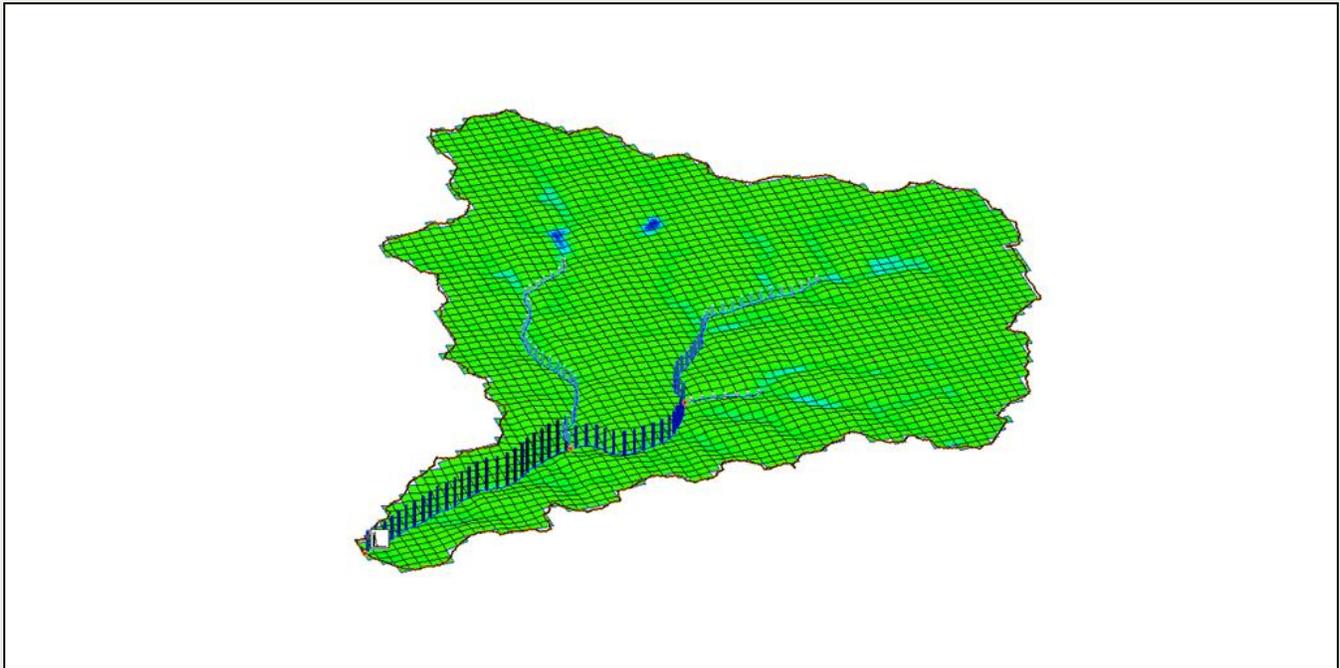




WMS 11.4 Tutorial

GSSHA Initial Overland Flow Model Setup

Set up and run a basic GSSHA model with overland flow



Objectives

Learn how to delineate a watershed and then set up and run a GSSHA model with a simple overland flow simulation *without* using the Hydrologic Modeling Wizard.

Prerequisite Tutorials

- Images
- DEM Basics
- Projections / Coordinate Systems

Required Components

- WMS Core
- GSSHA Model

Time

- 15–30 minutes

1	Introduction.....	2
2	Importing DEM Data	2
3	Computing Flow Directions and Flow Accumulations	2
4	Delineating the Watershed.....	3
5	2D Grid Generation.....	4
6	Job Control Setup.....	5
7	Uniform Index Map Setup	6
8	Roughness Table Setup.....	6
9	Defining Uniform Precipitation.....	7
10	Save and Run the GSSHA Model.....	7
11	Visualizing Overland Flow Results	7
11.1	Visualizing the Outlet Hydrograph	7
11.2	Examining the Summary File.....	8
11.3	Visualizing Depth Contours	9
12	Conclusion	10

1 Introduction

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

2 Importing DEM Data

The process of downloading a DEM using web services was covered in the previous tutorial. This tutorial will use a DEM which has already been downloaded.

1. Open a new instance of WMS.
2. Select *File* | **Open...** to bring up the *Open* dialog.
3. Browse to *GSSHAModeling1* and select "start.wms" then click **Open** to import the starting project and exit the *Open* dialog.

When opening a DEM, it is often necessary to reproject the DEM to UTM coordinates in order to run TOPAZ. For the purposes of this tutorial, the projections have already been set for the project.

3 Computing Flow Directions and Flow Accumulations

Start with computing the flow directions and flow accumulations by doing the following:

1. Select the **Drainage Module** .
2. Select *DEM* | **Compute Flow Direction/Accumulation...** to open the *Flow Direction/Accumulation Run Options* dialog.
3. Click **OK** to accept the defaults and close the *Flow Direction/Accumulation Run Options* dialog and open the *Units* dialog.

4. Make certain *Basin Areas* is set to “Square miles” and click **OK** to close the *Units* dialog. This will open the *Model Wrapper* dialog and TOPAZ will begin to run.

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

5. Once TOPAZ finishes running, click **Close** to exit the *Model Wrapper* dialog.
6. Select the **Create outlet point**  button. Click where the outlet is displayed in Figure 1 below to place an outlet at this location.

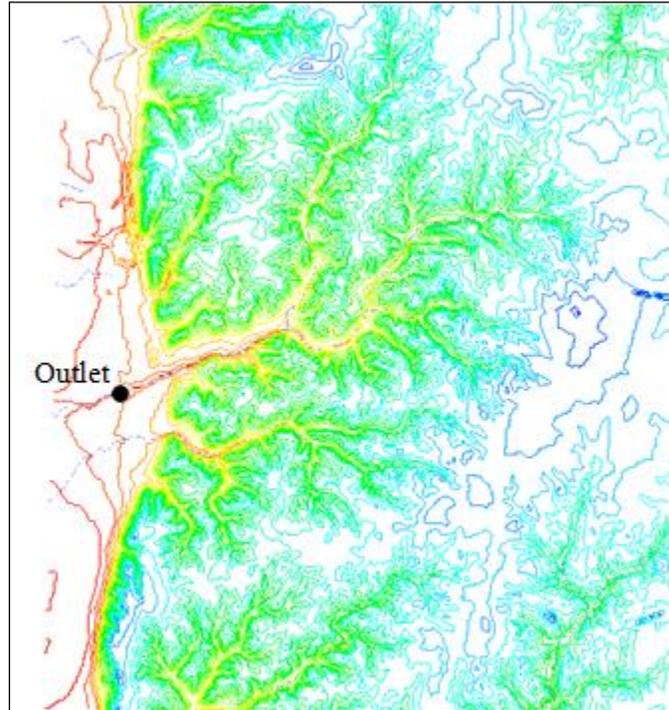


Figure 1 Outlet location

4 Delineating the Watershed

With the flow accumulation computed the watershed can be delineated by doing the following:

1. Select *DEM | Delineate Basins Wizard* to open the *Stream Feature Arc Options* dialog.
2. Set the *Threshold* value to “1”.
3. Click **OK** to close the *Stream Feature Arc Options* dialog. This will open the *Units* dialog.

The delineate basin wizard automates the steps in the previous tutorial of converting the DEM streams to arcs, defining the basin from the DEM, converting the basin boundary to a polygon, and then computing the basin data. The “Developing a GSSHA Model Using the Hydrologic Modeling Wizard” tutorial covers how to run the Hydrologic Modeling Wizard as a guide to GSSHA model set up.

4. Click **OK** to close the *Units* dialog.

The watershed should now look like .

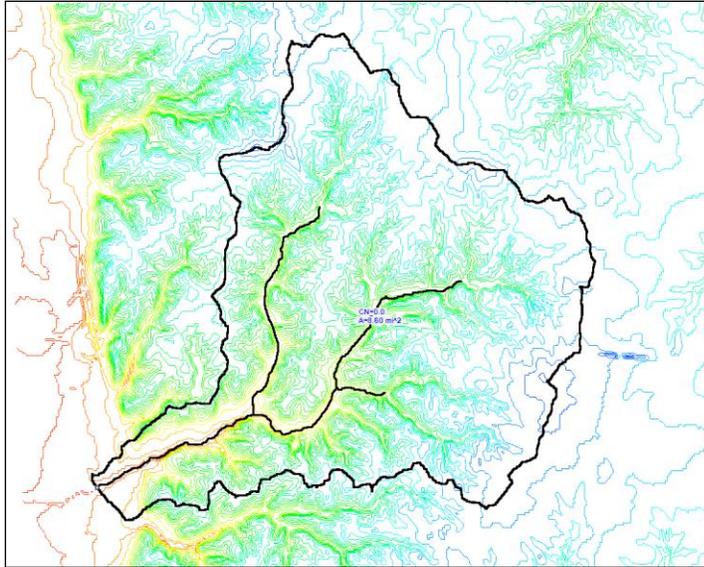


Figure 2 Delineated watershed

5. Click **File** | **Save As** to open the *Save As* dialog.
6. Browse to *GSSHAModeling1*.
7. Enter “JudysBase.wms” as the *File name* and click **Save** to save the project under the new name and exit the *Save As* dialog.

5 2D Grid Generation

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

(http://gsshawiki.com/index.php?title=GSSHA_Software_Primer)

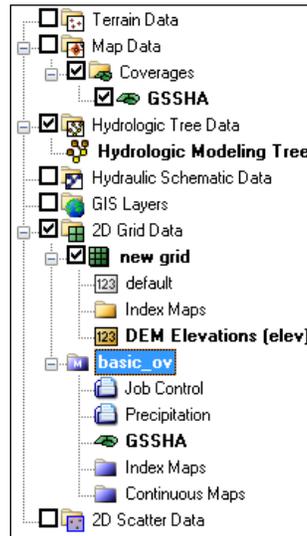
1. Switch to the **Map Module** .
2. Click on the **Select Feature Polygon**  tool, right-click anywhere within the watershed polygon and select  **Create Grid....**
3. Select **Yes** to create a grid for GSSHA. This will open the *Create Grid* dialog.
4. Set the *Base cell size* to a value of “90”.
5. Click **OK** to exit the *Create Grid* dialog and open the *Background Elev Interpolation* dialog.
6. Click **OK** to accept the defaults and exit the *Background Elev Interpolation* dialog.
7. Click **Yes** to delete the existing DEM.

The 2D grid representation of the watershed is now visible.

8. Switch to the **2-D Grid Module** .

9. Select *GSSHA* | **Save Project File...** to open the *Save GSSHA Project File* dialog.
10. Navigate to *GSSHAModeling1*.
11. Enter “basic_ov.prj” as the *File name* and click **Save** to save the GSSHA project file and exit the *Save GSSHA Project File* dialog.

Notice in the Project Explorer that a 2D Grid named “ new grid” is visible in the tree and a GSSHA model (“ basic_ov”) has been initialized.



6 Job Control Setup

When creating a GSSHA grid, the *GSSHA Job Control Parameters* were initialized using some default values. To get the overland flow running, it is necessary to define some realistic values.

1. Select *GSSHA* | **Job Control...** to open the *GSSHA Job Control Parameters* dialog.
2. Under *Outlet information*, ensure the *Slope* is set to “0.001”. This ensures water will run off the grid.
3. Under *Computation parameters*, for *Total time (min)* enter “1500”, and for *Time step (sec)* enter “10”.
4. Click on **Output Control...** to open the *GSSHA Output Control* dialog.
5. Under the *Hydrograph* section, set the *Output units* to *English (cfs)*.

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

6. Under the *Hydrograph* section, for the *Write frequency*, enter “15” (min).
7. Select **OK** to exit the *GSSHA Output Control* dialog.
8. Click **OK** to exit the *GSSHA Job Control Parameters* dialog.

7 Uniform Index Map Setup

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

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

Notice that an “ Index Maps” folder was created under the GSSHA model in the Project Explorer. A grid may have several index maps as was demonstrated in a previous tutorial.

1. Select *GSSHA | Maps...* to bring up the *GSSHA Maps* dialog.
2. Select the **Data Calculator...** button to open the *Data Calculator* dialog.
3. Enter “1” for *Expression*.
4. In the *Result* box, enter “Uniform”.
5. Turn *Index map* on.
6. Click **Compute**.
7. If asked to select cells or nodes, select **Cells**.
8. Select **Done** to exit the *Data Calculator* dialog and return to the *GSSHA Maps* dialog.
9. Click **Done** to exit the *GSSHA Maps* dialog.

Notice that the “ Uniform” index map is assigned to two different places in the Project Explorer: one in the “ Index Maps” folder under the 2D grid and the other under the GSSHA model (“ basic_ov”).

8 Roughness Table Setup

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

1. Select *GSSHA | Map Tables...* to open the *GSSHA Map Table Editor* dialog.
2. Using the *Using index map* drop-down menu, select “Uniform”.
3. Click the **Generate IDs** button.
4. In the *Surface roughness* edit field, enter a value of “0.05”.
5. Click **Done** to close the *GSSHA Map Table Editor* dialog.

The tabs on the *GSSHA Map Table Editor* dialog list some of the mapping tables that can be set up for different watershed processes that can be modeled in GSSHA. Other processes will be set up in later tutorials. The two steps for assigning values for each cell are first to assign index map IDs to the grid cells using an index map (in the case of roughness, a uniform index map was used) and then to assign roughness values to each ID used in the index map in the *GSSHA Map Table Editor*. For this simple model with

uniform roughness, the use of an index map may seem like extra work with selecting all the cells in the model and assigning the same roughness value to each cell. However, when there are several land use or soil type characteristics representing variations in roughness values, infiltration, and other spatial properties, it is essential to manage the GSSHA model data using index maps to represent the spatial variation and then to assign specific values to the index map IDs in the mapping tables.

9 Defining Uniform Precipitation

In addition to roughness, precipitation must be defined to evaluate a basic overland flow model. GSSHA has several options for defining precipitation, which will be explored in a later tutorial. The purpose of this tutorial is to get enough water on the grid to examine and correct any problems with overland flow.

1. Select **GSSHA | Precipitation...** to open the *GSSHA Precipitation* dialog.
2. Enter an *Intensity (mm/hr)* of “40”.
3. Enter a *Duration (min)* of “120”.
4. Select **OK** to close the *GSSHA Precipitation* dialog.

10 Save and Run the GSSHA Model

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

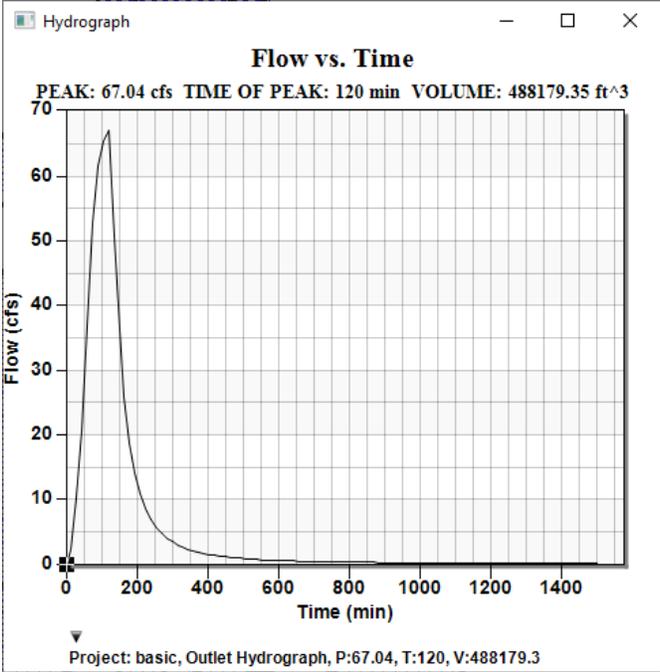
1. Select **GSSHA | Save Project File...** to bring up the *Save GSSHA Project File* dialog.
2. Enter “basic.prj” as the *File name* and **Save** the project to exit the *Save GSSHA Project File* dialog.
3. Select **GSSHA | Run GSSHA...** to open the *GSSHA Run Options* dialog.
4. Turn the *Suppress screen printing* option off.
5. Select **OK** to close the *GSSHA Run Options* dialog and open the *Model Wrapper* dialog.
6. Once the simulation has finished running, click **Close** to exit the *Model Wrapper* dialog and read in the solution.

11 Visualizing Overland Flow Results

Notice that after closing the *Model Wrapper*, WMS automatically read in some files. WMS stores the results of a run together as a solution set. There can be many solution sets in the Project Explorer, but they must be for the same 2D grid. These solution sets are useful because modifications can be made to job control parameters, index maps, boundary conditions, or mapping tables and used to compare results between model runs.

11.1 Visualizing the Outlet Hydrograph

As soon as the results are read, notice a small hydrograph icon at the outlet of the watershed. This icon is used to access the plot of the outflow hydrograph.

1. Click on the **Select hydrographs**  tool and double-click on the small hydrograph icon  near the outlet to bring up the *Hydrograph* dialog. The hydrograph should appear similar to  Figure 3 below.
2. Close  the *Hydrograph* dialog when finished viewing.

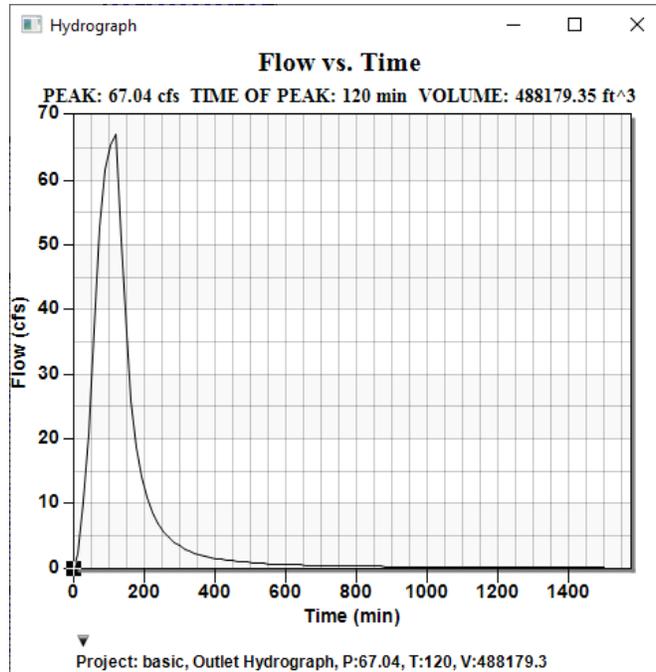


Figure 3 Runoff hydrograph

11.2 Examining the Summary File

A summary file was generated with the GSSHA solution. To view this:

1. Under  "basic (GSSHA)" in the Project Explorer, double-click on  "Summary File" to open the *View Data File* dialog. If the *Never ask this again* option has previously been turned on, then it will skip to the default text editor.
2. Select the desired text editor, then click **OK** to exit the *View Data File* dialog and open the Summary File in the selected editor.

Look through the summary file, in particular note the following:

- Total precipitation volume
- Total discharge
- Volume remaining on the surface
- Mass conservation error, etc.

Notice from these numbers that most of the water remained on the watershed grid in this simulation instead of running off the grid.

3. When done, close  the summary file.

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

- The simulation did not run long enough,

- And/or there are problems with the elevation grid that result in water ponding on the grid

Next, look at the water depths to examine the surface runoff behavior.

11.3 Visualizing Depth Contours

1. In the **2-D Grid Module** , select *Display* |  **Display Options...** to open the *Display Options* dialog.
2. Turn on *Contours* if it is not already on.
3. Select **OK** to close the *Display Options* dialog.
4. In the Project Explorer, under “ basic (GSSHA)”, right-click on “ depth” and select **Contour Options...** to open the *depth Contour Options* dialog.
5. Make sure the *Contour Method* drop-down is set to “Normal Linear”.
6. Select the **Legend...** button to open the *Contour Legend Options* dialog.
7. Turn on *Display legend*.
8. Click **OK** to close the *Contour Legend Options* dialog.
9. Click **OK** to close the *depth Contour Options* dialog.

In the *Properties* window (normally to the right side of the WMS screen), a set of time steps appear.

10. In the *Properties* window on the right side of the WMS application, click on the first time step and use the down arrow key (on the keyboard) to click through the time steps. Notice the depth contours varying in color as the time steps change.

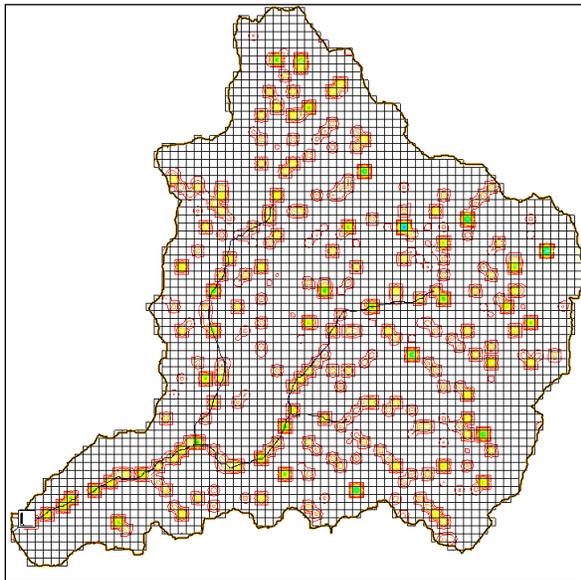


Figure 4 Contoured grid showing digital dams.

It is fairly easy to see that the water has ponded on the grid (see the circular spots on the grid and in the above figure). These puddles are the result of digital dams. Digital dams are local depressions that exist as a result of a lack of a grid’s elevation resolution. The lack of runoff is not due to a short simulation time but rather due to the presence of digital dams. Each grid cell in the model is 90m x 90m or 8100 meters squared, so a large

amount of water can be retained by each “digital dam” cell and any cells flowing to this cell.

Since not a lot of water runs off the watershed, but ponds in digital dams, this situation should be fixed to correct the model. Often, when there are many digital dams, the simulation will not run to completion. These digital dams are artificial depressions in the 2D Grid caused by a lack of resolution in both the original DEM and the derived grid. Fixing digital dams is the subject of the [“Correcting Overland Flow Problems”](#) tutorial.

12 Conclusion

This concludes the “GSSHA Initial Overland Flow Model Setup” tutorial. This tutorial covered how to delineate a watershed from a DEM, how to set up a GSSHA model, and how to run and view the results of a GSSHA model.