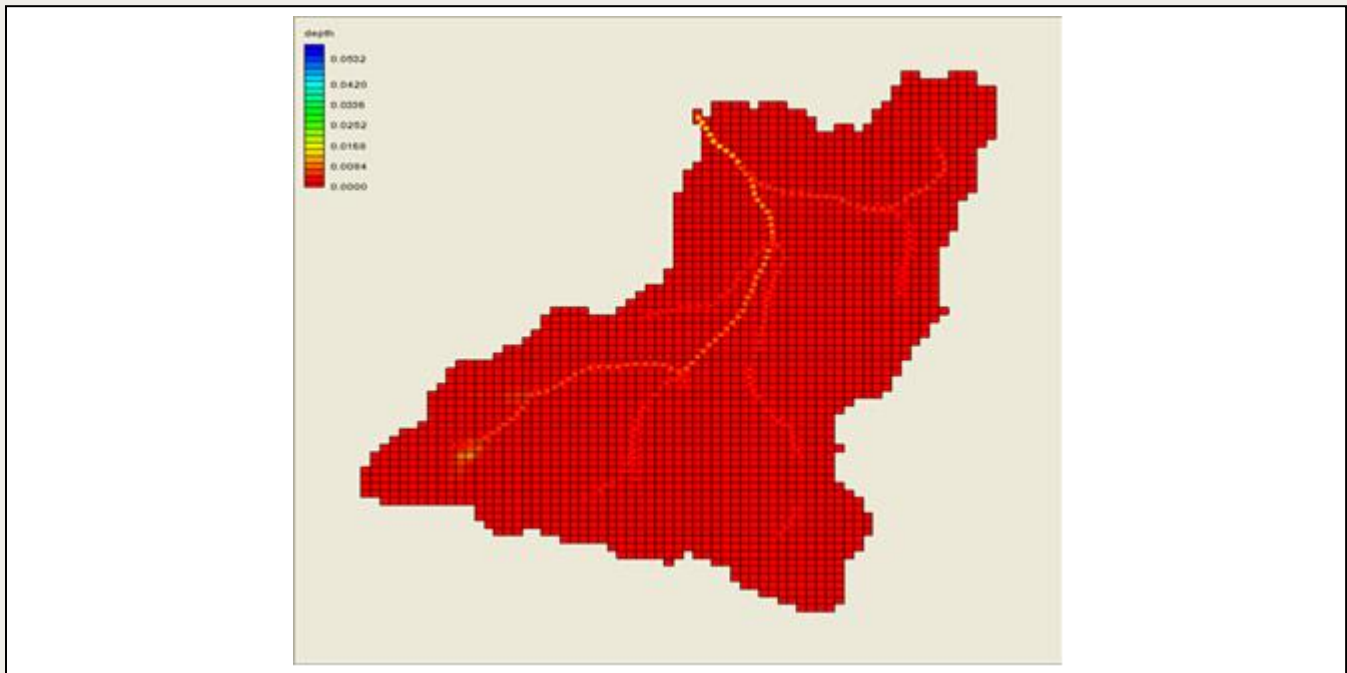


*WMS 11.2 Tutorial****Developing a GSSHA Model Using the Hydrologic Modeling Wizard***

Learn how to set up a basic GSSHA model using the WMS interface

**Objectives**

Step through the hydrologic modeling wizard to build a basic GSSHA model. Run the model through the GSSHA analysis engine. Read the GSSHA results and view an animation of the results.

**Prerequisite Tutorials**

- DEM Delineation

**Required Components**

- WMS Core
- GSSHA Model

**Time**

- 30–60 minutes






1	Introduction.....	2
2	Getting Started.....	2
3	Select Model.....	3
4	Define and Smooth Streams.....	3
4.1	Define Channel Geometry .....	4
4.2	Redistribute Vertices.....	5
4.3	Smooth Stream Thalwegs .....	5
5	Create 2D Grid .....	7
6	Job Control .....	7
7	Define Land Use and Soil Data.....	7
8	Hydrologic Computations.....	8
8.1	Create Index Maps .....	8
8.2	Define Model Parameters Using Index Map Tables.....	8
8.3	Additional Model Parameters.....	10
9	Define Precipitation .....	10
10	Clean Up Model.....	10
11	Run GSSHA.....	11
12	Post-Processing and Visualization .....	11
12.1	Outlet Hydrograph .....	11
12.2	Summary File.....	12
12.3	Grid Depth Contours.....	12
12.4	Stream Depth Contours .....	12
12.5	Creating an Animation Film Loop.....	14
13	Conclusion .....	15

## 1 Introduction

Gridded Surface Subsurface Hydrologic Analysis (GSSHA) is a physically based, distributed parameter, hydrologic model developed by the Engineering Research and Development Center of the Army Corps of Engineers. A detailed user's manual and additional GSSHA tutorials can be found at [www.gsshawiki.com](http://www.gsshawiki.com). In this tutorial, a GSSHA model will be created using data for Park City, UT and the Hydrologic Modeling Wizard. This tutorial uses a previously delineated watershed; to learn more about how to delineate a watershed using the Hydrologic Modeling Wizard, please see the "DEM Delineation" tutorial.

## 2 Getting Started

Get started by opening an existing project.

1. Open WMS, or click **New**  to reset WMS to its defaults.
2. If WMS was already open, a dialog may appear asking to save changes. Click **Don't Save** to clear all data.
3. Select **File / Open**  to bring up the *Open* dialog.
4. Browse to *spatial\spatial\WMS* and select "ParkCity.wms"
5. Select **Open** to close the *Open* dialog.
6. In the Project Explorer, under the  "Coverages" folder, right-click on  "Drainage" and select **Zoom To Layer**.
7. In the Project Explorer, turn off  "GIS Data".

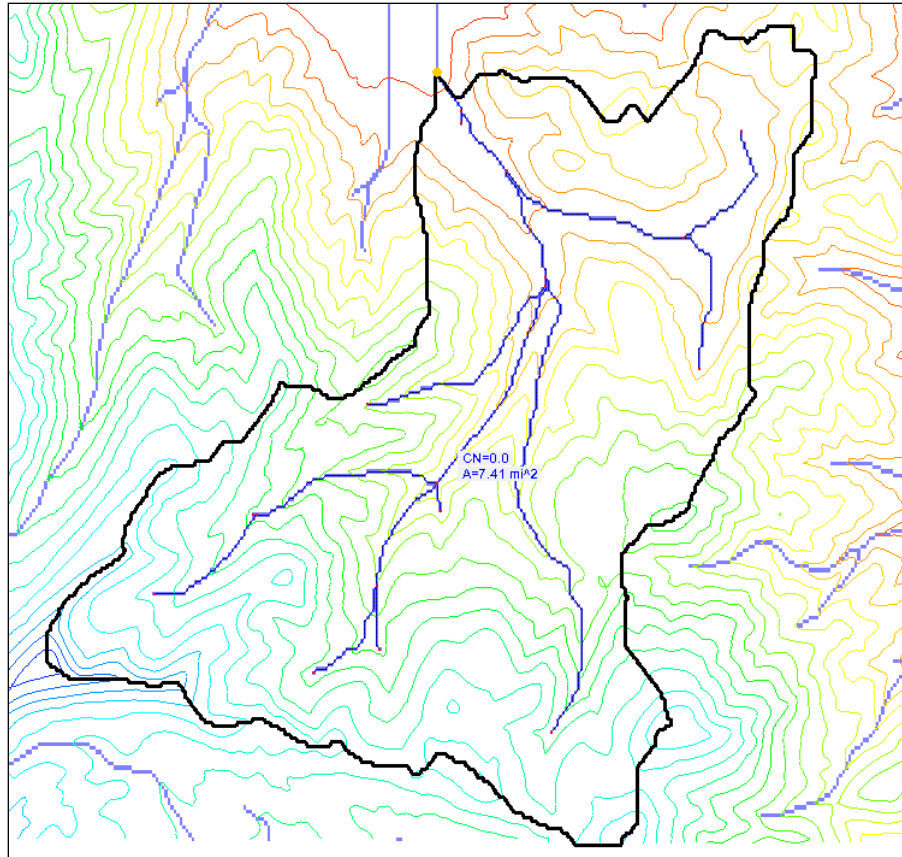



Figure 1 Initial project

### 3 Select Model

A watershed has already been delineated for the project. This allows skipping the first few steps of the Hydrologic Modeling Wizard and starting directly into the building the GSSHA model. For information on how to delineate a watershed, see the “DEM Delineation” tutorial.

To start building a GSSHA model:

1. Click on the **Hydrologic Modeling Wizard**  to open the *Project Filename* dialog.
2. From the list on the left, click on *Select Model*.
3. Use the drop-down menu under *Select the desired model* to select “GSSHA”.
4. Click on the **Initialize Model Data** button.
5. Click **Next >** to advance to the next step, *Define and Smooth Streams*.

### 4 Define and Smooth Streams





All the streams defined so far in GSSHA are generic and they do not have any defined shape or geometry. Also, because of anomalies in the DEM elevations, some stream bed elevations will not decrease between upstream and downstream cells when the 2D grid is

created. This “stream smoothing” problem creates issues when running a GSSHA simulation. In this step, define the channel cross section and roughness properties, and also check and smooth the channel bottom.

After defining the channel data and smooth streams, WMS assigns the smoothed stream elevations to the 2D grid wherever streams intersect 2D grid cells. This prevents having adverse slopes between grid cells on the 2D grid.

## 4.1 Define Channel Geometry

For this example, assume that all the stream segments have the same cross section. Enter different cross sections for individual stream arcs by selecting them and following the same procedure outlined in this section.

1. Select *Display* | **Display Options**  to bring up the *Display Options* dialog box.
2. Select “DEM Data” on the left, and turn off the options for *Stream* and *Flow Accumulation*.
3. Select **OK** to close the *Display Options* dialog.
4. In the Project Explorer, under the “ Terrain Data” folder, turn off “ ned\_35172081 (Converted)”.
5. In the *Define and Smooth Streams* dialog, choose the **Select a stream branch**  tool.
6. Click on stream segment #1 as shown in Figure 2 to select all the stream arcs. (If needed, use the scroll bar on the mouse to zoom in)
7. In the *Define and Smooth Streams* dialog, click on the **Set Selected Arc Attributes...** button to bring up the *Properties* dialog.
8. In the yellow *All* row of the spreadsheet:
  - Change the *Type* to “Trapezoidal channel”.
  - For *Manning’s n* enter 0.025
  - For *Depth (m)* enter 1.0
  - For *Bottom width (m)* enter 3.0
  - For *Side slope (H:V)* enter 1.45
9. Select **OK** to close the *Properties* dialog.
10. Select **Yes** if a message appears asking to renumber links.
11. Select **All** when asked to renumber all arcs or only the selected arcs.

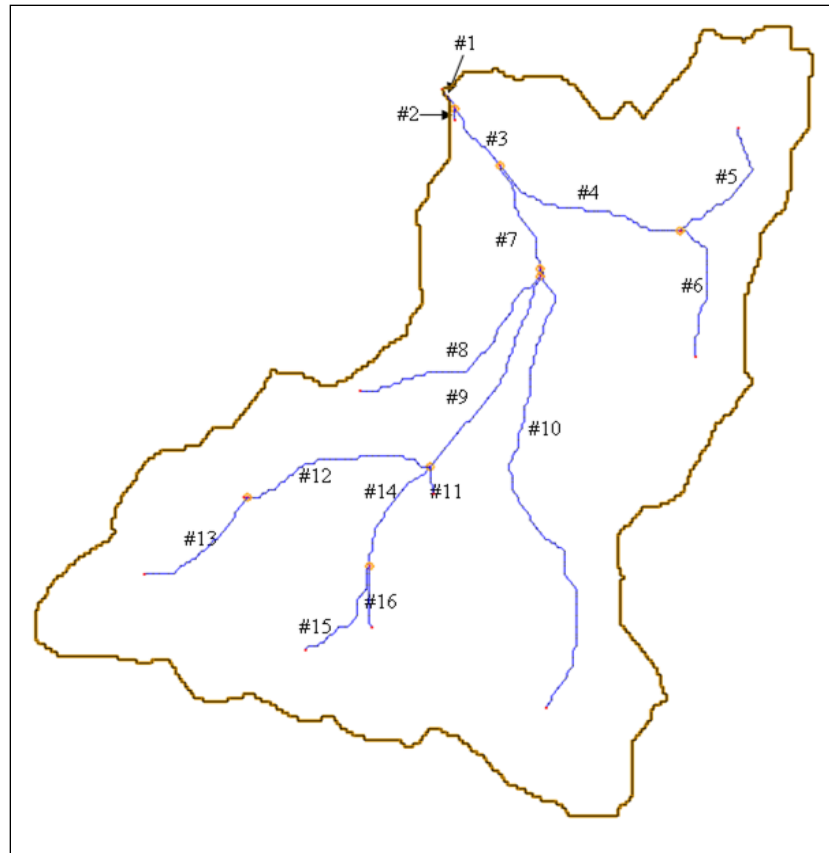


Figure 2 GSSHA stream segments


## 4.2 Redistribute Vertices

Before smoothing the streams, it is a good idea to redistribute the vertices on the streams because GSSHA requires a constant computational length between vertices along the 1-D streams. Also, make the spacing similar to the grid resolution.


1. In the *Define and Smooth Streams* dialog, click on the **Redistribute Vertices on All Streams...** button to open the *Redistribute Vertices* dialog box.
2. For *Average spacing* enter "90.0".
3. Turn on the option to *Use cubic spline*.
4. Select **OK** to close the *Redistribute Vertices* dialog.

## 4.3 Smooth Stream Thalwegs

After redistributing vertices, the next step is to smooth the thalweg elevations of the continuous branches of the channel until all have been smoothed.

1. Select *Display / Display Options*  to open the *Display Options* dialog.
2. In the "Map Data" tab, turn on the option for *Adverse Slopes*.
3. Select **OK** to close the *Display Options* dialog.
4. Click anywhere in the Graphics Window that's not an arc to deselect the stream arcs.

Notice that stream segments with adverse slopes are displayed using the color defined in the display options, and the streams that contain adverse slopes are bolded.

5. In the *Define and Smooth Streams* dialog, choose the **Select one or more streams**  tool.
6. Select stream arc #3 as shown in Figure 2.
7. Click on the **Smooth Selected Stream Segments...** button to bring up the *Smooth GSSHA Streams* dialog.

In the *Smooth GSSHA Streams* dialog, shown in Figure 3, notice a profile of the selected arcs. Notice that the segment has a general downward trend, but there are some places where the streambed slope is adverse. Even though GSSHA is able to handle adverse slopes, most of the adverse slopes are a result of the elevation data. It is not desirable to include adverse slopes that do not exist naturally in the model. This problem can be mitigated by making slight changes to the vertex and node elevations along the segment.

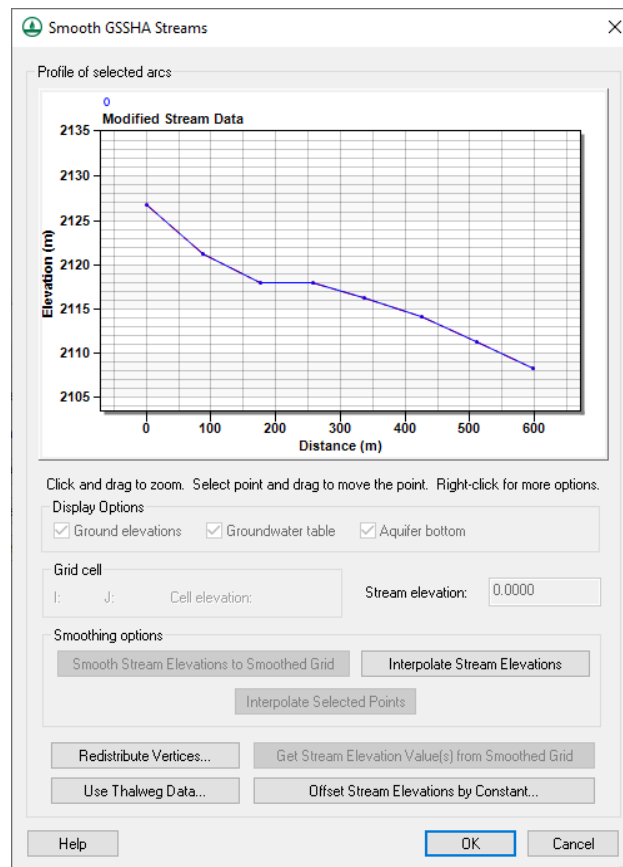


Figure 3 Smooth GSSHA Streams

8. Click the **Interpolate Stream Elevations** button as many times as needed to generate a smooth stream segment with no uphill flow.
9. Select **OK** to close the *Smooth GSSHA Streams* dialog.

If uphill flow cannot be eliminated in this manner, it is possible to edit individual points by selecting a point on the plot (hover directly over the point on the plot and move it up or down). Then, drag the point to a new position or edit the stream elevation value. Be especially careful to make sure the nodes next to the outlet are not adverse. Zoom into individual points by clicking and dragging a box around a section of the plot. It is also

possible to pan in any direction by moving the scroll bars along the sides of the plot. Right-clicking on the plot will produce a menu with several choices, including options to output data from the plot, set display options, and frame the plot to its maximum extents.

10. Repeat steps 5–8 for stream segments #4, 5, 6, and 14 as shown in Figure 2.
11. Click **Next >** to advance to the next step, *Create 2D Grid*.

## 5 Create 2D Grid

---

Now that the streams are smoothed, the next step is to create a 2D grid. When creating the grid, WMS assigns the smoothed stream elevations to the 2D grid wherever streams intersect 2D grid cells.

1. Make sure that the *Enter cell size* option is selected.
2. For the *X-dimension* enter a *Cell size* of “90” meters (the *Y-dimension* is automatically set to the same value as the X-dimension).
3. Click on the **Create 2D Grid** button to bring up the *Background Elev Interpolation* dialog.
4. Select *Interpolate GSSHA coverage streams* then select **OK** to interpolate elevations for each grid cell from the background DEM, and close the *Background Elev Interpolation* dialog.
5. Click **Yes** to delete the existing background DEM.
6. Click **Next >** to advance to the next step, *Job Control*.

## 6 Job Control

---

For the job control parameters, set the following:

1. Set the *Starting date* to “01/01/2008”.
2. Set the *Starting time* to “12:00:00 PM”.
3. Set the *Ending date* to “01/03/2008”.
4. Set the *Ending time* to “12:00:00 PM”.
5. Set the *Time interval* to 10 seconds.
6. Click on the **Set Job Control Data** button.
7. Click **Next >** to advance to the next step, *Define Land Use and Soil Data*.

## 7 Define Land Use and Soil Data

---

Since there is already land use and soil shapefiles imported, it is possible to convert these to feature data in the map module, which will be used to develop hydrologic input parameters.

1. Verify that “salt\_lake\_city.shp” is set to “Land Use” under the *Type* column.
2. Make sure that “SSURGO\_Soil.shp” is set to “Soil Type” under the *Type* column.
3. Click on the **Create Coverages...** button to bring up the *GIS to Feature Objects Wizard* dialog.

4. Select **Next >** to proceed to step 2 of 3.

Notice that WMS automatically set the LUCODE in the shapefile to be mapped to the Land use parameter in WMS.

5. Select **Next >** to proceed to step 3 of 3.
6. Select **Finish** to return to step 1 of 3 of the *GIS to Feature Objects Wizard*.
7. Click **Next >** to proceed to step 2 of 3.
8. Click **Next >** to proceed to step 3 of 3 of the wizard.
9. Click **Finish** to close the *GIS to Feature Objects Wizard* and return to the *Define Land Use and Soil Data* dialog.

WMS automatically maps HYDGRP to SCS soil type, TEXTURE to Texture, KSAT to Hydraulic conductivity, MOISTURE to Initial moisture, FIELD CAP to Field capacity, and WILTINGPT to Wilting point.

10. Click **Next >** to advance to the next step, *Hydrologic Computations*.

## 8 Hydrologic Computations

---

### 8.1 Create Index Maps

---

1. Click on the **Compute Index Mapping Tables...** button, to open the *GSSHA Maps* dialog.
2. For *Input coverage (1)* choose "Land Use" and set the *Coverage attribute* to "Id".
3. Enter an *Index map name* of "Land Use".
4. Click on the **Coverages** → **Index Map** button.
5. For *Input coverage (1)* choose "Soil Type" and set the *Coverage attribute* to "Texture".
6. Enter an *Index map name* of "Soil Type".
7. Click on the **Coverages** → **Index Map** button.
8. Turn on the *Input coverage (2)* option to create a combined index map based on both land use and soil type coverages.
9. For *Input coverage (2)* choose "Land Use" and set the *Coverage attribute* to "Id".
10. Enter an *Index map name* of "Combined".
11. Click on the **Coverages** → **Index Map** button.
12. Select **Done** to close the *GSSHA Maps* dialog and open the *GSSHA Map Table Editor* dialog.

Three index maps should be visible on the data tree under the 2D Grid Data. The next step is to define the index mapping parameters.

### 8.2 Define Model Parameters Using Index Map Tables

---

1. Switch to the *Roughness* tab (it is usually selected by default).
2. For the *Using index map* field select "Land Use".



3. Click on the **Generate IDs** button.

IDs for each of the land use type polygons that overlay the 2D grid are generated and displayed.

4. Now switch to the *Infiltration* tab.
5. Select **Yes** to turn the infiltration option on, and open the *GSSHA Job Control Parameters* dialog.
6. Under *Infiltration*, select the *Green & Ampt with soil moisture redistribution* option.
7. Select **OK** to close the *GSSHA Job Control Parameters* dialog and return to the *GSSHA Map Table Editor* dialog.
8. For the *Using index map* field select "Combined".
9. 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.

1. Switch to the *Initial Moisture* tab.
2. For the *Using index map* field select "Soil Type".
3. Click on the **Generate IDs** button.
4. Click on the **Import table...** button at the bottom part of the *GSSHA Map Table Editor*, to bring up the *Open* dialog.
5. Browse to the *spatial* folder.
6. **Open** the file "gssha.cmt" to close the *Open* dialog and return to the *GSSHA Map Table Editor* dialog.

**NOTE:** This table has typical values for the watershed parameters based on standard soil type and land use classification. Importing this table will populate model input parameters with the values in the file. Model input parameters should always be verified. One important thing to remember is that the values from the table are not absolute and the modeler needs to take ownership of the input values and adjust as necessary for the specific conditions of a given area. Generally these values are best estimated either by field measurement or by model calibration. Values obtained from this table should only be used as an initial estimate of the corresponding parameters. In this particular tutorial, none of the values need to be changed.

7. Switch back to the *Roughness* tab.

Notice the preliminary values for roughness that were entered based on the land use data.

8. Switch to the *Infiltration* tab.

The values listed in this spreadsheet are based on soil texture only. Notice that the index map contains a land use and soil type for each index map ID. One would expect the infiltration values for sand in an agricultural area to be different from the infiltration value for sand in a residential area. Similarly, loam in a forest environment will have different infiltration values than loam in an industrial area. Based on the knowledge of how land use affects infiltration properties of the soil, make suitable adjustments to the default parameters when working on personal models.

9. Switch to the *Initial Moisture* tab.

If wanting to adjust the values for initial moisture, enter these values here (make sure that the values entered here are smaller than the values for porosity in the *Infiltration* tab).

10. Select **Done** to close the *GSSHA Map Table Editor* dialog and return to the *Hydrologic Computations* dialog.

### 8.3 Additional Model Parameters

---

1. Click on the **Edit Parameters...** button to open the *GSSHA Job Control Parameters* dialog.
2. Under *Channel routing computation scheme* choose *Diffusive Wave*.
3. Click on the **Output Control...** button to bring up the *GSSHA Output Control* dialog.
4. Under the *Gridded data sets* section, turn on the *Cumulative infiltration depth* and *Infiltration rate* options.
5. Under the *Link / Node data sets* section turn on the *Channel depth* and *Channel flow* options.
6. Change the *Write frequency* of these datasets to “15” min.
7. In the *Hydrograph* section, change the *Write frequency* to “5” min.
8. Set the hydrograph *Output units* to be *English (cfs)* (all computations in GSSHA are performed using metric units, but the outlet hydrograph will be automatically converted from m<sup>3</sup>/s to cfs).
9. Select **OK** to close the *GSSHA Output Control* dialog.
10. Select **OK** to close the *GSSHA Job Control Parameters* dialog.
11. Click **Next >** in the *Hydrologic Modeling Wizard* to advance to the next step, *Define Precipitation*.

## 9 Define Precipitation

---

1. Click on the **Define Precipitation...** button to bring up the *GSSHA Precipitation* dialog.
2. For *Rainfall event(s)* select “Hyetograph”.
3. Click on the **Define Distribution...** button to bring up the *XY Series Editor* dialog.
4. Change the *Selected Curve* to “typel-24hour” by using the drop-down menu.

This is the temporal distribution of an SCS Type I-24 hour storm.

5. Select **OK** to close the *XY Series Editor* and return to the *GSSHA Precipitation* dialog.
6. Enter an *Average Depth* of “63.5” mm.
7. Select **OK** to close the *GSSHA Precipitation* dialog and return to *Define Precipitation*.
8. Click **Next >** to advance to the next step, *Clean Up Model*.

## 10 Clean Up Model


---

Defining most of the parameters and components required for the model are done.

1. Click on the **Clean Up Model** button to bring up the *GSSHA Model Check* dialog.
2. Click **Done** to close the *Model Check* dialog and bring up the *Model Wrapper* dialog.
3. Click **Close** when the model finishes to close the *Model Wrapper* dialog and return to the *Clean Up Model* dialog.
4. Click on the **Save** button to open the *Save GSSHA Project File* dialog.
5. Browse to the *spatial\GSSHA\* directory.
6. Enter "ParkCity.prj" and select **Save** to close the *Save GSSHA Project File* dialog and return to the *Clean up Model* dialog.
7. Select **Close** to close the *Clean Up Model* dialog.

## 11 Run GSSHA

---

1. Select the **2-D Grid Module** .
2. Select *GSSHA / Run GSSHA...* to bring up the *GSSHA Run Options* dialog.
3. Turn off the *Suppress screen printing* option.
4. Select **OK** to close the *GSSHA Run Options* dialog and bring up the *Model Wrapper* dialog.

The discharge at each time step is displayed in the model wrapper.

5. Click **Close** when the simulation is complete to close the *Model Wrapper* dialog. WMS will read in the solution files.


## 12 Post-Processing and Visualization

---


This section of the tutorial describes different methods of viewing and visualizing GSSHA model results. GSSHA outputs multiple solution files including an outlet hydrograph file, a summary file, and solution datasets, which appear in two separate folders in the WMS Project Explorer. All 2D grid datasets are displayed in the solution folder that is part of the GSSHA model in the 2D Grid Data section of the Project Explorer. The link/node datasets that GSSHA outputs when channel routing occurs appear in a different solution folder in the 2D Scatter Data section of the Project Explorer.

### 12.1 Outlet Hydrograph

---

1. Choose the **Select hydrographs**  tool.
2. Double-click on the hydrograph icon that is displayed next to the outlet grid cell in the Graphics Window to open the *Hydrograph* dialog.




This opens the runoff hydrograph in a plot window. If desired, export the time series data for the hydrograph by right-clicking on the hydrograph plot window and selecting **Export/Print...**

3. Close the *Hydrograph* plot window by clicking on the X in the upper right corner of the window.
4. Select *Display | Display Options...*  to bring up the *Display Options* dialog.

5. In the *2D Grid Data* tab turn off the display of everything except *Cells* and *Contours*.
6. In the *Hydrologic Modeling Data* tab turn off the display of *Hydrograph Icons*.
7. Select **OK** to close the *Display Options* dialog.

## 12.2 Summary File

---




1. In the “ 2D Grid Data” folder of the Project Explorer, expand the “ ParkCity (GSSHA)” model folder, if necessary.
2. Double-click on “ Summary File” to open the *View Data File* dialog.
3. Click **OK** to view the file in the selected application.

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. Close the summary file when done viewing it.





## 12.3 Grid Depth Contours


---

1. Turn off the “ Coverages” folder in the Project Explorer.
2. Under “ ParkCity (GSSHA)”, right-click on the “ depth” dataset in the Project Explorer and select **Contour Options...** to bring up the *depth Contour Options* dialog.
3. Change the *Contour Method* to “Color Fill”.
4. Click on the **Legend...** button to bring up the *Contour Legend Options* dialog.
5. Turn on the option to *Display legend*.
6. Select **OK** to close the *Contour Legend Options* dialog.
7. Select **OK** to close the *depth Contour Options* dialog.
8. Click on the first time step shown in the *Properties* window.
9. Use the down arrow key (on the keyboard) to cycle through the time steps and view the overland flow depth. In this particular model overland flow depth values first become evident approximately 10 hours after the start of the simulation at 10:00 PM on 01/01/2008. Skip ahead to this time step to view depth values.

## 12.4 Stream Depth Contours

---

1. Select the “ Stream depth” dataset in the “ ParkCity (GSSHA)” solution folder under the “ 2D Scatter Data” folder in the Project Explorer.
2. Select *Display / Display Options...*  to bring up the *Display Options* dialog.
3. Turn off *Auto z-mag* and set the *Z magnification* to “1.0”, on the bottom left-hand side of the dialog.
4. In the *2D Scatter Data* tab turn off the *Symbols* option and turn on the *Contours* option.
5. Change the *Radius* to “20”.

6. Set the *Z magnification* to “750” (this only applies to the link/node stream depth contours).
7. On the *2D Grid Data* tab under *Cells*, select *Smoothed*.
8. Select **OK** to close the *Display Options* dialog.
9. Use the  **Rotate** tool to position the grid so that it appears similar to Figure 4. If necessary, adjust the scatter setting radius and z-magnification display settings to make the display clearer.

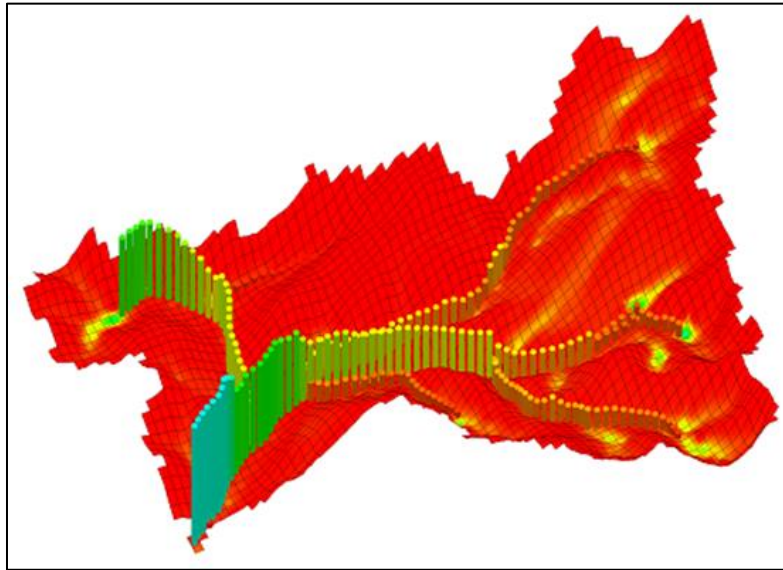


Figure 4 Rotated watershed

10. Click on the first time step shown in the *Properties* window.
11. Use the down arrow key (on the keyboard) to cycle through the time steps and view the channel flow depth. In this particular model channel flow depth values first become evident approximately 10 hours after the start of the simulation at 10:00 PM on 01/01/2008. Skip ahead to this time step to view depth values. Notice the changing bar diagrams. These bars represent the flood wave magnitude along the stream at each time step.

Figure 5 shows the overland flow depth contours changing simultaneously with the flood wave bar diagrams. This gives a clearer idea of the natural process of overland flow.

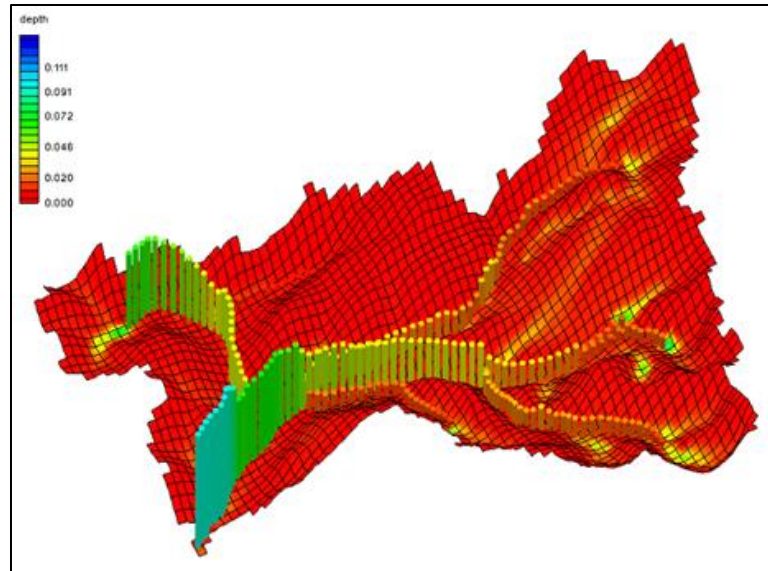




Figure 5 Grid and stream depth contours

## 12.5 Creating an Animation Film Loop

These processes can also be visualized in a film loop. WMS creates two types of file loops. One type is an MP4 file and the other type can be viewed in Google Earth so that the flood animation is displayed over a Google Earth image.

1. Select *Display / View / Plan View* .
2. Under "ParkCity (GSSHA)", right-click on the "depth" dataset in the Project Explorer and select **Contour Options...** to bring up the *depth Contour Options* dialog.
3. Change the *Contour Method* to "Normal Linear".
4. Select **OK** to close the *depth Contour Options* dialog.
5. In the **2-D Grid Module**  select *Data / Film Loop...* to bring up the *Film Loop Setup - General Options* dialog.
6. Choose the *Create New Filmloop* option.
7. Under *Select Film Loop Type* select the *Scalar/Vector Animation* option and click **Next >** to proceed to *Film Loop Setup – Time Step Options*.
8. Change *Run Simulation from* to "9:00:10 PM" to "11:30:10 AM".
9. For *Number of Frames* enter 30.
10. Click **Next >** to proceed to *Film Loop Setup – Display Options*.
11. Click **Finish** to close the *Film Loop Setup – Display Options* dialog (do not change the clock options).

WMS will take some time to create the film loop. It will automatically begin playing once it is completely saved. By default, the directory for the video files is in the same folder as the project file. This can be changed using the options under *Filename* in the first step of the *Film Loop Setup* dialog. If the data seems somewhat jumpy while scrolling through the time steps (or making a movie) this is because output time step is too large.

## 13 Conclusion

---

This exercise demonstrated how to set up a basic GSSHA model, including channel routing and how to use the Hydrologic Modeling Wizard. The wizard guided through the processes of creating a 2D grid, entering GSSHA job control data, defining and smoothing streams, and developing hydrologic input parameters. Options for post-processing GSSHA model results were also introduced.