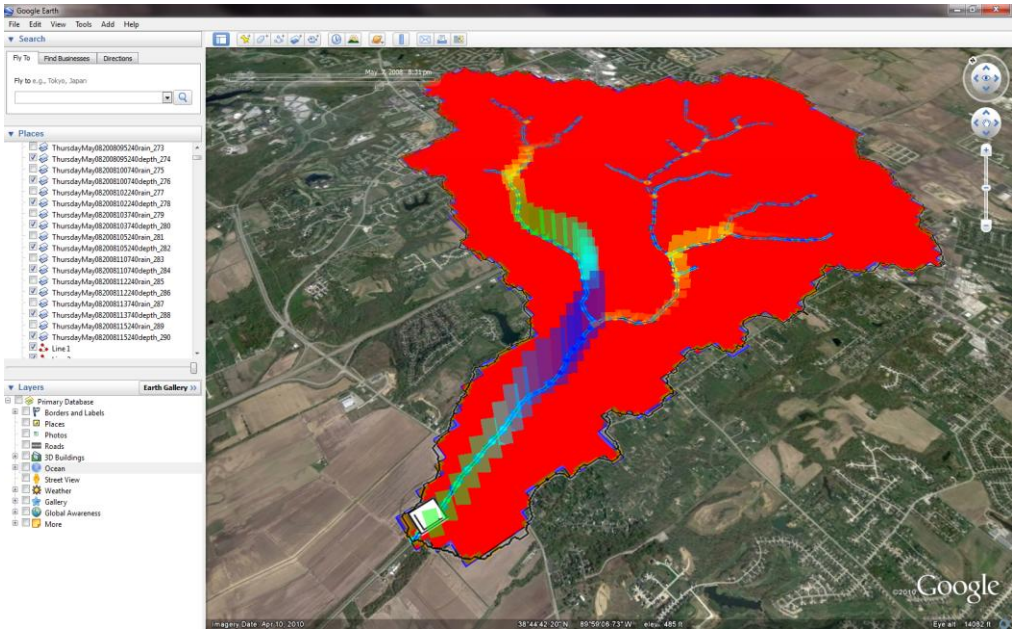
WMS

v. 9.1

WMS 9.1 Tutorial

GSSHA – Modeling Basics – Post-Processing and Visualization of GSSHA Model Results

Learn how to visualize GSSHA model results



Objectives

This tutorial demonstrates different ways of visualizing the output from GSSHA. You will learn how to view contours and channel depths, create animations, export these animations to Google Earth, and how to compare observed and computed data after a GSSHA model run.

Prerequisite Tutorials

- GSSHA – Modeling Basics – Developing a GSSHA Model Using the Hydrologic Modeling Wizard in WMS

Required Components

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

Time

- 20-30 minutes

AQUAVEO™

1 Contents

2 Introduction

In this tutorial, you will explore different post-processing options. You have already been using some of these post-processing tools in the previous tutorials. This tutorial explains and leads you through exploring some of the post processing tools in WMS.

3 Open an Existing Project

1. Start WMS. If you are continuing from a previous tutorial close and restart WMS.
2. Locate the ***images*** folder in your tutorial files. If you have used default installation settings in WMS, the tutorial files will be located in ***\My documents\WMS 9.1\Tutorials\.***
3. In the *2D Grid Module*, select ***GSSHA | Open Project File.*** Browse to and open the file ***\GSSHA Distributed Hydrologic modeling\ Visualization\visualization.prj.***

This Judy's Branch project models distributed roughness, infiltration, and distributed precipitation. Notice several precipitation gages covering the watershed area.

4. Because of the land use and soil type coverages as well as the precipitation gages, the display looks cluttered and the screen refreshes slowly. Turn off the display of all items under *Coverages* in the WMS *Project Explorer* window. Then turn the display of the *GSSHA* coverage back on. Select the *Frame* button to zoom into the GSSHA coverage after the GSSHA coverage is turned on. See the following figure:

4 Read Solution

WMS stores the results of a run together as a solution set (the input data is not a part of the solution, only the output data). There can be many solution sets in the project explorer but they must be for the same grid and streams. For example, you could vary roughness parameters in a single GSSHA model and have solution sets for each set of roughness parameters and then you can compare results for each set of input parameters.

Since this project has already been run, you do not have to run it in order to view the results. However, the results are not read when you read the project file, so you need to tell WMS to read the solution files.

1. In the *2D grid* module, select ***GSSHA | Read Solution..***
2. A dialog box pops up here that allows you to select a simulation (though the current project is there by default). Select *OK* in this dialog to start reading the solution.
3. Notice that solutions are added to the project explorer under the *2D Grid Data* and the *2D Scatter Data* folders.

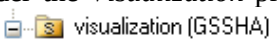
5 Visualization

5.1 Visualizing the Outlet Hydrograph

As soon as the result is read, you can see a small hydrograph icon at the outlet of your watershed. Clicking on this icon shows the outflow hydrograph in a plot window.

1. In the 2-D grid module, click on the "***Select Hydrograph***" tool and double click on the small hydrograph icon near the outlet.
2. Clicking on the icon opens the hydrograph in a plot window.
3. There are several options to control the display of the hydrograph plot, including options to view and export the plotted values by right clicking on the hydrograph plot and selecting the appropriate menu item.
4. The simplest way to copy the plot's data to a spreadsheet (as you have seen in previous tutorials) is to choose the *View Values* option and then *copy\paste* selected values to a spreadsheet.
5. *Close* the Hydrograph plot window when you are done.

5.2 Reading the Summary File

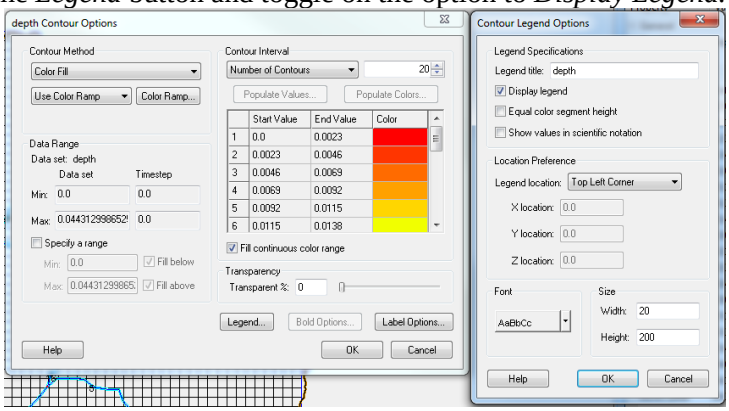
Notice that there is a folder named *visualization* under the *Visualization* project folder. The solution folder has a symbol 's' for solution 

. Notice the summary file under this project's solution.

1. Double-click on the *Summary File* under the solution folder.
2. If WMS asks for your editor just click *OK*.
3. Look through the summary file. It is good to check things like the mass balance error and the volume remaining on the surface to know that GSSHA is simulating the processes correctly.
4. When you are done, close the summary file

5.3 Visualizing Depth Contours

1. In the *Project Explorer*, right-click on *Depth* under the *visualization* folder.
2. Select *Contour Options* (this menu item can also be found under the *Display* menu or by using the Contour Options macro )
3. Under the Contour Method select the option to *Color fill* from the drop down box.
4. Click the *Legend* button and toggle on the option to *Display Legend*.



5. Click *OK* on the *Contour Legend Options* dialog.
6. Click *OK* on the *Contour Options* dialog.
7. In the *Properties* window (to the right side of the WMS window), a set of time steps appear. If the time steps are not showing, either the *Depth* data set is not selected in the project explorer or something else has been selected since selecting the Depth dataset. If this happens, click somewhere outside the watershed boundary and select the *Depth* data set with your left mouse button.
8. Click on first time step and use the down arrow key (on the keyboard) to cycle through the time steps. About half way through the time steps, notice that the depth contours vary in color at different time steps. You do not have to go through the time steps consecutively; you can select any time step to make it active. This model is actually not that interesting for surface depth because it "drains" very well so it takes a few time steps before you see any changes and even then the changes are modest.
9. Notice that the legend is also displayed.

You can also adjust the lighting (***Display | Display Options | Lighting Options***) and the vertical exaggeration (***Display | View | Z Magnification***).

Try panning  , zooming  , and rotating  .

5.4 Visualizing Stream Flow Results

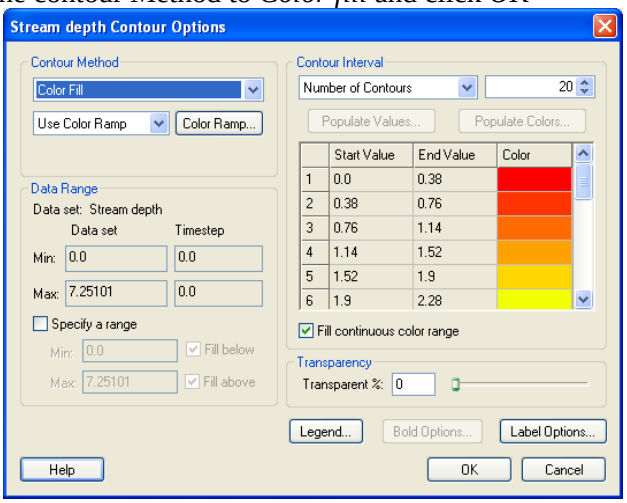
While the streams are connected to the overland flow plane, they represent a separate but coupled model. The depth and flow outputs from this stream hydraulic model are in a format called the link/node data set format. These files hold a data value for every node (arc segment) of every link at the same time step as that for the gridded output data. Two of the most common files in this format are the channel depth file (*.cdp) and the channel discharge file (*.cdq.).

These stream data sets can be visualized in WMS. You can create depth contours, flood histograms, or a movie of how these values vary with time. In this tutorial, you will see some of these WMS visualization tools.

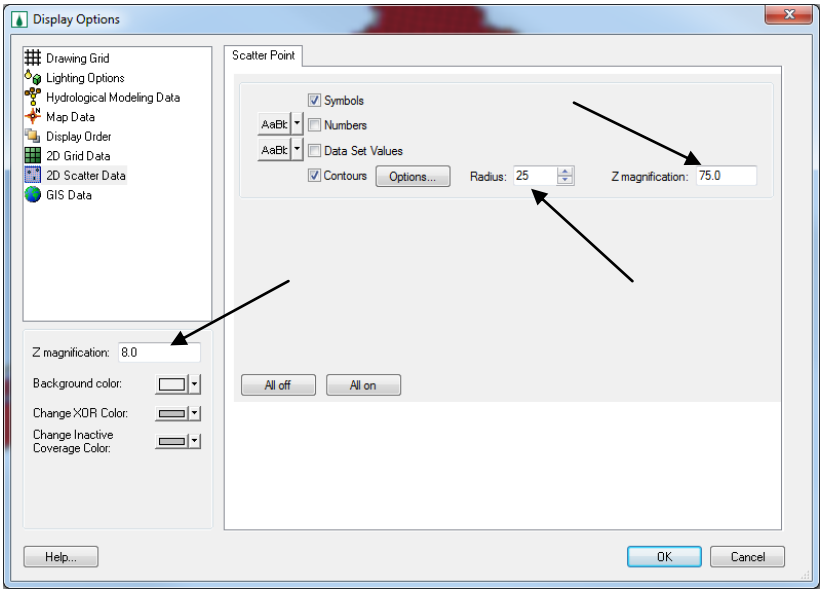
In the *2D Scatter Data* folder in the project explorer, you'll see two data sets, *Stream Depth, and Stream Flow*. The Stream Depth and Stream Flow data sets are link/node data sets. In order to visualize stream depth and\or stream flow, follow these steps:

Visualizing Depth Contours

1. In the project explorer click on *Stream Depth* to select it.
2. Select ***Display | Display Options***. In the 2D Scatter Data window, toggle off the display of *Symbols* and toggle on the display of *Contours* and click on *Options*.
3. Change the contour Method to *Color fill* and click *OK*

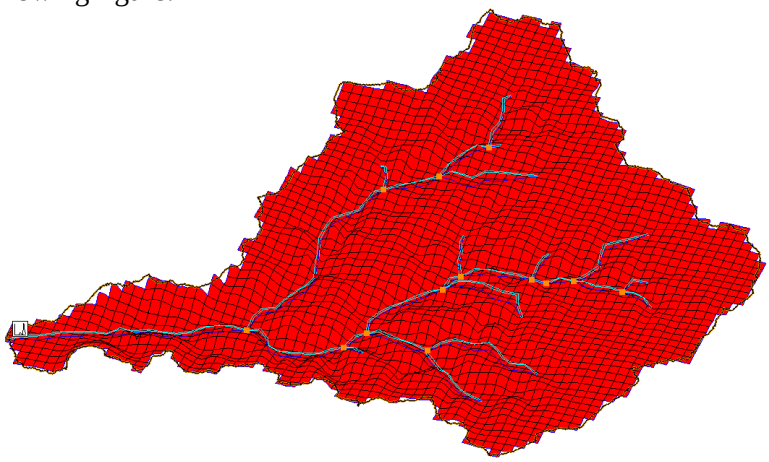


4. In the *Display Options dialog*, change the *Radius* to 25 and Z-magnification to 75.

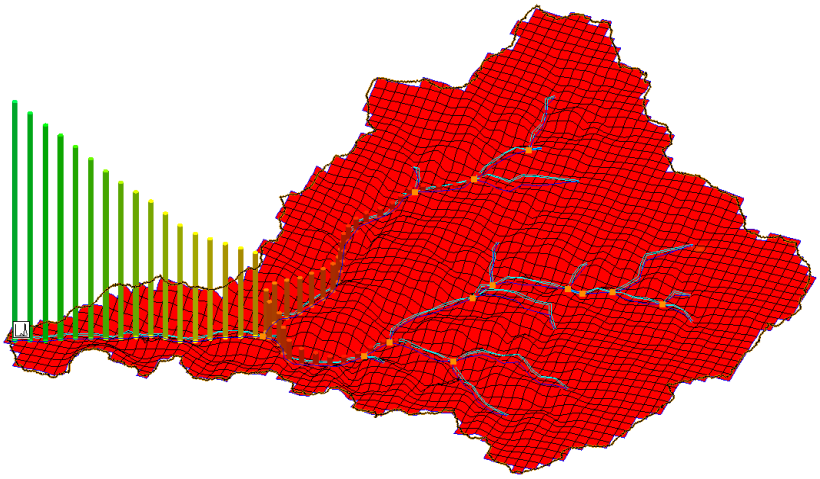


5. On the left side of the dialog, uncheck the *Auto z-mag* box, set the Z-magnification to 8 and Click *OK*

6. Select the "*Rotate*" tool. Rotate the watershed by clicking on the left mouse button and dragging in the window until it looks similar to the following figure:



7. Now click on the *Stream Flow* in the Project Explorer to select it and cycle through the different time steps (shown on the properties window at the right side of WMS main window). Notice the bar diagrams representing the flood wave being generated and traveling along the streams to the outlet.

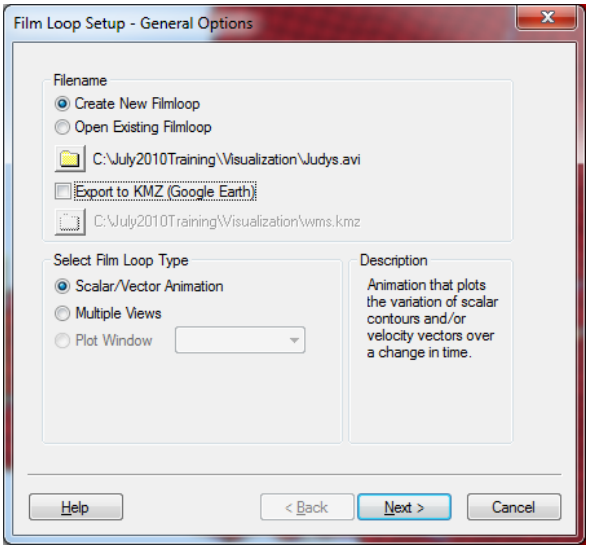


Notice the overland flow depth contour changing with the flood wave bar diagrams. This gives a clearer idea of the link between overland flow and stream flow.

5.5 Creating an Animation Film Loop

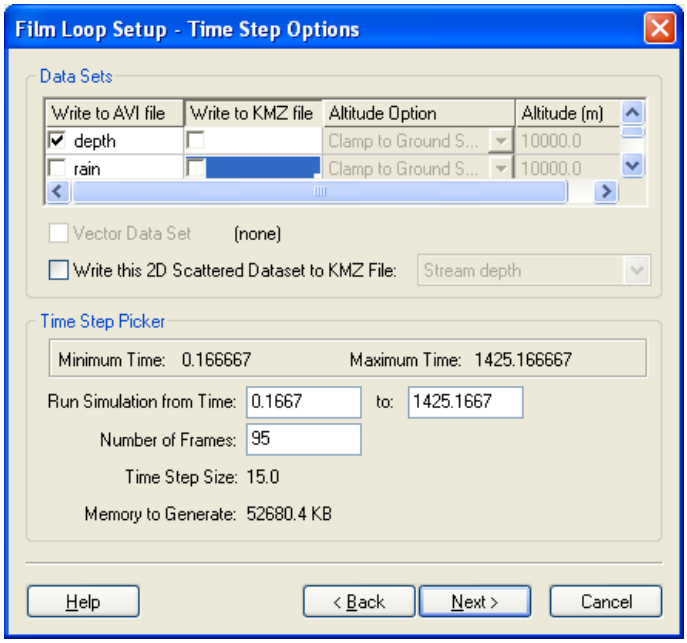
These processes can be animated using the film loop command.

1. In the 2D-Grid Module, select ***Data | Film Loop…***
2. Make sure to select "*Create New Filmloop*" and choose the location to save your AVI movie file by clicking on the *Browse* button. Save your movie as ***\GSSHA Distributed Hydrologic modeling\Personal \Visualization\Judys.avi***
3. We will NOT export Google Earth animation at this moment. Make sure the *Export to KMZ* option is checked OFF.



4. Select the *Scalar\Vector Animation* option and click *Next*.

5. Under *Write to AVI file*, check *depth* data set to select it. Turn off all the boxes under *Write to KMZ file.*
6. Turn off the *Write this 2D Scattered Dataset to KMZ File* option.



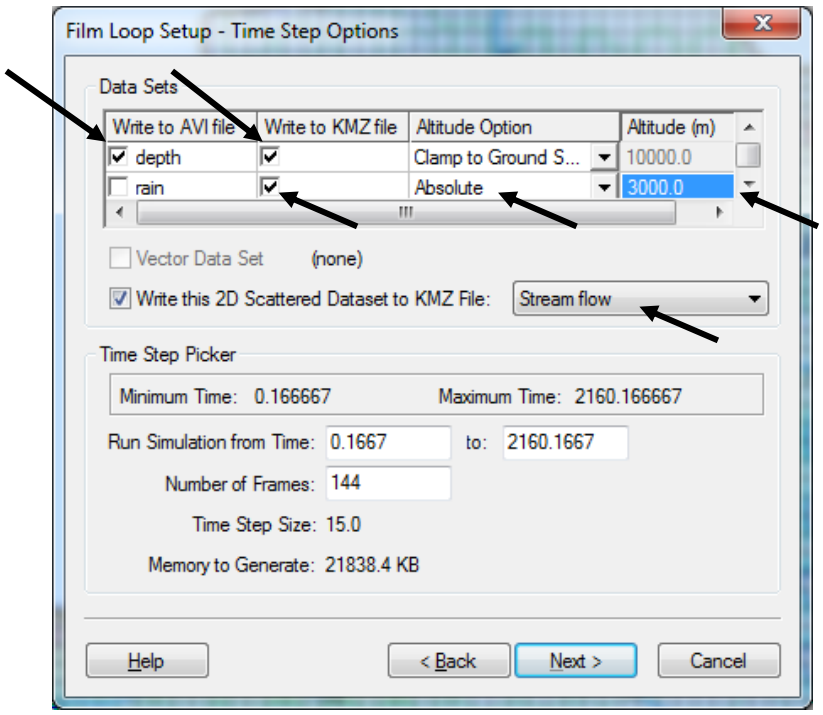
7. Click *Next*.
8. Do not change any clock options. Click "*Finish*". WMS takes some time to create the movie. The movie begins playing once the entire movie is saved. In the movie, you can see the overland flow depth contours and stream flood waves animated simultaneously. The AVI file is saved and can be put inside a presentation or played separately as needed. Close the animation player when you are done viewing it.

5.6 Creating a Google Earth Animation

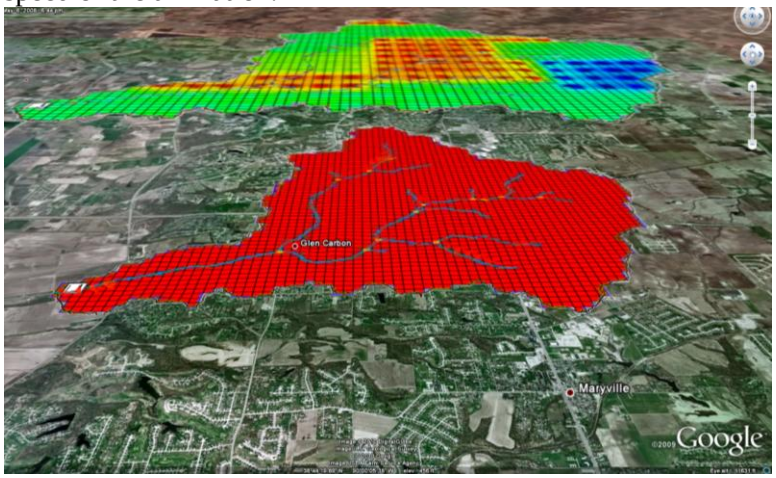
You can export an animation to Google Earth and display your animation in its real-world location.

1. Change the display to plan view by clicking *Plan View*  . You cannot export a movie to Google Earth in an oblique view.
2. In the 2D-Grid Module  , select ***Data | Film Loop…***
3. Choose the location to save your AVI movie file by clicking on the *Browse* button  . Save your movie as ***\GSSHA Distributed Hydrologic modeling\Personal\Visualization\Judys.avi***
4. Select the option *Export to KMZ (Google Earth)*. Define the location to save the KMZ file as ***\GSSHA Distributed Hydrologic modeling\ Personal\Visualization\ Judys.kmz***
5. Toggle the on "*Scalar\Vector Animation*" option and click Next.
6. Under *Write to AVI file,* turn on the *depth* data set to select it.
7. Turn on the toggle boxes under *Write to KMZ file* corresponding to both the *Depth* and *Rain* datasets.

8. For the *Rain* dataset, select *Absolute* under *Altitude Option* and enter 3000 meters for *Altitude.*
9. Check on the option *Write this 2D Scattered Dataset to KMZ File* and select *Stream flow*.



10. Click *Next*.
11. Do not change any clock options. Click "*Finish*". WMS will take some time to (3-5 minutes) to create the movie. After the movie is completed, WMS opens the Google Earth animation.
12. Once in Google Earth, tilt the Google Earth view to see the animated Stream Flow lines. Play around with some of the Google Earth display options, such as changing the transparency of the data and changing the speed of the animation.



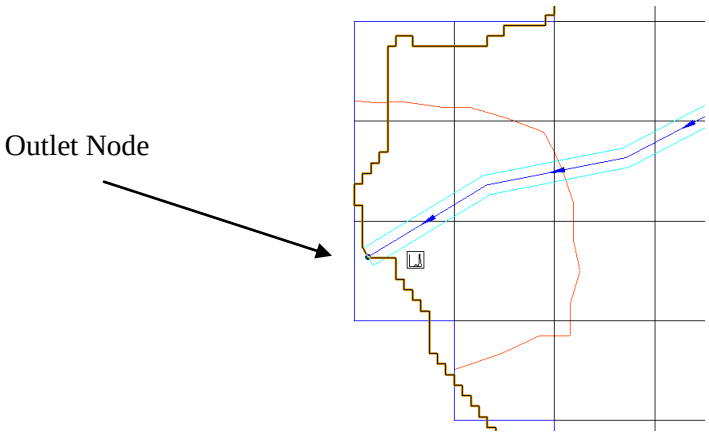
The upper layer represents the distributed rainfall input and the lower layer represents overland runoff.

6 Comparing the simulation results with the Observed data

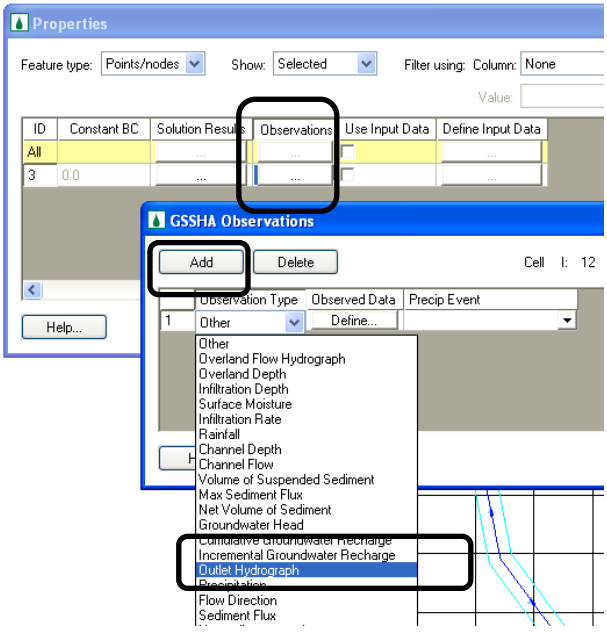
If you have some measured data at the watershed outlet or at any stream node upstream in the watershed, you can enter the measured data and compare the simulation result with that data.

We will use an arbitrary observed flow hydrograph at the outlet and compare it with simulation results.

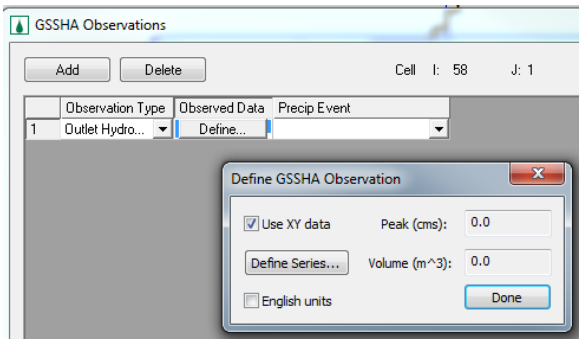
1. Zoom in around the watershed outlet.
2. In the map module, select the *Select Feature Point\Node* tool and double click the outlet node.



3. In the *Properties* dialog that opens, scroll all the way to the right and under *Observations* select the browse button.



4. In *GSSHA Observations* dialog, click *Add* and select *Outlet Hydrograph* under *Observation Type*.
5. Click the *Define* button.
6. Check *Use XY data option* and click the *Define Series* button.



7. The observed flow data can be found at ***\GSSHA Distributed Hydrologic modeling\Raw Data\JudysBranch\Judys_ObsFlow.txt.***
8. In Notepad, open the ***Judys_ObsFlow.txt*** file and copy and paste the data into the XY Series editor in WMS. You should see the hydrograph plotted on the XY-series editor.
9. Click OK, click *Done*, and click OK. This will close all but the properties dialog.
10. Click the Browse button under *Solution Results* (just to the left of *Observations).*
11. You can now compare the observed flow data with the simulation results in the *GSSHA Solution Analysis* dialog by clicking on the *Observed (Outlet Hydrograph)* and the *visualization (GSSHA) Stream flow (Channel Flow)* check boxes.