WMS

v. 11.0

WMS 11.0 Tutorial

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. It instructs 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

- Developing a GSSHA Model Using the Hydrologic Modeling Wizard

Required Components

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

Time

- 20–30 minutes

AQUAVEO™

1 Introduction

This tutorial explores different post-processing options. Previous tutorials have already used some of these post-processing tools. This tutorial demonstrates more of the post-processing tools in WMS.

1.1 Getting Started

Open a WMS project file for the Judy's Branch 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 "visualization.prj" file.
4. Click **Open** to import the project.

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

Because of the land use and soil type coverages as well as the precipitation gages, the display looks cluttered and the screen refreshes slowly.

5. In the Project Explorer, turn off the display of " Map Data" then turn on the " GSSHA" coverage.
6. Click the **Frame** macro to zoom into the GSSHA project.

The Main Graphics Window should appear similar to Figure 1.

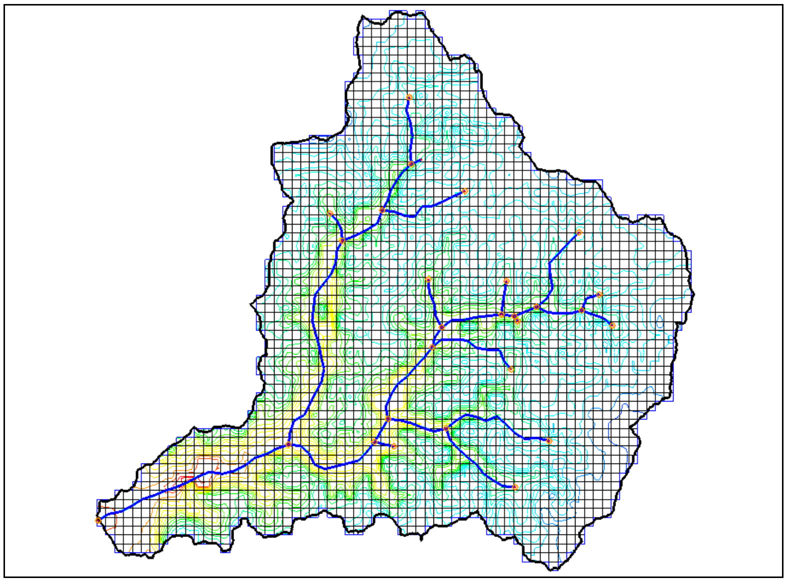


Figure 1 Initial project

1.2 Reading a 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, vary roughness parameters in a single GSSHA model and have solution sets for each set of roughness parameters and then compare results for each set of input parameters.

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

1. In the 2D grid module, select *GSSHA* | **Read Solution** to open the *Read GSSHA Solution* dialog.

The solution file for the current project should already be seleced.

2. Click **OK** to close the *Read GSSHA Solution* dialog and import the solution into WMS.

Notice that solutions are added to the Project Explorer under the " 2D Grid Data" and the " 2D Scatter Data" folders.

2 Visualizing the Outlet Hydrograph

After the results have been imported, a small hydrograph icon is visible at the outlet of the watershed. Clicking on this icon shows the outflow hydrograph in a plot window.

1. Using the **Select Hydrograph** tool, double-click on the small hydrograph icon near the outlet to bring up a plot window.

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.

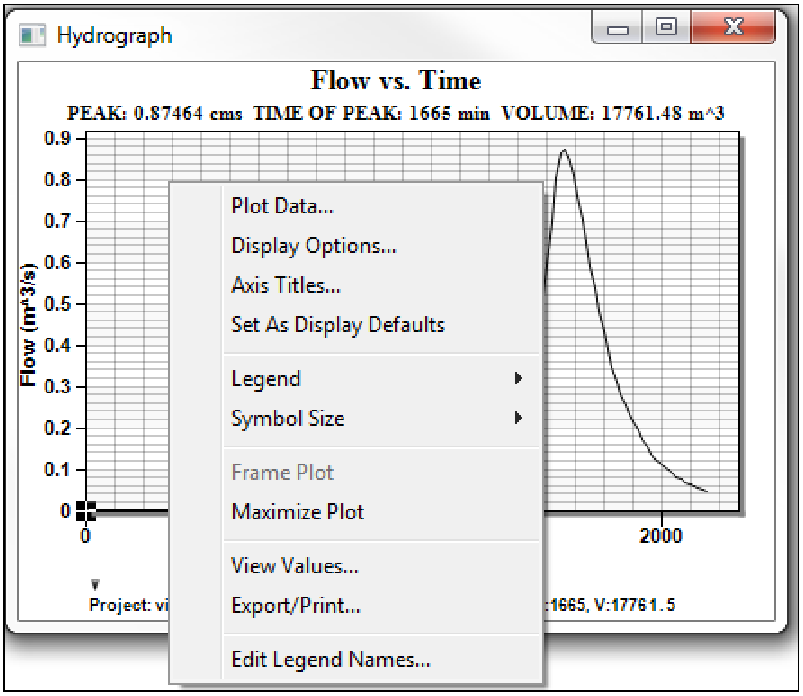


Figure 2 Right-click menu in the hydrograph plot window

The simplest way to copy the plot's data to a spreadsheet is to open the *View Values* dialog then copy and paste selected values to a spreadsheet.

2. When done reviewing the hydrograph , close the hydrograph plot window.

3 Reading the Summary File

Notice that there is a solution folder named " visualization (GSSHA)" under the " visualization" project folder. Notice the summary file under this project's solution.

1. Double-click on the " Summary File" under the solution folder.
2. If the *View Data File* dialog appears, select a text editor and click **OK**.

After the text eidtor opens, 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.

3. When done, close the summary file and return to WMS.

4 Visualizing Depth Contours

Changing the contours display options can increase the visibility of dataset features.

1. In the Project Explorer, right-click on " Depth" under the " visualization (GSSHA)" folder and select **Contour Options** to open the *depth Contour Options* dialog.

This dialog can also be reached through the *Display* | **Contour Options** menu command or by using the **Contour Options** macro.

2. Change *Contour Method* to "Color fill".
3. Click the **Legend** button to open the Contour Legend Options dialog.
4. Turn on the *Display Legend* option.
5. Click **OK** to close the *Contour Legend Options* dialog.
6. Click **OK** to close the *Contour Options* dialog.

In the *Properties* window to the right side of the WMS window, a set of time steps will appear. If the time steps are not showing, either the " Depth" dataset 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" dataset.

7. 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 (Figure 3). It's not required to go through the time steps consecutively; 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 noticing any changes and even then the changes are modest.

Notice that the legend is also displayed.

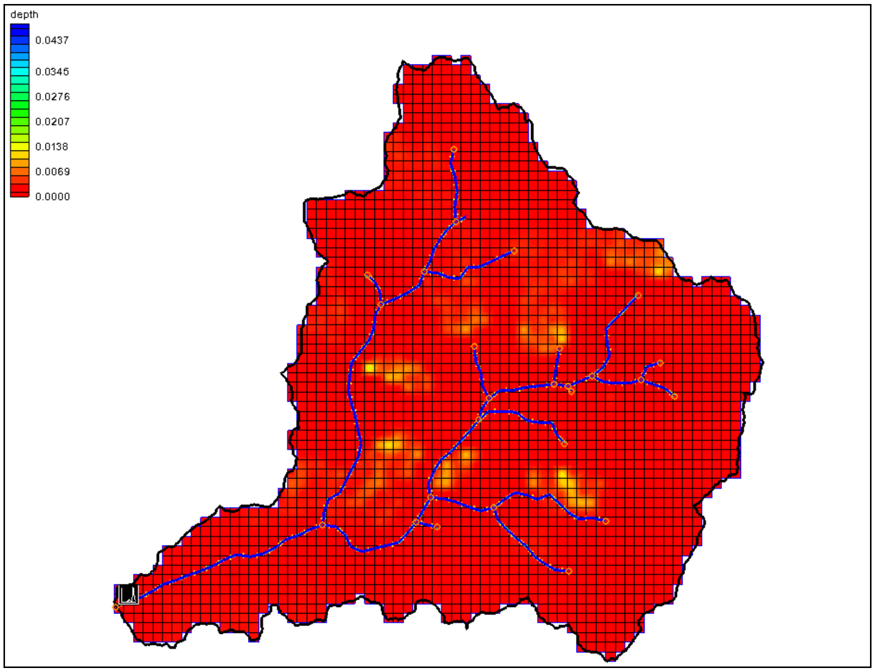


Figure 3 Depth dataset with contours changed to color fill

5 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 dataset 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 datasets can be visualized in WMS by creating depth contours, flood histograms, or a movie of how these values vary with time.

In the " 2D Scatter Data" folder in the Project Explorer, notice two datasets: " Stream depth", and " Stream flow". The " Stream depth" and " Stream flow" datasets are link/node datasets. In order to visualize stream depth and\or stream flow, follow these steps:

1. In the Project Explorer, click on " Stream depth" to make it active.
2. Select *Display* | **Display Options** to open the *Display Options* dialog.
3. On the left side of the dialog, uncheck the *Auto z-mag* box, set the *Z-magnification* to "8".
4. Select *2D Scatter Data* from the list on the left.
5. Turn off the *Symbols* option and turn on the *Contours* option.
6. Change the *Radius* to "25" and the *Z-magnification* to "75".
7. Click **Options** next to *Contours* to open the *Stream depth Contour Options* dialog.
8. Change the *Contour Method* to "Color fill".
9. Click **OK** to close the *Stream depth Contour Options* dialog.
10. Click **OK** to close the *Display Options* dialog.
11. Using the **Rotate** tool, rotate the watershed in the Graphics Window until it looks similar to Figure 4.

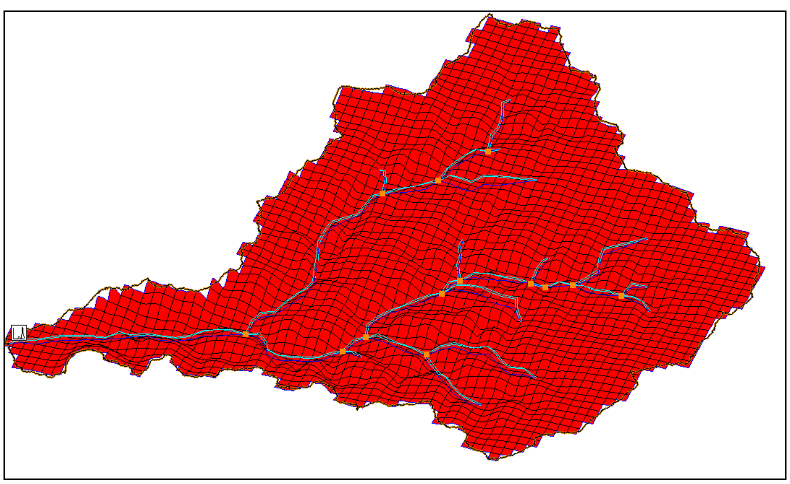


Figure 4 Rotated view of the stream depth dataset

12. In the Project Explorer, click on the " Stream flow" to make it active.
13. Cycle through the different time in the Properties window.

Notice the bar diagrams representing the flood wave being generated and traveling along the streams to the outlet.

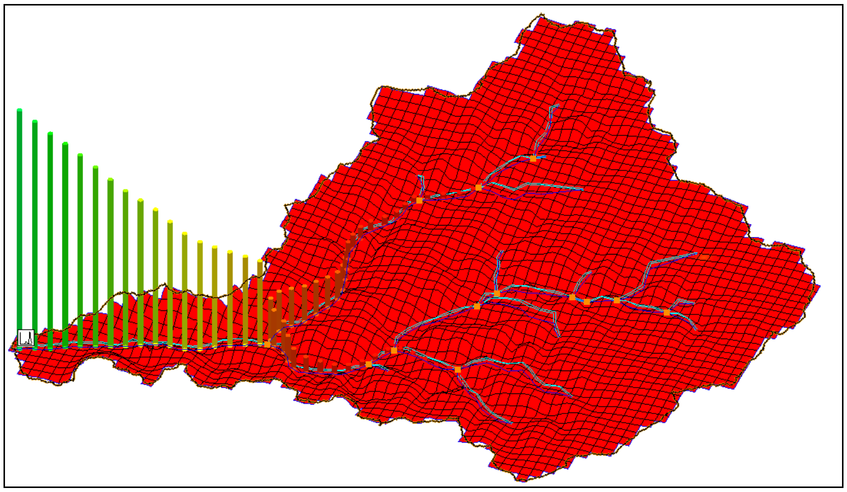


Figure 5 Flood wave diagram

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.

6 Creating an Animation Film Loop

These processes can be animated using the film loop command.

1. Make the **2D Grid Module** active.
2. Select *Data* **| Film Loop** to open the *Film Loop Setup* wizard.
3. Select *Create New Filmloop* and uncheck *Export to KMZ (Google Earth)*.
4. Click the **Browse** button to open a *Save* dialog.
5. For the *File name* enter "Judys.avi" then click **Save**.
6. Click **Next** to go to the *Time Step Options* of the *Film Loop Setup* wizard.
7. Under *Write to AVI file*, check *depth* dataset to select it.
8. Turn off all the boxes under *Write to KMZ file*.
9. Turn off the *Write this 2D Scattered Dataset to KMZ File* option.
10. Click **Next** to go to the *Disply Options* of the *Film Loop Setup* wizard.
11. Do not change any options and click **Finish** to close the *Film Loop Setup* wizard and generate the animation.

WMS takes some time to create the movie. The movie begins playing once the entire movie is saved. The movie shows 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.

12. Close the animation player when done viewing it.

7 Creating a Google Earth Animation

Export an animation to Google Earth and display the animation in its real-world location.

1. Click the **Plan View** macro to switch the view.

A movie in an oblique view cannot be exported to Google Earth.

2. Make the **2D Grid Module** active.
3. Select *Data* | **Film Loop** to open the *Film Loop Setup* wizard.
4. Select *Create New Filmloop*.
5. Click the **Browse** button to open a *Save* dialog.
6. For the *File name* enter "Judys.avi" then click **Save**.
7. Turn on the *Export to KMZ (Google Earth)* option.
8. Under *Export to KMZ (Google Earth)*, click the **Browse** button to open a *Save* dialog.
9. For the *File name* enter "Judys.kmz" then click **Save**.
10. Click **Next** to go to the *Time Step Options* of the *Film Loop Setup* wizard.
11. Under *Write to AVI file,* check *depth* dataset to select it.
12. Turn on the *Depth* and *Rain* datasets under *Write to KMZ file*.
13. For the *Rain* dataset, select "Absolute" under *Altitude Option* and enter "3000" meters for *Altitude.*
14. Turn on the *Write this 2D Scattered Dataset to KMZ File* option and select the "Stream flow" option.
15. Click **Next** to go to the *Disply Options* of the *Film Loop Setup* wizard.
16. Do not change any options and click **Finish** to close the *Film Loop Setup* wizard and generate the animation.

WMS will take some time to (3–5 minutes) to create the movie. After the movie is completed, WMS opens the Google Earth animation.

Once in Google Earth, it may be neceassry to tilt the Google Earth view to see the animated stream flow lines (Figure 6). 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.

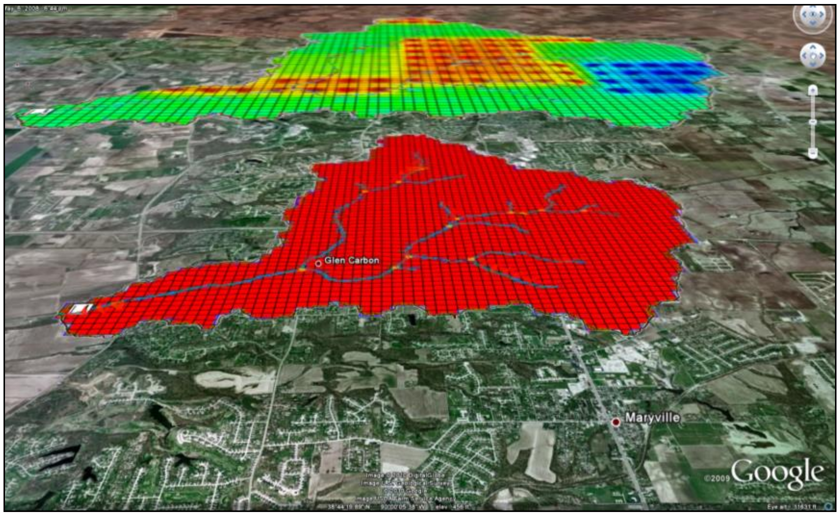


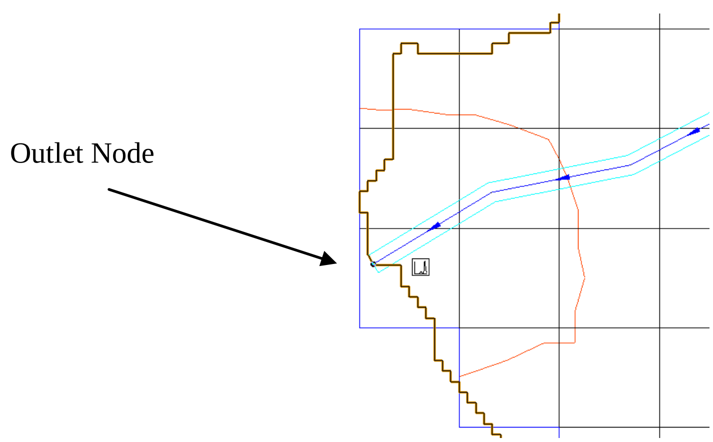
Figure 6 Google Earth animation showing the upper layer representing the distributed rainfall input and the lower layer representing overland runoff

8 Comparing Simulation Results with Observed Data

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

Use an arbitrary observed flow hydrograph at the outlet and compare it with simulation results.

1. **Zoom** in around the watershed outlet.
2. Make the **Map Module** active.
3. Using the **Select Feature Point\Node** tool, double-click the outlet node to open the *Properties* dialog.



4. Scroll to the right until the *Observations* column heading is visible and select the ... button for this column to open the *GSSHA Observations* dialog.
5. Click **Add** to create a new row.
6. Under *Observation Type*, select “Outlet Hydrograph”.

7. Click the **Define** button to open the *Define GSSHA Observation* dialog.
8. Check *Use XY data* option and click the **Define Series** button to open the *XY Series Editor* dialog.
9. In Notepad or another text editor, open the "Judys_ObsFlow.txt" file.
10. Select all the data then copy and paste the data into the *XY SeriesEditor* in WMS.

The hydrograph is plotted on the *XY Series Editor*.

11. Click **OK** to close the *XY Series Editor*.
12. Click **Done** to close the *Define GSSHA Observation* dialog.
13. Click **OK** to close the *GSSHA Observations* dialog.
14. Back in the *Properties* dialog, click the  button under *Solution Results* to open the *GSSHA Solution Analysis* dialog.

Now to compare the observed flow data with the simulation.

15. Turn on the *Observed* and the *visualization (GSSHA) Stream flow* check boxes.

The generated hydrographs should be similar to Figure 7.

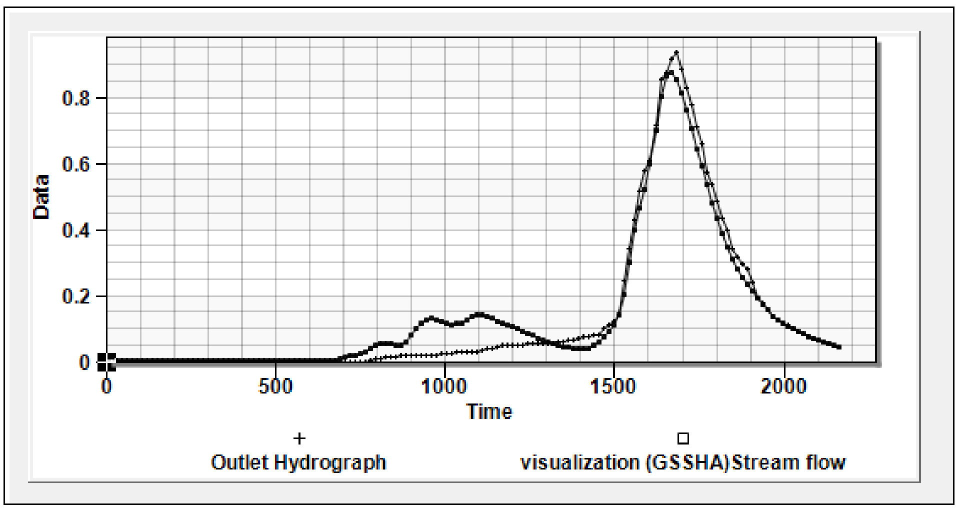


Figure 7 Comparing observed data to solution data

9 Conclusion

This concludes the "Post-Processing and Visualization of GSSHA Model Results" tutorial. Continue exploring the post-processing and visualization tools, or exit the program.

