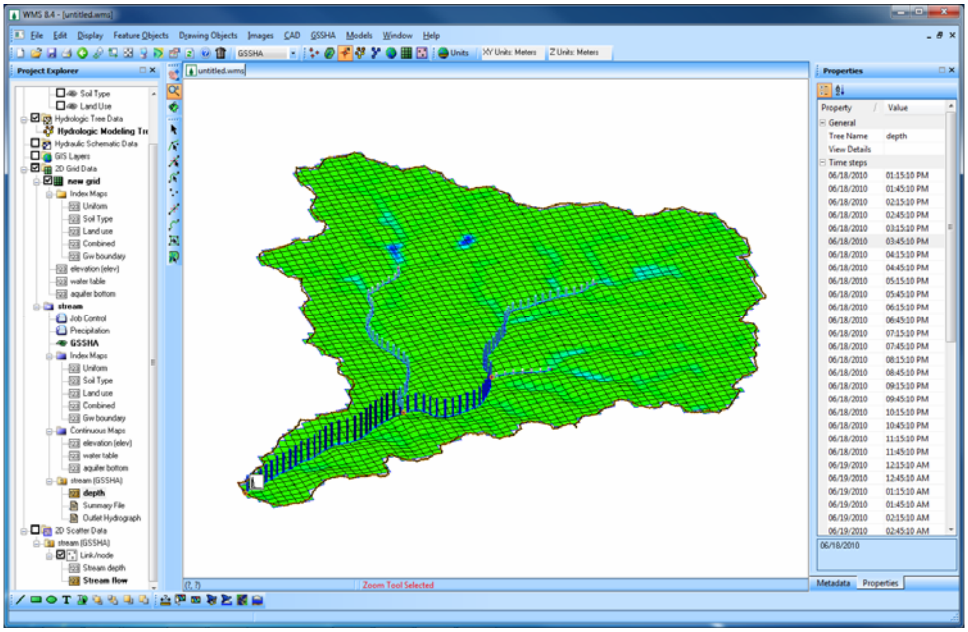
WMS

v. 8.4

WMS 8.4 Tutorial

GSSHA – Modeling Basics – GSSHA Initial Overland Flow Model Setup

Setup and run a basic GSSHA model with overland flow



Objectives

Learn how to delineate a watershed and then setup and run a GSSHA model with a simple overland flow simulation without using the hydrologic modeling wizard.

Prerequisite Tutorials

- GSSHA – WMS Basics – Loading DEMs
- Contour Options
- Images
- Coordinate Systems

Required Components

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

Time

- 30-60 minutes

AQUAVEO™

1 Contents

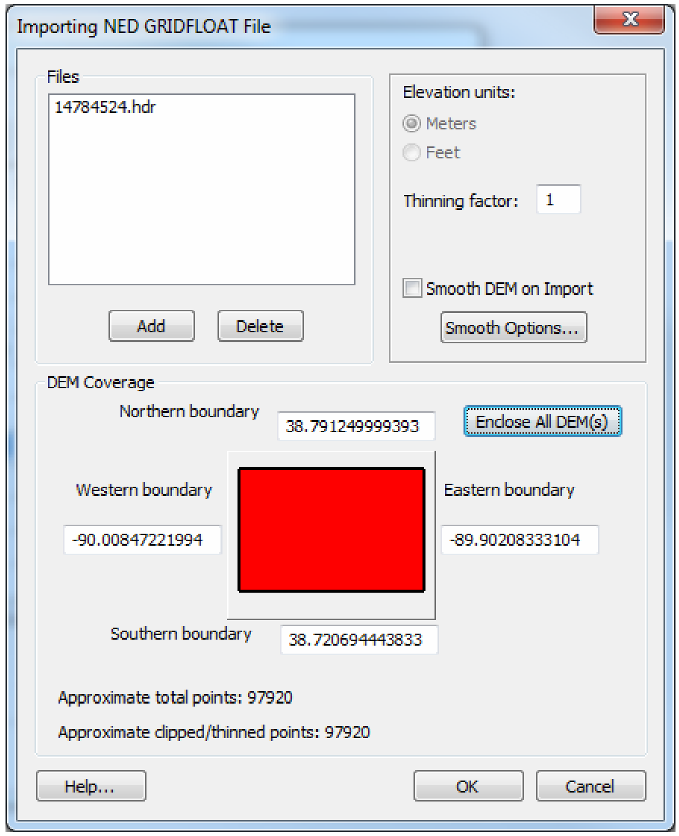
2 Introduction

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 you will use a DEM to create the watershed boundary polygon and the streams using the same delineation process you learned 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 as you would subdivide a lumped parameter model. GSSHA models require that you have a single watershed boundary polygon.

3 Importing DEM Data

You learned how to download a DEM using web services in the previous workshops. In this workshop, you will open a DEM which has already been downloaded.

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



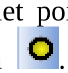
4. Select *Yes* when prompted for coordinate conversion.
5. Set the horizontal *Object Projection* to *Geographic, NAD83, ARC DEGREES* and the vertical *Object Projection* to *NAVD 88(US), Meters* if different.
6. Select *Specify* option for the *Project projection.* Then select *Global projection* and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

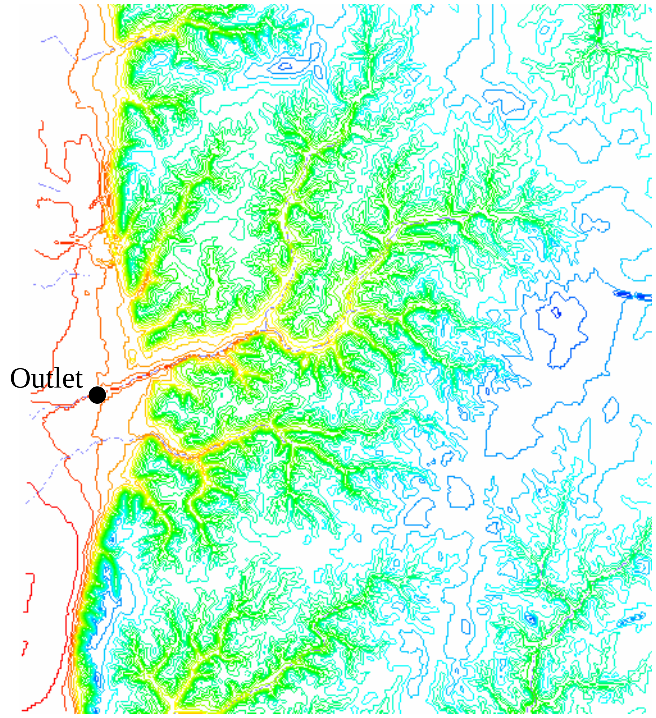
You should now be able to see the DEM contours plotted on WMS main window.

4 Computing Flow Directions and Flow Accumulations

1. In the *Drainage Module* , select ***DEM | Compute TOPAZ Flow Data*** and Click *OK* twice to accept the default TOPAZ input.

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

2. Click on *Close* after the computations are complete.
3. You need to create an outlet point to delineate a watershed. Select the *Create Outlet Point* button . Place the outlet at approximately the location shown in the figure below.

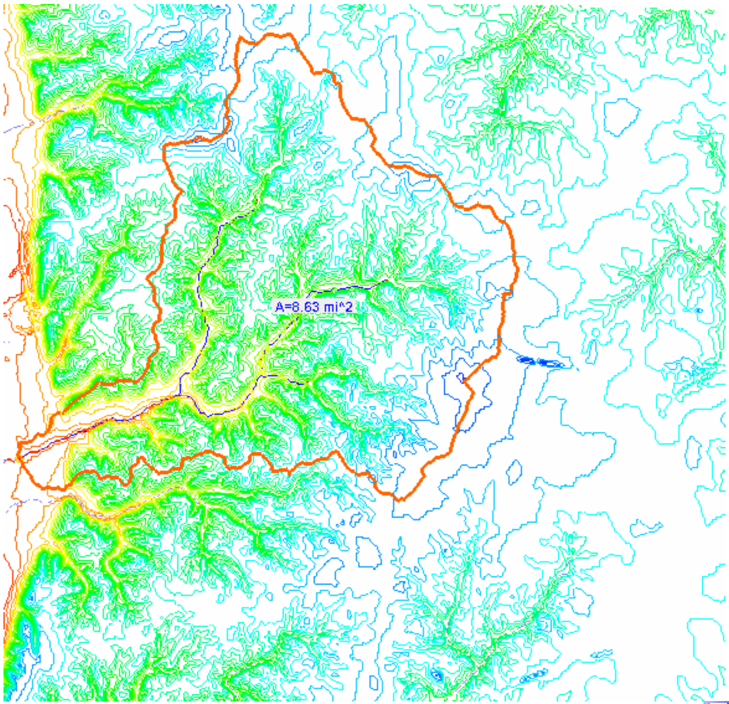


5 Delineating the Watershed

1. Select ***DEM | Delineate Basins Wizard***.
2. Make sure that the Stream Threshold value is 1.0 sq miles. Click the *Delineate Watershed button.*

The delineate basin wizard 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. In a future tutorial, you will learn how to run the hydrologic modeling wizard as a guide to GSSHA model set up.

3. Click *Close* to close the *Delineate Watershed* dialog after the delineation process completes.
4. Your watershed should look somewhat like this:



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

6 2D grid generation

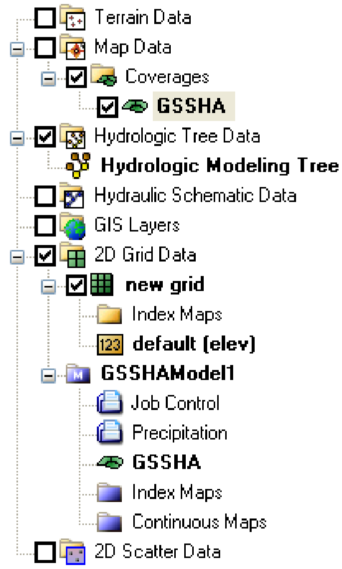
A basic GSSHA model can be generated from a delineated basin boundary in a drainage coverage and a DEM. When delineating your watershed, it is helpful to use a value for the flow accumulation threshold when generating stream arcs from flow accumulation cells so the characteristics of your streams are correctly represented in your GSSHA model. In the following steps, you 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 *Map Module* .
2. Click on *Select Feature Polygon Tool* and right click anywhere within the watershed polygon. Select *Create Grid*. Select *Yes* to create a grid for GSSHA.
3. Make sure that *Base Cell Size* option is checked on and enter 9**0** (meters in this case) as the cell size and click *OK.*
4. Click OK to interpolate grid cell elevation from the DEM, and select *Yes* when prompted to delete the DEM data.

You can now see the 2D grid representation of the watershed. Also notice that under *Coverages* in the data tree, the *Drainage* coverage has been changed to a *GSSHA* coverage.

5. To save your model as a GSSHA project, go to the ***2D Grid module*** and select ***GSSHA | Save Project File***. Save your GSSHA project file to *C:\Training\Personal\BasicGSSHA\ GSSHA\basic_ov.prj***.**

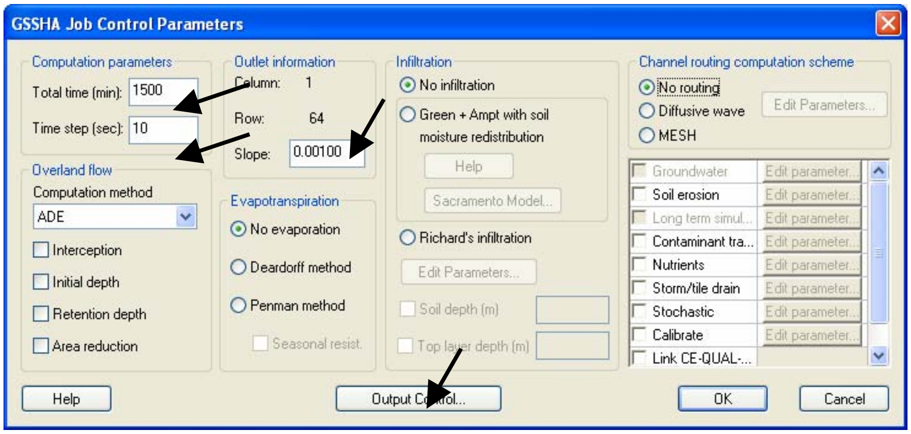
Notice in the project explorer that a 2D Grid called new grid is visible in the tree and a GSSHA model (GSSHAModel1) has been initialized.



7 Job Control Setup

When you created a GSSHA grid, the GSSHA Job Control parameters were initialized using some default values. To get the overland flow running you will need to define some realistic values.

1. In the 2D Grid module select ***GSSHA | Job Control***
2. Make sure the outlet slope is set to 0.001. This ensures water will run off the grid.
3. Enter a *time step* of 10 (seconds) and a *total run time* of 1500 (minutes). Make sure the values are same as those shown in the following figure.



4. Click on *Output Control* and choose your *Output Units* to be ***English*** in the lower part of the Output Control dialog.

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

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

6. Select *OK*

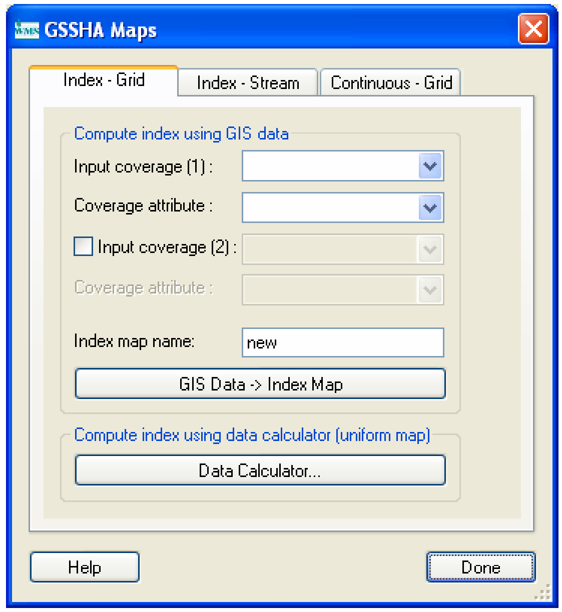
8 Uniform Index Map Setup

Once the Job Control parameters are set to some realistic values, there are two main parameter sets to set up 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.

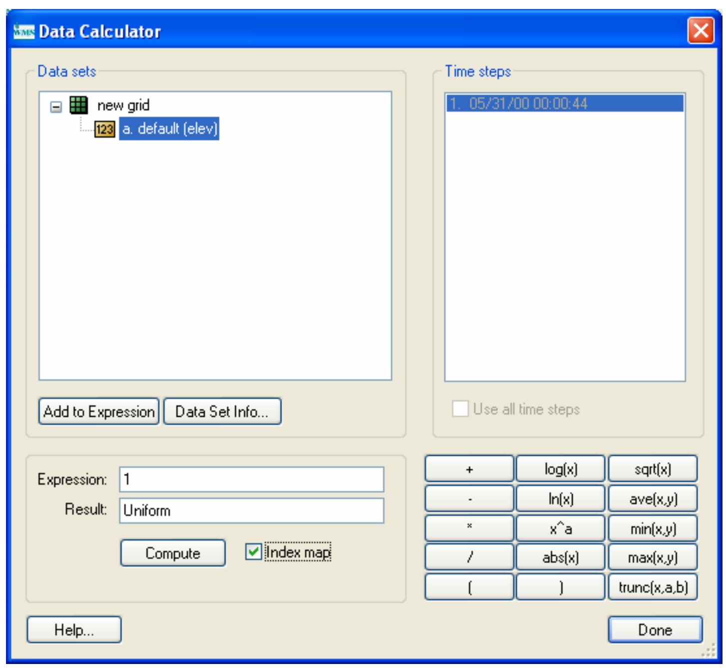
To set up overland flow roughness coefficients, an index map must be set up that describes the spatial variation of the roughness and the roughness values themselves must be assigned in a mapping table. In this example, you will create a spatially uniform set of roughness values so you can examine and 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. In this tutorial, you will add a uniform index map to this folder.

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



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

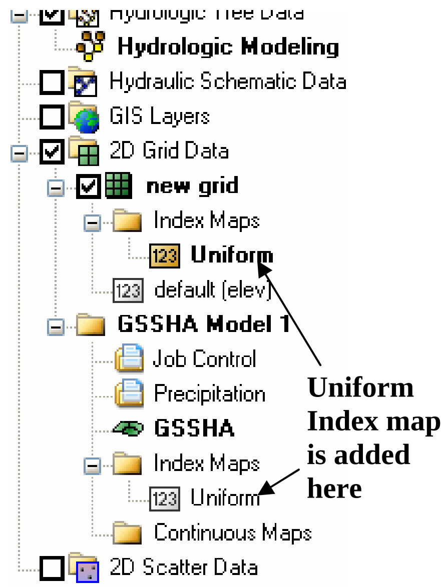


3. In the *Expression* box, type 1.
4. In the *Result* box, type "*Uniform*"
5. Check the *Index map* option
6. Select *Compute*.
7. Select ***Done***. This takes you back to the Index Map dialog.

8. Select ***Done*** again 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 (*GSSHAModel1*).

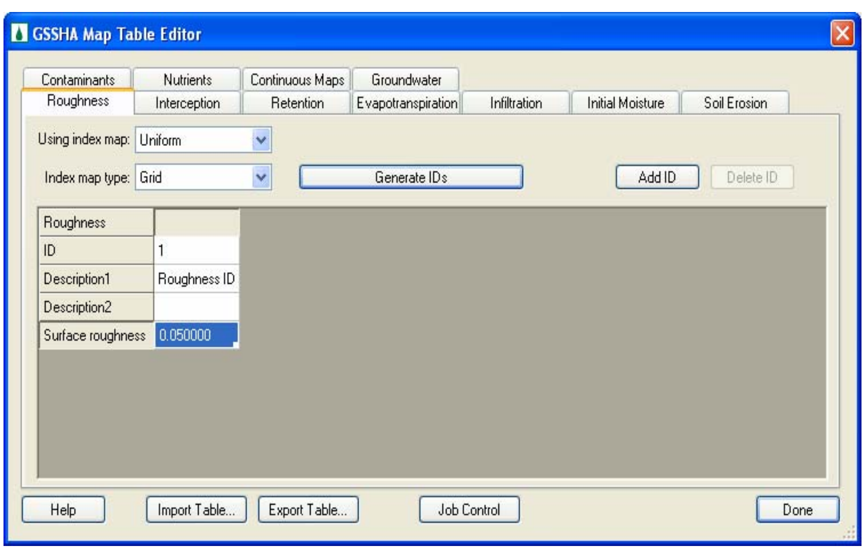
The actual index map is stored in the *Index Maps* folder under the 2D grid and it is also assigned to the active GSSHA model. You can *Assign* or *Remove* index map(s) from a GSSHA model, and if there are multiple GSSHA models, the same index map may be used for each. Similarly, multiple index maps may be used in multiple GSSHA models (or an index map may be removed from all GSSHA models).



9 Roughness Table Setup

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

1. Select ***GSSHA | Map Tables***.
2. Select the *Roughness tab* (it should be selected by default).
3. In the *Using Index Map* combo box select "*Uniform*".
4. Select ***Generate IDs*** button.
5. In the Surface Roughness edit field, enter a value of 0.05.



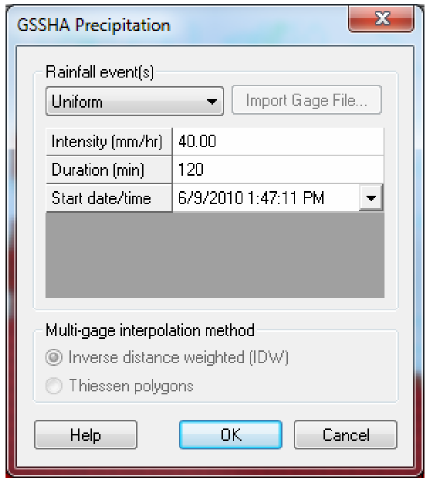
6. Select ***Done***

The tabs on the Mapping Table dialog list some of the mapping tables that can be set up for different watershed processes that can be modeled in GSSHA. We will set up other processes in later tutorials. The two steps for assigning values for each cell are first to assign index map ID's to your grid cells using an index map (In the case of roughness, you used a uniform index map) 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 since you can select all the cells in your model and assign the same roughness value to each cell. However, when you have several land use or soil type characteristics representing variations in roughness values, infiltration, and other spatial properties, it is essential to manage your GSSHA model data using index maps to represent the spatial variation and then to assign specific values to your index map IDs in the mapping tables.

10 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. You will explore these options in a later workshop, but you will begin by setting up a simple uniform precipitation event for a short duration. 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*** …
2. Select the Uniform Rainfall in the Rainfall event(s) combo box.
3. Enter the rainfall intensity of 40 (mm\hr).
4. Enter the rainfall duration of 120 (minutes).
5. If you are simulating an actual event and know the date and time of the storm occurrence, you can enter them, but for this synthetic, uniform storm just use the defaults.
6. Select *OK*.



11 Save 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...***
2. Save the GSSHA project as ***C:\Training\Personal\BasicGSSHA\GSSHA\basic_ov.prj.***

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

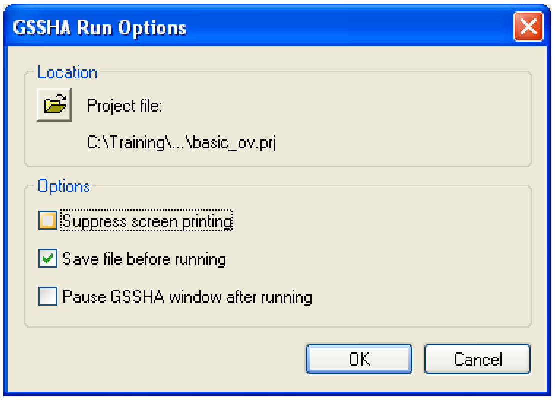
Extension	***Description***
prj	Project file
ele	Elevation file
msk	Watershed mask
cmt	Mapping table file
cif	Channel input file
gst	Grid-Stream file
idx	Index map
dep	Overland depth map (output)
cdp	Channel depth file (output)
cdq	Channel discharge file (output)
map	WMS map file (not used by GSSHA)

3. Open Windows Explorer and go to the folder ***C:\Training\Personal\BasicGSSHA\GSSHA.***
4. Open ***basic_ov.prj*** using note pad (***Right click*** on the file, select ***Open With*** and select ***Notepad*** to open)

5. This is the GSSHA project file. This file tells GSSHA which watershed processes to run and lists the input and output files used in the GSSHA project. Notice that there are files listed in this file with similar names but different extensions (as discussed in the above table).
6. Close *Notepad* and *Windows Explorer* and return to *WMS*.

12 Running the Model

1. Select ***GSSHA* | *Run GSSHA***. Turn the *Suppress screen printing* option off.

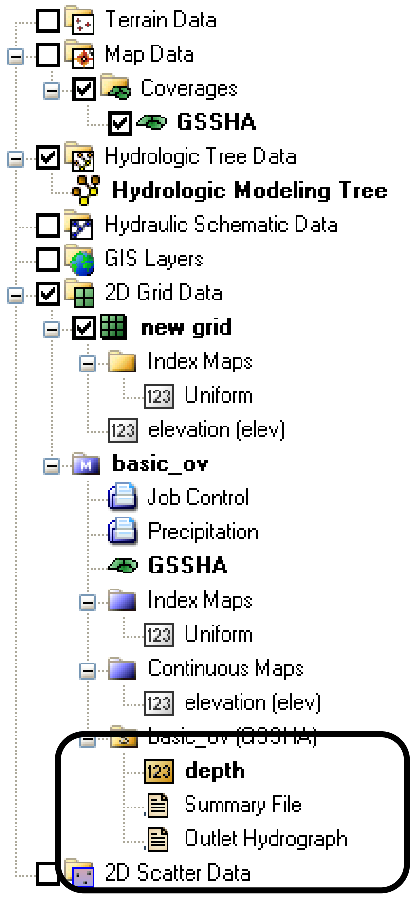


2. Select *OK*.

You will see the time steps being computed and discharge at each time step in the model wrapper. Click "*Close*" after the computation is complete, which will take some time depending on your computer speed.

13 Visualizing Overland Flow Results

Since you have just run your first simulation, it would be nice for you to see what happened! Notice that after you closed 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 you can modify job control parameters, index maps, boundary conditions, or mapping tables and you can compare results between model runs.



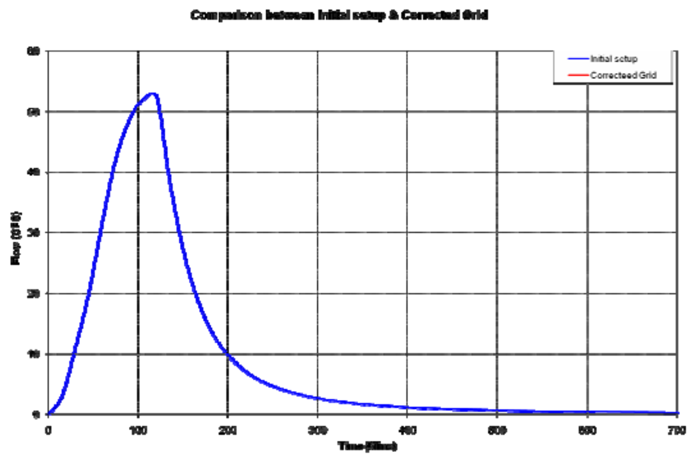
You can see in the above figure that your model is named *basic_ov* and that one of the folders within the model has a letter 'S', representing 'Solution'. The *Depth* dataset under the solution folder represents overland flow depth data. The *Summary file* is a text document that has detailed information about your model inputs, results, mass balances, and errors (if any). The *Outlet Hydrograph* is used to access a plot of the outlet hydrograph.

13.1 Visualizing the Outlet Hydrograph

As soon as the results are read you can see a small hydrograph icon at the outlet of your watershed. This icon is used to access the plot of the outflow hydrograph.

1. In the 2-D grid module, click on the "Select Hydrograph" tool and double click on the small hydrograph icon near the outlet.
2. This will open the hydrograph in a plot window.
3. Right click on the hydrograph plot and choose *View Values*. This will display the hydrograph ordinates and corresponding times. You can copy and paste these data to a spreadsheet program.
4. Select the data values under *Flow (cfs)* column, right click and select *Copy*.

5. Open the spreadsheet ***C:\Training\InitialGSSHAComparison.xls*** and paste the hydrograph ordinates in the *Data* tab, in the *Initial Setup & Corrected Grid* section, under the column *Initial Setup*.
6. When pasting data in the spreadsheet, paste your data in the white areas only. Notice some additional tabs in this spreadsheet that can be used to view the results. Select the *Initial_Corrected* tab to view a plot of your hydrograph.
7. Return to WMS and *Close* the *View Values* and *Hydrograph plot* windows after you are done. Your hydrograph will look something like this:



13.2 Examining the Summary File

1. Double-click on Summary File under the solution folder.
2. If WMS asks for your editor, select Notepad and click *OK*.
3. Look through the summary file, in particular note the following:
    - Total precipitation volume
    - Total discharge
    - Volume remaining on the surface
    - Mass conservation error etc
4. Notice from these numbers that most of the water remained on the watershed grid in this simulation instead of running off the grid.
5. When you are done you can close the summary file.

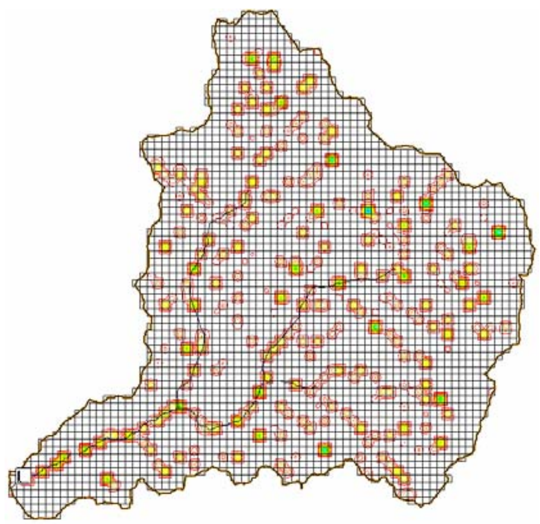
There could be a couple of reasons for the water remaining on the grid instead of being converted to runoff:

- The simulation did not run long enough,
- There are problems with the elevation grid that result in water ponding on the grid, or
- Both of the above reasons

Let's look at the water depths to examine the surface runoff behavior.

13.3 Visualizing Depth Contours

1. In the 2D Grid module , select ***Display | Display Options***
2. Turn on the 2D Grid *Contours*.
3. Select *OK*.
4. In the data tree, right-click on *Depth* under the solution folder.
5. Select *Contour Options*.
6. Make sure the *Contour Method* drop-down box is set to *Normal Linear*.
7. Select the *Legend…* button.
8. Turn on the legend Display.
9. Click *OK*.
10. Click *OK*.
11. In the properties window (normally to the right side of the WMS screen), a set of time steps appear.
12. In the Properties window on the right side of the WMS application, click on first time step and use the down arrow key (on the keyboard) to toggle through the time steps. You can see the depth contours varying in color as the time steps change.



It is fairly easy to see that the water has ponded on the grid (see the circular spots on your 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. Your lack of runoff is not due to a short simulation time but rather due to the presence of digital dams. Each grid cell in your 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, your simulation will not run to completion. The problem is usually due to digital dams. 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 next tutorial.

