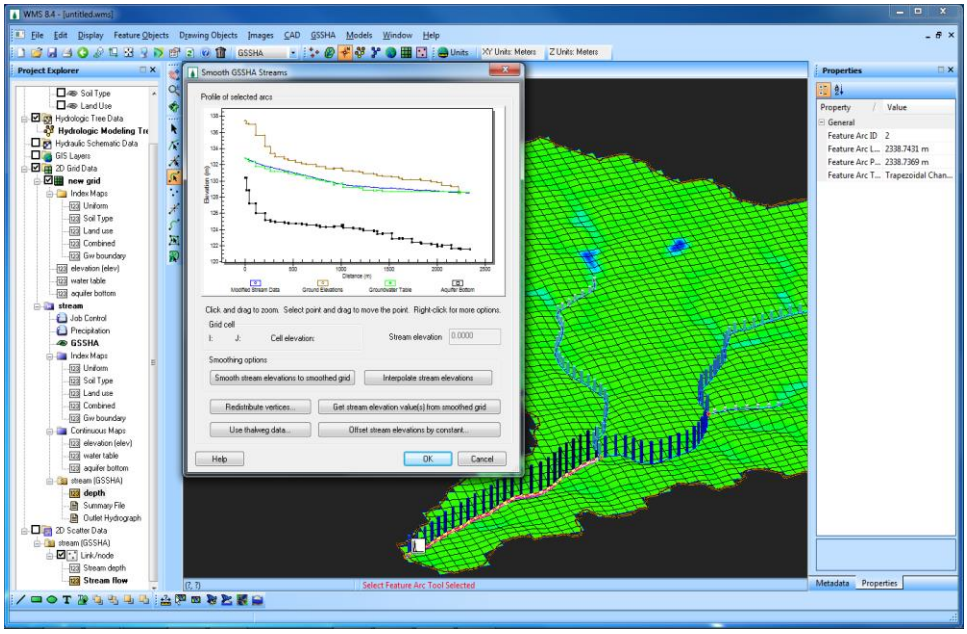
WMS

v. 11.0

WMS 11.0 Tutorial

Advanced Groundwater Modeling in GSSHA

Add advanced groundwater features to an existing GSSHA model



Objectives

This tutorial demonstrates how to add advanced groundwater interactions such as stream interaction, wells, and specified head boundary conditions to an existing GSSHA model.

Prerequisite Tutorials

- Groundwater Modeling in GSSHA

Required Components

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

Time

- 30–60 minutes

AQUAVEO™

1 Introduction

This tutorial discusses additional tools that can be used when creating a groundwater simulation using GSSHA. It will demostrate adding streams, wells, and additional boundary conditions to an existing GSSHA project.

2 Open an Existing GSSHA Project

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

1. Make the **2D Grid Module** active.
2. Select *GSSHA* | **Open Project File…** to bring up the *Open* dialog.
3. Locate the *data files* folder for this tutorial, and select the file "GWAdv.prj" file.
4. Click **Open** to import the project.
5. In the Project Explorer, turn off the display of "Map Data" then turn on the "GSSHA" coverage.

The project should appear similar to Figure 1:

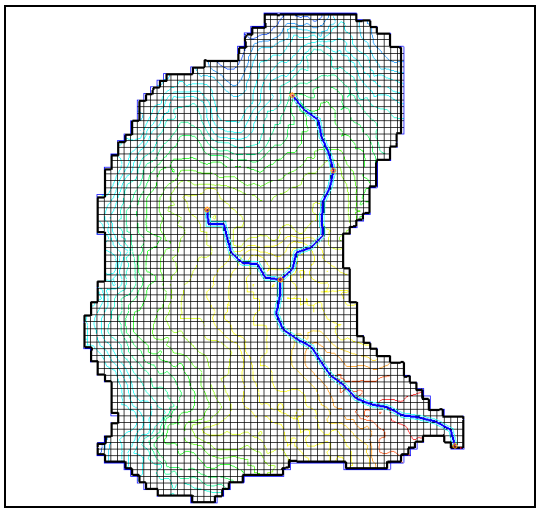


Figure 1 Initial project

3 Adding Streams to the Groundwater Model

The basic groundwater model developed in the previous workshop did not have the stream routing option turned on. This tutorial shows how to 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, subsurface flow from the channel bottom will also be enabled.

1. In the Project Explorer, select the "GSSHA" coverage to make it active.
2. Using the **Select Feature Line Branch Tool**, double-click at the downstream most arc of the channel network (the one that is connected to the watershed outlet) to open a *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 "0.5" m, *Bottom Width* to "1" m 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".
6. Click **OK** to close the *Properties* dialog.
7. Click **Yes** at the warning message.
8. Click **All** at the warning message to renumber the arcs.
9. With the **2D Grid Module** active, select *GSSHA* | **Job Control** to open the *GSSHA Job Control Paramters* dialog.
10. Select the *Diffusive Wave* option under *Channel routing computation scheme*.
11. Click **OK** to close the *GSSHA Job Control Paramters* dialog.
12. In the Project Explorer, under "2D Grid Data", select the "Gw Boundary" index map.

When selecting the index map, a single-colored index map will be visible on the WMS display which represents a generic groundwater boundary condition. This map needs to be regenerated for the river flux boundary condition just defined on the stream arcs.

13. Right-click on the "Gw boundary" index map and select **Regenerate**.

Notice the different-colored cells that represent the channel network.

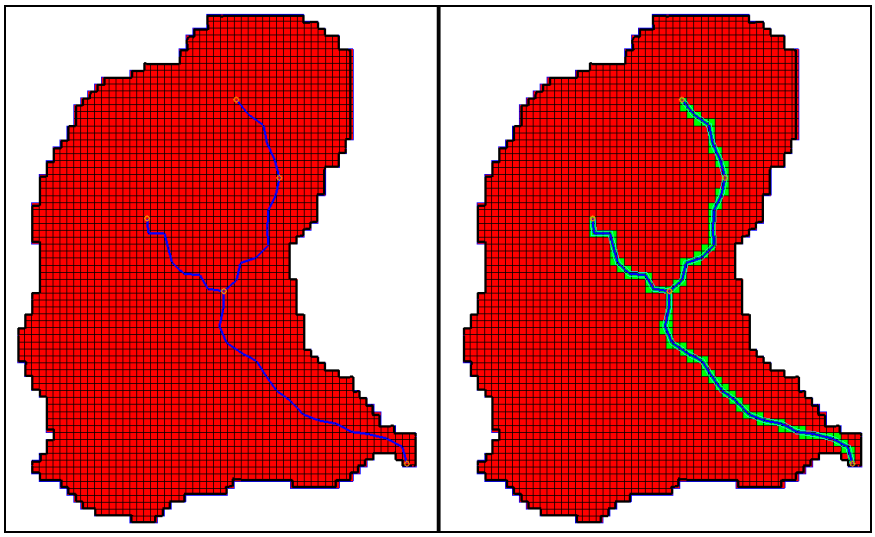


Figure 2 Stream added to index map

3.1 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, the final groundwater elevation from the previous workshop will be used as a hot start file for the starting groundwater head of this model.

1. In the Project Explorer, select the "2D Grid Data"folder to make it active.
2. Right-click on the "new grid" folder directly under "2D Grid Data" and select **Import Scalar dataset** to open the *File Formats* dialog.
3. Select *GRASS ASCII grid file* and click **OK** to close the File Formats dialog and bring up an *Open* dialog .
4. Select the file "GWHotStart.ggd" and click **Open** to import the scalar data.

In the Project Explorer, under "2D Grid Data" notice a "GWAdv" GSSHA model.

5. Right-click the "Continuous Maps" folder under "GWAdv" and select A*ssign* | **GWHotStart**.

This will list the "GWHotStart" dataset imported as a hot start file as one of the continuous maps for the "GWAdv" model.

6. Select *GSSHA* | **Job Control** to open the *GSSHA Job Control Paramters* dialog.
7. Next to the *Groundwater* option on the right of the dialog, click *Edit Paramters* to open the *GSSHA Groundwater* dialog.
8. Select "GWHotStart" for the *Water Table* option.

9. Click **OK** to close the *GSSHA Groundwater* dialog.
10. Click **Ok** to close the *GSSHA Job Control Paramters* dialog.

3.2 Save and Run the Model

Now to save and run the GSSHA model.

1. Select *GSSHA* | **Save Project File** to open the *Save GSSHA Project File* dialog.
2. Enter "GWAdv_final.prj" as the *File name* and click **Save**.
3. Select *GSSHA* | **Run GSSHA** to open the *GSSHA Run Options* dialog.
4. Click **OK** to start the *Model Wrapper* dialog.
5. When the model finishes running, click **Close** to exit the *Model Wrapper* and to load in the solution.

3.3 Visualize the Results

Once done running, review the results.

1. Using the **Select Hydrographs** tool, double-click on the hydrograph icon at the outlet location to bring up a hydrograph plot.

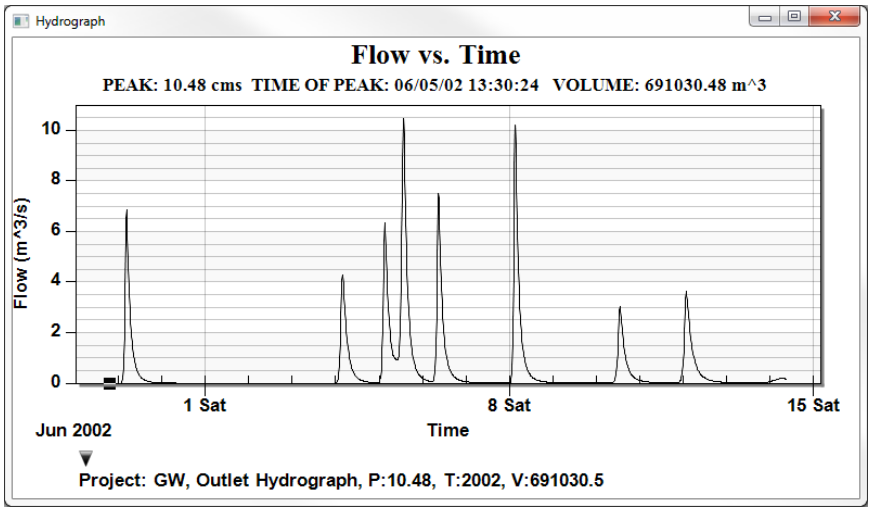


Figure 3 Hydrographs plot

2. When done reviewing the hydrograph, close the window.
3. In the Project Explorer, right-click the "groundwater_head" dataset and select **Contour Options** to open the *Contour Optionds* dialog.
4. Change the *Contour Method* to "Color Fill" and click **OK** to close the *Contour Options* dialog.
5. In the Project Explorer, select the "groundwater_head" dataset and toggle through the time steps to see how the groundwater head varied with time.

4 Add Wells to the Groundwater Model

GSSHA can be used to simulate the effects of pumping wells on groundwater heads. In this section, two wells will be added into the model being built.

1. Select *GSSHA* | **Save Project File** to open the *Save GSSHA Project File* dialog.
2. Enter "GWAdv_wells.prj" as the *File name* and click **Save**.
3. In the Project Explorer, select the "GSSHA" coverage to make it active.
4. Using the **Create Feature Point** tool, add points in the two locations shown in Figure 4.

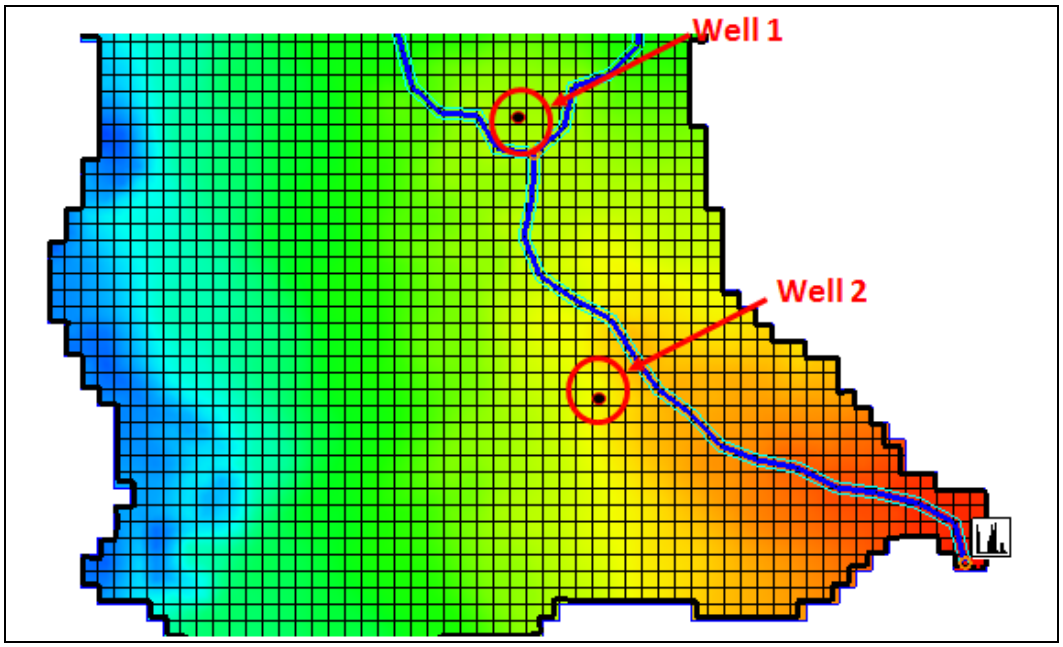


Figure 4 Well locations

5. Using the **Select Feature Point/Node** tool, double-click on Well 1 to open the *Properties* dialog.
6. In the *Groundwater BC* column, select "Static Well".
7. Enter "2.55" m^3/s for the *Pumping Rate*.
8. Click **OK** to close the *Properties* dialog.
9. Using the **Select Feature Point/Node** tool, double-click on Well 2 to open the *Properties* dialog.
10. In the *Groundwater BC* column, select "Dynammic Well".
11. Under *Pump Rate*, click on the ... button to open the *XY Series Editor* window.
12. Outside of WMS, browse and open the spreadsheet "PumpingRate.xls".
13. Copy all the data and paste it into the *XY Series Editor*.
14. Click **OK** to close the *XY Series Editor* dialog.
15. Click **OK** to close the *Properties* dialog.

16. In the Project Explorer, right-click on the "Gw boundary" index map and select **Regenerate**.

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.

4.1 Import the Hotstart File for the Groundwater Table

1. In the Project Explorer, select the "2D Grid Data"folder to make it active.
2. Right-click on the "new grid" folder directly under "2D Grid Data" and select **Import Scalar dataset** to open the *File Formats* dialog.
3. Select *GRASS ASCII grid file* and click **OK** to close the *File Formats* dialog and bring up an *Open* dialog .
4. Select the file "GWHotStart.ggd" and click **Open** to import the scalar data.
5. In the Project Explorer, right-click on the hot start file "GWHotStart (2)" and select **Rename**.
6. Enter "GWAdv" as the new name and press *Enter*.

In the Project Explorer, under "2D Grid Data", notice the "GWAdv_wells" GSSHA project.

7. Right-click the "Continuous Maps" folder under "GWAdv_wells" and select A*ssign* | **GWAdv**.

This will list the "GWAdv" dataset imported as a hot start file as one of the continuous maps for the "GWAdv_wells" model.

8. Select *GSSHA* | **Job Control** to open the *GSSHA Job Control Paramters* dialog.
9. Next to the *Groundwater* option on the right of the dialog, click *Edit Paramters* to open the *GSSHA Groundwater* dialog.
10. Select "GWAdv" for the *Water Table* option.
11. Click **OK** to close the *GSSHA Groundwater* dialog.
12. Click **OK** to close the *GSSHA Job Control Paramters* dialog.

4.2 Save and Run the Model

The model is now ready to run.

1. Select *GSSHA* | **Save Project File** to open the *Save GSSHA Project File* dialog.
2. Enter "GWAdv_wells_final.prj" as the *File name* and click **Save**.
3. Select *GSSHA* | **Run GSSHA** to open the *GSSHA Run Options* dialog.
4. Click **OK** to start the *Model Wrapper* dialog.
5. When the model finishes running, click **Close** to exit the *Model Wrapper* and to load in the solution.

4.3 Visualize the Results

Once done running, review the results.

1. Using the **Select Hydrographs** tool, double-click on the hydrograph icon at the outlet location to bring up a hydrograph plot.
2. When done reviewing the hydrograph, close the window.
3. In the Project Explorer, right-click the "groundwater_head" dataset and select **Contour Options** to open the *Contour Optionds* dialog.
4. Change the *Contour Method* to "Color Fill" and click OK to close the *Contour Options* dialog.
5. In the Project Explorer, select the "groundwater_head" dataset and toggle through the time steps to see how the groundwater head varied with time.

5 Add Outside Boundary Conditions

In all of the groundwater models built so far, it has been assumed that the watershed boundary is a "No Flow" boundary condition. In this exercise, a specified head boundary condition will be added to the watershed boundary.

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

Start with creating a new arc that represents the specified head boundary.

1. Select *GSSHA* | **Save Project File** to open the *Save GSSHA Project File* dialog.
2. Enter "GWAdv_Bdry.prj" as the *File name* and click **Save**.
3. In the Project Explorer, select the "Geology" dataset to make it active.
4. In the Project Explorer, select the "GSSHA" coverage to make it active.
5. Using the **Create Feature Arc** tool, draw an arc starting from the point shown in Figure 5 and intersect each grid cell along the boundary until reaching the end point.

While drawing the boundary arc, follow these guidelines:

- Do not intersect the arc being drawn with the watershed boundary arc. This will end the arc at the point of intersection.
- Make sure to intersect each grid cell along the edge of the watershed model.
- Make sure to 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 the view on the section of the working area on screen.

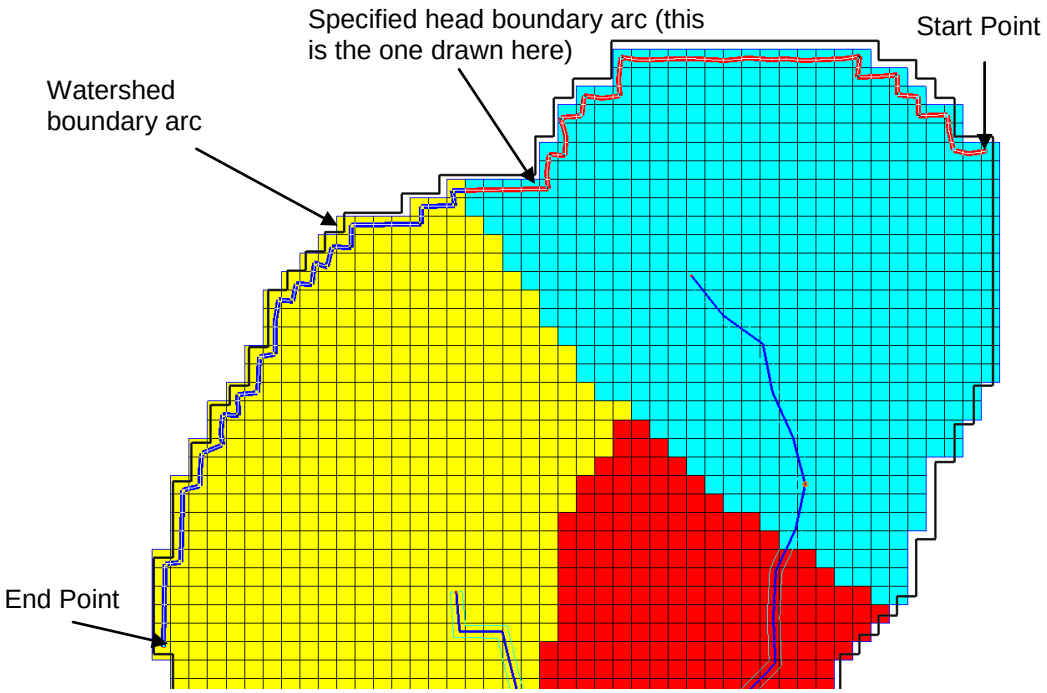


Figure 5 Feature arc path

6. Once the arc has been traced, using the **Select Feature Arc** tool, double-click on the arc to open the *Properties* dialog.
7. Change *Groundwater BC* to "Constant head".

GSSHA will read the head elevation from the water table data.

8. Click **OK** to close the *Properties* dialog.
9. In the Project Explorer, right-click on the "Gw boundary" index map and select **Regenerate**.

If properly created, the specified head boundary will be displayed for the "Gw boundary" index map as shown in Figure 6.

NOTE: if a few cells are missed along the boundary, edit the location of the vertices along the arc using **Select Feature Vertex** tool and regenerate the "Gw boundary" index map.

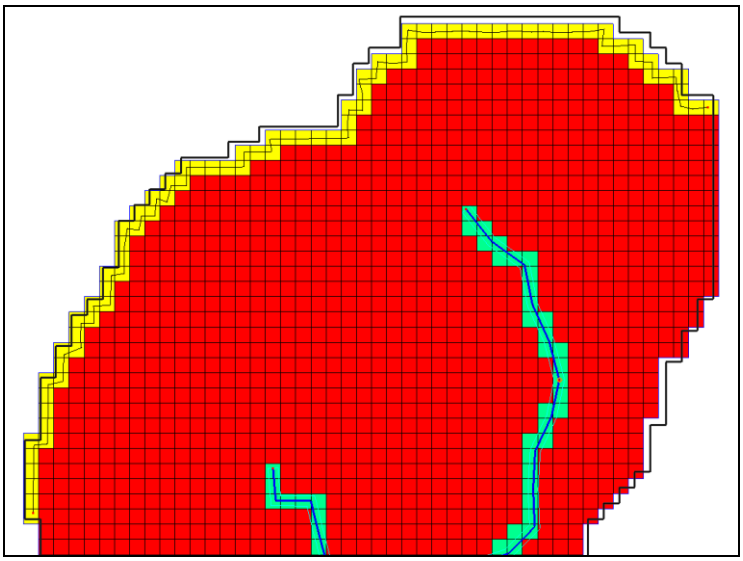


Figure 6 Added boundary condition

5.1 Import Hotstart File for Groundwater Elevations

1. In the Project Explorer, select the "2D Grid Data"folder to make it active.
2. Right-click on the "new grid" folder directly under "2D Grid Data" and select **Import Scalar dataset** to open the *File Formats* dialog.
3. Select *GRASS ASCII grid file* and click **OK** to close the *File Formats* dialog and bring up an *Open* dialog .
4. Select the file "GWHotStart.ggd" and click **Open** to import the scalar data.
5. In the Project Explorer, right-click on the hot start file "GWHotStart (2)" and select **Rename**.
6. Enter "GWAdvWells" as the new name and press *Enter*.

In the Project Explorer, under "2D Grid Data", notice the "GWAdv_Bdry" GSSHA project.

7. Right-click the "Continuous Maps" folder under "GWAdv_Bdry" and select A*ssign* | **GWAdvWells**.

This will list the "GWAdvWells" dataset imported as a hot start file as one of the continuous maps for the "GWAdv_Bdry" model.

8. Select *GSSHA* | **Job Control** to open the *GSSHA Job Control Paramters* dialog.
9. Next to the *Groundwater* option on the right of the dialog, click *Edit Paramters* to open the *GSSHA Groundwater* dialog.
10. Select "GWAdvWells" for the *Water Table* option.
11. Click **OK** to close the *GSSHA Groundwater* dialog.
12. Click **OK** to close the *GSSHA Job Control Paramters* dialog.

5.2 Save and Run the Model

The model is now ready to run.

1. Select *GSSHA* | **Save Project File** to open the *Save GSSHA Project File* dialog.
2. Enter "GWAdv_Bdry_final.prj" as the *File name* and click **Save**.
3. Select *GSSHA* | **Run GSSHA** to open the *GSSHA Run Options* dialog.
4. Click **OK** to start the *Model Wrapper* dialog.
5. When the model finishes running, click **Close** to exit the *Model Wrapper* and to load in the solution.

5.3 Visualize the Results

Once done running, review the results.

1. Using the **Select Hydrographs** tool, double-click on the hydrograph icon at the outlet location to bring up a hydrograph plot.
2. When done reviewing the hydrograph, close the window.
3. In the Project Explorer, right-click the " groundwater_head" dataset and select **Contour Options** to open the *Contour Optionds* dialog.
4. Change the *Contour Method* to "Color Fill" and click OK to close the *Contour Options* dialog.
5. In the Project Explorer, select the " groundwater_head" dataset and toggle through the time steps to see how the groundwater head varied with time.

6 Conclusion

This concludes the "Advanced Groundwater Modeling with GSSHA" tutorial.