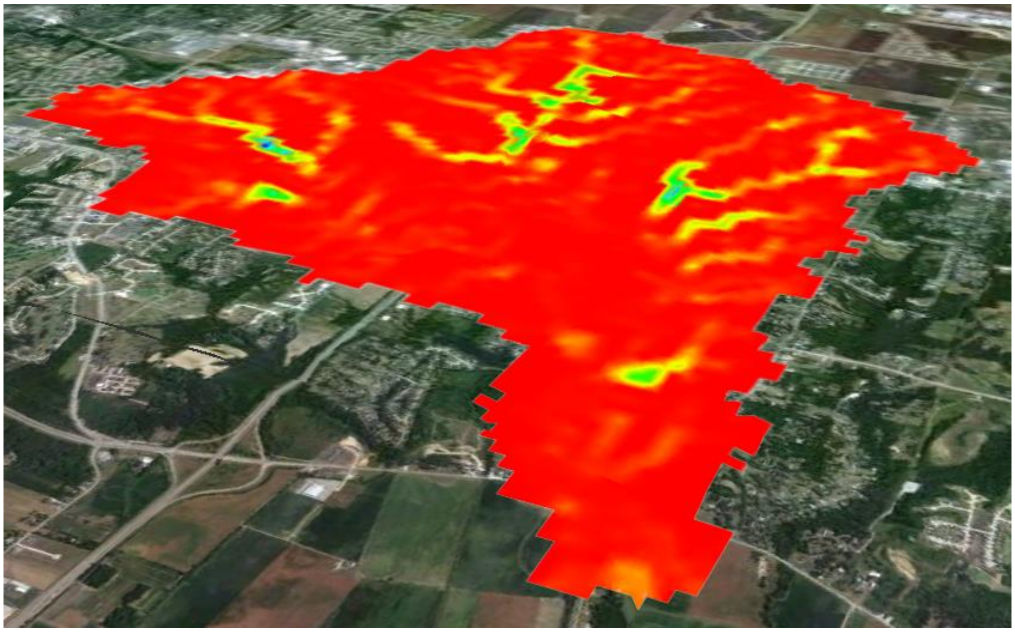
v. 10.1

WMS 10.1 Tutorial

GSSHA – Applications – Snowmelt Modeling in GSSHA

Set up and run a snowmelt simulation in GSSHA



Objectives

Learn how to set up a simulation using the snow modeling capabilities within GSSHA. This tutorial, demonstrates how to accurately simulate snow accumulation, melting, and runoff within a watershed model using the data from the long term simulation tutorial.

Prerequisite Tutorials

- GSSHA – Applications – Long Term Simulations in GSSHA

Required Components

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

Time

- 20-30 minutes

1 Contents

1 **Contents** 2
2 **Introduction** 2
3 **Open an Existing GSSHA Project** 2
4 **Setting up the Snow Model Simulation** 3
5 **Save and Run the Model** 3
6 **Viewing the Results** 4

2 Introduction

In this tutorial, set up and run the snow model simulation within GSSHA. Begin with an existing project file. The snow model runs under long-term simulations and therefore correct hydrometeorological (HMET for short) data is required. Currently there are three snow models within GSSHA to choose from:

1. Hybrid Energy Balance (Hybrid)
2. Energy Balance, and
3. Temperature Index (TI)

The Hybrid and TI snow models are the most commonly applied models. The TI model is the most accurate when calibration data in the form of snow water equivalent (SWE) is available. The Hybrid snow model is the default snow model and computes the amount of melt within the snow pack on an energy balance while also accounting for the heat transfer within the snow pack.

Melt-water moves both vertically and laterally through the snow pack, essentially delaying the water as it moves through the watershed. The melt-water flows through the snow pack and overland, and can be infiltrated or directly evaporated. Infiltration is limited due to frozen soil, simulated with a temperature index method.

By accurately simulating the accumulation and melt of the snow pack as well as the melt-water transport, the GSSHA model can accurately replicate the response of watersheds where snow is a contributing factor.

3 Open an Existing GSSHA Project

Open a WMS project file for the Judy's Branch watershed. This model has been set up to simulate approximately a year of data (Sept 2006 – Sept 2007), capturing the snow accumulation and melt within the watershed. Modify the project and perform a long term simulation with snow included.

1. In the ***2D Grid Module*** select ***GSSHA* | *Open Project File…***
2. Locate the ***SnowModel*** folder for this tutorial. If needed, download the tutorial files from www.aquaveo.com.
3. Browse and open the file ***SnowModel\longterm_snow.prj***

4 Setting up the Snow Model Simulation

After having defined a long term simulation, setting up a snowmelt model is simple if the input data is correct. The snowmelt parameters can be defined using the snowmelt options button in the GSSHA Job Control dialog. This section shows how to set up snowmelt data after defining a long term simulation.

1. Select **GSSHA | Job Control…**
2. Select the *Edit parameter...* button from the ***Snowmelt options*** line of the parameter window.
3. Enter the following data in the GSSHA Snowmelt Options window:
    a. ***Hydraulic conductivity of the snow pack***: **0.00555 m/s**
    b. ***Dry adiabatic lapse rate*** (turn this option ON): **-5 deg C/km**
    c. ***Elevation of HMET gage***: **140.0 m**
    d. ***Base temperature to begin melt*** (turn this option ON): **5 deg C**
4. Keep all the other parameters at their default values and select the *OK* button.
5. Select the *Output Control* button in the *GSSHA Job Control Parameters* dialog.
6. Scroll down to the bottom of the *Gridded datasets* output files and turn on the option to export the *Snow water equivalent* gridded dataset.
7. Select the *OK* button twice to go back to the main WMS window.

These steps told GSSHA to write temporal and spatial snow water equivalent data (in meters) to a storm water equivalent file, begin melt of the snow pack at 5.0 deg C, simulate the melt-water from the snowpack as flow through the snowpack, and account for orographic effects by lowering the temperature as elevation increases.

It is important to note that the Hybrid snow model is automatically activated when temperatures drop below 0 deg C and either precipitation or an existing snowpack are present. Output for the snow model is only activated when the option to export the Snow water equivalent gridded dataset is turned on. The hydraulic conductivity of the snow pack (in this case 0.00555 m/s) gives the rate at which the flow moves through the snowpack. The default value for the temperature at which snow melts is 0 deg C, unless otherwise specified in the base temperature to begin melt (in this case 5.0 deg C). The GSSHA model assumes a 0.0 deg C/km dry adiabatic lapse rate (no change in temperature with elevation) unless the dry adiabatic lapse rate option and the elevation of the HMET gage is defined.

More options exist for modeling snow, such as the Temperature Index snow model, but these options will not be examined in this tutorial. For more information on advanced snow modeling capabilities please see the GSSHA wiki site (http://www.gsshawiki.com).

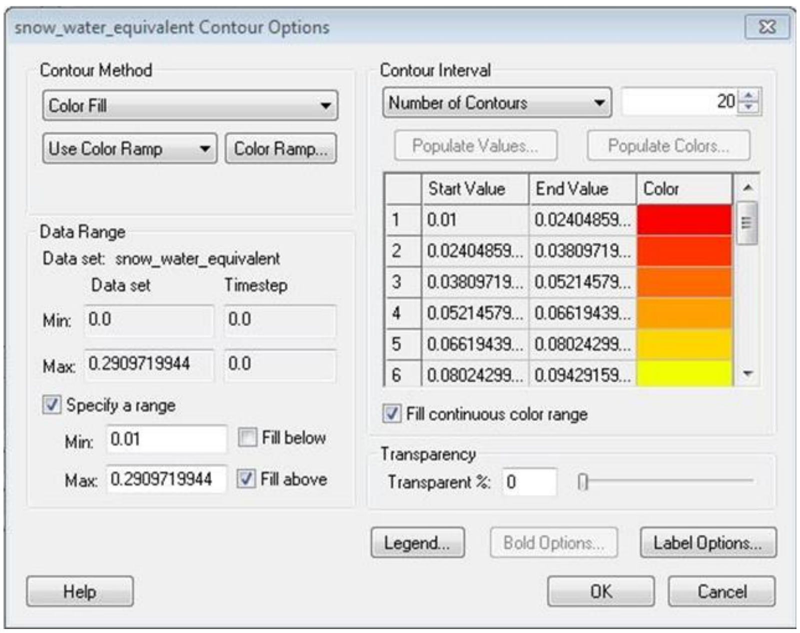
5 Save and Run the Model

1. Save the GSSHA project ***SnowModel\snowmelt_final.prj***
2. Select ***GSSHA | Run GSSHA*** and click *OK*. The model will take several minutes to complete running. When the model has finished running, click on the *Close* button to close the model wrapper and WMS will read the output.

6 Viewing the Results

To visualize the snow-related results, do the following:

1. In the ***2D Grid Module***, select ***Display* | *Display Options***.
2. Turn on the *2D Grid Contours*.
3. Select *OK*.
4. In the data tree, right-click on ***snow_water_equivalent*** under the solution folder.
5. Select ***Contour Options***.
6. Make sure the *Contour Method* drop-down box is set to *Color Fill*.
7. Check the *Specify a range* bullet in the lower left corner of the dialog box.
8. Set the Min to **0.01** and leave the Max at the value given.
9. Uncheck the *Fill below* bullet. The *Contour Options* dialog box should look like the following:



10. Select the *Legend…* button.
11. Turn on the *Display legend* toggle.
12. Click *OK.*
13. Click *OK.*
14. In the properties window (normally to the right side of the WMS screen), locate the set of time steps that appear when selecting the ***snow_water_equivalent*** dataset. Click on the time steps and use the down arrow key (on the keyboard) to toggle through the time steps. Notice the snow water equivalent contours varying as the time steps change. Notice the following:
    a. See initial snow water around **10/17/2006**.
    b. The initial snow pack will start to melt around **11/07/2006**.

c. The snow pack begins to accumulate once again and major melting becomes evident from **4/10/2007** through **5/13/2007**.

