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
3210 N. Canyon Road
Suite 300
Provo, Utah 84604
Tel.: (801) 691-5528
e-mail: 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

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 a different name as *C:\Training\Personal\ Groundwater\GW.pr.j*
3. In the *Project Explorer*, toggle off the display of all the map data and toggle the *GSSHA* coverage on.

1.2 Creating Groundwater Datasets

To simulate groundwater in a GSSHA model, you need to define the following index maps and tables:

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

You will create the aquifer bottom and water table elevation maps from x, y, z points stored in a .csv file. These points 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 XYZ data of the aquifer bottom. These data can be derived from borehole data that contain information about depth to different layers in the subsurface.

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 *Start import at row* value to 2. Do not turn on the *Heading Row* toggle.
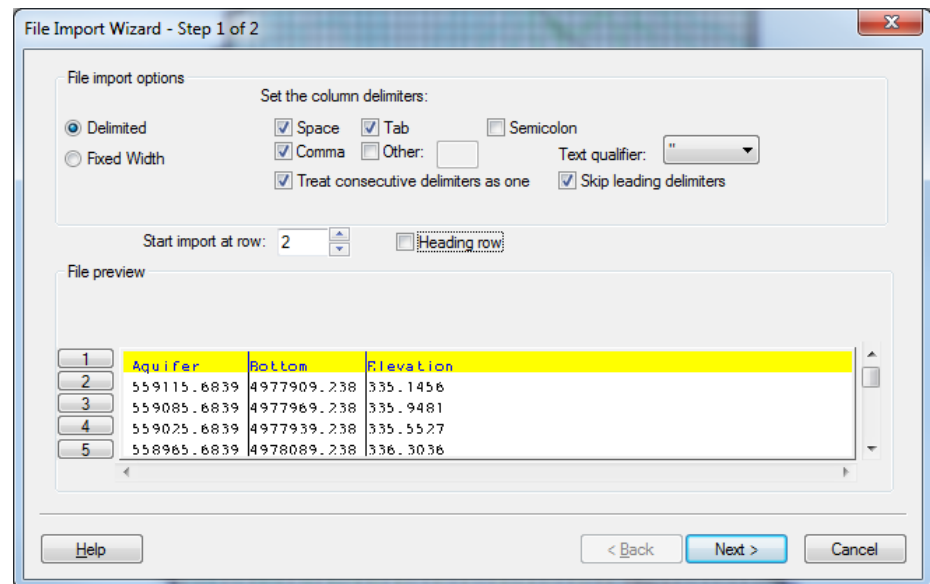
3. Make sure the Comma delimiter is toggled on and click *Next*.
4. Make sure the WMS data type is set to *2D Scatter Points*.
5. Make sure the first column is mapped to *X*, the second column to *Y*, and the third column to *Data set*.
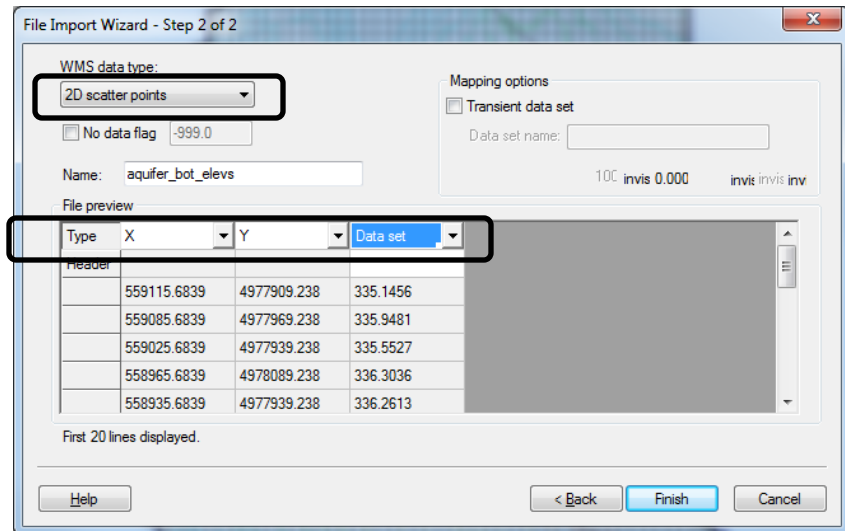
6. Click *Finish* to import the dataset as a 2D Scatter Point dataset
7. A new scatter point dataset named *aquifer_bot_elevs* should appear in the *Project Explorer* and the points are plotted.
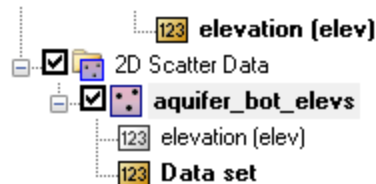
8. Make sure *Data set* is selected as the active dataset.
9. Right click on the *aquifer_bot_elevs* scatter set and choose *Interpolate->...To Grid.*
10. Change the interpolated data set name to *Aquifer Bottom*. DO NOT toggle on the option to *Map elevations*.
11. Click OK to create a new grid data 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*
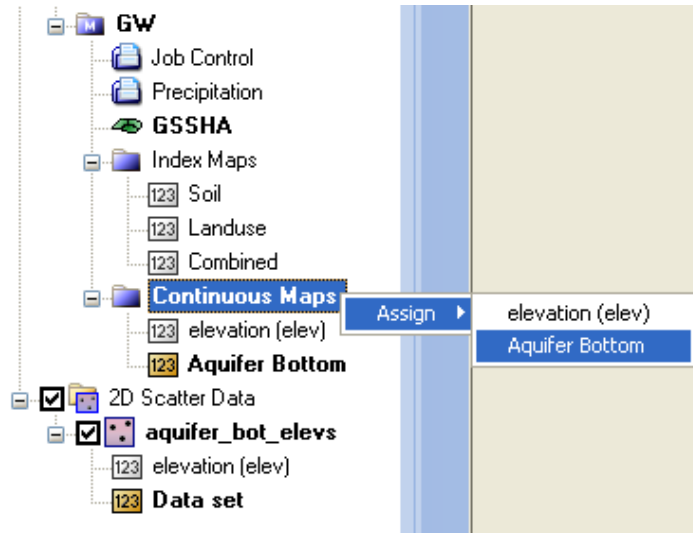
1.2.2 Initial Water Table Elevations

Ideally, you will have observation well data from which to derive an initial representation of the water table. These xyz data can be interpolated to the grid to develop the required continuous map in the same way the aquifer bottom map was created. In lieu of measured values you can develop an initial water table by assuming an initial water table and running GSSHA to derive a water table map. In this example you will use xyz data that represent well water surface elevations scattered throughout your 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 *Start import at row* value to 2. Do not toggle on the *Heading row* toggle. Make sure the Comma delimiter is toggled on and click *Next.*
3. Make sure the *WMS data type* is set to *2D Scatter Points*. 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.
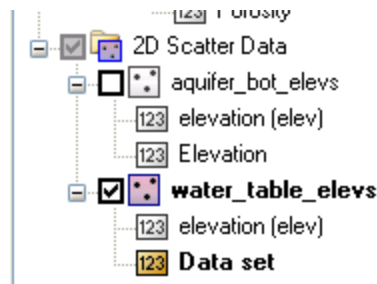

6. Select the *Data set* data set (under *water_table_elevs*) to make sure it is selected as the active dataset.

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

After you interpolate the water table XYZ data to define 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 which is not desirable. You will need to check this and fix any anomalies before using the water table elevation map with your model.

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 *Data | Data Calculator*.
2. Double click *elevation (elev)* data which will insert a dataset symbol such as *d1* in the *Expression* box. Note that the dataset symbol might be different in your case.
3. Then click the - (minus) button or type it from the keyboard.
4. Double click the *InitialWaterTable* dataset. Type *Elev-Water table* in *Result* field.
5. Do NOT select the toggle *Index Map*.
6. Your data calculator will look something like this:
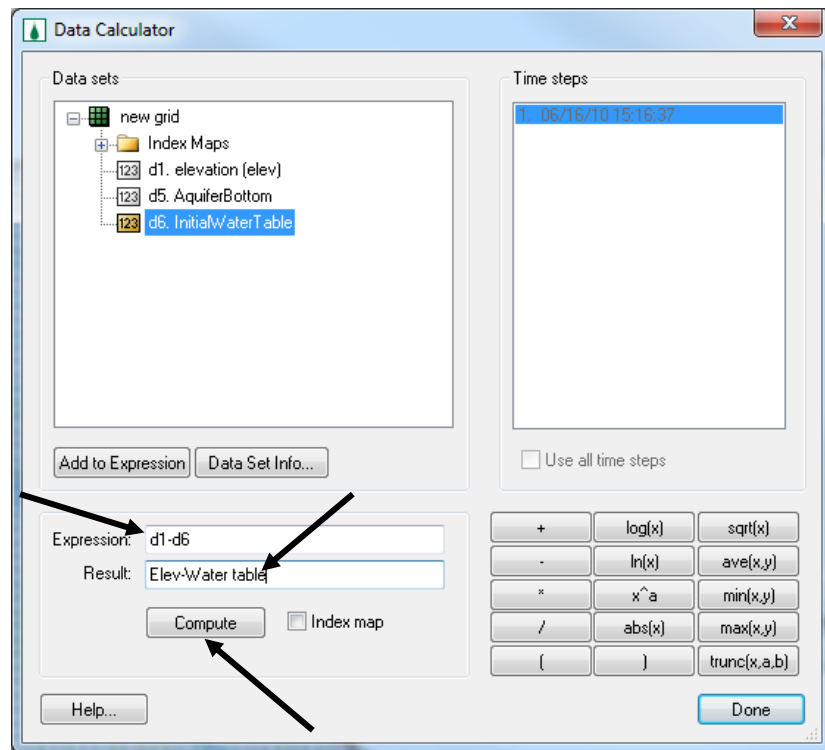
7. Click *Compute* and then *Done*.
8. Right click on the *Elev-Water Table* map in the *Project Explorer* and select *Properties*.
9. This will open the *Data Set Info* dialog. You can see a Histogram of the values in the data you selected.
10. Double click on the plot area and select the maximize button in the other dialog that opens.
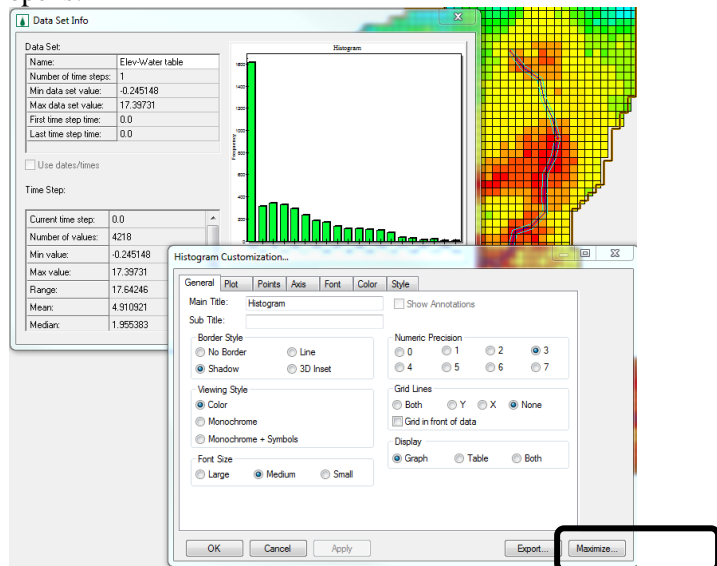
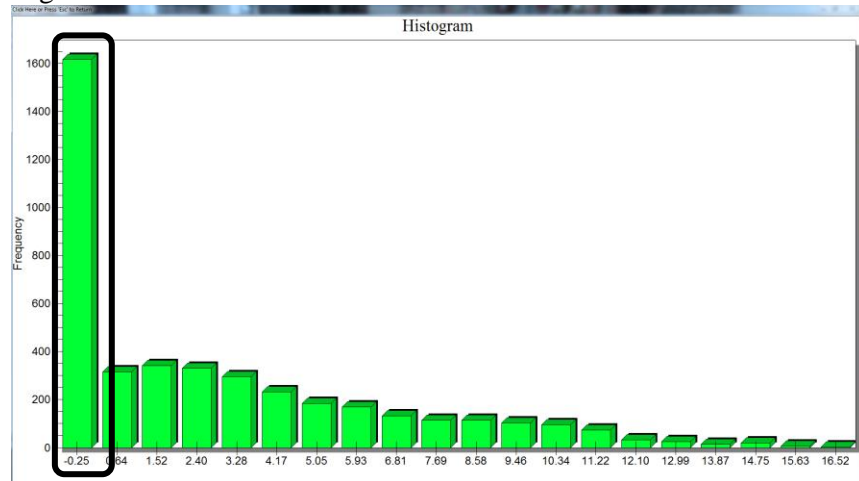11. In the maximized histogram plot, check to see if there are any negative values. In this example you will have a negative value. This means that the water table elevation is greater than the ground elevation at these cells. You can change the contour options to display the locations where the negative values occur.

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

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 pick the maximum and minimum value from two datasets and assign them to the new dataset.

1. We want to adjust the water table dataset so that:
   - The aquifer bottom is always at least 1.0m below the ground surface
   - The water table is always at least 0.1m below the ground surface
   - The water table is always below the ground surface and above the aquifer bottom.
2. To do this, we will have to first use the *'min' function* to select the lesser of the aquifer bottom and the ground surface elevation -1.0 values. This ensures that the aquifer bottom is at least 1.0m below the ground surface.
3. We will use the *'min' function* again to select the lesser of the water table and the ground surface elevation - 0.1 values. This ensures that the groundwater table is at least 0.1m below the ground surface.

4. Then we will use the *max function* between the result from the previous two steps. This ensures that the aquifer bottom is at least 0.1m below the water table.

5. See the following figure and try to see how the equation works before you proceed to the next step.
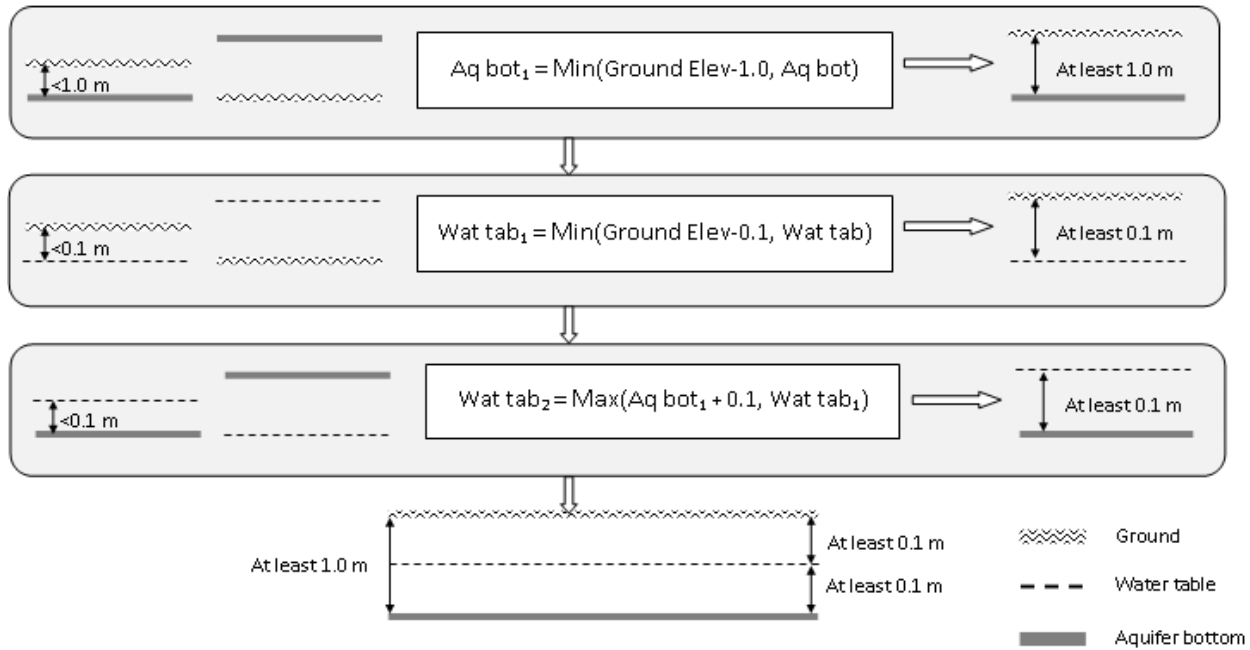
6. Open the data calculator.

7. Click on the *min(x,y)* function which will put an expression *min(??, ??)* in the *Expression* box.

8. Fill in the first position by *elevation(elev)-1.0* and second position by *aquifer bottom*.

9. Enter name *AquiferBottom1* in the *Result* field. See the following figure.
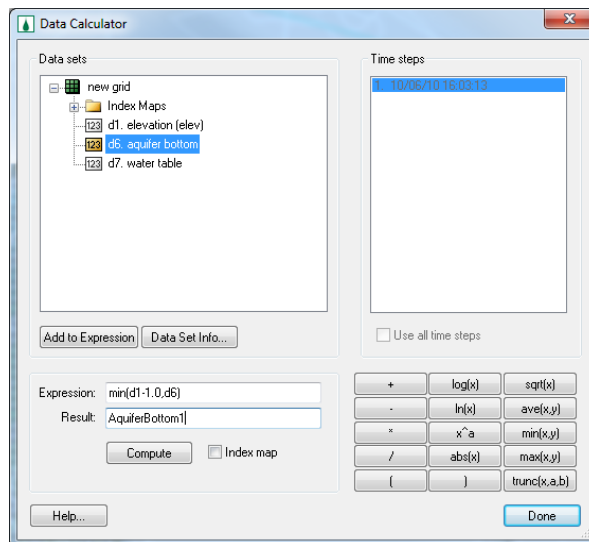
10. Click *Compute*.
11. Similarly, use the equations for the other two cases (See below) and compute new datasets that makes sure that:
    - ground is always on the top,
    - water table is in between ground and
    - aquifer bottom and aquifer bottom is always the lowest.

Equation II: *WaterTable1 = min(elevation(elev)-0.1, Water table)*
Equation III: *WaterTable2 = max(Aquiferbottom1+0.1, WaterTable1)*

Once you compute the updated water table map, it should automatically get associated with the current GSSHA model. If *WaterTable2 and/or AquiferBottom1* are listed under the *Continuous Maps* folder under the GSSHA model *GW,* then they are already assigned. If they are not listed, follow the following steps:

1. Right click on the *Continuous Maps* folder underneath the *GW* folder in the 2D Grid Data section of the *Project Explorer*.
2. Select *Assign-> WaterTable2*. Repeat the same process for *AquiferBottom1*.
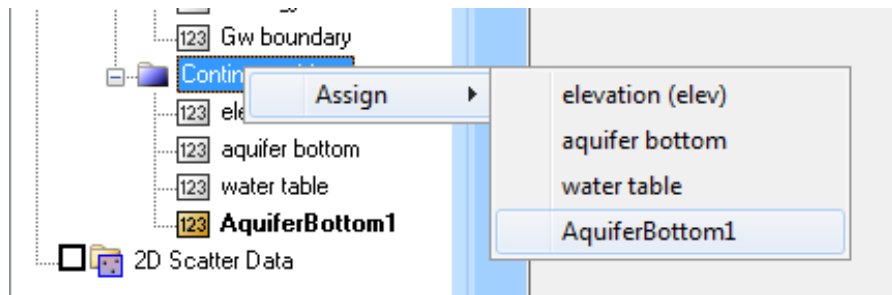
1.2.5 Hydraulic Conductivity and Porosity

To define the hydraulic conductivity and porosity you 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. In the *Project Explorer*, right click on *Coverages* and select *New Coverage*.
2. Select *Soil Type* for *Coverage type*.
3. Change the name to *Geology* and select *OK*. This will add a coverage named *Geology* under the *Coverages* folder.
4. Select *File | Open* and browse to open the following shape file: *C:\Training\RawData\Groundwater\Geology.shp*.
5. Make sure that the *Geology* coverage is selected and *Geology.shp* is active.
6. In the *GIS Module*, select *Mapping | Shapes -> Feature objects*.

7. Click *Yes* to use all the polygons.
8. In the *GIS to feature object wizard*, use the default mapping and click *Finish* on the wizard.
9. Once done mapping, you may delete the shape file from the *Project Explorer* and turn off the display of *Geology* coverage. Select the *GSSHA* coverage to make this coverage active.
10. In 2D Grid module, select *GSSHA | Maps*
11. Select *Geology* as the input coverage and *Texture* as the coverage attribute.
12. Change the index map name to *Geology*.
13. Select "*Coverages->Index Map*". When the index map is finished, click *Done* to close the *GSSHA Maps* dialog.
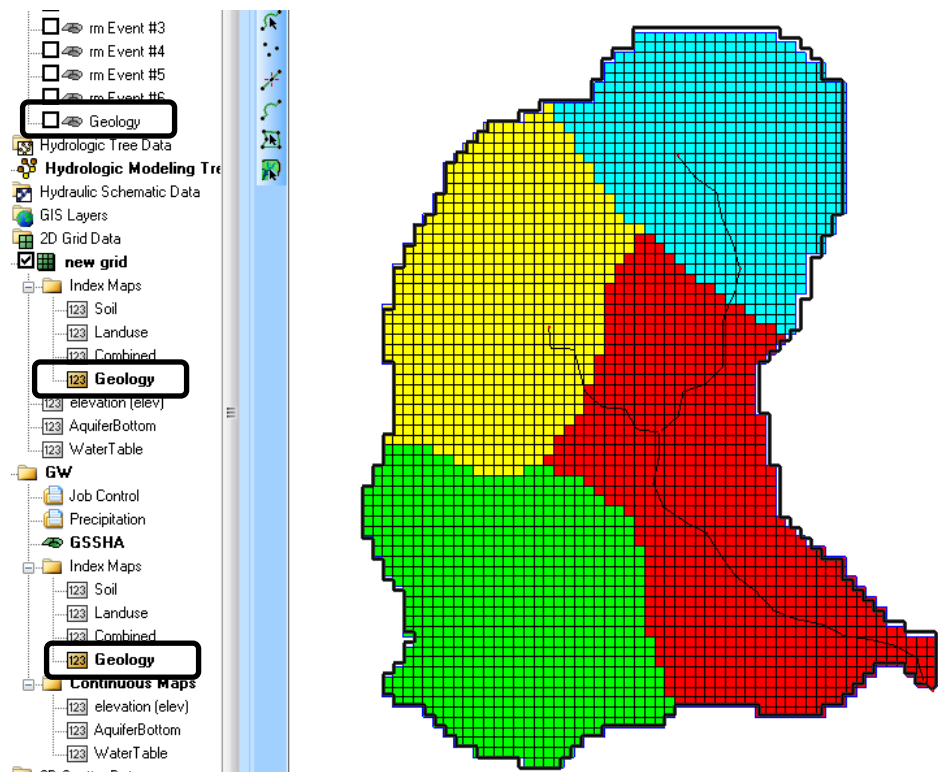
1.3 Groundwater Job Control

1. In the GSSHA *Job Control* (*GSSHA | Job Control*), toggle on the *Groundwater* option and click the *Edit parameter* button. This 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, GSSHA assumes a no-flow groundwater boundary condition around the perimeter of the watershed.

2. In the *GSSHA Groundwater* dialog, set the *Aquifer bottom* Data Set to *AquiferBottom1* and the *WaterTable* Data Set to *WaterTable2*. We will define the *Hydraulic Conductivity* and the *Porosity* Data in 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

You have not defined the hydraulic conductivity and porosity maps for groundwater model yet. You can use the geology index map to define these values in a groundwater mapping table.

1. Select the ***GSSHA | Map Tables…*** menu command. In the *GSSHA Map Table Editor* switch to the *Groundwater* tab.
2. Select *Geology* as your index map and generate IDs. You should see four different IDs. 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

3. After you are finished entering the values, select the *Done* button to close the *GSSHA Map Table Editor* dialog.

1.5 Verifying Long Term Simulation Data

Before running the groundwater model, make sure that the long term simulation is setup correctly.

1. In the GSSHA *Job Control* dialog, click *Edit parameters* for the *Long term simulation*.
2. Make sure the *Latitude* is 44.81, *Longitude* is 267.83, *GMT* is -6 hours, *Minimum Event Discharge* is 0.1 cms, and *Soil Moisture depth* is 0.25 m.
3. Click on the browse button ⬚ for the *HMET Data file*. Browse and select ***C:\Training\Groundwater\HMETData.txt***.
4. Make sure continuous simulation format is set to *WES*. Click *OK*.
5. Select *OK* in the Job Control dialog.

1.6 Setting Output Control

1. In GSSHA *Job control*, click on the *Output Control* button.

2. Turn off the toggle next to *Surface Depth* and toggle on the *Groundwater elevations* and *Cumulative groundwater recharge*.
3. Change the *Write frequency* to be 180 minutes for the grid data and leave the *hydrograph write frequency* as it is.
4. Select *OK* and *OK*.

1.7 Saving and Running the GSSHA Model

You 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 capacity 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*.

1.8 Viewing Groundwater Model Results

1. Once the model has finished running, close the model wrapper to read the solution.
2. Visualize the outflow hydrograph. Copy and Paste the outflow hydrograph ordinates to the *Data* sheet under the *Basic GW Model* column of *C:\Training\Groundwater\GWResults.xls*.
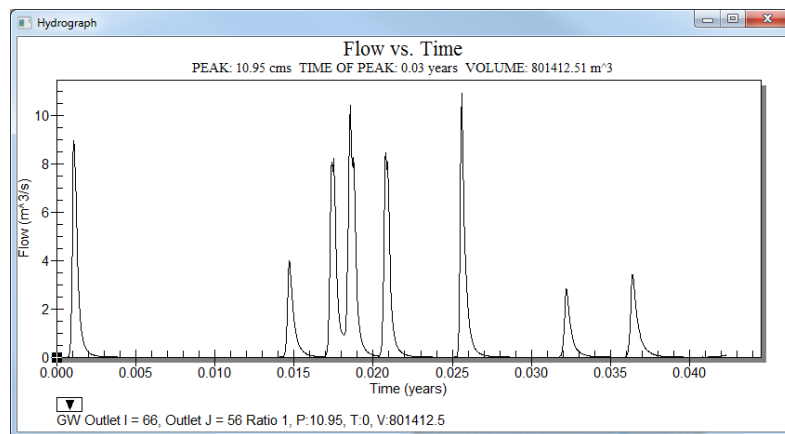
3. Open the summary file and scroll down to find the *SIMULATION TOTALS* at the end of the file. Copy the information in the SIMULATION TOTALS section of the file to the *Simulation Totals* sheet under the *Basic GW Model* column of *C:\Training\Groundwater\GWResults.xls*.
4. Select the g*roundwater_head* data set in the *Project Explorer* and toggle through the time steps to see how the groundwater head varied with time.

5. Similarly, select *gw_recharge_cum* (Cumulative groundwater recharge) dataset and toggle through the time steps to see the amount of recharge in various parts of the watershed as the simulation progresses.
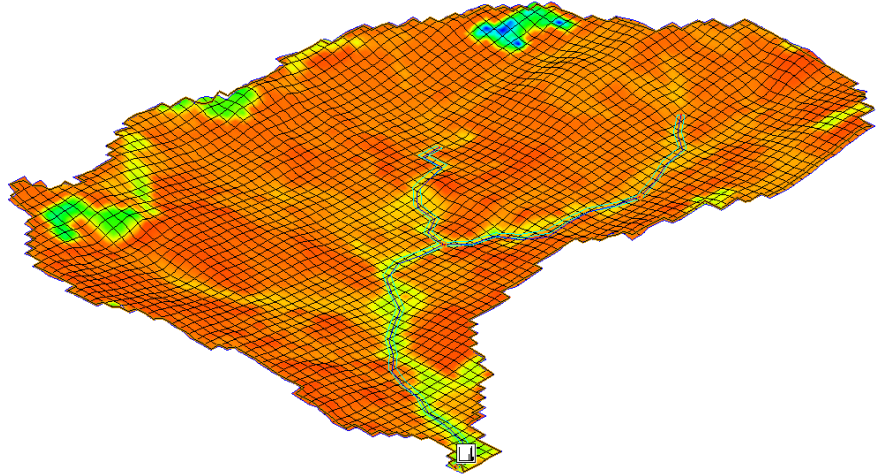
1.9 Exporting a Hot start file for next case

Using the output dataset from one model as an input to another model is called a *Hotstart* in GSSHA. In the coming workshop, you will use the final groundwater elevation in this workshop as a hotstart as the starting groundwater head of next model.

For this, you will have to export the final groundwater head data and import it later in another model.

1. Select the g*roundwater_head* data set in the *Project Explorer* and select the final time step in the properties window.
2. With the last time step selected, right click on g*roundwater_head* in the project explorer and select *Export Dataset.*
3. Select the file format to be *GRASS ASCII grid file* and click OK.
4. Save the file as *C:\Training\Personal\Groundwater\GWHotStart.ggd.*

We will use this file in next workshop.

CHAPTER 2

Advanced Groundwater Modeling in GSSHA

In the previous workshop you learned how a basic groundwater simulation can be developed in GSSHA. In this workshop, you learn about some more advanced topics in groundwater simulation.

2.1 Open an Existing GSSHA Project

You will continue working with the same model that you developed in the 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.

2.2 Adding Streams to the groundwater model

The basic groundwater model that you developed in the previous workshop did not have the stream routing option turned on. In this workshop, you will turn on stream routing and define a groundwater boundary condition for the streams. Because a significant amount of water goes into the subsurface flow from the channel, you will also enable subsurface flow from the channel bottom.

1. Click on the *GSSHA* coverage in the *Project Explorer* 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.

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. For all segments, turn on the *Sub-surface losses/gains* option and enter 55 cm for *Sediment Thickness* and 25 cm/hr for *sediment hydraulic conductivity*.

5. Change the *Groundwater BC* to *Flux River.* See the following figure.
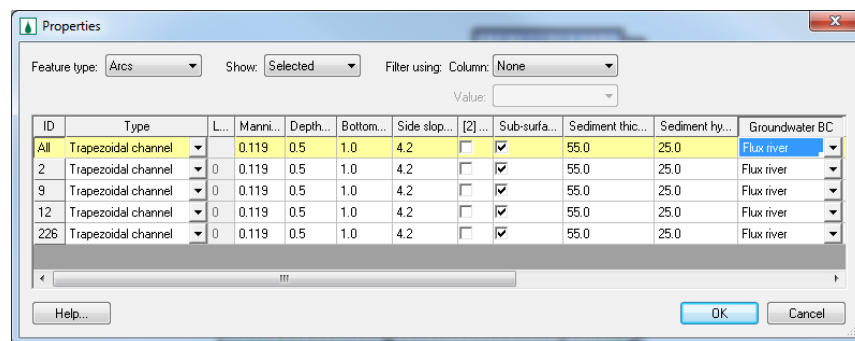
3. Click OK.

4. In the GSSHA *Job Control* dialog (*2D Grid Module*, *GSSHA | Job Control*), 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 the *Gw Boundary* index map. As you select it, you will see a single-colored index map on the WMS display 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 different-colored cells that represent the channel network.

2.3 Import a Hotstart file for groundwater elevations

Using the output dataset from one model as an input to another model is called a *Hotstart* in GSSHA. In this workshop, you will use the final groundwater elevation from the previous model as a hotstart file for the starting groundwater head of this model.

1. In the *Project Explorer*, click on the *2D Grid Data* folder to select it.

2. Right click on the *new grid* folder directly under the *2D Grid data* and select *Import Scalar dataset*
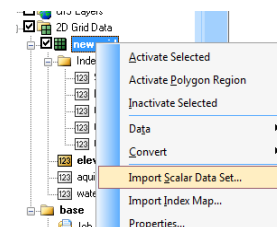
3. Select *GRASS ASCII grid file* and click *OK*.
4. Browse and open the file *C:\Training\Personal\Groundwater\ GWHotStart.ggd*.

 NOTE that in the *Open* dialog, you will need to change the file type to *All Files (*.*)*.

5. In the *Project Explorer*, under *2D Grid data* you should see a *GWAdv* GSSHA model. Right click the *Continuous Maps* folder under this model and *assign* the *GWHotStart* dataset. This will list the *GWHotStart* dataset which you imported as a hotstart file as one of the continuous maps for the *GWAdv* model.
6. In the GSSHA *Job Control*, Click on *Edit Parameters* button next to *Groundwater*.
7. Assign *GWHotStart* for the *Water Table*.
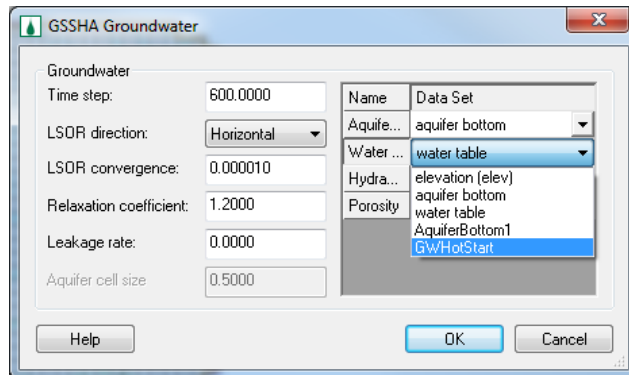
8. Click *OK* and *OK* again.

2.4 Save and Run the Model

1. Save the project as *C:\Training\Personal\Groundwater\GWAdv.prj*.
2. Run GSSHA and visualize the results.

2.5 Visualize the Results

1. Once done running, open the hydrograph plot and copy the ordinates to the *With Streams* column in the *Data* tab in the spreadsheet *C:/Training\Groundwater\GWResults.xls*.
3. Compare the values and hydrograph plot with the previous case where you did not have the streams turned on.
4. Step through the time steps of the *groundwater_head* map to see how the water surface changes during your simulation.
5. Open the summary file and scroll down to find the *SIMULATION TOTALS* at the end of the file. Copy the information in the SIMULATION TOTALS section of the file to the *Simulation Totals* sheet under the *With Streams* column of *C:\Training\Groundwater*

GWResults.xls. Observe any differences in the final values and think about why these differences might have occurred in your current model.

2.6 Add Wells to the Groundwater Model

GSSHA can be used to simulate the effects of pumping wells on groundwater heads. In this section, you will add two wells in the model that we have been building.

If you closed the project or did not finish the model, 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 *Project Explorer*, 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.
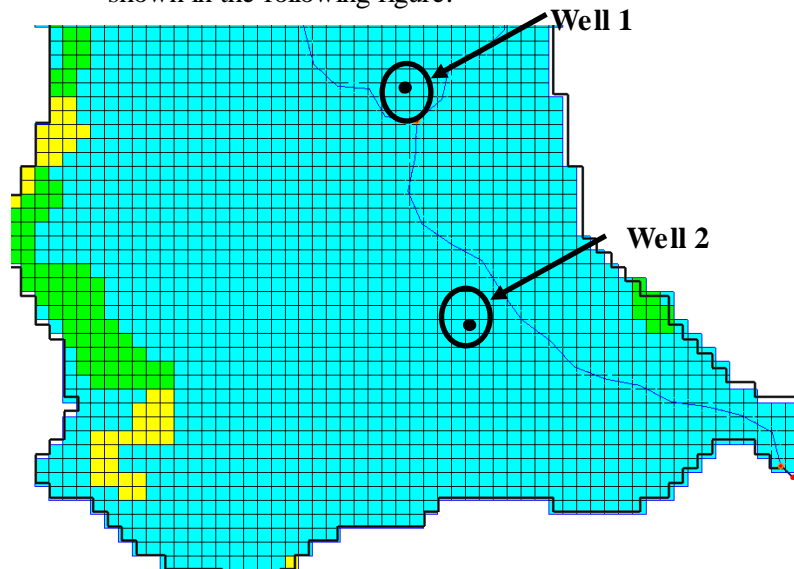
5. Click on the *Select Feature Point/Node* Tool and double click on *Well 1* (see above figure). This will open the *Properties* dialog.
6. In the *Groundwater BC* column, select *Static Well.* Enter 2.55 m^3/s for the *Pumping Rate*.
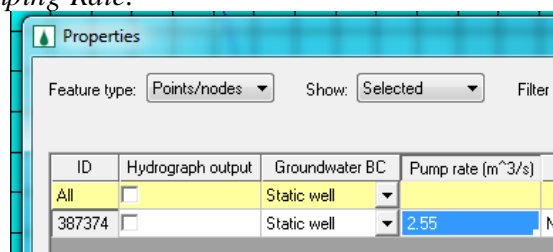
7. Click OK.
8. Similarly, double click on *Well 2* and select *Dynamic well*. Click on the open button to the right of the Dynamic well selection, under *Pump rate,* which will open the *XY Series Editor* window.
9. Outside of WMS, browse and open the spreadsheet *C:\Training\ Groundwater\PumpingRate.xls.* Copy all the data and paste it into the *XY Series Editor*.
10. Click *OK* and *OK* again.
11. Select the *Gw Boundary* Index map in the *Project Explorer* and *Regenerate* it.
12. This will create an index map for the wells. This new index map will be selected, causing a change in the display of the boundary condition index map. All the boundary conditions can be viewed by selecting the *Gw Boundary* index map.

2.7 Save and Run the Model

1. Save the project as *C:\Training\Personal\Groundwater\GWAdvWells.prj*
2. Run GSSHA and visualize the results.

2.8 Visualize the Results

1. When GSSHA has finished running, open the hydrograph plot and copy the ordinates to the *With Wells* column in the *Data* tab in the spreadsheet *C:/Training\Groundwater\GWResults.xls.*
2. Compare the results with the results from the previous models which did not simulate wells.
3. Step through the maps of *groundwater_head* and see how the water surface changes over the period of your simulation.
4. Open the summary file and scroll down to find the *SIMULATION TOTALS* at the end of the file. Copy the information in the SIMULATION TOTALS section of the file to the *Simulation Totals* sheet under the *With Wells* column of *C:\Training\Groundwater\ GWResults.xls*. Observe any differences in the final values and think about why these differences might have occurred in your current model.

2.9 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. In this exercise, we will define a specified head boundary condition to the watershed boundary and re-run the model.

The grid cells that have a specified head boundary condition need to be selected using an arc. You can select these cells either by modifying an existing arc or by creating a new arc.

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

If you closed or did not complete the previous model, you may open it from *C:\Training\Groundwater\GWAdvWells.prj.*

1. Save the GSSHA project as *C:\Training\Personal\Groundwater\ GWAdvBdry.prj.*
2. Turn off the display of all the coverages except the *GSSHA* coverage.
3. In the *Project Explorer*, click on the *GSSHA* coverage to select it.
4. Select the *Create Feature Arc Tool* and draw an arc starting from the point shown in the following figure and intersect each grid cell along the boundary until you reach the end point.
5. While drawing the boundary arc, follow these guidelines:
   - Do not intersect the arc you are drawing with the watershed boundary arc. This will end the arc at the point of intersection.
   - Make sure that you intersect each grid cell along the edge of the watershed model.
   - Make sure you make a continuous selection of grid cells (do not draw the arc so that a cell is missed in between).

   - It is a good idea to use the zoom tool or the mouse scroll wheel (press the wheel down to pan or scroll to zoom in/out) while creating the arc to center your view on the section of the screen where you are working.
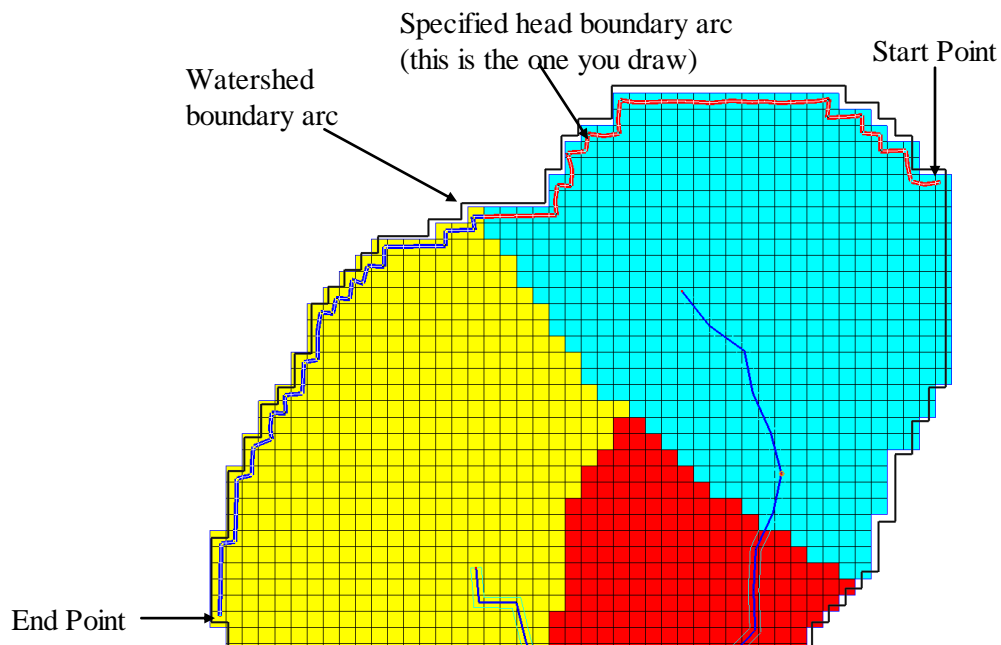
6. Once you have traced the arc, click on *Select Feature Arc Tool* and double click on the Specified head boundary arc to open the *Properties* dialog.
7. Change the Groundwater BC property for the arc to *Constant head.* GSSHA will read the head elevation from the water table data.
8. Click OK.
9. Select the *Gw Boundary* Index map in the *Project Explorer* and *Regenerate* it.
10. If properly created, your specified head boundary will be displayed for the *Gw boundary* index map as shown in the following figure.
    NOTE: if you missed a few cells along the boundary, you may edit the location of the vertices along the arc using *Select Feature Vertex* tool and regenerate the *Gw boundary* index map.
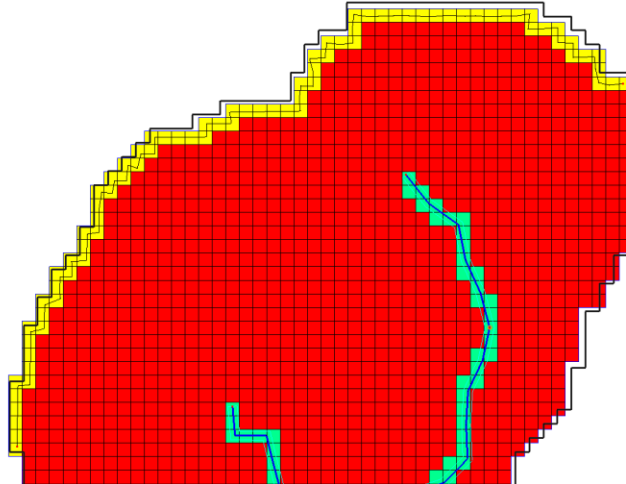
2.10 Save and Run the Model

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.

2.11 Visualize the Results

1. When GSSHA has finished running, open the hydrograph plot and copy the ordinates to the *Static Head BC* column in the *Data* tab in the spreadsheet *C:/Training\Groundwater\GWResults.xls.*
2. Compare the results with the results from the previous models which did not simulate the outside boundary conditions.
3. Step through the maps of *groundwater_head* and see how the water surface changes over the period of your simulation.

4. Open the summary file and scroll down to find the *SIMULATION TOTALS* at the end of the file. Copy the information in the SIMULATION TOTALS section of the file to the *Simulation Totals* sheet under the *Static Head BC* column of **C:\Training\Groundwater\ GWResults.xls**. Observe any differences in the final values and think about why these differences might have occurred in your current model.

CHAPTER   3

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 redistribute Superlink Vertices*.

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, you will need to import a background image that has the pipe layout. Trace the pipe arcs so that the network will be similar to that shown in the following image.


Storm drain Pipes
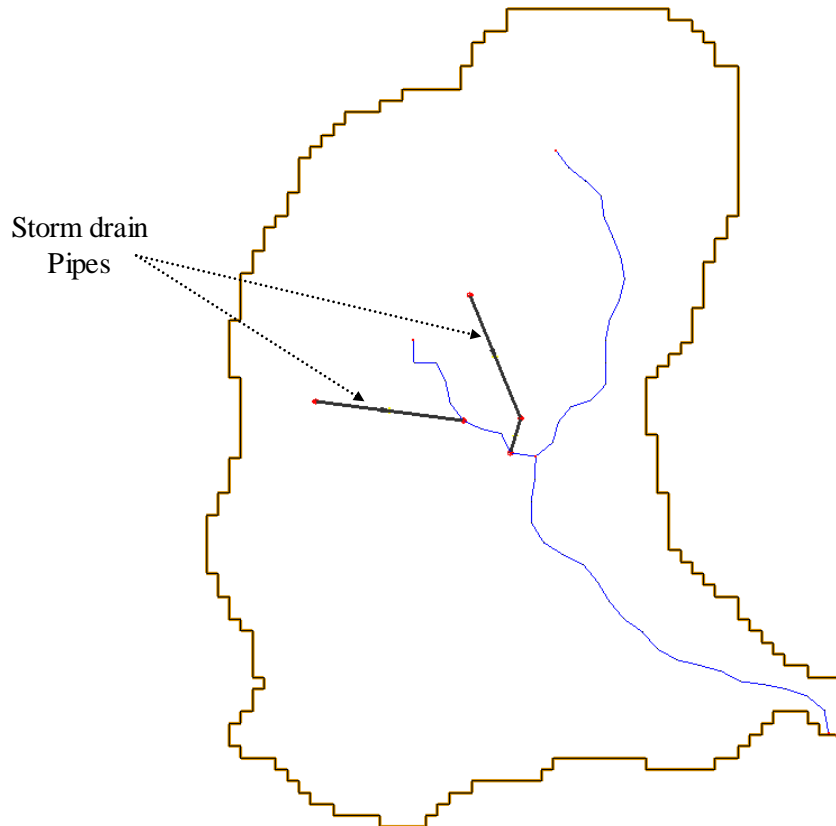
1. Select **File/Open** and open file **C:\Training\Subsruface\Images\ StormDrains.jpg**.
2. You might NOT see a lot of change when the image is loaded because the image exactly resembles your watershed. But you should have seen the pipe arcs displayed and some text showing the pipe node elevations.
3. Turning off the display of 2D grid helps visualizing the image better.
4. Click on the *GSSHA* coverage to make it active.
5. Zoom into the area where the pipes are shown in the background image.
6. 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.

7. Take the following points into consideration while drawing the pipe network:

   - Draw the arcs from downstream to upstream direction.

   - 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, see the following figure for example.
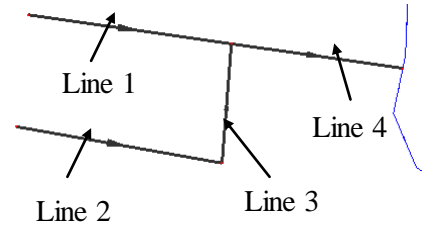
   - The pipes will drain into the natural streams.
   - The pipes do not form a closed loop (WMS does not support closed pipe networks at this moment).

8. 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.

9. 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 side of WMS main window.

10. Repeat the same process to define the elevation of each (all begin and end points of the pipes). The elevation of the nodes where the pipe arcs meet streams are automatically assigned by WMS.

11. Turn off the display of background image.

12. Still in the *Map module,* select all the pipes arcs using *Select feature Arc tool* and select **Feature Object/Attirbutes.**

13. 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.
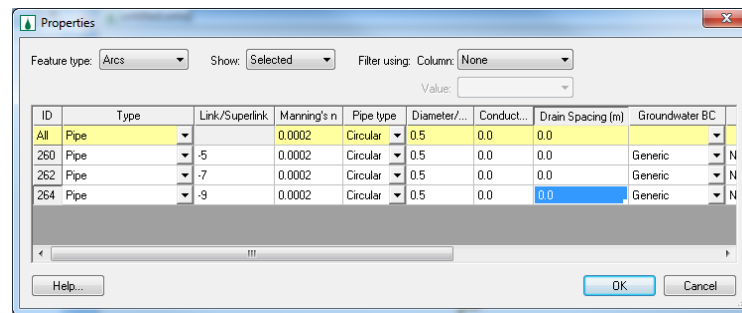
14. After defining these attributes, Click OK.

15. 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,

16. Select the nodes where the pipes are connected to the stream using the *Select Feature Point/Node tool* and edit their attributes. Set the attributes as follows:

    - *Manhole area* to 1.0 m^2
    - *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.

17. Click OK.

18. 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^2
    - *Inlet type* to *6 grate inlets*
    - *Weir length* to *0.1 m and*
    - *Orifice diameter* to *0.1 m.*

19. 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. Turn on the display of 2D grid data if it is turned off.
2. Once done running, have a look at the hydrograph plot. The peak discharge does not vary by a lot but you can see the difference in the runoff volumes as compared to the base model.
3. Under the solution folder, select the dataset *groundwater_head* and toggle through the time steps to see the variation in GW head contours.
4. Open the summary file and look at the amount of water getting in and out of the superlinks (storm drain pipes).

## 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. Turn off the display of 2D grid data.
4. Import a background image located at *C:\Training\Subsurface\ TileDrains.jpg*.
5. Once the background image loads, zoom into the area where the pipes shown. Trace the tile drain network (red arcs in the background image) as you did in the previous section for the storm drains.

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

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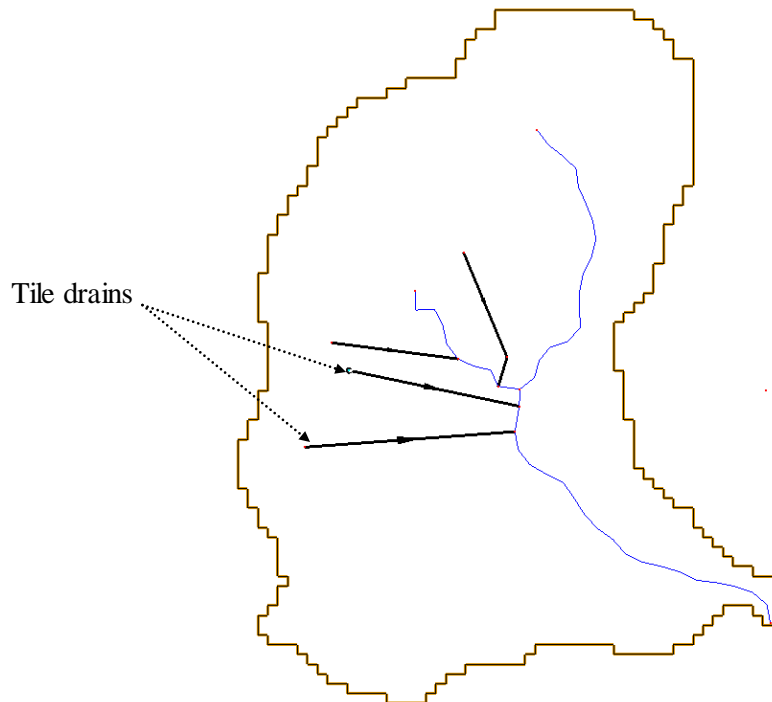
Tile drains

7. Enter the elevation of all the nodes for the tile drains that you just drew. The node elevations are displayed in the background image.
8. You may now turn off the display of background image.
9. Still in the *Map module,* select all the ***tile drain arcs*** using *Select feature Arc tool* and select ***Feature Object|Attirbutes.***
10. 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.
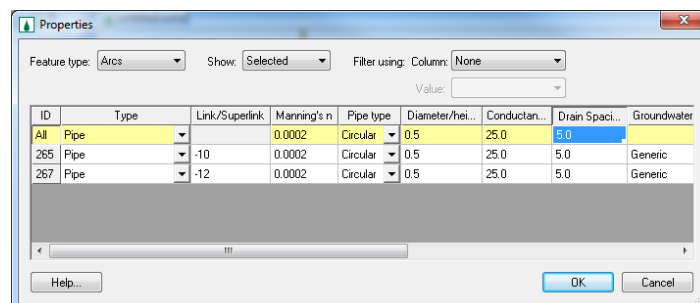
11. After defining these attributes, Click OK.

12. 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.*

13. Similarly, select the nodes connected to the stream using the *Select Feature Point/Node tool* and edit their attributes to the following
    * *Manhole area* to 1.0 m^2
    * *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.*

14. Click OK after you are done defining the attributes

15. Select the remaining nodes in network and edit their attributes. Set the attributes as follows:
    * *Manhole area* to 1.0 m^2
    * *Inlet type* to *6 grate inlets*
    * *Weir length* to *0.1 m and*
    * *Orifice diameter* to *0.1 m.*

16. 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 plot. Compare the peak flow and runoff volumes with those in the previous models.
2. Look at how the groundwater head varies.
3. See if there is any difference in the overland flow depth contours
4. Open the summary file and see the amount of water getting in and out of the superlinks.
5. Notice the interaction between superlinks, streams and groundwater.