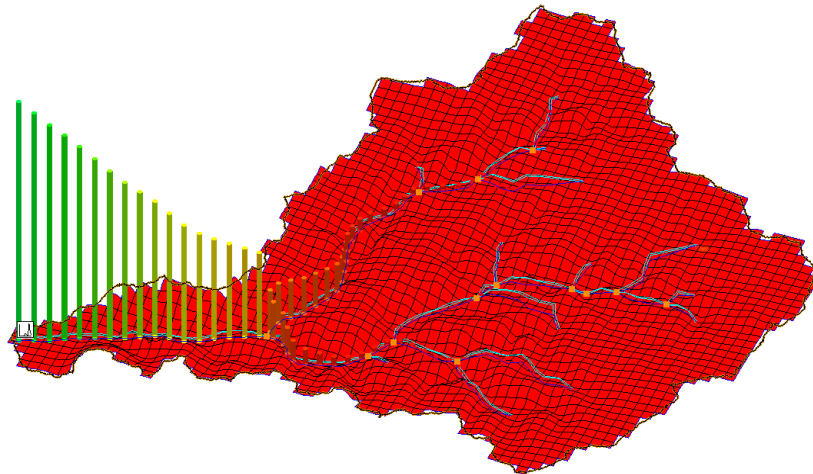
4. In the *Display Options* dialog, change the *Radius* to 25 and *Z-magnification* to 25.



5. On the right side of the dialog, set the Z-magnification to 8 and Click *OK*
6. Select the “*Rotate*” tool . Rotate the watershed by using the left mouse click 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 and cycle through the different time steps (shown on the properties window at the right side of WMS main window). You can see the bar diagrams representing flood wave generating and getting accumulated to create peak flow at the outlet

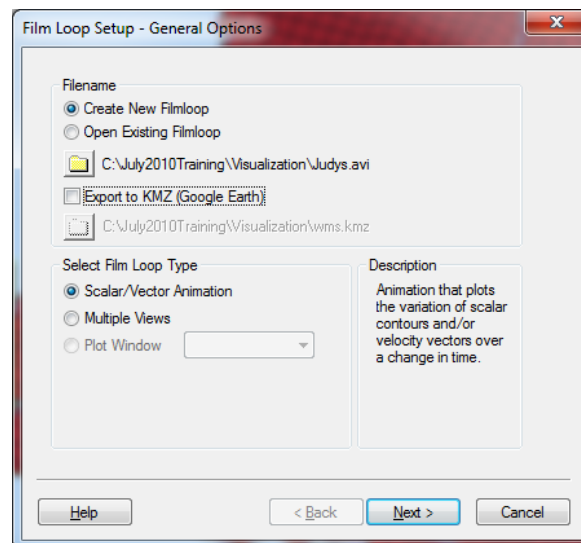


You can see the overland flow depth contour changing along with the flood wave bar diagrams. This gives a clearer idea of the natural process of overland flow.

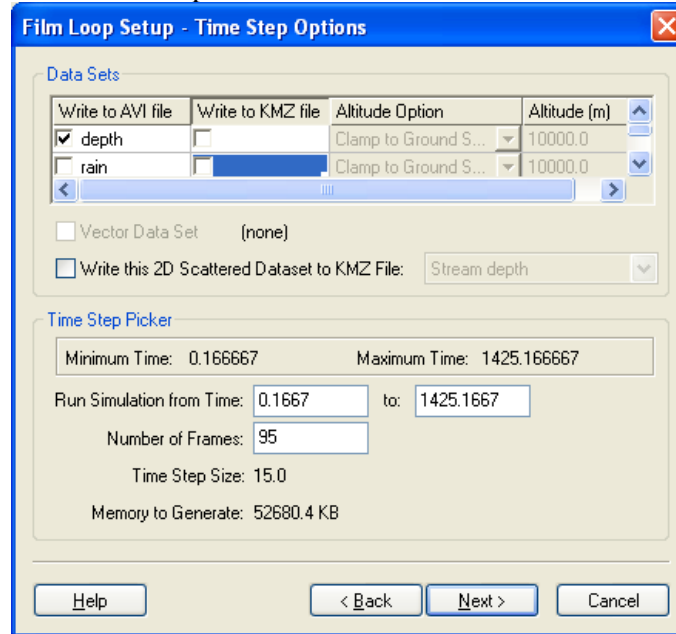
7.3.5 Creating an Animation Film Loop

These processes can be animated and a film loop created.

1. In the 2D-Grid Module , select **Data / Film Loop...**
2. Make sure to select “*Create new Film Loop*” 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. So, make sure *Export to KMZ* option is checked OFF.






4. Toggle the on “*Scalar\Vector Animation*” option and click Next.
5. Under *Write to AVI file*, check *depth* data set to select it. Check off all the tick marks under *Write to KMZ file*
6. Check off the option *Write this 2D Scattered Dataset to KMZ File*



7. Click Next.
8. Do not change any clock options. Click “*Finish*”. WMS might take some time to create the movie. It will begin playing once the entire movie is saved. 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.

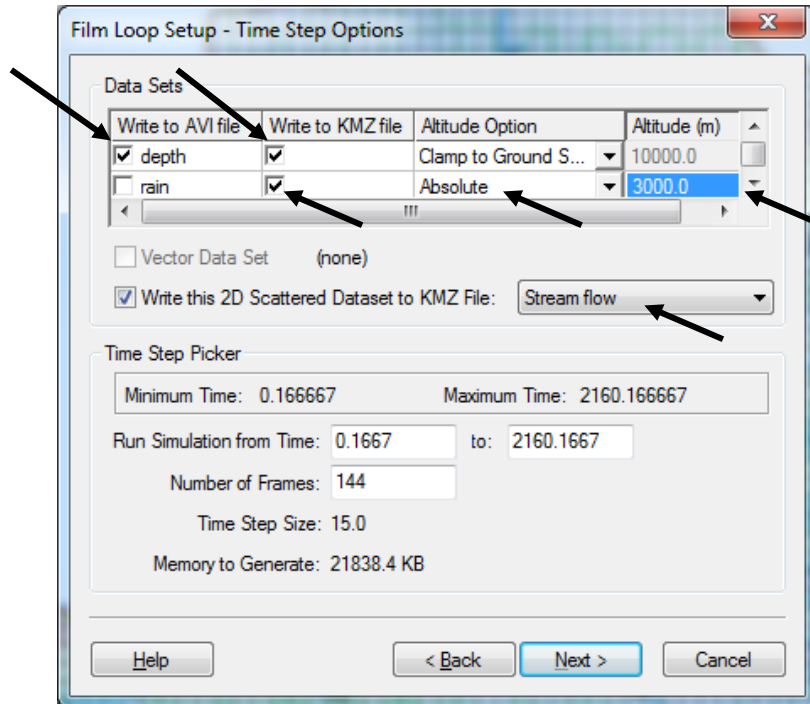
7.3.6 Creating a Google Earth Animation

You can export this film loop to Google Earth and have your flood animation displayed over a Google Earth image.

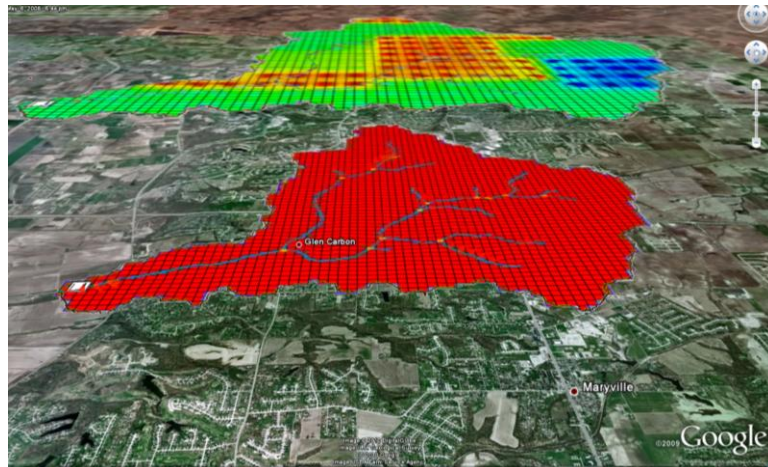
1. Change the display to plan view by clicking *Plan View* . You cannot export movie to Google Earth in an oblique view.
2. In the 2D-Grid Module , select *Data / Film Loop...*
3. Choose the location to save your AVI movie file by clicking on the *Browse* button . Save your movie as *C:\Training\Personal\Visualization\Judys.avi*
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*, check on the *depth* data set to select it.

7-8 GSSHA Modeling Using WMS – Volume 2

7. Check on the tick mark under *Write to KMZ file* corresponding to *both Depth and Rain* dataset.
8. For *Rain* datasets, select *Absolute* under *Altitude Option* and enter 3000m for *Altitude*.
9. Check on the option *Write this 2D Scattered Dataset to KMZ File* and select *Stream flow* option.



10. Click Next.
11. Do not change any clock options. Click “*Finish*”. WMS might take some time to (3-5 minutes) to create the movie and will open the Google Earth animation once the entire movie is saved.
12. You can tilt the Google Earth view to see the Stream Flow animation. You can also change the transparency of the data and change the speed etc.





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

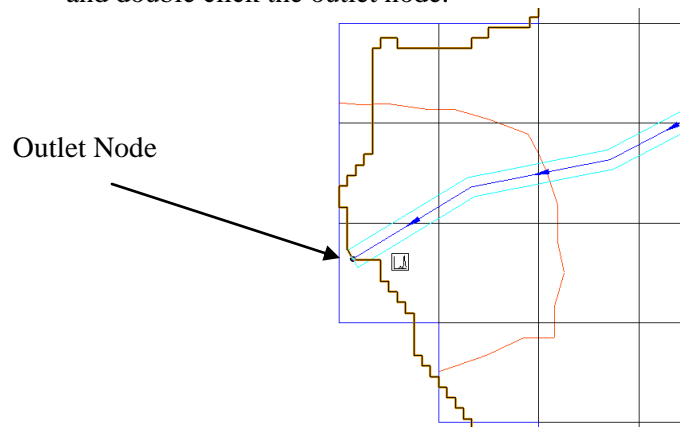
7.4 Comparing the simulation results with the Observed data

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

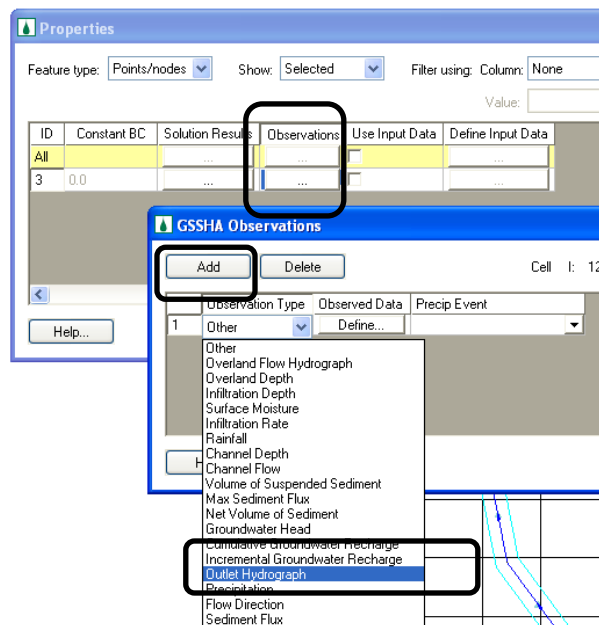
We will use an arbitrary observed flow hydrograph at the outlet and compare it with simulation results.

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.

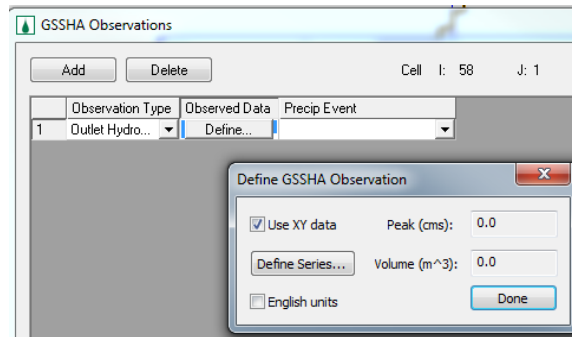


3. In the *Properties* dialog that opens, scroll all the way to the right and under *Observations* select the browse button.



7-10 GSSHA Modeling Using WMS – Volume 2

4. In *GSSHA Observations* dialog, click *Add* and select *Outlet Hydrograph* under *Observation Type*.
5. Click the *Define* button.
6. Check *Use XY data* option and click the *Define Series* button.



7. The observed flow data can be found at *C:\Training\Raw Data\JudysBranch\Judys_ObsFlow.txt*.
8. In Notepad, open this file, copy and paste data into XY Series editor in WMS. You can see the hydrograph plotted on the XY-series editor.
9. Click OK, click *Done*, and click OK. This will close all but the properties dialog.
10. Click *Browse* button under *Solution Results* (just on the left of *Observations*).
11. You can now compare the observed flow data with the simulations results in the *GSSHA Solution Analysis* dialog.

