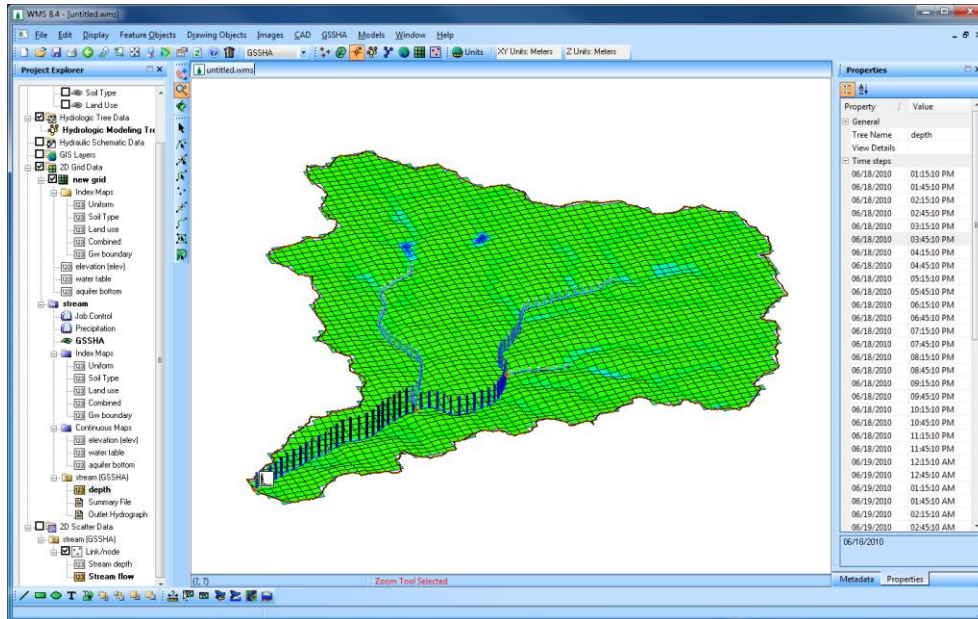


## WMS 10.0 Tutorial

# GSSHA – Modeling Basics – Developing a GSSHA Model Using the Hydrologic Modeling Wizard in WMS

Learn how to setup a basic GSSHA model using the hydrologic modeling wizard



## Objectives

This tutorial shows how to setup a working GSSHA model with overland flow, infiltration, and channel routing using steps in the hydrologic modeling wizard. Setting up these processes was covered in previous tutorials, but the wizard helps you setup a basic, less error-prone model that includes all these processes with minimal effort.

## Prerequisite Tutorials

- GSSHA – Modeling Basics – Stream Flow
- GSSHA – WMS Basics – Creating Feature Objects and Mapping their Attributes to the 2D Grid

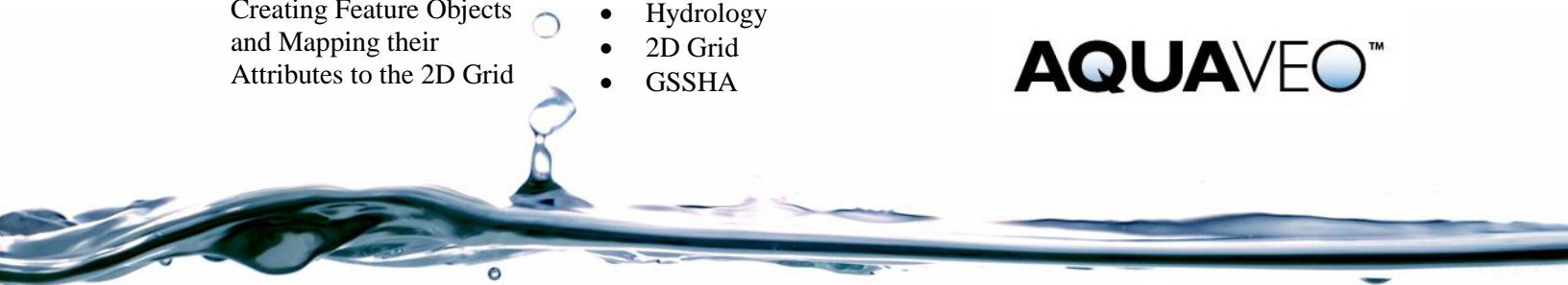
## Required Components

- Data
- Drainage
- Map
- Hydrology
- 2D Grid
- GSSHA

## Time

- 30-60 minutes

**AQUAVEO™**



# 1 Contents

---

1	Contents .....	2
2	Introduction .....	2
3	Starting the Hydrologic Modeling Wizard .....	3
4	Defining the Project Boundary and Projections .....	3
5	Importing the DEM Data .....	4
6	Computing Flow Direction and Flow Accumulation .....	5
7	Delineating the Watershed .....	6
8	Initiating GSSHA .....	7
9	Defining Stream Channels .....	7
9.1	Defining stream cross sections .....	7
9.2	Smoothing the stream thalwegs .....	8
10	Creating a Grid .....	10
11	Defining Job Control .....	10
12	Define Land use and Soil Data .....	10
13	Creating Index maps and defining Model Parameters .....	12
14	Defining Model Parameters .....	13
15	Editing Parameters .....	14
16	Defining a Meteorological Model .....	15
17	Cleaning and Checking the Model .....	15
18	Save the Model .....	15
19	Run the Model .....	15

## 2 Introduction

---

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

In this tutorial, you will create a GSSHA model with the Hydrologic Modeling Wizard using data in the area of Park City, UT. This model will use the same basic simulation options (2D overland flow, infiltration, and 1D stream flow) you have just set up in the previous tutorials and will serve as a review of what you have learned so far.

Locate the **|GSSHA Distributed Hydrologic modeling** folder in the files for this tutorial. If needed, download the tutorial files from [www.aquaveo.com](http://www.aquaveo.com).

The raw data for this Park City model can be found at **|GSSHA Distributed Hydrologic modeling|Raw Data|ParkCity**.

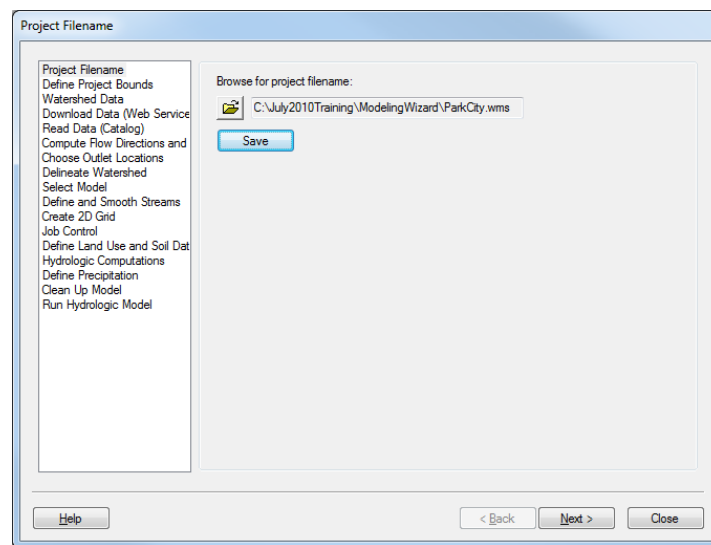
### 3 Starting the Hydrologic Modeling Wizard


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

2. Click on the “*Hydrologic Modeling Wizard*” button  near the menu



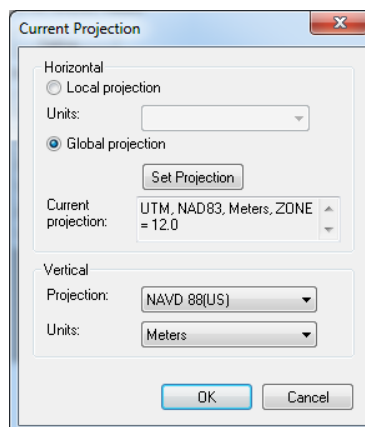
bar to start the Hydrologic Modeling Wizard. This wizard will guide you through the basic set up of a GSSHA model (or any other hydrologic model supported by WMS). You can start at the beginning of the wizard or jump in and out at any step as needed. All of the WMS menus\tools work while the wizard is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



3. Click on the *Browse* button  to specify the project name and location to save. Enter *\GSSHA Distributed Hydrologic modeling\Personal\ModelingWizard\ Parkcity.wms* as the project name and click *Save*.
4. Click on the *Save* button in the wizard.

### 4 Defining the Project Boundary and Projections


1. In the modeling wizard, click *Next* to go to the *Define Project Bounds* step of the wizard. Here you need to define your project projection and project bounds.
2. Click on the *Define* button under *Project projection*.



3. Set the Projection and Datum to be *UTM NAD 83, Zone 12* and set the horizontal and vertical units to *meters*. Set the vertical projection to *NAVD 88 (US)* and click *OK*.

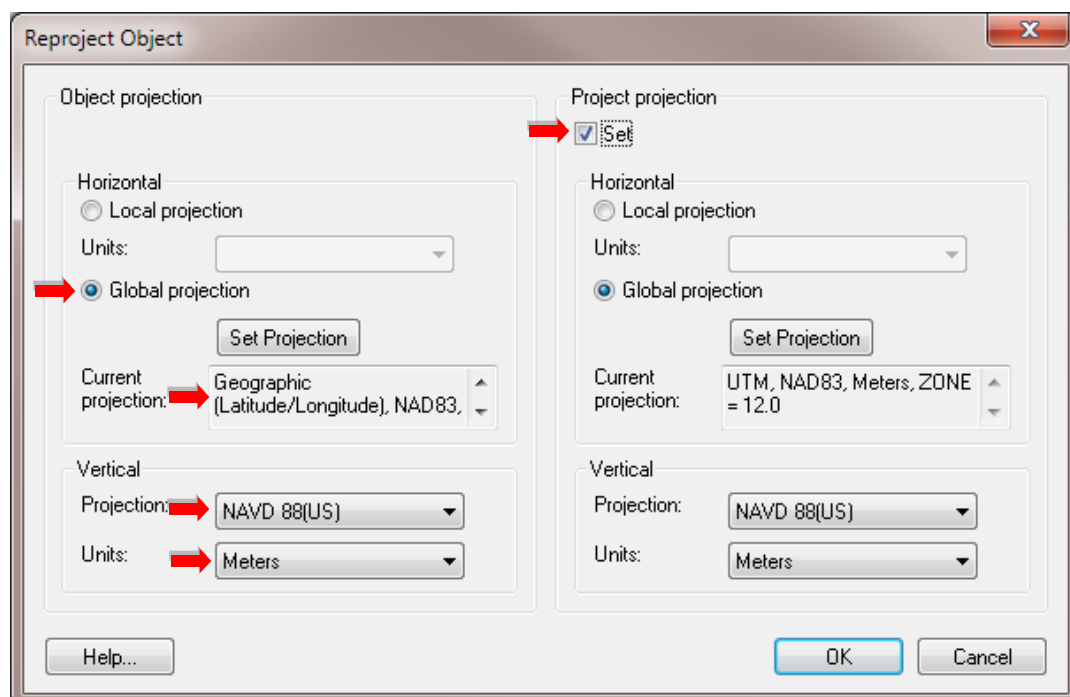
## **5 Importing the DEM Data**

---

1. Click *Next* on the wizard to import watershed data. The data can be downloaded from the web, from catalog files with listings of local data and their boundaries, or data can be imported from any location on your computer or local network. Here you will import the data from a file on your computer.
2. Next to the ***Open file(s)*** text in the wizard, click on the browse button . Open the file ***\\GSSHA Distributed Hydrologic modeling\\Raw Data\\ParkCity\\DEM\\ned\_35172081.hdr***.
3. Click *OK* to verify the data you have selected and Click *Yes* if prompted to *Reproject* the data.

**NOTE:** *Skip steps 4 and 5 if you are not prompted to Reproject the data*

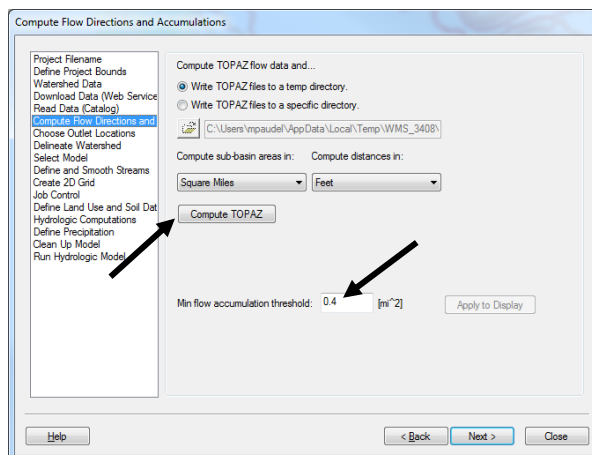
4. In the *Reproject Object* dialog (See below), make sure that the *Object Projection* coordinate is *Geographic NAD 83(US)* and that vertical units are set to meters (use *NAVD 88 (US)* for the vertical system). Then, if needed, check on the *Edit Project coordinate system* under the *Project projection* option.

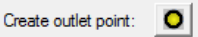


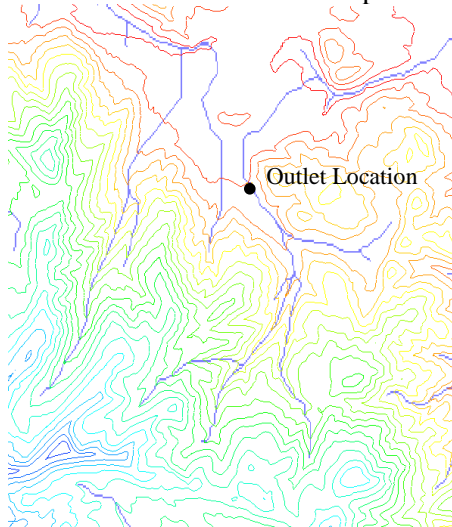
5. Set the *New Projection* Horizontal system to *UTM NAD 83 (US)*. Make sure that the UTM zone is 12 and the vertical system is NAVD 88 (US) in the new projection. The horizontal units for both the current and the new projections should be meters. Click *OK* and WMS will convert the projection of the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

## 6 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 0.4 sq mile.



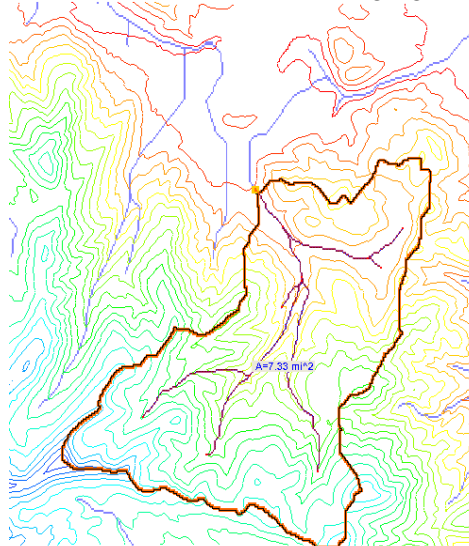
3. Click on the “*Compute Topaz*” button which will compute the flow directions and accumulations and infer the streams based on DEM data. Click on “*Close*” after computations are complete. If you move the modeling wizard dialog to the side of the WMS window, you can see blue lines added to the display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




## **7 Delineating the Watershed**

---

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



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

## 8 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*.

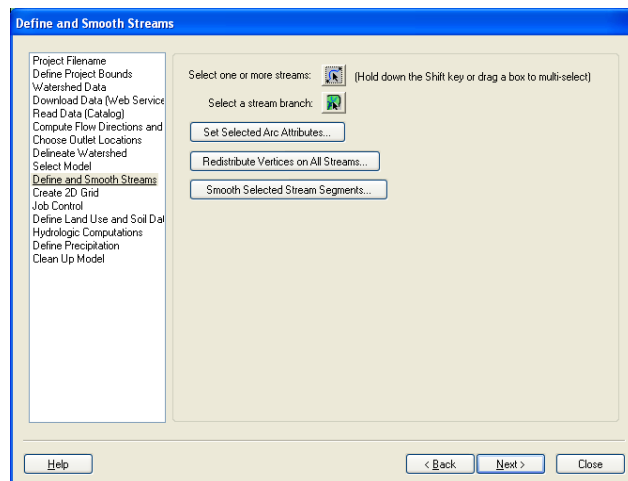
This generates the basic parameters necessary for GSSHA. Notice that the *Drainage* coverage in the project explorer has been converted to a *GSSHA* coverage and a default GSSHA model (GSSHAModel1) has been created under the 2D Grid Data folder.


## 9 Defining Stream Channels

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

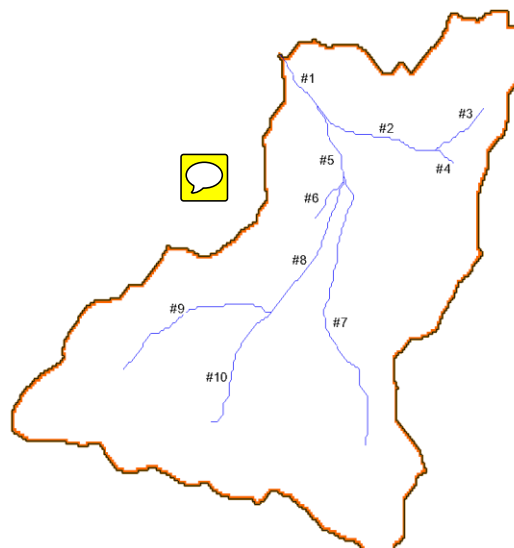
### 9.1 Defining stream cross sections

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



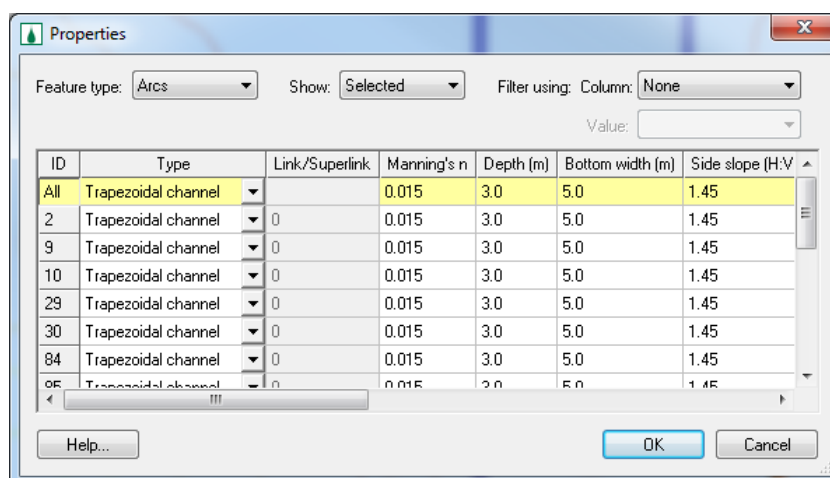
2. In the wizard, click on the *Select feature line branch tool*  and click on stream segment #1 (most downstream stream segment that is connected to the watershed outlet, you might need to zoom to the outlet area) in the following figure.
3. Select the *Set Selected ArcAttributes* button.





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

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




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

## 9.2 Smoothing the stream thalwegs

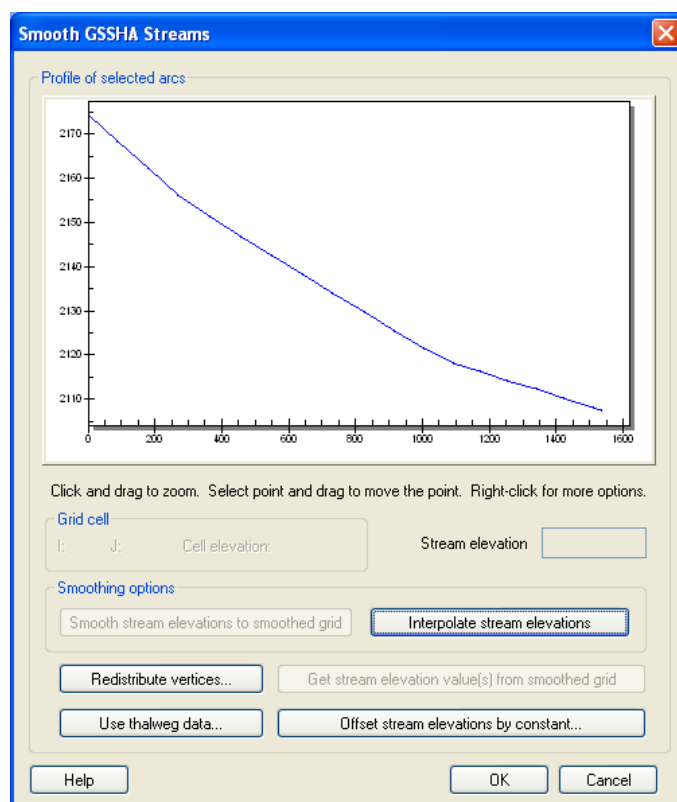
1. With the entire stream network still selected, click on the *Redistribute Vertices on All Streams* button in the wizard.
2. Enter 90 for the Spacing and turn on the option to *Use Cubic Spline*.
3. Click *OK*.



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

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

In the Smooth GSSHA Streams dialog you will see a profile of the arcs you have selected. Notice that while the segment has a general downward trend, in some places the streambed is significantly adverse. While GSSHA is able to handle adverse slopes, it is not desirable that adverse slopes should be present in a model where they do not exist naturally. You can solve this adverse slope problem by making slight changes to the node elevations along each segment.



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

Once the stream segment you have selected is smooth, select a new stream segment or combination of segments to smooth. Repeat the smoothing process outlined in steps 4 through 7 until no stream arcs in the basin have adverse slopes. Your streams are now ready for use in the GSSHA model.

## **10 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.

	X-dimension	Y-dimension
Cell size:	90.000	90.000

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

## **11 Defining Job Control**




---

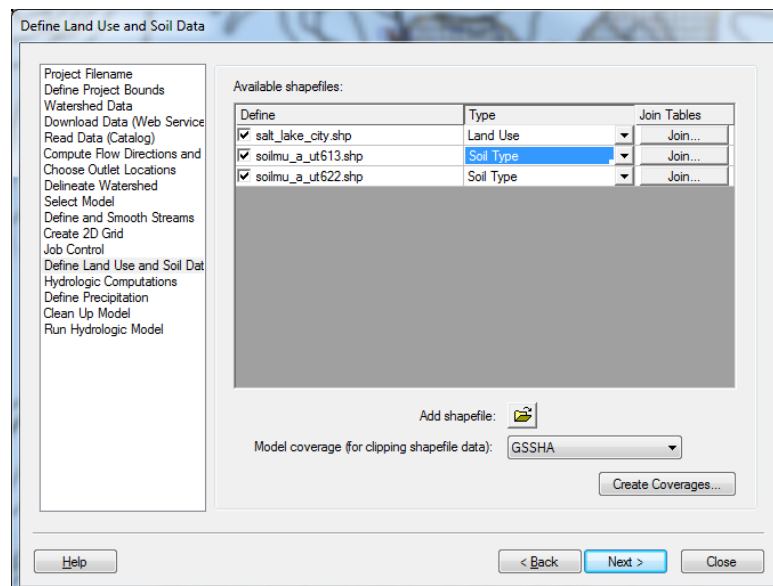
1. Click *Next* in the wizard dialog.
2. Enter start and end dates of (07\01\2010, 12:00:00PM to 07\02\2010, 12:00:00PM) with a computational time interval of 10 sec. for the model simulation. Click on *Set Job Control Data*.
3. Click *Next*.

## **12 Define Land use and Soil Data**

---

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

1. Click on the **Add shapefile** button  in the lower section of the *Define Land Use and Soil Data* window. Browse to and open the file `|GSSHA Distributed Hydrologic modeling\Raw Data\ParkCity\Luse\ salt_lake_city.shp`. Notice that the land use shape file is added to the “GIS Layers” in the WMS Project Explorer window.
  2. Click on the **Add Shapefile** button again  and open the file `|GSSHA Distributed Hydrologic modeling\Raw Data\ParkCity\SSURGO_Soil\soil_ut613\spatial\soilmu_a_ut613.shp`. Notice that this soil shapefile does not overlay your watershed completely. Because the shapefile does not overlay the entire watershed, you need to open an adjacent soil dataset so you have soil data that covers your entire watershed.
  3. Click on the **Add Shapefile** button again  and open the file `|GSSHA Distributed Hydrologic modeling\Raw Data\ParkCity\SSURGO_Soil\soil_ut622\spatial\soilmu_a_ut622.shp`. This will cover the entire watershed.
- Since these are the raw SSURGO data, you need to join the attributes to them. The NRCS SSURGO attributes are joined using a menu command available in the project explorer.
4. 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*. Turn all the toggles ON, use the default parameters, and click *OK*.
  5. In the Wizard, make sure that the *Type* field is correctly selected for the corresponding shapefiles. The *Type* field should be set to *Land Use* for `salt_lake_city.shp` and to *Soil Type* for the other shapefiles.

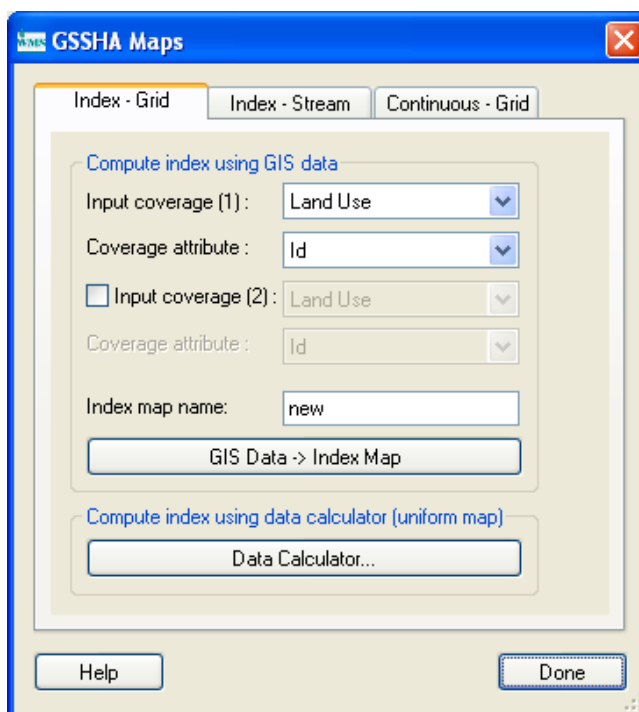


6. In the wizard, click on *Create coverages* to create land use and soil coverages and map the corresponding shape file polygons to their respective coverages. Because the shapefiles use standard attribute names, the mapping should be defaulted correctly, so accept the defaults in the *GIS to Feature Objects Wizard* by clicking *Next* and *Finish* when prompted.

7. After both the coverages are created, toggle off the display of your soil shapefiles (all the .shp files under the *GIS Layers* folder) and toggle off the *Land Use* and *Soil Type* coverage display under the *Coverages* folder in the *WMS Project Explorer*. Doing this cleans up your display and makes the model display faster.
8. Click *Next* on the modeling wizard.

## 13 Creating Index maps and defining Model Parameters

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

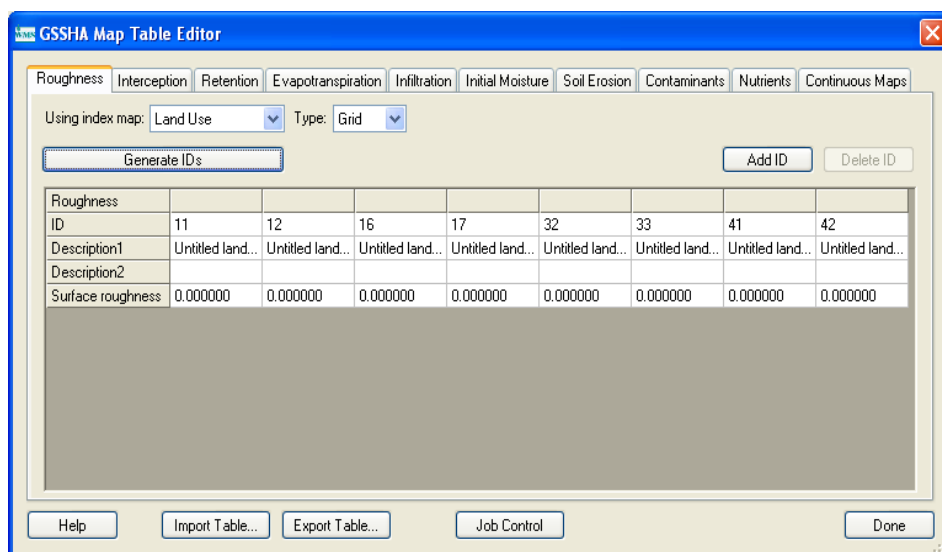


2. Select *Land Use* as the Input Coverage and *ID* as the Coverage Attribute. Enter *Land Use* as the Index Map Name. Then click on the **Coverages -> Index Map** button.
3. Select *Soil type* as the Input Coverage and *Texture* as the Coverage Attribute. Enter *Soil Type* as the Index Map Name. Then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

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

## 14 Defining Model Parameters

- In the *GSSHA Map Table Editor* dialog, switch to the *Roughness* tab (it is 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 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 `|GSSHA Distributed Hydrologic modeling|tables|gssha.cmt`.

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

Values obtained from this table should be used only as initial parameter estimates.

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

## **15 Editing Parameters**

---

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

7. Click *Next* on the modeling wizard to define the precipitation data.

## **16 Defining a Meteorological Model**

---

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

## **17 Cleaning and Checking the Model**

---

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

## **18 Save the Model**

---

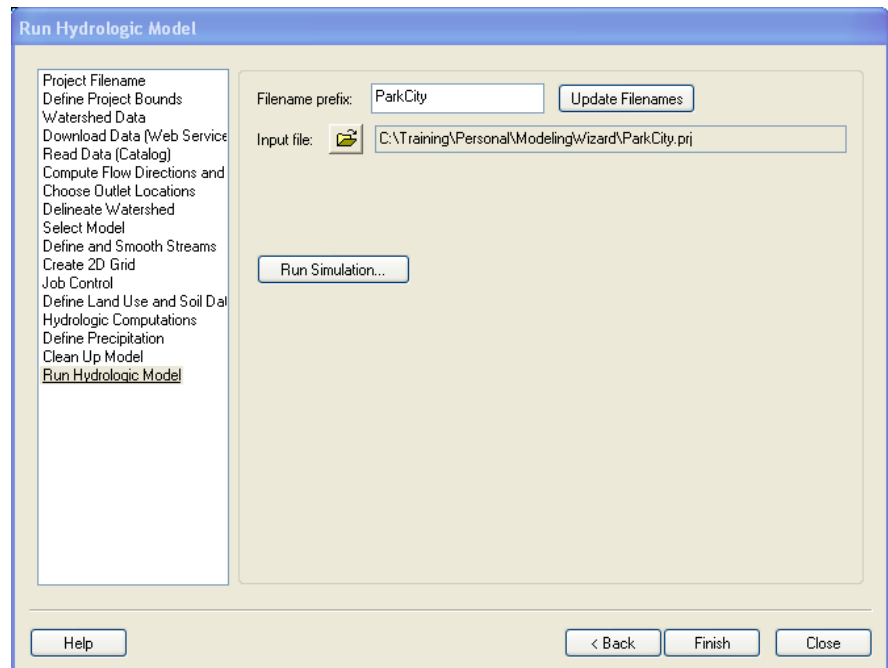
1. Click on the *Save* button to save the GSSHA project file. The GSSHA project file is in a different format than the WMS project file, so even though you have saved the WMS project, this step is necessary to save the GSSHA project file.
2. Save your project as *|GSSHA Distributed Hydrologic modeling\Personal\ModelingWizard\ Parkcity.prj*.

## **19 Run the Model**

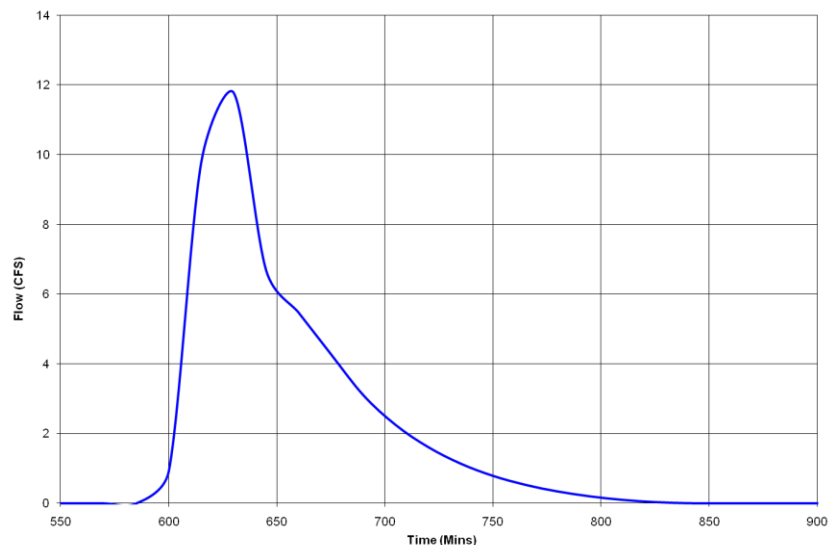
---

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





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



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