

---

***Introduction to Spatial Hydrologic  
Modeling in the US Army Corps of  
Engineers:***

***GSSHA Using the Watershed Modeling  
System***

***Tutorials – Volume 5***

***GROUNDWATER***



*WMS 8.4 Compatible – Volume 5*

Copyright © 2010 Aquaveo, LLC

All Rights Reserved

Unauthorized duplication of the *WMS* software or documentation is strictly prohibited.

AQUAVEO LLC MAKES NO WARRANTIES EITHER EXPRESS OR IMPLIED REGARDING THE PROGRAM *WMS* AND ITS FITNESS FOR ANY PARTICULAR PURPOSE OR THE VALIDITY OF THE INFORMATION CONTAINED IN THIS USER'S MANUAL

The software *WMS* is a product of the Aquaveo, LLC. For more information about this software and related products, contact Aquaveo at:

Aquaveo  
75 South 200 East, Suite 201  
Provo, Utah 84606  
Tel.: (801) 691-5528  
e-mail: [wms@aquaveo.com](mailto:wms@aquaveo.com)  
WWW: <http://www.aquaveo.com/>

For technical support, contact Aquaveo's tech support number at (801) 691-5530 (Monday-Friday, 8am-5pm Mountain Time)

# TABLE OF CONTENTS

<b>1</b>	<b>GROUNDWATER MODELING IN GSSHA</b> .....	<b>1-1</b>
1.1	OPEN AN EXISTING GSSHA PROJECT .....	1-1
1.2	CREATING GROUNDWATER DATASETS .....	1-1
1.3	GROUNDWATER JOB CONTROL .....	1-9
1.4	GROUNDWATER MAP TABLE .....	1-10
1.5	VERIFYING LONG TERM SIMULATION DATA .....	1-10
1.6	SAVING AND RUNNING THE GSSHA MODEL .....	1-10
1.7	VIEWING GROUNDWATER MODEL RESULTS .....	1-11
<b>2</b>	<b>ADVANCED GROUNDWATER MODELING IN GSSHA</b> .....	<b>2-1</b>
2.1	ADDING STREAMS TO THE GROUNDWATER MODEL .....	2-1
2.2	IMPORT A HOTSTART FILE FOR GROUNDWATER ELEVATIONS .....	2-2
2.3	SAVE AND RUN THE MODEL .....	2-3
2.4	ADD WELLS TO THE GROUNDWATER MODEL .....	2-3
2.5	IMPORT HOTSTART FILE FOR GROUNDWATER ELEVATIONS .....	2-5
2.6	SAVE AND RUN THE MODEL .....	2-5
2.7	ADD OUTSIDE BOUNDARY CONDITIONS .....	2-6
2.8	SAVE AND RUN THE MODEL .....	2-8
2.9	CHANGE THE GROUNDWATER MODELING EQUATION .....	2-8
2.10	UPDATE THE MAPPING TABLE FOR RICHARD'S PARAMETERS .....	2-9
2.11	SAVE AND RUN THE MODEL .....	2-10
<b>3</b>	<b>SUBSURFACE TILE AND STORM DRAINS</b> .....	<b>3-1</b>
3.1	OPEN AN EXISTING GSSHA PROJECT .....	3-1
3.2	ADDING STORM DRAINS THE GROUNDWATER MODEL .....	3-1
3.3	SAVE AND RUN THE MODEL .....	3-4
3.4	RESULTS VISUALIZATION .....	3-5
3.5	ADDING SUBSURFACE TILE DRAINS THE GROUNDWATER MODEL .....	3-5
3.6	SAVE AND RUN THE MODEL .....	3-7
3.7	RESULTS VISUALIZATION .....	3-7

---


# Groundwater Modeling in GSSHA

In this tutorial you will learn how to set up a groundwater model in GSSHA. You will begin with an existing long term simulation project with all the parameters for a basic surface water simulation already defined.

## 1.1 Open an Existing GSSHA Project

---

Open a GSSHA project file for the Eight Mile Creek watershed.

1. In the *2D Grid Module*  select *GSSHA | Open Project File...* Browse and open the file *C:\Training\Groundwater\Base.prj*
2. Save the project with different name as *C:\Training\Personal\Groundwater\GW.prj*
3. In the project explorer, toggle off the display of all the map data and toggle back GSSHA coverage on.

## 1.2 Creating Groundwater Datasets

---

To simulate groundwater in a GSSHA model, we will need to define the following parameters:

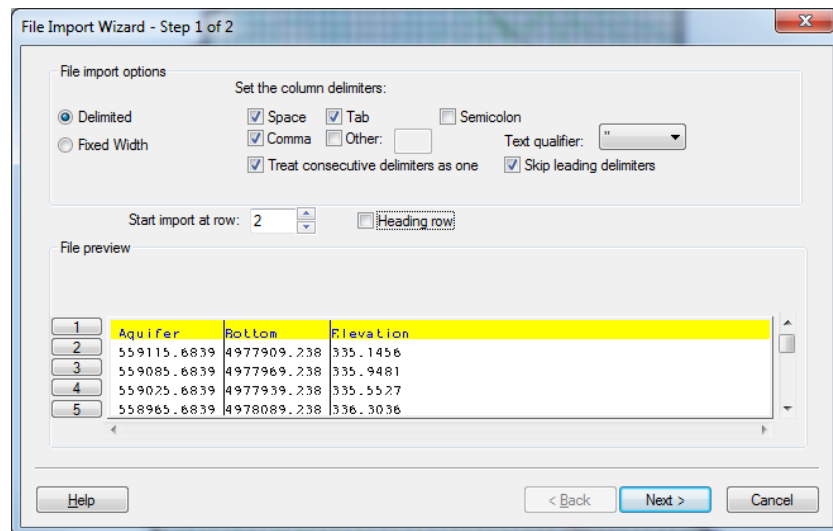
- Aquifer bottom map,
- Initial water table elevation map,
- Boundary condition map
- Subsurface hydraulic conductivity map
- Subsurface porosity map.

We will create the aquifer bottom and water table elevation maps from x,y,z points that we have stored in a .csv file that were derived from borehole logs. The hydraulic conductivity and porosity inputs will be developed using a geology shape file.

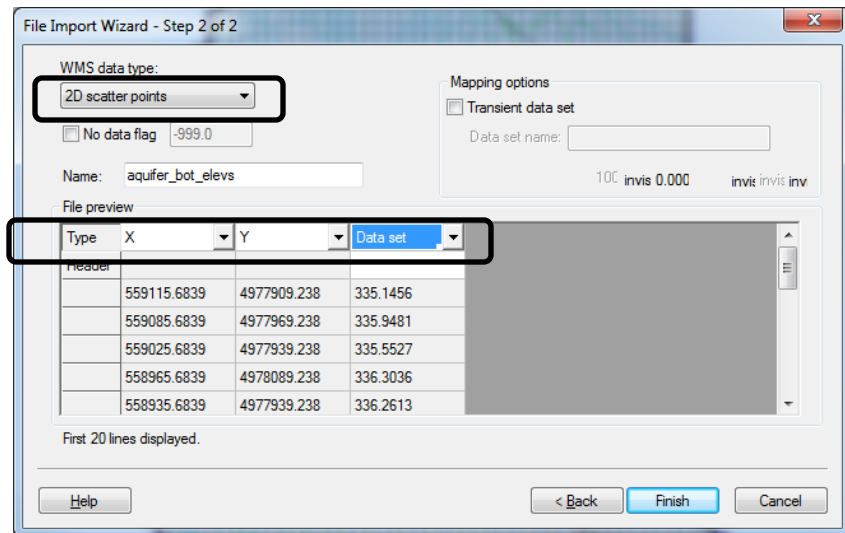
### 1.2.1 Aquifer Bottom

The aquifer bottom map is created from x y z data of the aquifer bottom. These data are typically derived from borehole data that contain information about depth to different layers in the system.

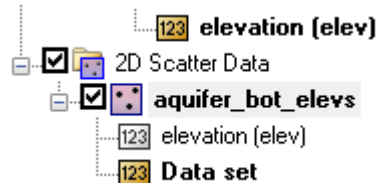
1. Select **File / Open** in the WMS window, browse and open file **C:\Training\RawData\Groundwater\aquifer\_bot\_elevs.csv**
2. In the **File Import Wizard**, change the **starting import at row** to 2. Do not check **Header Row Toggle**



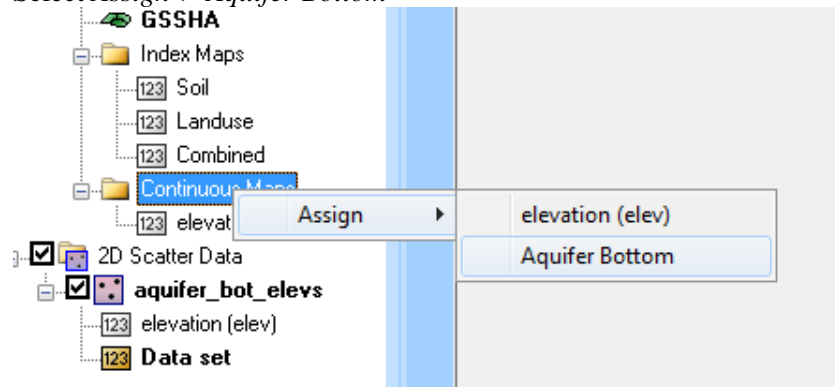
3. Make sure the Comma delimiter is toggled on and click **Next**
4. Make sure the WMS data type is set to **2D Scatter Points**
5. Make sure the first column is mapped to **X**, the second column to **Y**, and the third column to **Data set**



6. Click *Finish* to import the dataset as a 2D Scatter Point dataset
7. A new scatter point dataset named *aquifer\_bot\_elevs* should appear in the Project Explorer and the points are plotted over the watershed.



8. Make sure *Data set* is selected as the active dataset.
9. Right click on the *aquifer\_bot\_elevs* scatter set and choose *Interpolate->...To Grid*
10. Change the interpolated data set name to *Aquifer Bottom*. DO NOT toggle on the option to *Map elevations*.
11. Click OK to create a new grid data set from the aquifer bottom scatter points
12. You should now see a new continuous map named *Aquifer Bottom* among the other continuous datasets for the grid.
13. Right click on the *Continuous Maps* folder underneath the *GW* folder in the 2D Grid Data section of the Project Explorer.
14. Select *Assign-> Aquifer Bottom*

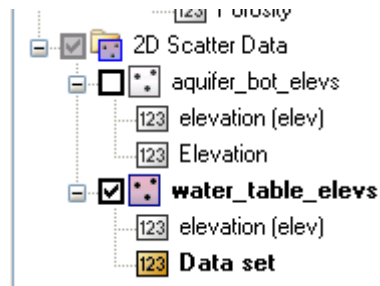


### 1.2.2 Initial Water Table Elevations

---

Ideally, you will have observation well data from which to derive an initial water table. These xyz data can be interpolated to the grid to develop the required map. In lieu of measured values you can develop an initial water table by assuming an initial water table and running the model to steady state to derive a water table map. In this example you will use xyz data that represent well elevations scattered throughout our domain.

1. In the WMS window select **File / Open**, browse and open file **C:\Training\RawData\ Groundwater\water\_table\_elevs.csv**
2. In the *File Import Wizard*, change the *starting import at row* to 2, make sure the Comma delimiter is toggled on and click *Next*.
3. Make sure the *WMS data type* is set to *2D Scatter Points* and map the first column to *X*, the second column to *Y*, and the third column to *Data set*
4. Click *Finish* to import the dataset as a 2D Scatter Point dataset
5. A new scatter point dataset named *water\_table\_elevs* should appear in the Project Explorer and the points are plotted over the watershed.



6. Make sure *Data set* is selected as the active dataset.
7. Right click on the *water\_table\_elevs* scatter set and choose *Interpolate->...To Grid*
8. Change the interpolated data set name to *InitialWaterTable*. DO NOT toggle on the option to Map elevations.
9. Click OK to create a new grid data set from the water table scatter points
10. You should now see a new continuous map named *InitialWaterTable* among the other continuous datasets of the grid.

### 1.2.3 Checking Water Table Elevation Data

---

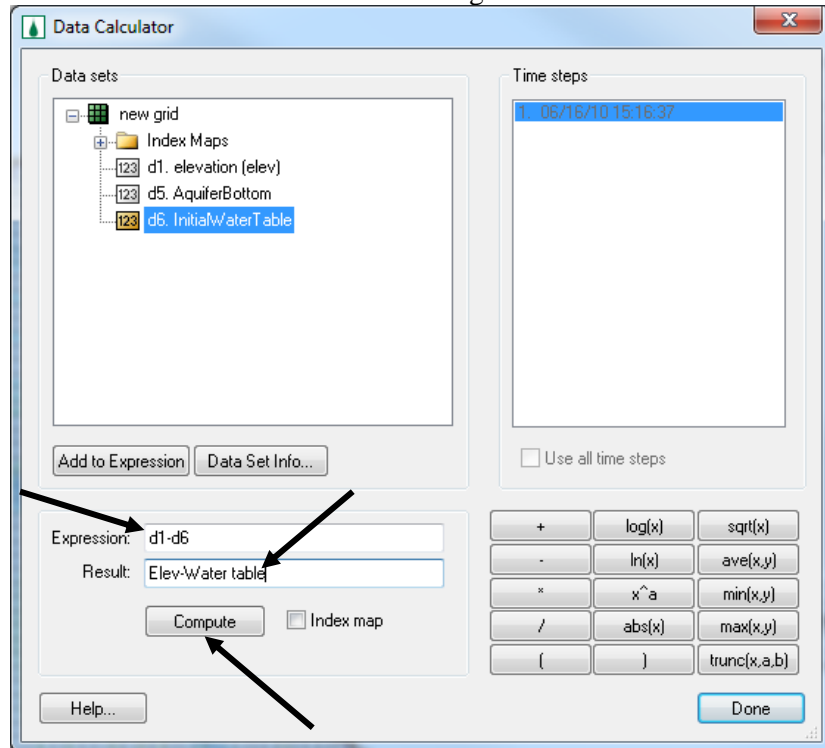
While interpolating the scatter data sets to delineate the water table elevation map, the water table elevation at some locations could be higher than the ground elevation or lower than the aquifer bottom. We will need to check this before using the water table elevation map.

One of the easiest ways to check this is to use the data calculator where you can subtract the water table map from the elevations map or subtract the aquifer bottom from the water table elevations.

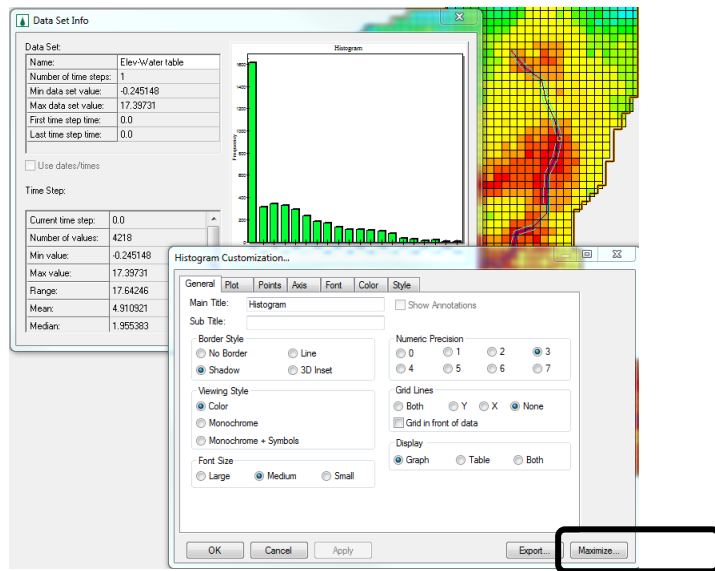
1. In the *2D Grid* module, select **GSSHA/Maps**. Select *Data Calculator*.



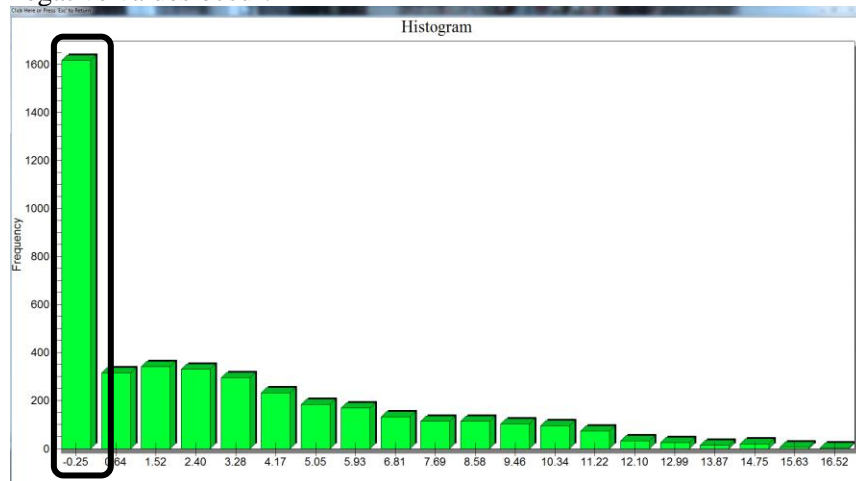
2. Double click *elevation (elev)* data which will insert a symbol such as *d1* in the *Expression* box. Please note that the symbol such as *d1* might be different in your case.
3. Then click the - (minus) button or type it from the keyboard.
4. Double click *InitialWaterTable*. Type *Elev-Water table* in *Result* field.
5. Do NOT select the toggle *Index Map*.
6. Your data calculator will look something like this.



7. Click *Compute*, *Done* and *Done* again.
8. Right click on the *Elev-Water Table* map in the project explorer and select *Properties*.
9. This will open the *Data Set Info* dialog. You can see a Histogram of the values in the data you selected.
10. Double click the plot area and select maximize button in another dialog that opens.



11. In the maximized histogram plot, check to see if there are any negative values. In this example you will have a negative value which means that the water table elevation is greater than ground elevation at these locations. You can change the contour options to display the locations where the negative values occur.



12. Similarly, use the expression *InitialWaterTable - Aquifer Bottom* in the data calculator to see if any cell has water table below the aquifer bottom. Then see the properties of new data set to see if there are any negative value. There are negative values in this case also which means that the water table elevation is lower than the aquifer bottom at some locations.

### 1.2.4 Adjusting the Water Table Elevation Data

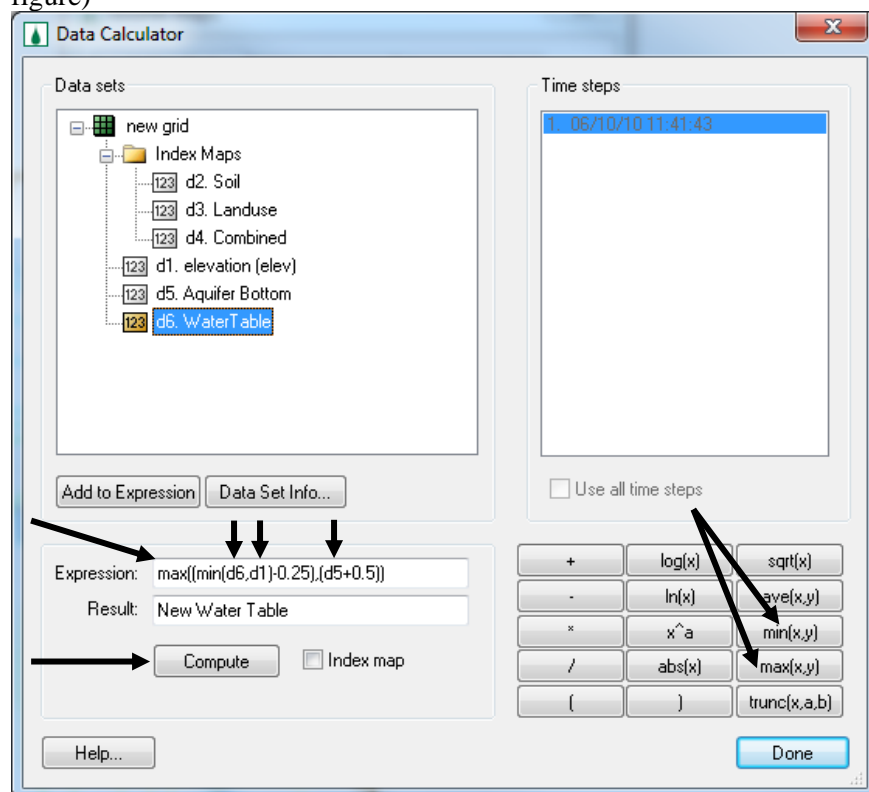
There are different ways to get around this issue. In this exercise, we will use the *max(x,y)* and *min(x,y)* functions available in the data calculator. These functions

will pick the maximum and minimum value from the two datasets and assign it to index map.

1. We want to adjust the water table such that it is
  - At least 0.25m below the ground surface
  - At least 0.5m above the aquifer bottom
2. So, we will have to first use the *min function* to select a minimum between the water table and the ground surface elevation. Then, subtract 0.25m from this so that the water table is at least 0.25m below the ground surface.
3. Then we will use the *max function* between the result from the previous step and the aquifer bottom and add 0.5m to it. This ensures the water table to be at least 0.5m higher than the aquifer bottom
4. You may create a new dataset for each step or use the following formula directly:

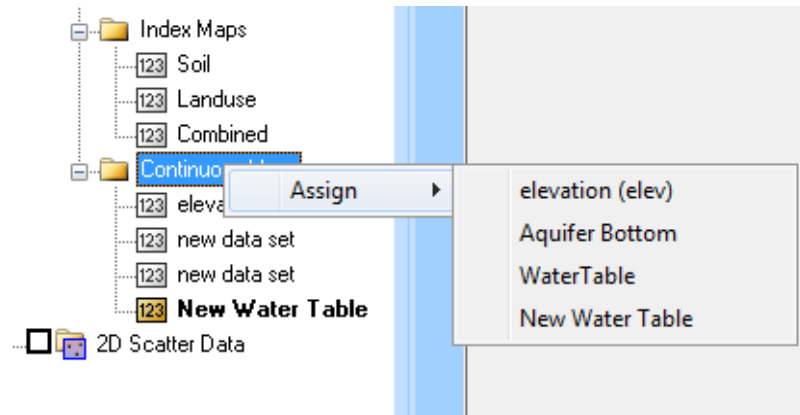
$$\text{Max}((\text{min}(\text{InitialWaterTable}, \text{ground elevation})-0.25), (\text{aquifer bottom} +0.5))$$

5. Open the data calculator.
6. Type equation in the expression field of the figure below into the Expression field.
7. Enter *New Water Table* in the *Result* field. Click *Compute*. (See following figure)



Once you compute the updated water table map, you need to associate the new datasets with the GSSHA model.

8. Right click on the *Continuous Maps* folder underneath the *GW* folder in the 2D Grid Data section of the Project Explorer.
9. Select *Assign-> New Water Table*

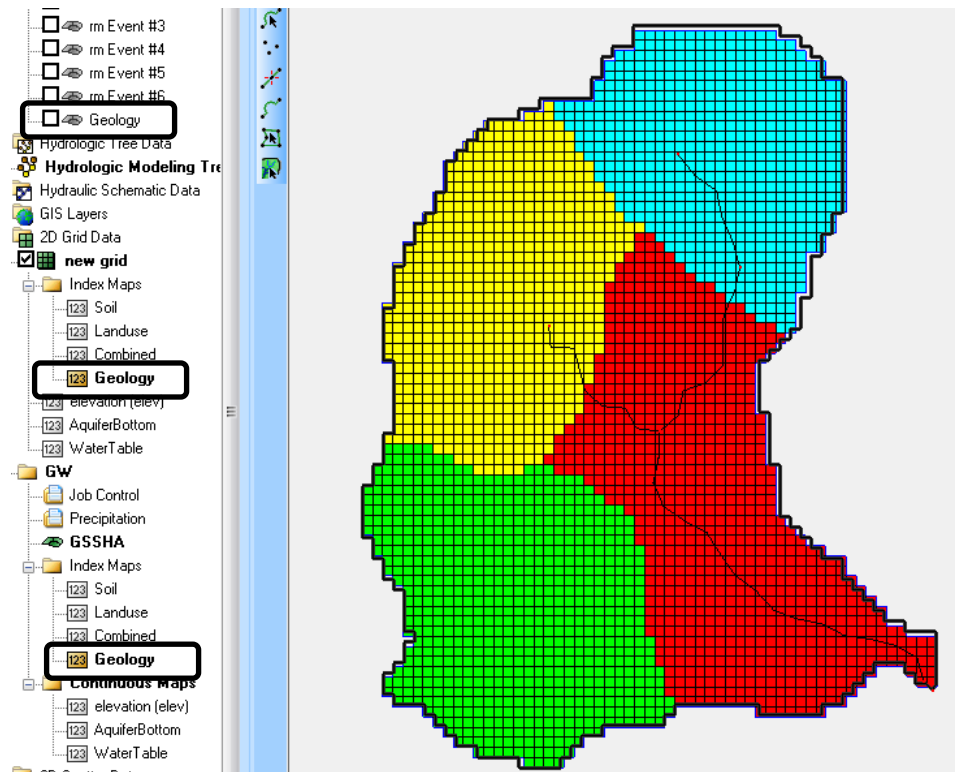


### 1.2.5 Hydraulic Conductivity and Porosity

---

To define the hydraulic conductivity and porosity we will use a geology shapefile which has information about the underground soil type distribution. In GSSHA hydraulic conductivity and porosity are used to derive the values of transmissivity and storage.

1. Create a new *Soil Type Coverage* and change the name to *Geology*.
2. Select **File / Open** and browse to open a shape file **C:\Training\RawData\Groundwater\Geology.shp**
3. Make sure that the *Geology* coverage is selected and *Geology.shp* is active.
4. While, in the *GIS Module*, select **Mapping/Shapes -> Feature objects**
5. Click yes to use all the polygons.
6. Map *Geology.shp* to the *Geology* coverage you created in step 1. In the *GIS to feature object wizard*, use the default mapping and click finish on the wizard. Once done mapping, you may delete the shape file from the project explorer.
7. Select **GSSHA / Maps**
8. Select *Geology* as the input coverage and *Texture* as the coverage attribute. Change the index map name to *Geology* and select “*Coverages->Index Map*”. Then click *Done* to close the *GSSHA Maps* dialog.



### 1.3 Groundwater Job Control

1. In the GSSHA *Job Control*, toggle on the *Groundwater* option and click the *Edit parameter* button which will open the *GSSHA Groundwater* dialog.

**Note:** As you toggle the groundwater option, a groundwater boundary condition index map named *Gw boundary* is created automatically in the project explorer. Also, you should remember that no-flow ground water condition is assumed around the perimeter of the watershed.

2. In the *GSSHA Groundwater* dialog, set the *Aquifer bottom* Data Set to *AquiferBottom* and the *WaterTable* Data Set to *New WaterTable*. We will define the *Hydraulic Conductivity* and the *Porosity* Data from the groundwater mapping table in the next step.
3. Set the Time Step to be 600
4. Make sure the LSOR direction is set to horizontal
5. Make sure the LSOR convergence is 0.00001
6. Make sure the Relaxation coefficient is 1.2
7. Make sure the Leakage rate is set to 0.0
8. Click OK to close *GSSHA Groundwater* dialog
9. Click OK to close the *Job Control* dialog

## 1.4 Groundwater Map Table

---

We have not defined the hydraulic conductivity and porosity for the groundwater flow yet. Since, we are using the geology index map, this will be done in the mapping table.


1. In the GSSHA *Map Tables* table switch to the *Groundwater* tab.
2. Select *Geology* as your index map and generate IDs. You should have seen four different IDs populated. Enter the values as shown in the following table

Parameter	Loamy Sand	Sandy Loam	Loam	Silt Loam
Hydraulic Conductivity	5.98	2.18	1.32	0.68
Porosity	0.437	0.453	0.463	0.501

## 1.5 Verifying Long Term Simulation data

---

Before we run the groundwater model, let us make sure that the long term simulation is properly set.

1. In the GSSHA *Job Control* dialog, click *Edit parameters* for *Long term simulation*.
2. Make sure the *Latitude* is 44.81, *Longitude* is 267.83, *GMT* is -6, *Minimum Event Discharge* is 0.1, *Soil Moisture depth* is 0.25.
3. Click on the browse button  for *HMET Data file*. Browse and select *C:\Training\Groundwater\HMETData.txt*
4. Make sure continuous simulation format is set to *WES*. Click *OK*
5. Click on the *Output Control* button. Check off the toggle next to *Surface Depth* and toggle on the *Groundwater elevations*.
6. Change the *Write frequency* to be 1440 mins for the grid data and leave the *hydrograph write frequency* the same.
7. Click *OK* and *OK* again.

## 1.6 Saving and Running the GSSHA Model

---

We are now ready to save and run the GSSHA model

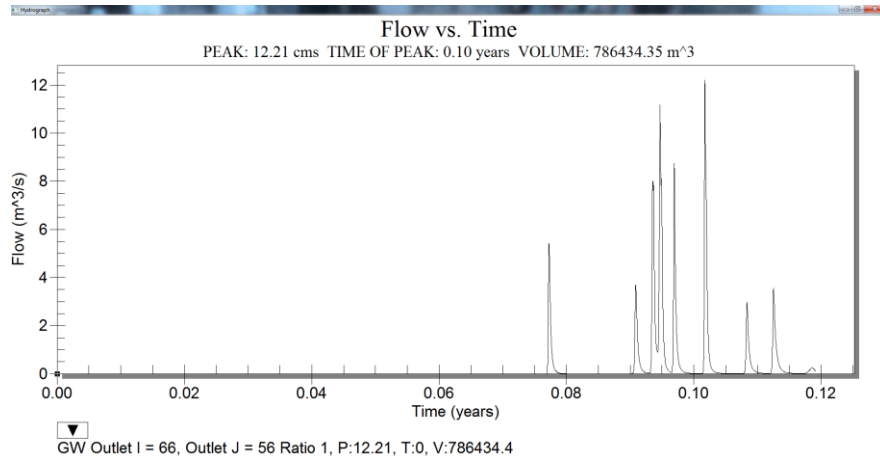
1. Save the project as *C:\Training\Personal\Groundwater\GW.prj*
2. Select *GSSHA/Run GSSHA*

The model should take about 10-15 minutes to run, depending on the speed of your computer.

If you would like to view the solution without waiting for the model to run to completion, click the *Abort* button, select **GSSHA | Read Solution** and open the solution located at *C:\Training\Groundwater\GW.prj*.

## 1.7 Viewing Groundwater Model Results

1. Once the model has finished running, close the model wrapper to read the solution.
2. Visualize the results using different tools such as the hydrograph, plot, summary file etc.



3. Export the outflow hydrograph ordinates to the *Data* sheet under the Basic GW Model column of *C:/Training/Groundwater/GWResults.xls*.
4. Click on *Groundwater elevations* dataset in the project explorer and toggle through the time steps to see how the groundwater head is varying.





---

## ***Advanced Groundwater Modeling in GSSHA***

In the previous workshop you learned how a basic groundwater simulation can be developed in GSSHA. Here you will learn about some of the advanced topics in groundwater simulation.


You will continue working with the same model that you developed in previous workshop. If your model in the previous workshop did not work right or you closed it, you may open it from *C:\Training\Groundwater\GW.prj*

1. Save the project as *C:\Training\Personal\Groundwater\GWAdv.prj*
2. Turn off the display of all the coverages except GSSHA coverage.

### **2.1 Adding Streams to the groundwater model**

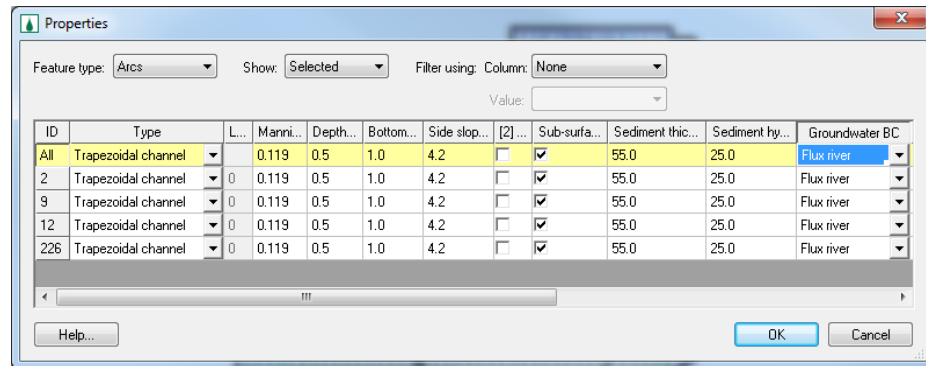
---

The basic groundwater model that you developed in the previous workshop did not have streams turned on. Here we will allow the stream routing and also define a groundwater boundary condition for the streams. Since, a significant amount of water goes into the subsurface flow from the channel, we will also simulate subsurface flow from the channel bottom.

1. Click on the *GSSHA* coverage in the data tree to select it.
2. Click on the *Select Feature Line Branch Tool*  and double click at the downstream most arc of the channel network (the one that is connected to the watershed outlet). This will open the channel *Properties* dialog with all stream arcs selected.

## 2-2 GSSHA Modeling Using WMS – Volume 5

3. For all segments, change the channel type to *Trapezoidal channel*, *Manning's n* to 0.119. *Depth* to 0.5m, *Bottom Width* to 1m and *Side Slope* to 4.2.
4. Check on the *Sub-surface losses/gains* options and enter 55cm for *Sediment Thickness* and 25cm/hr for *sediment hydraulic conductivity*.
5. Change the *Groundwater BC* to *Flux River*. See the following figure.

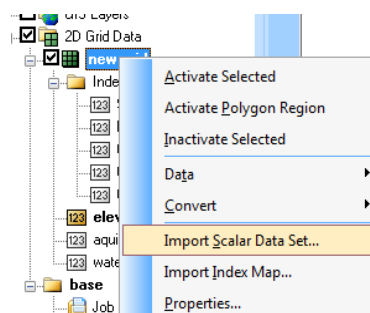


3. Click OK.
4. In the GSSHA *Job Control* dialog, select the *Diffusive Wave* option for channel routing. Click OK to close the *Job Control* dialog.
5. In the project explorer, under *2D Grid Data*, click on *Gw Boundary* index map. As you select this index map, you will see red colored index map on the desktop which represents a generic groundwater BC. This map needs to be regenerated for the river flux boundary condition you just defined on the stream arcs.
6. Right click on the *Gw boundary* index map and select *Regenerate*. You can now see green colored cells under the channel network.

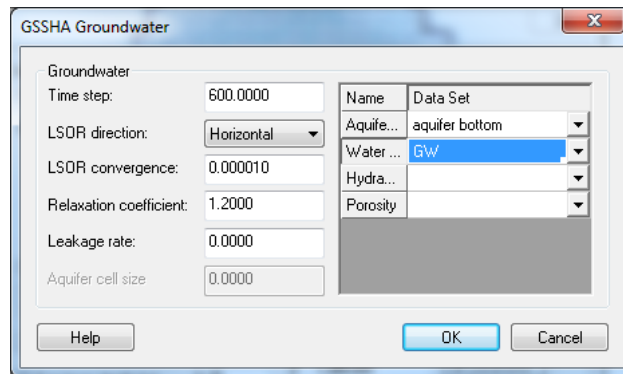
## 2.2 Import a Hotstart file for groundwater elevations

Using the output from one model as an input to another model is called a *Hotstart* in GSSHA. In this workshop, we will use the final groundwater elevations file written out by GSSHA in the previous workshop as a hotstart file for the starting groundwater head.

1. In the project explorer, click on the *2D grid data* to select it.
2. Right click on the *new grid* folder directly under the *2D Grid data* and select *Import Scalar dataset*



3. Select *GRASS ASCII grid file* and click OK.
4. Browse and open file *C:\Training\Personal\Groundwater\GW.wte*. Please note that in the *Open* dialog, you might need to change the file type to *All Files (\*.\*)*.
5. In the project explorer, under *2D Grid data* you should see *GWAdv GSSHA* project. Right click the *Continuous Maps* folder under this model and assign the *GW* dataset.
6. Now you can see the *GW* which you imported as a hotstart file is listed as one of the continuous map.
7. In the *GSSHA Job Control*, Click on *Edit Parameters* button next to *Groundwater*.
8. Assign *GW* for the *Water Table*.



9. Click OK and OK again.

## 2.3 Save and Run the Model

---

1. Save the project as *C:\Training\Personal\Groundwater\GWAdv.prj*
2. Run GSSHA and visualize the results.
5. Once done running, copy the hydrograph ordinates to the *Data* sheet under *With Streams* column of *C:\Training\Groundwater\GWResults.xls*.
3. Compare the results.
4. Step through the maps of groundwater head and see how the water surface changes over the period of your simulation.


## 2.4 Add wells to the groundwater model

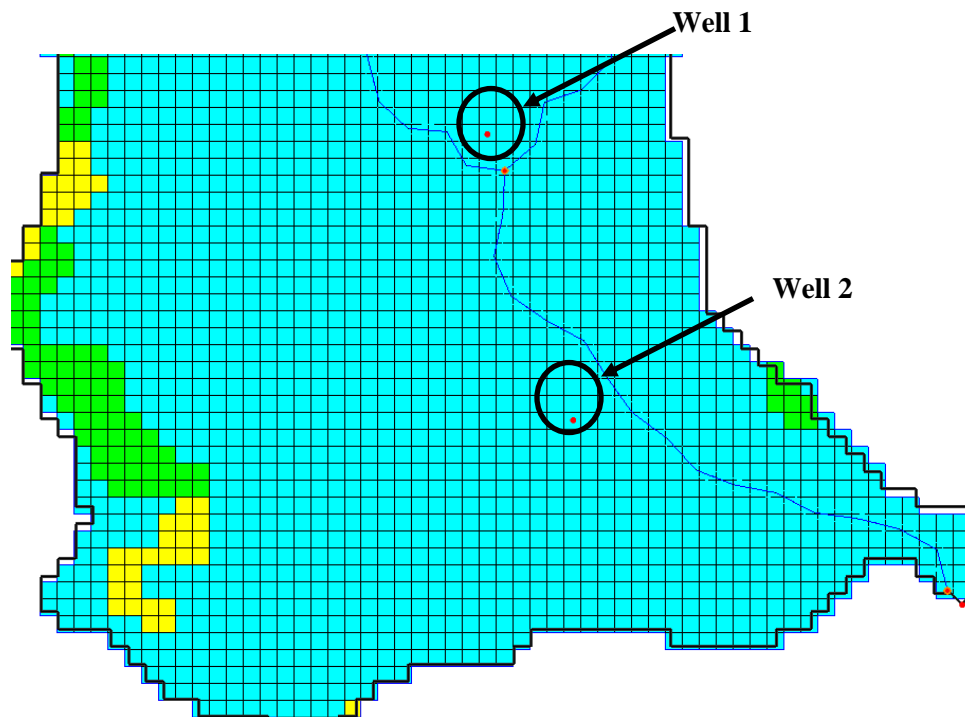
---


GSSHA can be used to simulate the effects of pumping wells on the groundwater heads. We will add two well in the model that we have been working.

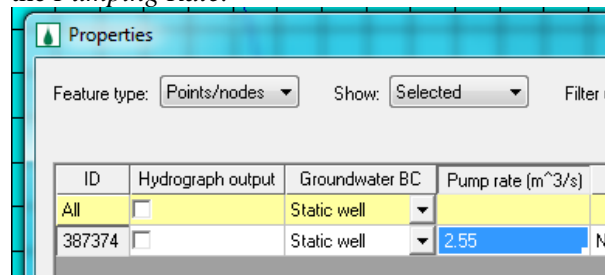
## 2-4 GSSHA Modeling Using WMS – Volume 5

If you closed the project or did not work quite well, you may open it from **C:\Training\Groundwater\GWAdv.prj**.

1. Save the project as **C:\Training\Personal\Groundwater\GWAdvWells.prj**.
2. Turn off the display of all the coverages except the *GSSHA* coverage.
3. In the data tree, click on the *GSSHA* coverage to select it.
4. Select the *Create Feature Point Tool*  and click on the two cells shown in the following figure.



5. Click on the *Select Feature Point/Node Tool*  and double click on *Well 1* (see above figure). This will open the *Properties* dialog.
6. In the *Groundwater BC* column, select *Static Well*. Enter 2.55 m<sup>3</sup>/s for the *Pumping Rate*.



7. Click OK.

8. Similarly, double click on *Well 2* and select it to be a *Dynamic Well*. Click on the open button to the right, under *Pump Rate* which will open the *XY Series Editor* window.
9. Outside of WMS, browse and open a spreadsheet *C:\Training\Groundwater\PumpingRate.xls*. Copy all the data and paste into the *XY Series Editor*.
10. Click OK and OK again.
11. This will create an index map for the wells. This index map will be selected because of which the display changes and you might not see the streams. Other boundary conditions are in *Gw Boundary* index map
12. Select the *Gw Boundary* Index map in the data tree and *Regenerate* it.

## 2.5 Import Hotstart file for groundwater elevations

---

1. In the project explorer, click on *2D grid data* to select it.
2. Right click on *new grid* folder which is right under *2D Grid data* and select *Import Scalar dataset*
3. Select *GRASS ASCII grid file* and click OK.
4. Browse and open the file *C:\Training\Personal\Groundwater\GWAdv.wte*. Please note that in the *Open* dialog, you might need to change the file type to *All Files (\*.\*)*.
5. In the project explorer, under *2D Grid data* you should see *GWAdvWells* in the GSSHA project. Right click on the *Continuous Maps* folder under this model and Assign the *GWAdv* dataset.
6. Now you can see *GWAdv* which you imported as a hotstart file has been listed as one of the continuous map.
7. In the GSSHA *Job Control*, Click on the *Edit Parameters* button next to *Groundwater*.
8. Assign *GWAdv* for the *Water Table*.
9. Click OK and OK again.

## 2.6 Save and Run the Model

---

1. Save the project as *C:\Training\Personal\Groundwater\GWAdvWells.prj*
2. Run GSSHA and visualize the results.
6. Once done running, copy the hydrograph ordinates to the *Data* sheet under *With Wells* column of *C:\Training\Groundwater\GWResults.xls*.
3. Compare the results.
4. Step through the groundwater head maps and see the results of the pumping wells.

## 2.7 Add outside boundary conditions


---

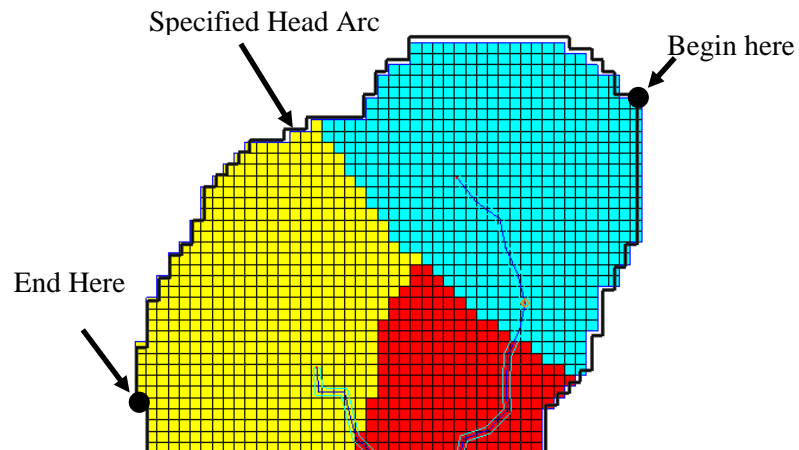
In all of the groundwater models built so far, we have been assuming that the watershed boundary is a *No Flow* boundary condition for the groundwater model. In this exercise, we will define different boundary conditions and re-run the model.

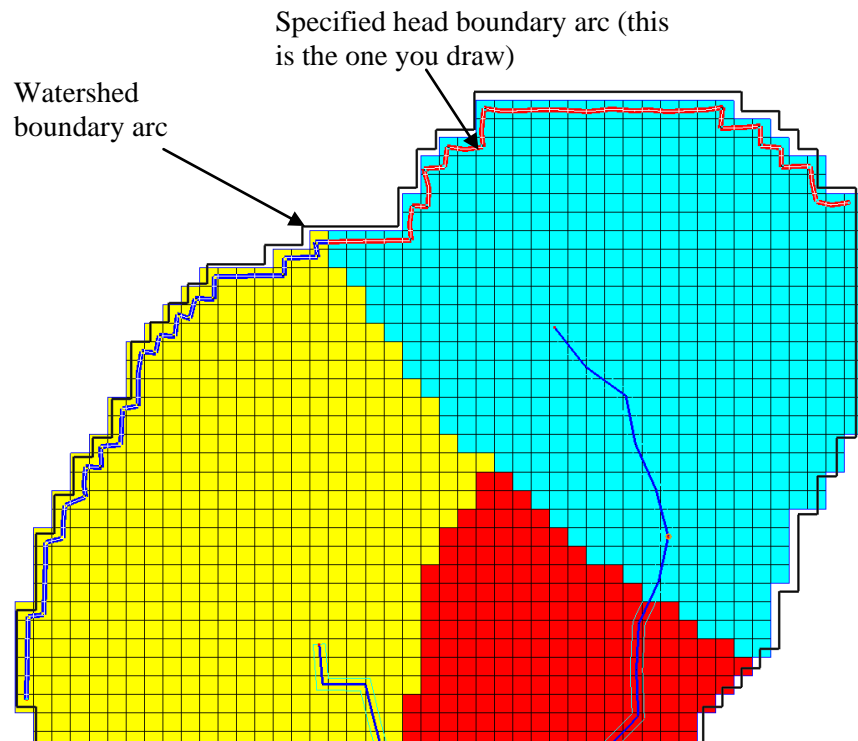
The grid cells that are specified head boundaries need to be selected by using an arc. You can do that either by using/modifying an existing arc or by creating a new arc.


In this workshop, we will create a new arc that represents the specified head boundary.

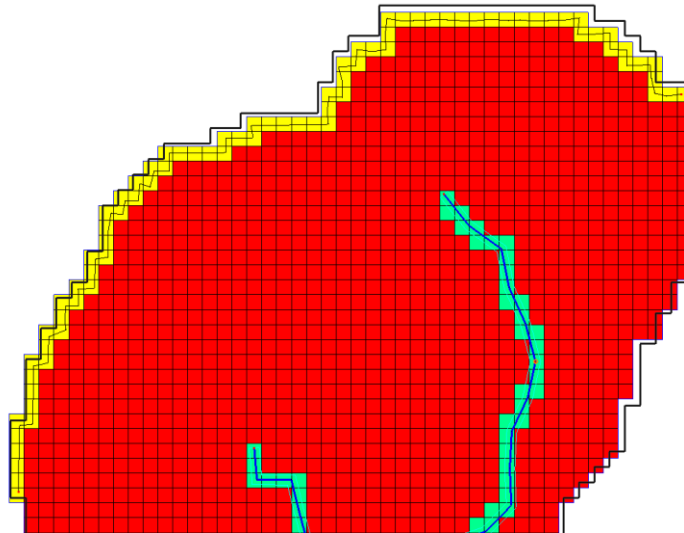
If you closed the project or did not work quite well, you may open it from **C:\Training\Groundwater\GWAdvWells.prj**.

1. Turn off the display of all the coverages except the *GSSHA* coverage.
2. In the data tree, click on the *GSSHA* coverage to select it.
3. Select *Create Feature Arc Tool*  and draw an arc starting from the point shown in the following and intersect each grid cell along the boundary till you reach the end point.
4. While drawing the take these points into consideration:
  - Do not intersect the arc you are drawing with the watershed boundary arc which will end the arc at the point of intersection.
  - Make sure that you intersect each grid cell along the edge of the watershed
  - Make sure you make a continuous selection of grids (do not draw the arc so that any cell is missed in between)





5. Once you have tracing the arc, click on *Select Feature Arc Tool*  and double click on the *Specified head arc* figure to open the *Properties* dialog.
6. Change the Groundwater BC to *Constant head*. GSSHA will read the head elevation from the water table data.
7. Click OK.
8. Select the *Gw Boundary Index* map in the data tree and *Regenerate* it.
9. If properly crated, your specified head boundary will be displayed as the following figure in *Gw boundary index* map.



## 2.8 Save and Run the Model

---

We are now ready to run the model.

1. Save the project as ***C:\Training\Personal\Groundwater\GWAdvBdry.prj***
2. Run GSSHA and visualize the results.
7. Once done running, copy the hydrograph ordinates to the *Data* sheet under *Static Head BC* column of ***C:\Training\Groundwater\GWResults.xls***.
3. Compare the results.

## 2.9 Change the groundwater modeling equation

---

So far we have been using the *Green and Ampt with soil moisture redistribution* equation to simulate the surface-groundwater interaction. Now we will use *Richard's Infiltration equation* and run the same model. We will then compare the results from these two equations.

If you closed your previous model, open it from ***C:\Training\Personal\Groundwater\GWAdvBdry.prj*** or from ***C:\Training\Groundwater\GWAdvBdry.prj***

Save the project as ***C:\Training\Personal\Groundwater\GWRich.prj***

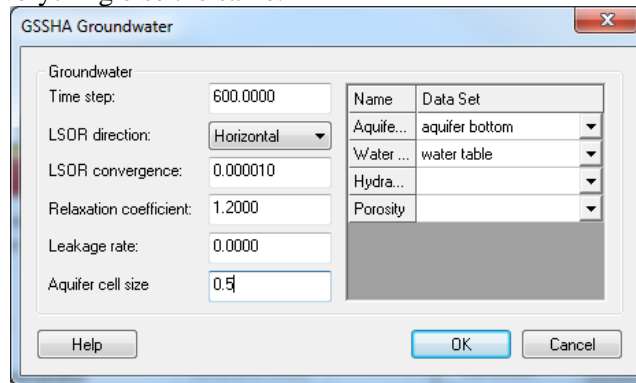
1. In the *GSSHA Job Control*, select *Richard's Infiltration* under *Infiltration* methods.
2. Click on the *Edit Parameters* button for Richard's equation.
3. Enter the following parameters in the *Richard's Parameters* dialog.

Weight	1.0
DTHETA Max	0.025



C Option	Havercamp
K Option	Geometric
Upper Option	Green Ampt
Max Iteration	1
Max num. Cells	13

4. Click OK.
5. Still in the *GSSHA Job Control* dialog, click on *Edit Parameters* button next to *Groundwater*.
6. In the *GSSHA Groundwater* dialog, enter 0.5m for *Aquifer cell size*. Leave everything else the same.

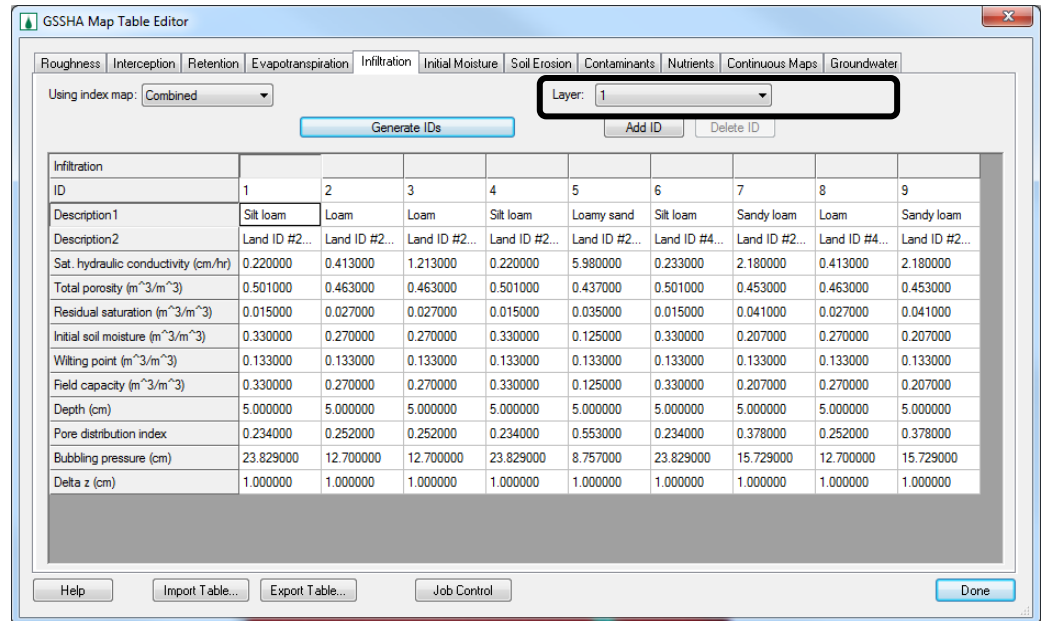


7. Click OK and OK again.

## 2.10 Update the mapping table for Richard's parameters

1. In the *GSSHA Map Tables*, select *Infiltration tab*.
2. Use *Combined* index map and generate IDs.
3. In *Layer* options, select *1*.
4. The parameters for each layer are listed in *C:\Training\Groundwater\RichParams.xls*. Browse and open this spreadsheet.
5. Copy the data for layer 1 and paste in the map table making sure that Layer 1 is selected.

## 2-10 GSSHA Modeling Using WMS – Volume 5



6. In the *Layers* drop down, select layers 2 and 3 and enter the values by copying from the spreadsheet.
7. assuming that all the three layers have same parameters).
8. Once you have defined the parameters for all the three layers, click *Done* to close the *GSSHA Map Table* dialog.

## 2.11 Save and Run the Model

1. Save the project as *C:\Training\Personal\Groundwater\GWRich.prj*
2. Run GSSHA and visualize the results.
8. Once done running, copy the hydrograph ordinates to the *Data* sheet under *Richards Equation* column of *C:\Training\Personal\Groundwater\GWResults.xls*.
3. Compare the results.

---


## Subsurface Tile and Storm Drains

In this workshop we will see how GSSHA can be used to simulate sub surface storm and tile drains. We will begin with an existing model that has long term simulation and ground water processes already defined and add storm and tile drains. In the first part, we will add a small storm drain network and run the model. We will then add tile drains and re-run the model.

### 3.1 Open an Existing GSSHA Project

---


Open a WMS project file for Eau Galle watershed.

1. In the *2D Grid Module*  select *GSSHA | Open Project File...*  
Browse and open the file *C:\Training\Subsurface\base.prj*
2. Save the project as *C:\Training\Personal\Subsurface\stormdrain.prj*
3. Turn off the display of all the coverages except the *GSSHA* coverage.

### 3.2 Adding Storm drains the groundwater model

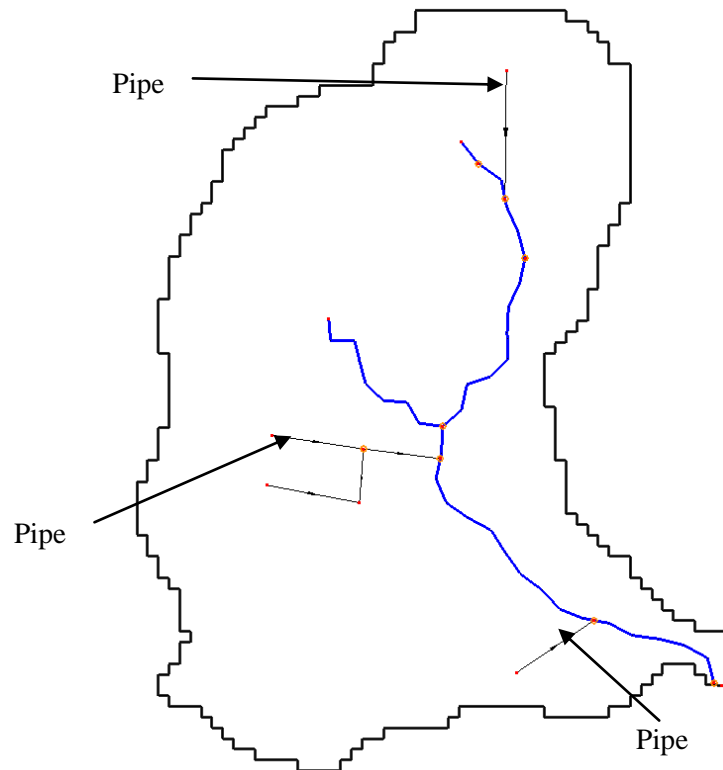
---


The storm drains are generally used to collect the overland flow and convey it to natural streams. In this workshop, we will create a simple storm drain network and run the model.

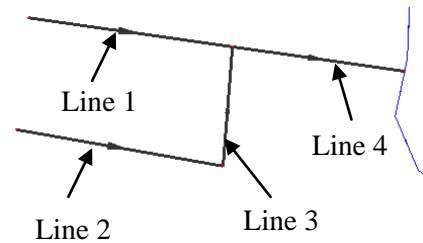
1. In the *2D Grid Module*  select *GSSHA |Job Control*
2. Check the toggle on for *Storm/tile drain*.
3. Click on *Edit Parameters* button next to the toggle you just turned on and select *Allow GSSHA to define node spacing*.
4. Click OK and OK again.


### 3.2.1 Creating Storm drain arcs

You will create a pipe network as shown in the following figure. To get the pipes at the right places, import a background image that has the pipe layout and trace the pipe arcs so that the network will be similar to that shown in the following image.



1. Select **File/Open** and open file **C:\Training\Subsurface\StormDrains.jpg**.
2. You might NOT see a lot of change when the image is loaded because the image exactly resembles your watershed. But you should have seen the pipe arcs displayed.
3. Click on the **GSSHA** coverage to make it active.
4. Select **Create Feature Arc tool**  and trace arcs over the pipe networks. Make sure you start and begin at the exact locations shown in the background image (start and end at the little red dots) so that the node elevations that you will define in a while will be proper.
5. Take the following points into consideration while drawing the pipe network:
  - Draw the arcs from downstream to upstream
  - Each arc should be a straight line, i.e. there should be no vertices in the line. If there is change in direction, double click to end the line and begin another line for another direction.

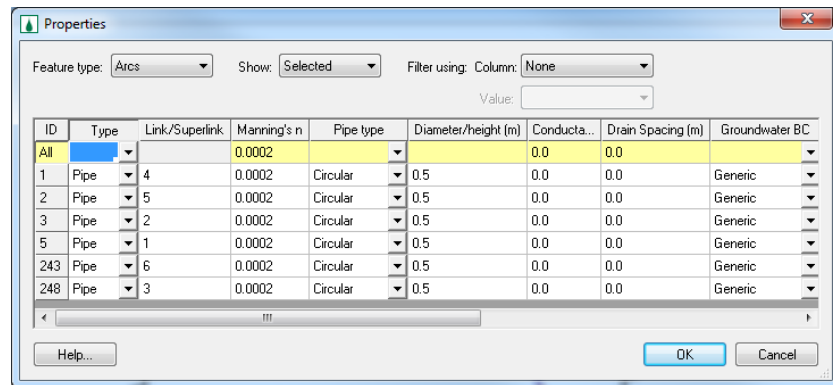



- The pipes will drain into the natural streams.
  - The pipes do not form a closed loop (WMS does not support closed pipe networks at this moment).
6. The elevation for the nodes in the pipes (super junctions) need to be at least 2m below the ground surface and should go into the ground water media. So we will redefine the super junction elevations so that they are at proper locations.
  7. Zoom into the area where the pipes begin and end. You should be able to see the elevation for each node displayed in the background image.
  8. While still in the *Map module*, click on the *Select Feature Point/Node tool*  and select each individual super node and assign the node elevation as shown in the background image. The elevation for the node can be changed in the *properties* window to the right.

**IMPORTANT:** While defining the pipe node elevations, there are two important considerations:

- The node should be at least 2m below the ground surface
  - The nodes and pipe arcs should not go into the groundwater media which means the elevation of the nodes must be higher than the top elevation of the groundwater elevation at that location.
  - The node elevations you have entered from the background image are selected meet these two conditions.
9. Repeat the same process to define the elevation of each (all begin and end points of the pipes).
  10. Still in the *Map module*, select all the pipes and select ***Feature Object/Attributes***.
  11. Set the
    - *Type* to *Pipe*
    - *Manning's n* to 0.0002
    - *Pipe type* to *Circular*
    - *Diameter* to 0.5m
    - *Conductance* to 0.0m and
    - *Drain Spacing* to 0m.

### 3-4 GSSHA Modeling Using WMS – Volume 5



12. After defining these attributes, Click OK.
13. Then select *Refresh button*  and zoom around the pipe network. Check the flow direction represented by an arrow head. If by chance you see any arc flowing in the opposite direction, right click on the arc and select *Reverse Directions*. Alternatively, you may delete that arc and recreate it in the proper direction,
14. You may turn off the display of 2D grids for better display of the pipes.
15. Select the nodes where the pipes are connected to the stream and edit their attributes. Set the attributes as follows:
  - *Manhole area* to 1.0 m<sup>2</sup>
  - *Inlet type* to *Empty to channel*
  - *Weir length* to 0.1 m and
  - *Orifice diameter* to 0.1 m.
  - Leave all other fields the same. If you want a hydrograph output at these nodes, you may toggle the option under *Hydrograph output* column.
16. Click OK.
17. Select remaining nodes (all other nodes in the pipe, not on the stream nodes) in network and edit their attributes. Set the attributes as follows:
  - *Manhole area* to 1.0 m<sup>2</sup>
  - *Inlet type* to *6 grate inlets*
  - *Weir length* to 0.1 m and
  - *Orifice diameter* to 0.1 m.
18. Click OK.

### 3.3 Save and Run the Model

You have now defined the storm drain network. Save the GSSHA project and run it.

1. Save the project as *C:\Training\Personal\Subsurface\stormdrain.prj*
2. Select *GSSHA/Run GSSHA*

### 3.4 Results visualization

1. Once done running, have a look at the hydrograph output. Copy and paste the hydrograph ordinates to a spreadsheet located at *C:\Training\Subsurface\drains.xls*
2. Look at how the groundwater head varies.
3. See if there is any difference in the overland flow depth contours
4. You should be able to see *Pipe Flow* scatter datasets in the project explorer. You can visualize this data in the similar way you visualize the stream flow data.

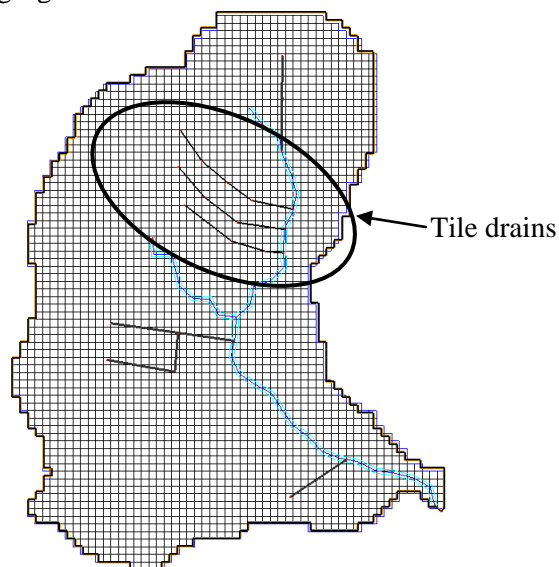
### 3.5 Adding subsurface tile drains the groundwater model

We will keep working with the same model and add tile drains to it. Tile drains are generally used in the agricultural fields to lower the water table by draining water into the channel. The way GSSHA treats tile drains and storm drains is the same but the conductance of the materials around the pipe should be high in case of tile drains.

1. Save the GSSHA model that you created in previous section as *C:\Training\Personal\Subsurface\ tiledrain.prj*
2. Turn off the display of all other coverages except the *GSSHA* coverage.
3. Import a background image located at *C:\Training\Subsurface\TileDrains.jpg*.
4. Once the background image loads, zoom into the area highlighted in the following figure and trace the tile drain network as you did in the previous section for storm drains.

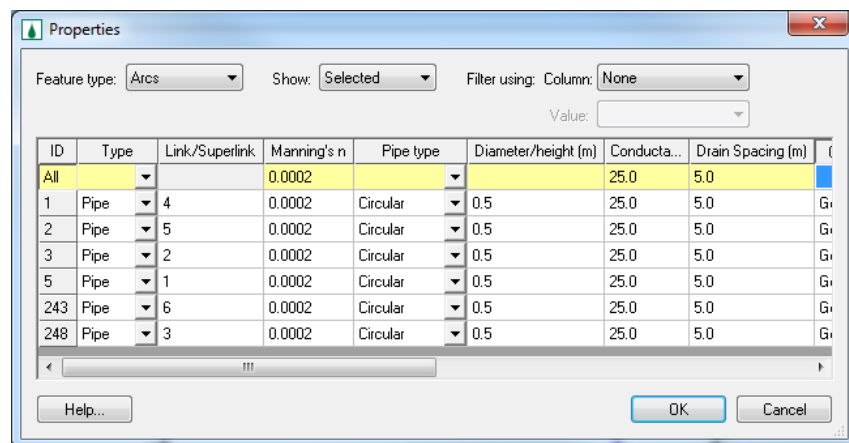
**Note:** You do not have to change anything for the storm trains that you added in the previous simulation. Just create the tile drain arcs.


5. After you are done tracing the arcs, the network should look similar to the following figure.



### 3-6 GSSHA Modeling Using WMS – Volume 5

6. Enter the elevation of all the nodes for the tile drains that you just drew. The node elevations are displayed in the background image.
7. Still in the *Map module*, select all the tile drains and select **Feature Object/Attributes**.
8. Set the attributes to the following.
  - *Type* to *Pipe*
  - *Manning's n* to 0.0002
  - *Pipe type* to *Circular*
  - *Diameter* to 0.5m
  - *Conductance* to 25m and
  - *Drain Spacing* to 5m.
  - Leave everything else the same as the defaults.



9. After defining these attributes, Click OK.
10. Then select the *Refresh button*  and zoom in around the tile drain network (if you already zoomed out). Check the flow direction represented by an arrow head. If by chance you see any arc flowing in the opposite direction, right click on the arc and select *Reverse Directions*.
11. Similarly, select the nodes connected to the stream and edit their attributes to the following
  - *Manhole area* to 1.0 m<sup>2</sup>
  - *Inlet type* to *Empty to channel*
  - *Weir length* to 0.1 m and
  - *Orifice diameter* to 0.1 m.
  - Again if you want the hydrograph output at these nodes, you may toggle the option under the *Hydrograph output column*.
12. Click OK after you are done defining the attributes
13. Select the remaining nodes in network and edit their attributes. Set the attributes as follows:
  - *Manhole area* to 1.0 m<sup>2</sup>
  - *Inlet type* to *6 grate inlets*
  - *Weir length* to 0.1 m and
  - *Orifice diameter* to 0.1 m.
14. Click OK.



### 3.6 Save and Run the Model

---

You have now defined the tile drain network. Your model has both storm and tile drains. Save the GSSHA project and run it.

1. Save the project as *C:\Training\Personal\Subsurface\tiledrain.prj*
2. Select *GSSHA/Run GSSHA*

### 3.7 Results visualization

---

1. Once done running, have a look at the hydrograph output. Copy and paste the hydrograph ordinates to a spreadsheet located at *C:\Training\Subsurface\drains.xls*
2. Look at how the groundwater head varies.
3. See if there is any difference in the overland flow depth contours
4. You should be able to see *Pipe Flow* scatter datasets in the project explorer. You can visualize this data in the similar way you visualize the stream flow data.