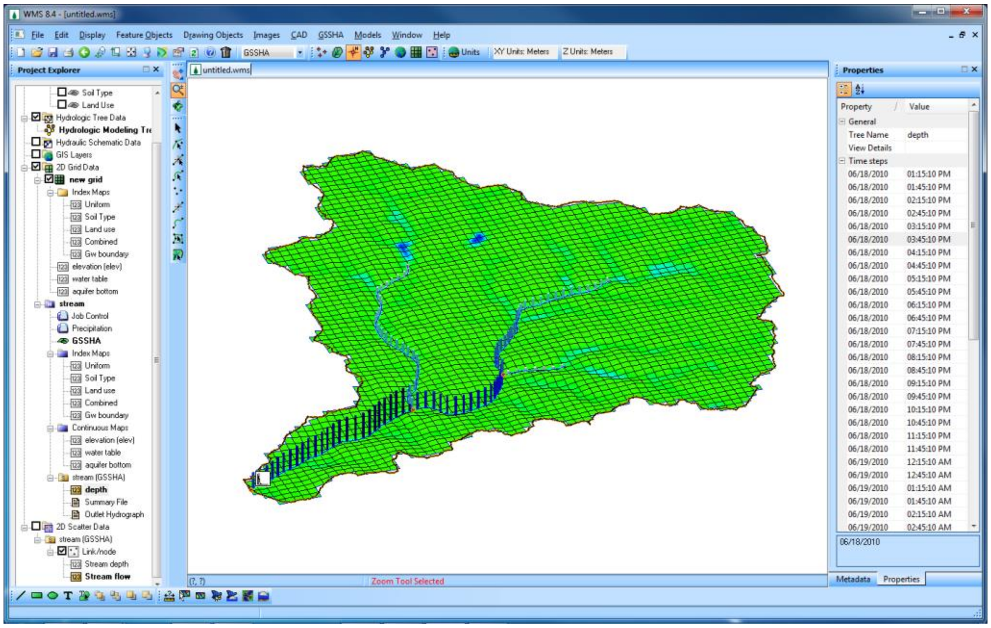
WMS

v. 10.1

WMS 10.1 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 set up 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

2 Introduction

Unless using the modeling tools on a day to day basis, it can be difficult to remember the step-by-step processes to develop a hydrologic model using WMS. The Primer on the GSSHA wiki is an excellent reference to help remember the process, but the WMS interface also includes a hydrologic modeling wizard to help guide through the basic model setup process. The wizard also streamlines and improves some of the processes used 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, 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) that were set up in the previous tutorials and will serve as a review.

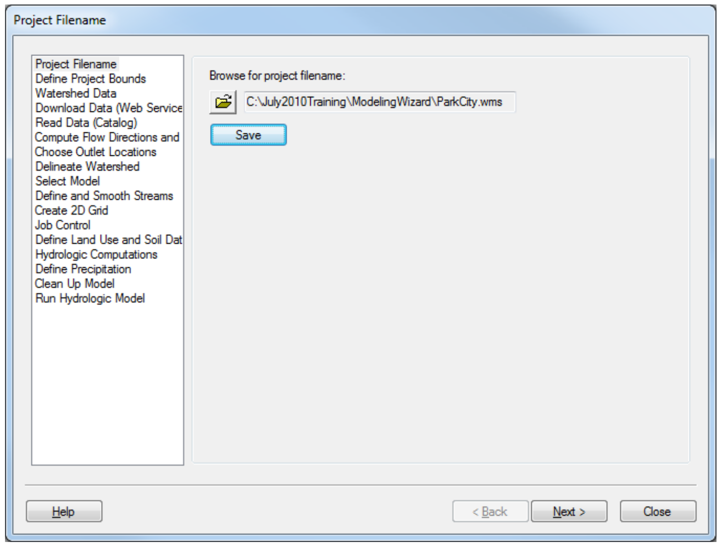
Locate the ***Raw Data***, ***Personal***, and ***Tables*** folders for this tutorial. If needed, download the tutorial files from www.aquaveo.com.

The raw data for this Park City model can be found at ***Raw Data\ParkCity.***

3 Starting the Hydrologic Modeling Wizard

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

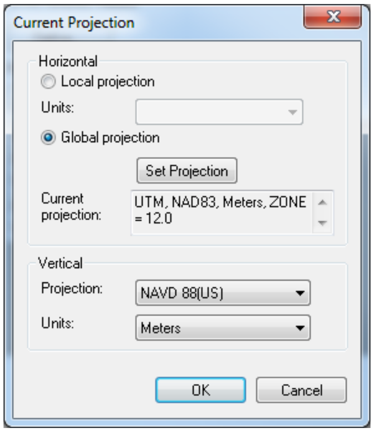
2. Click on the "***Hydrologic Modeling Wizard***" button  near the menu bar  to start the Hydrologic Modeling Wizard. This wizard will guide through the basic setup of a GSSHA model (or any other hydrologic model supported by WMS). Start at the beginning of the wizard or jump in and out at any step as needed. All of the WMS menus\tools work while the wizard is open, though the wizard window may need to be moved aside to see the WMS windows.



3. Click on the *Browse* button  to specify the project name and location to save. Enter ***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 define the 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

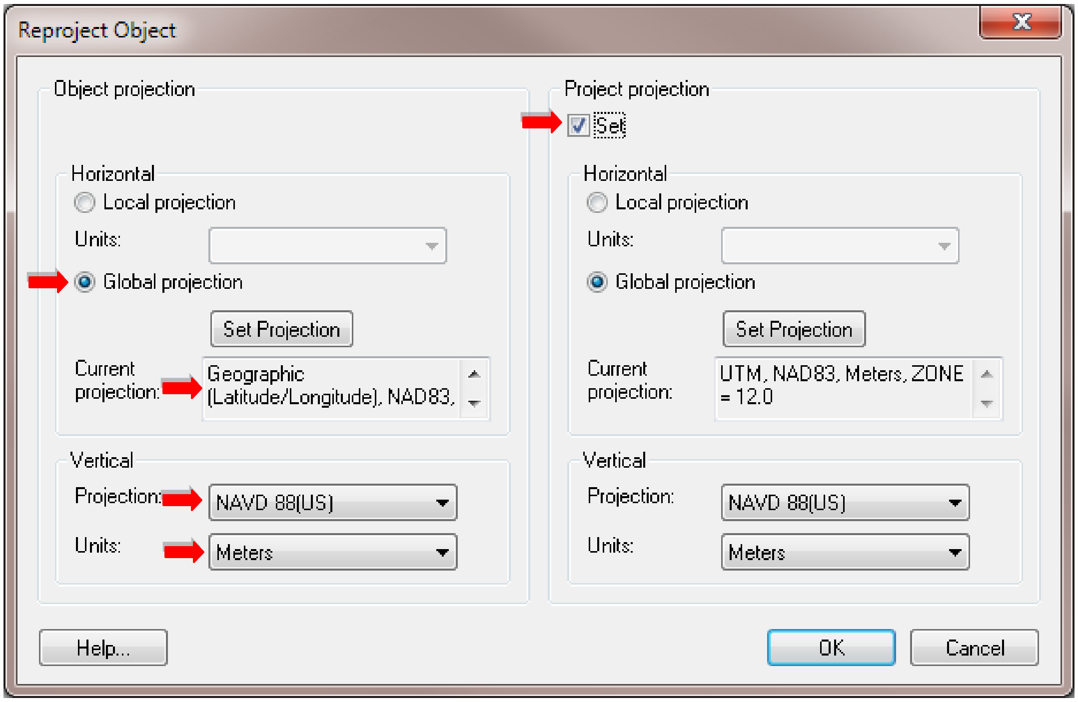
1. Click *Next* on the wizard to import watershed data. The data can be downloaded from the web, from catalog files with listings of local data and their boundaries, or data can be imported from any location on the computer or local network. Here import the data from a file on the computer.

2. Next to the ***Open file(s)*** text in the wizard, click on the browse button . Open the file ***Raw Data\ ParkCity\DEM\ ned_35172081.hdr.***

3. Click *OK* to verify the selected data and Click *Yes* if prompted to *Reproject* the data.

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

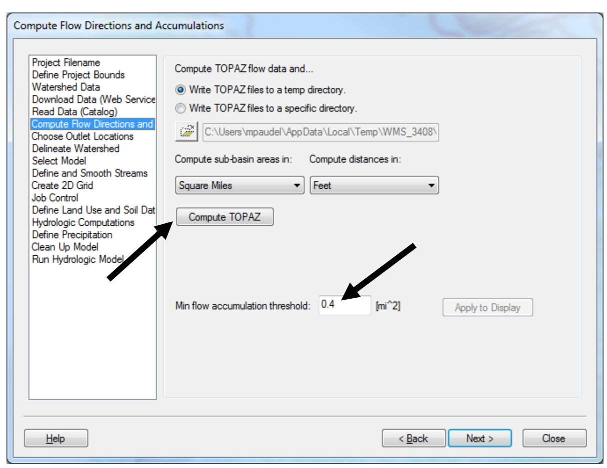
4. In the *Reproject Object* dialog (See below), make sure that the *Object Projection* coordinate is *Geographic NAD 83(US)* and that vertical units are set to meters (use *NAVD 88 (US)* for the vertical system). Then, if needed, check on the *Edit Project coordinate system* under the *Project projection* 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, the DEM contours should appear behind the modeling wizard in the WMS main window.

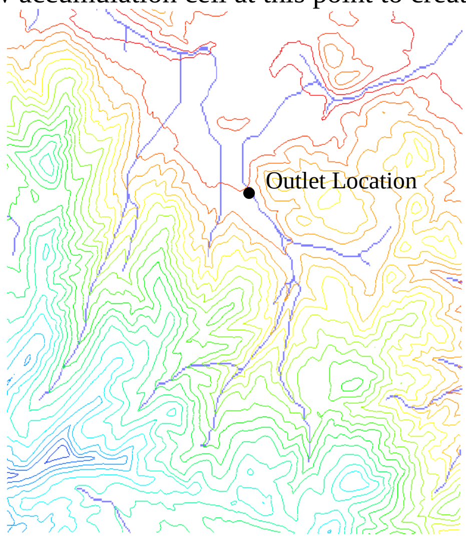
6 Computing Flow Direction and Flow Accumulation

1. In the Modeling Wizard, click *Next* and *Next* again until at the *Compute Flow Direction and Accumulation* part of the wizard.
2. Set the *Min flow accumulation threshold* to 0.4 sq mile.



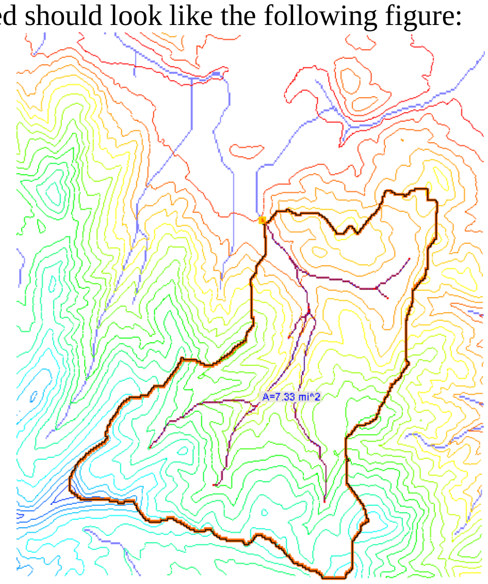
3. Click on the "*Compute Topaz*" button which will compute the flow directions and accumulations and infer the streams based on DEM data. Click on "*Close*" after computations are complete. Move the modeling wizard dialog to the side of the WMS window to see blue lines added to the display that represent areas of high flow accumulation.
4. Click Next in the modeling wizard and Select *Create Outlet Point* 

. When moving the mouse pointer out of the wizard dialog, it will now change to a small cross hair. Locate the point where the outlet should be for the watershed, 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* in the modeling wizard and click on the *Delineate Watershed* button.
2. The watershed should look like the following figure:



3. Save the 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*.

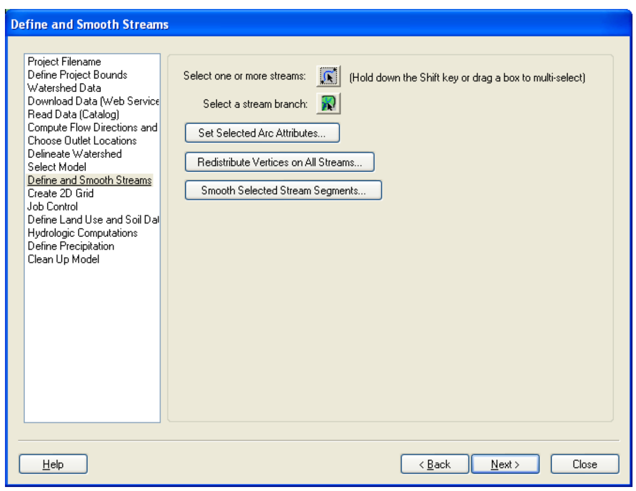
This generates the basic parameters necessary for GSSHA. Notice that the *Drainage* coverage in the project explorer has been converted to a *GSSHA* coverage and a default GSSHA model (GSSHAModel1) has been created under the 2D Grid Data folder.

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, define the channel cross section and roughness properties and also check and smooth the channel bottom.

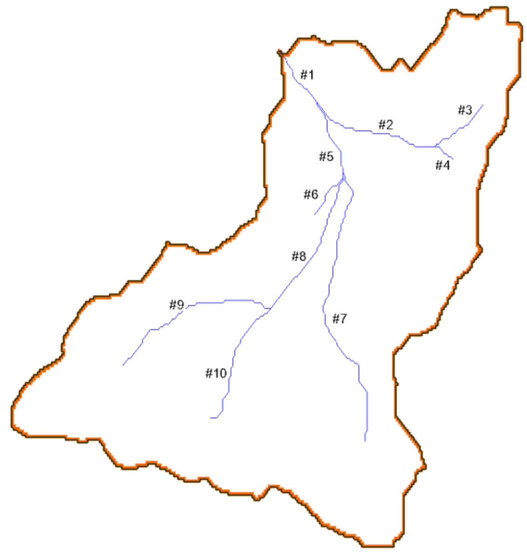
9.1 Defining stream cross sections

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



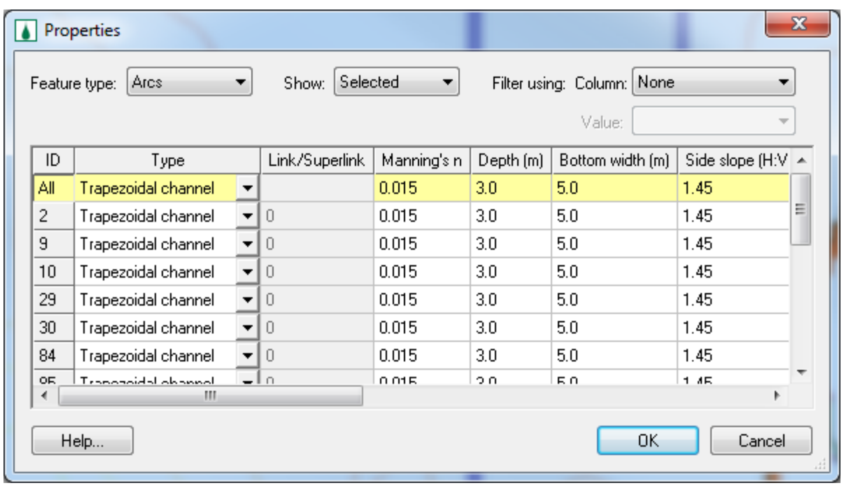
2. In the wizard, click on the *Select feature line branch tool*  and click on stream segment #1 (the most downstream stream segment that is connected to the watershed outlet) in the following figure.

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

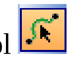


5. Click *OK*. Here assume that all the stream segments have the same cross section. However, as demonstrated previously, it may be necessary to define different stream properties for each arc.

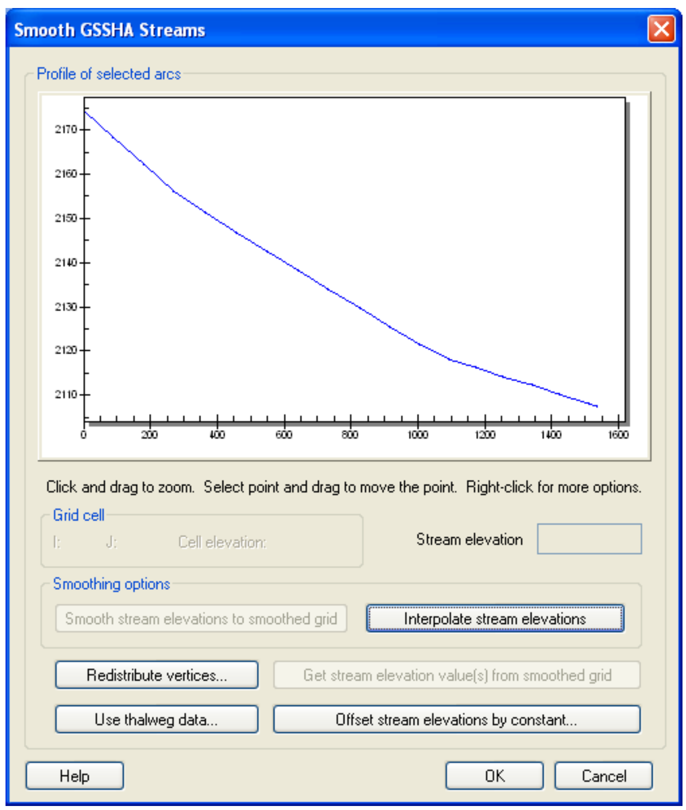
9.2 Smoothing the stream thalwegs

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 to be generated later. 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 the Shift key down and click) all arcs to the left (or right) until reaching the top. Make sure that there is no branch in the selection and the selection of streams is continuous.
5. With these streams selected, select *Smooth Selected Stream Segments* button on the wizard dialog.

The *Smooth GSSHA Streams* dialog shows a profile of the selected arcs. 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. Solve this adverse slope problem by making slight changes to the node elevations along each segment.



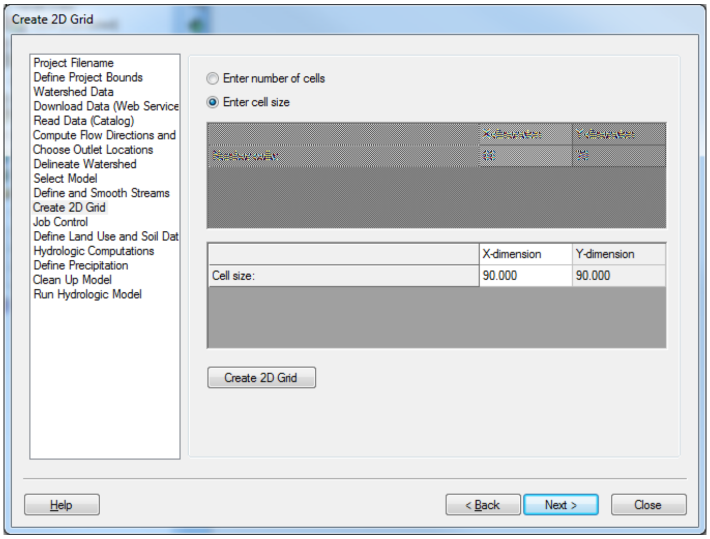
6. Select the *Interpolate Stream Elevations* button as many times as needed to generate a smooth stream segment with no uphill flow, then click *OK.*
7. If uphill flow cannot be eliminated in this manner, edit individual points by selecting the node and then dragging the point to a new position or editing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

10 Creating a Grid

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

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



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

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, 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 ***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.

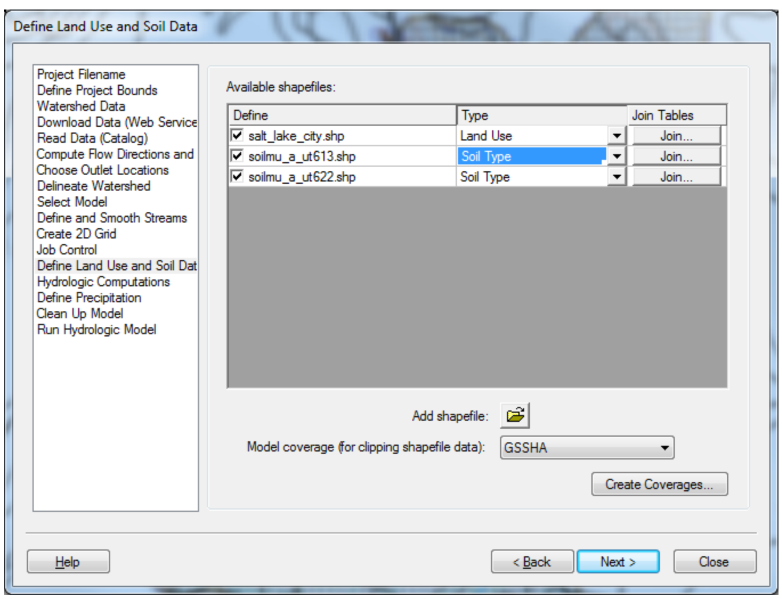
2. Click on the ***Add Shapefile*** button again  and open the file ***Raw Data\ParkCity\SSURGO_Soil\soil_ut613\spatial\soilmu_a_ut613.shp***. Notice that this soil shapefile does not overlay the watershed completely. Because the shapefile does not overlay the entire watershed, open an adjacent soil dataset to have soil data that covers the entire watershed.

3. Click on the ***Add Shapefile*** button again  and open the file ***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, 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.

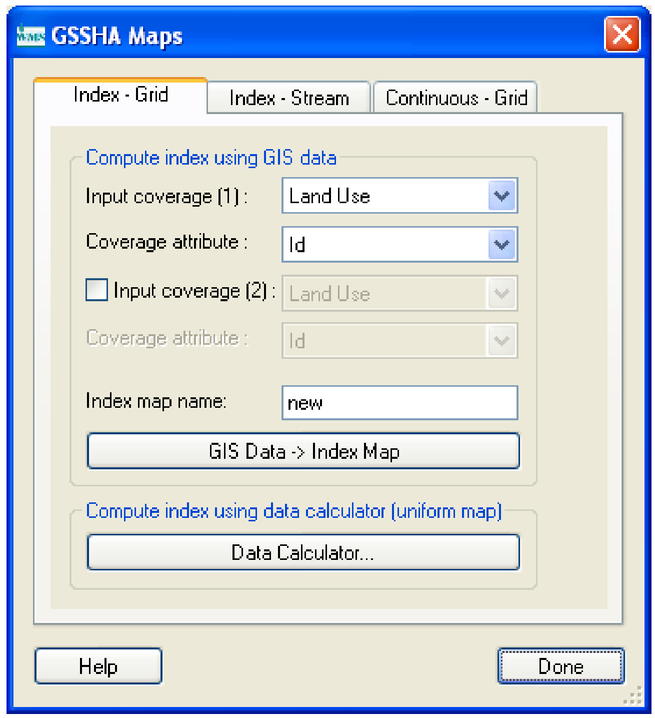


6. In the wizard, click on *Create coverages* to create land use and soil coverages and map the corresponding shape file polygons to the respective coverages. Because the shapefiles use standard attribute names, the mapping should be defaulted correctly, so accept the defaults in the *GIS to Feature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

7. After both the coverages are created, toggle off the display of the 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 the display and makes the model display faster.
8. Click *Next* in 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.

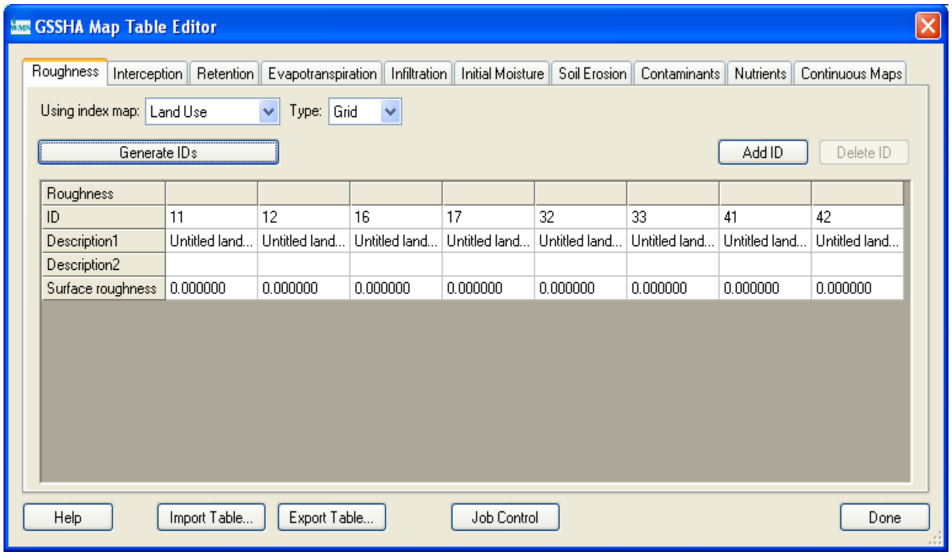


2. Select *Land Use* as the Input Coverage and *ID* as the Coverage Attribute. Enter *Land Use* as the Index Map Name. Then click on the ***Coverages → Index Map*** button.
3. Select *Soil type* as the Input Coverage and *Texture* as the Coverage Attribute. Enter *Soil Type* as the Index Map Name. Then click on the ***Coverages → Index Map*** button.
4. Toggle on the Input coverage (2) option to 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 the GSSHA model (Land Use, Soil Type, and Combined).

6. The *GSSHA Map Table Editor* dialog will pop up. Define the index mapping parameters.

14 Defining Model Parameters

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



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

    http:\\emrl.byu.edu\gsda\data_tips\tip_landuse_table.html

    to find a table that relates land use classification code IDs to land use descriptions.
11. For the columns with land use IDs of 11, 12, 16, and 17, reduce the values of Hydraulic conductivity to half of the values currently in the table. These land uses are impervious relative to bare soil, causing the infiltration rate to decrease. In a modeling situation, 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 entered here are smaller than the values for porosity in the *Infiltration* tab.
13. When done defining the parameters, click "*Done*".

15 Editing Parameters

1. Click on the *Edit Parameters* button in 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 datasets and the *Channel depth* and *Channel flow* link/node datasets.
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* in 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 "*TypeI-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

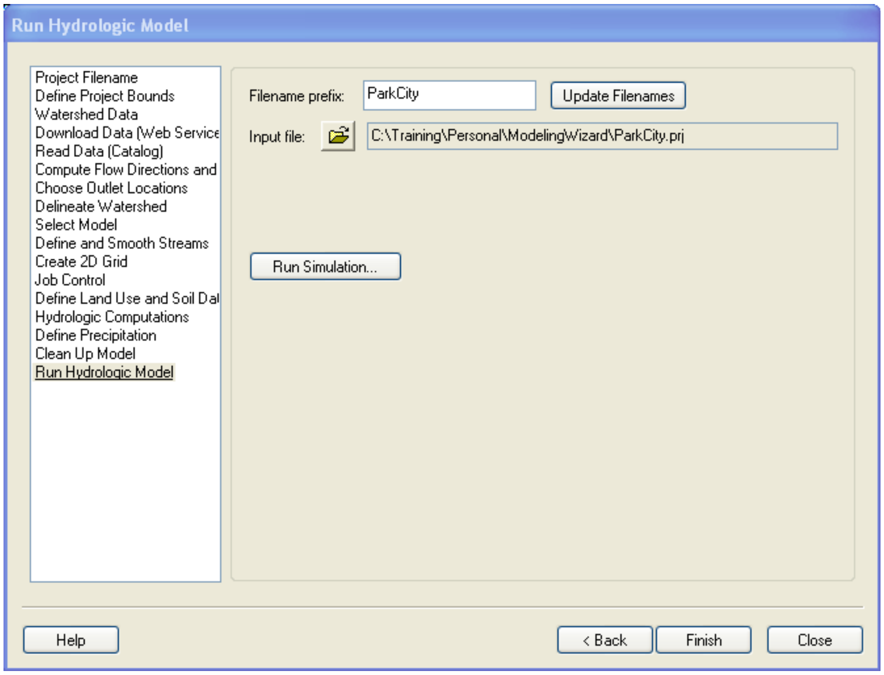
1. At this step in the hydrologic modeling wizard, most of the parameters and components required for the model are done. Click on the *Clean Up Model* button. This button does several things to clean up the display of the model. This button also prepares the model for running by running the Clean Dam 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 opens a GSSHA dialog where suitable corrections can be made 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

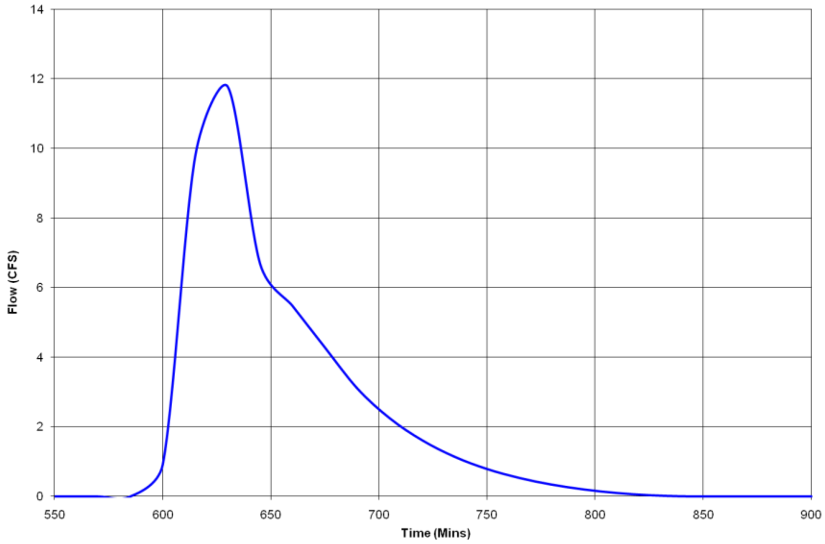
1. Click on the *Save* button to save the GSSHA project file. The GSSHA project file is in a different format than the WMS project file, so even though the WMS project is saved, this step is necessary to save the GSSHA project file.
2. Save the project as ***Personal\ModelingWizard\Parkcity.prj.***

19 Run the Model

1. Click *Next* in the modeling wizard. This will go to the *Run Hydrologic Model* wizard step.
2. Make sure the path to save the project is correct (***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, 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.



This tutorial demonstrated 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 a basic GSSHA model is set up, the model will be ready for modeling additional hydrologic processes.

