
***Introduction to the US Army Corps of
Engineers Spatial Hydrologic Modeling
System with GSSHA and WMS***

***GSSHA Using the Watershed Modeling
System***

Tutorials – Volume 3

GSSHA Applications

WMS 8.4 Compatible – Volume 3

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
75 South 200 East, Suite 201
Provo, Utah 84606
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

1	PRECIPITATION METHODS IN GSSHA	1-1
1.1	OPEN AN EXISTING GSSHA PROJECT	1-1
1.2	USING UNIFORM RAINFALL	1-1
1.3	USING A DESIGN STORM HYETOGRAPH	1-3
1.4	USING RAIN GAGES WITH THE INVERSE DISTANCE WEIGHTED METHOD OF INTERPOLATION	1-4
1.5	USING RAIN GAGES WITH THE THIESSEN POLYGON METHOD OF INTERPOLATION	1-7
1.6	USING NEXRAD RAINFALL DATA IN GSSHA	1-7
1.7	SUMMARY	1-13
2	ANALYZING THE EFFECTS OF LAND USE CHANGE	2-1
2.1	OPEN AN EXISTING GSSHA MODEL	2-1
2.2	CHANGING THE LAND USE TO RESIDENTIAL	2-2
2.3	CHANGING THE LAND USE FROM RESIDENTIAL TO INDUSTRIAL	2-8
2.4	ADDING A DETENTION BASIN	2-9
2.5	ADDING A WET LAND	2-13
2.6	ADDING AN INFILTRATION BASIN	2-15
2.7	ADDING A BUFFER STRIP	2-20
2.8	SAVING A GSSHA GROUP	2-22
2.9	RUNNING A GSSHA GROUP	2-23
3	LONG TERM SIMULATIONS IN GSSHA	3-1
3.1	OPEN AN EXISTING GSSHA PROJECT	3-1
3.2	FORMAT DATA FOR LONG-TERM SIMULATION	3-1
3.3	SETTING UP LONG-TERM SIMULATION	3-12
3.4	CHANGE THE OUTPUT CONTROL	3-14
3.5	SAVE AND RUN THE MODEL	3-14
3.6	RESULTS	3-14
4	SIMULATING SEDIMENT TRANSPORT	4-1
4.1	OPEN AN EXISTING GSSHA PROJECT	4-1
4.2	TURNING SOIL EROSION ON AND ADDING SEDIMENT	4-1
4.3	SPECIFYING SOIL EROSION PARAMETERS	4-3
4.4	SPECIFY THE OUTPUT PARAMETERS	4-4
4.5	SAVE AND RUN THE MODEL	4-4
4.6	VISUALIZING THE RESULTS	4-5
4.7	UPDATING CHANNEL ERODIBILITY PARAMETERS	4-9
5	SIMULATING CONSTITUENT TRANSPORT	5-1
5.1	OPEN AN EXISTING GSSHA PROJECT	5-1
5.2	ADDING CONTAMINANTS	5-1
5.3	DEFINING CONTAMINANTS IN THE CHANNELS	5-3
5.4	DEFINING CONTAMINANTS PARAMETERS	5-3
5.5	SAVE AND RUN	5-5
5.6	VIEW RESULTS	5-5
5.7	ADDING SEDIMENT TRANSPORT	5-6
5.8	SAVE AND RUN	5-8


5.9	VIEW RESULTS.....	5-8
6	OVERLAND FLOW BOUNDARY CONDITIONS IN GSSHA	6-1
6.1	HYDROGRAPH BOUNDARY CONDITION.....	6-1
6.2	VARIABLE STAGE BOUNDARY CONDITION	6-4

GSSHA Precipitation

In this tutorial, you will learn the different methods that precipitation data can be defined as storms in GSSHA. You will begin with an existing GSSHA project file. You will also see how NEXRAD data can be processed for GSSHA and view the difference in results while using various rainfall methods.

1.1 Open an Existing GSSHA Project


Open a WMS project file for Judy's Branch watershed.

1. In the *2D Grid Module*  select **GSSHA | Open Project File...** Browse and open the file *C:\Training\Precipitation\base.prj*
2. Turn off the display of the Soil Type and Land Use coverages by unselecting them in the Project Explorer.

This model already has overland roughness, infiltration and channel routing options defined. We will not need to define or adjust these parameters, but will focus on exploring different methods for defining precipitation.

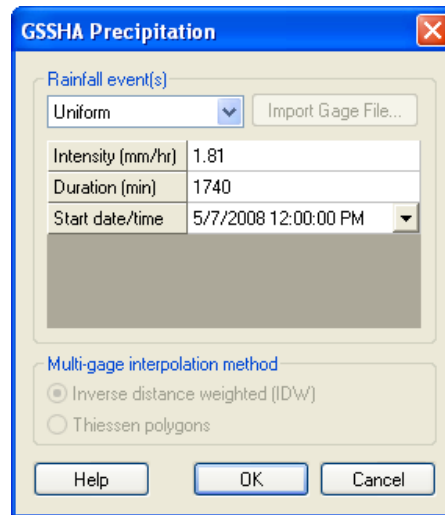
1.2 Using Uniform Rainfall

GSSHA has four different methods of defining rainfall precipitation. The method you use will be selected based on the availability of the data and purpose of the model. We will first use the uniform precipitation method, which generally is used to evaluate the initial set up of a model.

1. In the *2D Grid Module*  select **GSSHA / Precipitation**.
2. Under *Rainfall event(s)* select *Uniform*.
3. Enter 1.809 mm\hr for the intensity and 1740 minutes for duration. We are using total depth of 52.451 mm (2.065in) over a duration of 29 hours.

This precipitation depth is obtained from a real storm which we will use for comparison of the different methods. It was obtained from the NOAA site: (<http://hdsc.nws.noaa.gov/hdsc/pfds/>). The real storm total sums up to 2.065 inch over a duration of 29 hours.

4. Change the starting date as 05\07\2008 12:00:00 PM and click *OK*.



1.2.1 Changing the Job control

Since the rainfall will last for 29 hours, the total simulation time should be adjusted so that all runoff from the watershed will be captured.

1. Select **GSSHA / Job Control**. Then change the total time to 2880 mins (2 days). Make sure the simulation time step is 10 sec.
2. Click *OK*.

1.2.2 Save and Run the Model

We have now defined the uniform precipitation. Next we will save and run the model.

1. Select **GSSHA / Save Project file...** Save it as **C:\Training\Personal\Precipitation\Uniform.prj**.
2. Select **GSSHA | Run GSSHA...**

1.2.3 Visualization


1. Once GSSHA has finished running, click *Close*
2. You should have noticed that there is no runoff. This is because the rainfall intensity was small and all of the rain got infiltrated.

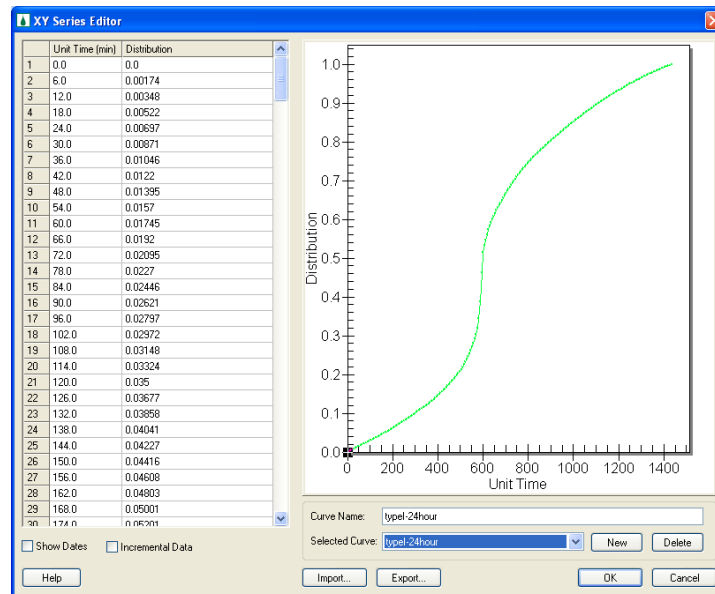
1.3 Using A Design Storm Hyetograph

Here we will see how typical rainfall distribution can be defined in GSSHA. We will be using the SCS synthetic rainfall distribution applied to the same total depth of 52.451mm (2.065in). In a similar fashion an actual temporal distribution could be defined if available. The actual temporal distribution of this storm will be used in the next section.

Now we can keep using the same GSSHA project and just change the precipitation method. Once the new precipitation has been defined, we will save the project with different name and run it.

1.3.1 Define the rainfall

1. Make sure you are in the *2D Grid Module* 
2. Select *GSSHA | Precipitation*. In the precipitation dialog, select *Hyetograph* for the Rainfall event(s) option.
3. Click on the *Define Distribution* button.
4. In the *XY Series Editor* dialog, select *typeI-24hour* for the *Selected Curve* option. See the following figure.



5. Click *OK*.
6. Enter **52.451 mm** for the *Average depth* field and make sure that the *Start date* is set to 05\07\2008 12:00:00 PM.
7. Click *OK*.

1.3.2 Save and Run the Model

We have now defined the hyetograph method of defining precipitation. Next we will save and run the model.

1. Select *GSSHA | Save Project file...* Save it as *C:\Training\Personal\Precipitation\Hyetograph.prj*.

2. Select *GSSHA* | *Run GSSHA...*



1.3.3 Visualization

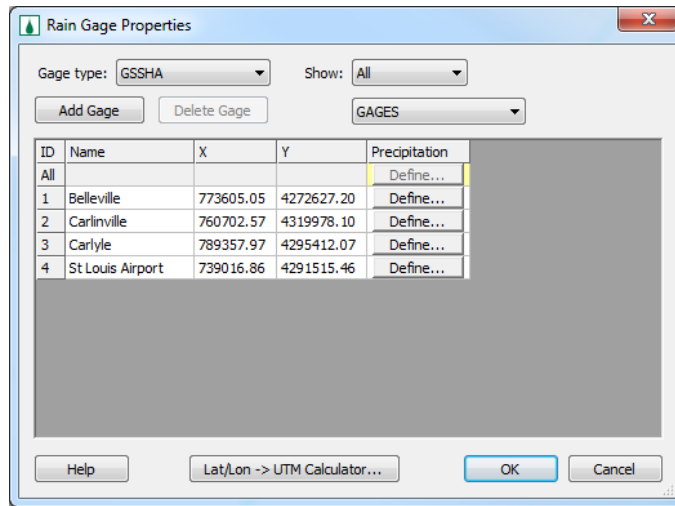
1. Once GSSHA has finished running, click *Close*
2. View the results (Hydrograph at the outlet).
3. For comparison, export the hydrograph ordinates as you did before and copy\paste in the spreadsheet *C:\Training\RainfallMethods.xls* under column *Type I*.
4. Close the plot window(s) when you are done copying.
5. Note the difference in the outlet flow hydrograph when comparing the *uniform* and *hyetograph* methods.

1.4 Using Rain Gages with the Inverse Distance Weighted Method of Interpolation

For this next simulation we will see how rain gages can be used to define precipitation in GSSHA. We will be using four gage locations in the vicinity of the Judy's Branch watershed; namely Belleville, Carlinville, Carlyle and St. Louis Airport gages.

1.4.1 Creating gages

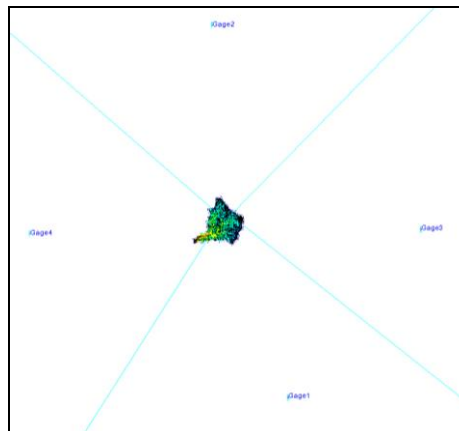
1. Right click on *Coverages* in the Project Explorer and select *New Coverage*
2. Change the type of the coverage to *Rain Gage* and click *OK* which will add coverage on the data tree under coverages
3. Click on the *Rain gage* coverage and choose *Create Feature Point Tool* . Click on the white area just outside your watershed boundary so that it will be easier to locate and make sure you are not too far away from the watershed. This will add a rain gage to the watershed. Create 3 additional gages surrounding the watershed. Do not worry about the exact location of the gages for now. We will edit their coordinates in the next step.
4. Choose *Select Feature Point\Node tool*  and double click any of the 4 gage that you just created.
5. In the Rain Gage properties dialog, make sure the *Gage Type* is set to *GSSHA* and select *All* for the *Show* option.



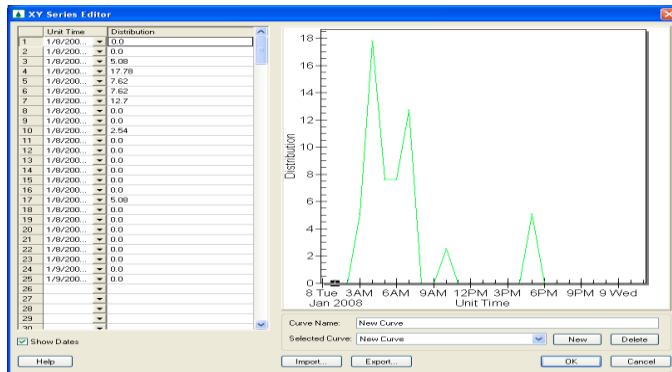
6. Edit the coordinates and the names of each gage as shown in the following table.


Gage name	Change gage name to	X	Y
Gage1	Belleville	773605.05	4272627.20
Gage2	Carlinville	760702.57	4319978.10
Gage3	Carlyle	789357.97	4295412.07
Gage4	St Louis Airport	739016.86	4291515.46

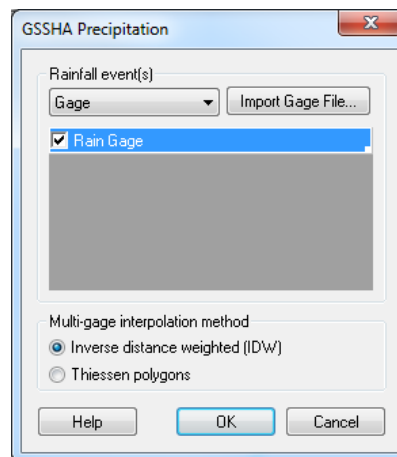
7. If you move *Rain Gage Properties* dialog to one side, you can see how WMS automatically draws Thiessen polygons (see the following figure).



8. On the *Rain Gage Properties* dialog, click on the *Define...* button for *Belleville* gage which will open the XY Series editor window. In the XY Series Editor make sure the option *Show Dates* is toggled *ON*.
9. Open the spreadsheet *C:\Training\RawData\JudysBranch\ RealStorm.xls* which has hourly precipitation records for these gages.
10. Copy and paste the columns with date and incremental distribution for *Belleville* to the XY series editor. Your XY Series will look something like this:



11. Click *OK*
12. Repeat the same process for all other gages.
13. Once done, click *OK* to close the Rain gage property dialog.
14. Switch to the 2D Grid Module  and select *GSSHA / Precipitation*.
15. Select Gage under Rainfall event(s) option and it will bring Rain Gage in the list. Check on to select Rain Gage.



16. Select Inverse Distance Weighted (IDW) for interpolation method.
17. Click *OK*

1.4.2 Save and Run Model

We have now defined the IDW method of interpolation for the gages to define the precipitation. Next we will save and run the model.

1. Select *GSSHA | Save Project file...* Save it as *C:\Training\Personal\Precipitation\IDW.prj*.
2. Select *GSSHA | Run GSSHA....*


1.4.3 Visualization

1. Once GSSHA has finished running, click *Close*
2. View the results (Hydrograph at the outlet).

3. For comparison, export the hydrograph ordinates and copy\paste to the spreadsheet *C:\Training\RainfallMethods.xls* under the column *IDW*.
4. See the difference in the outlet flow hydrograph in using the three different precipitation methods tested so far.

1.5 Using Rain Gages with the Thiessen Polygon Method of Interpolation

For this simulation we will use the same gages for precipitation, but with the Thiessen Polygon interpolation method.

1. Switch to the *2D Grid Module*  and select *GSSHA / Precipitation*
2. Leave everything the same other but change the *Interpolation method* to *Thiessen Polygons* and click *OK*.

1.5.1 Save and Run Model

We will save and run the model.

1. Select *GSSHA | Save Project file...* Save it as *C:\Training\Personal\Precipitation\Thiessen.prj*.
2. Select *GSSHA | Run GSSHA....*

1.5.2 Visualization

1. Once GSSHA has finished running, click *Close*
2. View the results (Hydrograph at the outlet).
3. For comparison, export the hydrograph ordinates and copy\paste into the spreadsheet *C:\Training\RainfallMethods.xls* under the column *Thiessen*.
4. See the difference in the outlet flow hydrograph using the different precipitation methods.


1.6 Using NEXRAD Rainfall Data in GSSHA

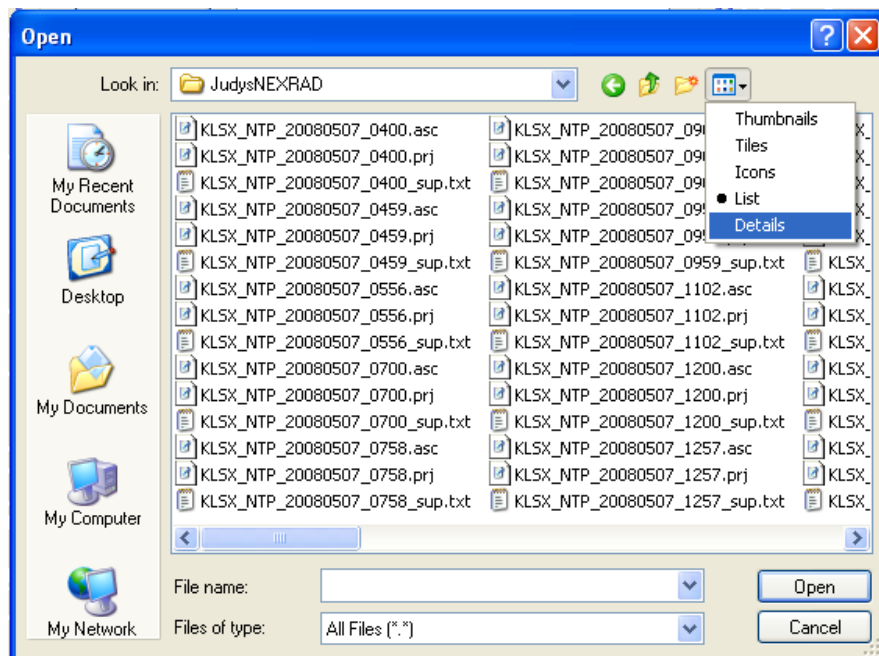
In this section, you will see how NEXRAD rainfall data can be used in GSSHA. You will begin with an existing GSSHA project file. You will see how NEXRAD data can be processed for GSSHA and view the difference in results between using gage-based and distributed rainfall.

1.6.1 Importing NEXRAD Rainfall Data

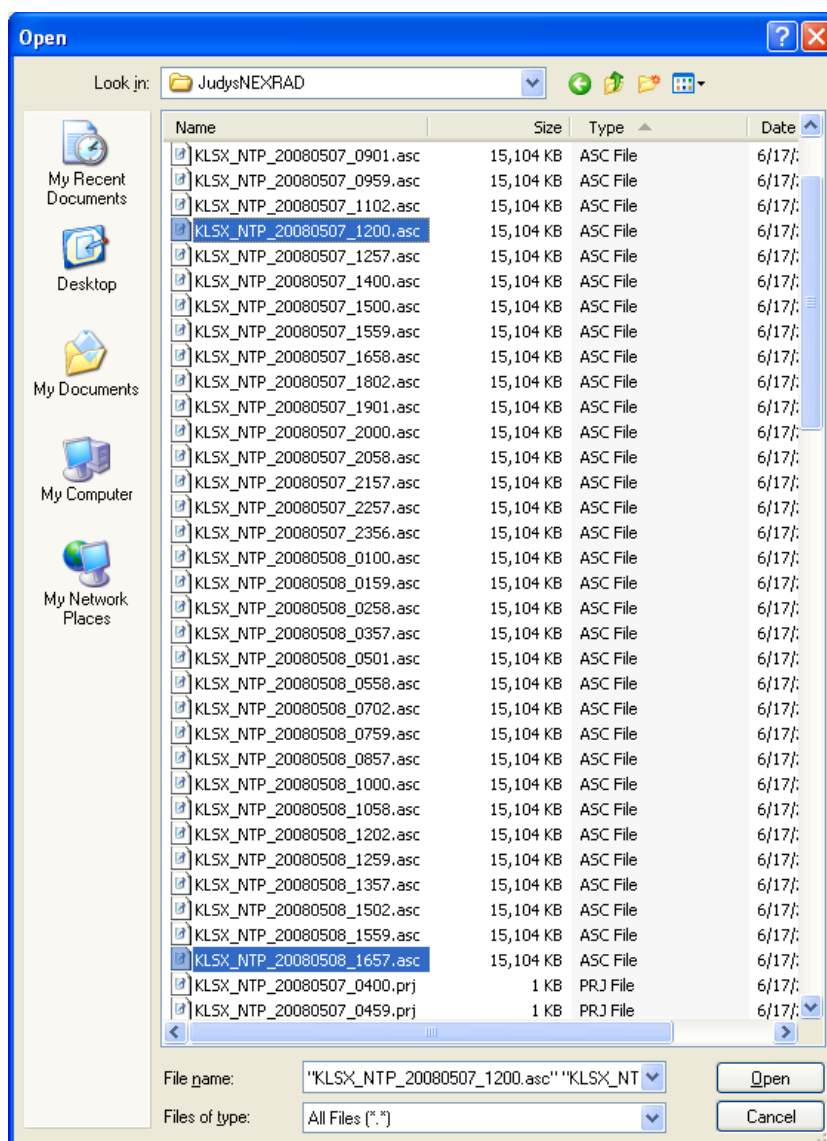
NEXRAD rainfall datasets have already been downloaded for this watershed. For information on how to obtain your own radar rainfall datasets, see http://www.xmswiki.com/index.php?title=GSDA:Obtaining_NEXRAD_Radar_Data_from_NCDC

1-8 GSSHA Modeling Using WMS – Volume 3

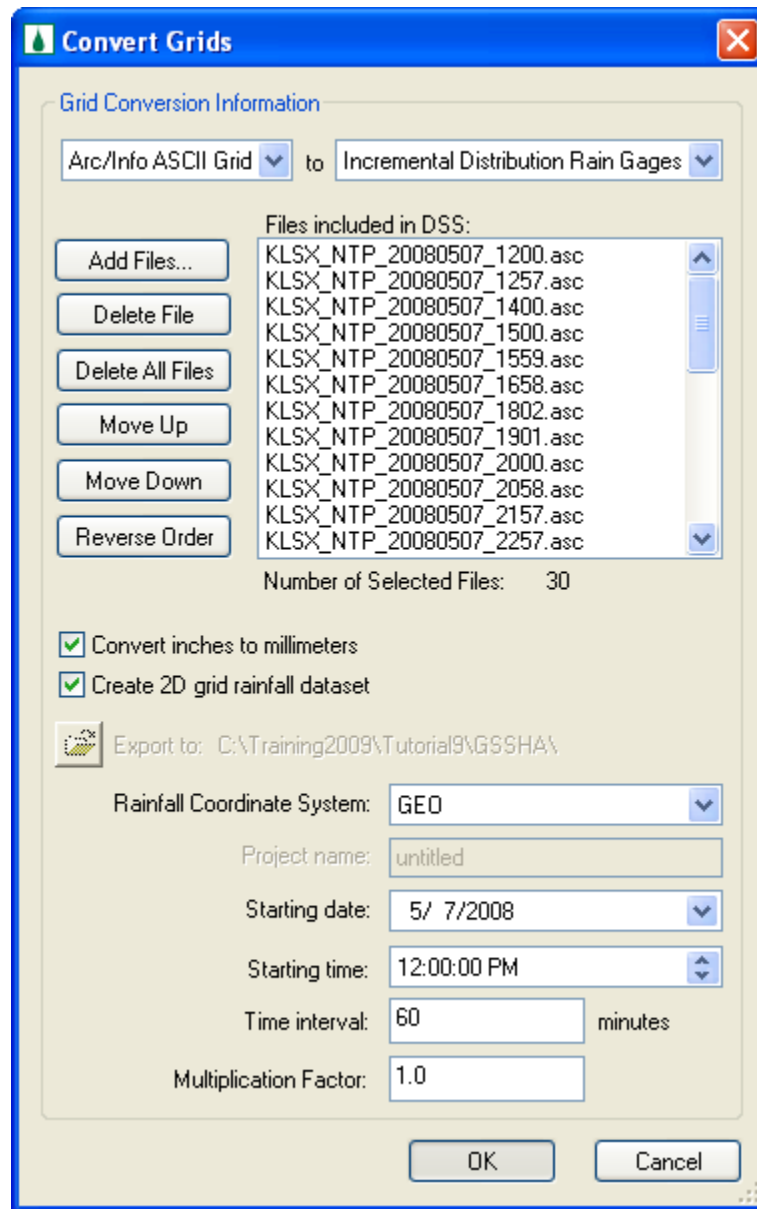
1. We will keep working with the same model. Check off the display of *Rain Gage Coverage* by unselecting it on the project explorer. Zoom to GSSHA coverage (Right click GSSHA coverage and select *Zoom to Layer*).
2. In the 2D Grid Module  select **GSSHA / Precipitation**. Under *Rainfall event(s)* select *Nexrad Radar*
3. Click on *Import Radar Data...* button which will open the *Convert Grids* dialog
4. In the *Convert Grids* dialog, make sure that *Grid Conversion Information* is set to *Arc\Info ASCII Grid to Incremental Distribution Rain gages*
5. Click the *Add Files...* button and browse to **C:\Training\RawData\JudysBranch\Nexrad** In the Open file browser, change the View Menu to *Details* (See following figure)



6. Click on the *Type* column heading to sort the files by Type
7. Select the **last** time grid which is *KLSX_NTP_20080508_1657.asc*



8. Hold the *Shift* key and select the **starting** time grid which is *KLSX_NTP_20080507_1200.asc*
9. Click *Open*
10. In the Convert Grids dialog, toggle on the option *Convert inches to millimeters* (if it is not already on)
11. Toggle on *Create 2D grid rainfall dataset* option. Make sure that the time interval is 1 hour (60 min)
12. Change the *Starting date* to **05\07\2008** and the *Starting time* to **12:00:00 PM** (See the following figure)



13. Click *OK* to save the grid file. It will take some time to create the gages for the NEXRAD rainfall method.
14. After a couple of minutes the conversion process will complete and a summary file will open up (Following figure) showing the date\time and rainfall depth (mm) at each time interval.
15. Review and close the summary file

NEXRAD RADAR DATA SUMMARY REPORT

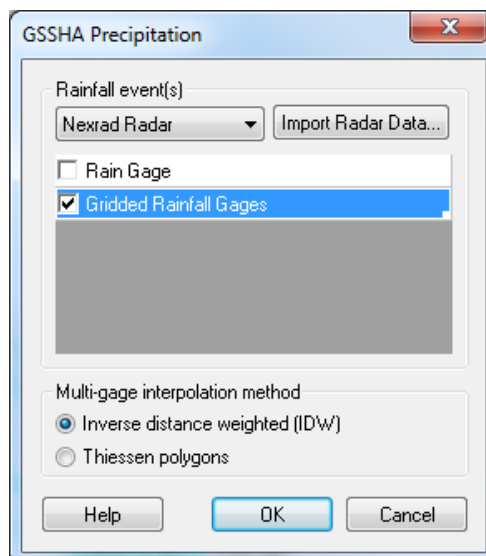
Basin Average Hyetograph

Date & Time	Time (min)	Incremental Depth (mm)	Cumulative Depth (mm)
wednesday, May 07, 2008, 13:00:00		0.000000	0.000000
wednesday, May 07, 2008, 14:00:00		60.000000	5.401149
wednesday, May 07, 2008, 15:00:00		120.000000	5.401149
wednesday, May 07, 2008, 16:00:00		180.000000	5.401149
wednesday, May 07, 2008, 17:00:00		240.000000	5.401149
wednesday, May 07, 2008, 18:00:00		300.000000	5.401149
wednesday, May 07, 2008, 19:00:00		360.000000	5.401149
wednesday, May 07, 2008, 20:00:00		420.000000	1.138621
wednesday, May 07, 2008, 21:00:00		480.000000	6.568966
wednesday, May 07, 2008, 22:00:00		540.000000	0.000000
wednesday, May 07, 2008, 23:00:00		600.000000	1.532759
Thursday, May 08, 2008, 00:00:00		660.000000	0.000000
Thursday, May 08, 2008, 01:00:00		720.000000	0.000000
Thursday, May 08, 2008, 02:00:00		780.000000	5.036207
Thursday, May 08, 2008, 03:00:00		840.000000	0.000000
Thursday, May 08, 2008, 04:00:00		900.000000	0.656897
Thursday, May 08, 2008, 05:00:00		960.000000	0.000000
Thursday, May 08, 2008, 06:00:00		1020.000000	0.000000
Thursday, May 08, 2008, 07:00:00		1080.000000	0.000000
Thursday, May 08, 2008, 08:00:00		1140.000000	2.408621
Thursday, May 08, 2008, 09:00:00		1200.000000	0.000000
Thursday, May 08, 2008, 10:00:00		1260.000000	0.000000
Thursday, May 08, 2008, 11:00:00		1320.000000	0.802874
Thursday, May 08, 2008, 12:00:00		1380.000000	3.795402
Thursday, May 08, 2008, 13:00:00		1440.000000	9.926437
Thursday, May 08, 2008, 14:00:00		1500.000000	6.568966
Thursday, May 08, 2008, 15:00:00		1560.000000	4.963218
Thursday, May 08, 2008, 16:00:00		1620.000000	1.386782
Thursday, May 08, 2008, 17:00:00		1680.000000	0.000000
Thursday, May 08, 2008, 18:00:00		1740.000000	0.000000

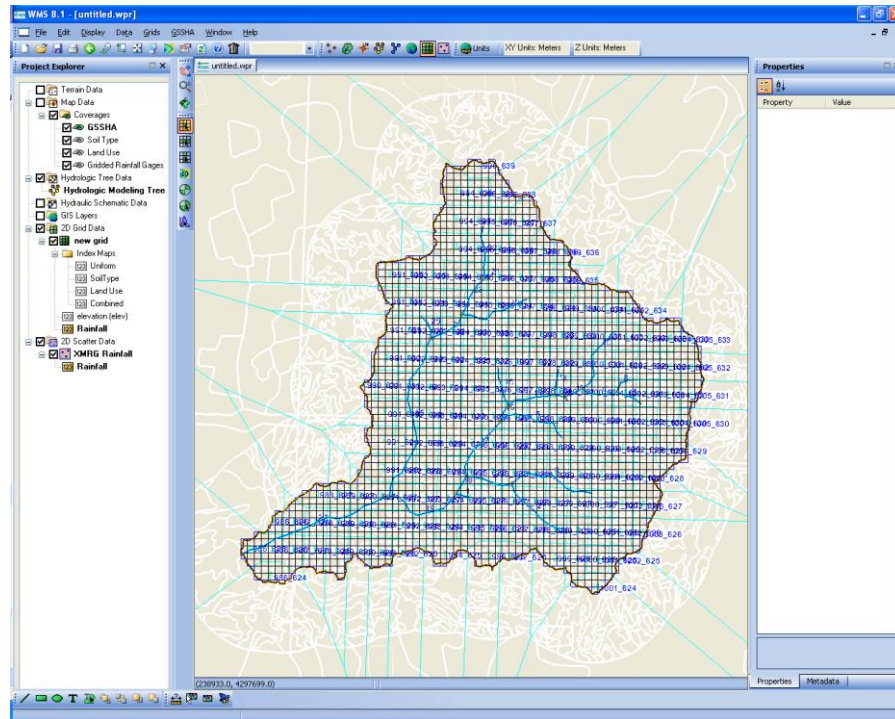
Total Storm Duration: 1740.000000 min
Total Storm Depth: 50.186897 mm

You should now see many gages covering the watershed and a network of polygons joining the gages. You should also notice a new *Gridded Rainfall Gages* coverage in the coverage tree.

16. The GSSHA Precipitation dialog will appear after the completion of the conversion process. Toggle on the *Gridded Rainfall Gages* option and click *OK*.
17. In the *GSSHA precipitation* dialog make sure that you toggle off the Rain Gage coverage and the change the interpolation method back to IDW.



18. The WMS window should look similar to the following figure.




1.6.2 Visualizing Meteorological Data

Before continuing, let us visualize the gridded rainfall data.

1. Toggle off the display of *Gridded Rainfall Gages* by unselecting it in the project explorer.
2. Right click on *Rainfall Cumulative* on the data tree under 2D Grid Data and select *Contour Options...* Select *Color Fill* for the *Contour Method*. Click *OK*
3. With the down arrow key on your keyboard, step through the time steps in the properties window on the right sidebar to see how the precipitation varies.

There are two rainfall datasets, one is incremental and the other is cumulative. You may choose to view the incremental rainfall data set in the same way you viewed the cumulative data set. Whichever dataset is selected will be used to create the film loop.

4. In *2D Grid Module*  select *Data | Film Loop...* Specify the folder where you want to have the animation saved (*C:\Training\Personal\Precipitation*), check off the option to *Export to KMZ (Google Earth)* and click *Next*
5. Toggle on either (*Rainfall Cumulative*) or (*Rainfall Incremental*) and toggle off all other datasets.
6. Toggle off the option to *write to KMZ file*.
7. Click *Next*


- Click *Finish*. WMS will now a few moments to create and save the animation file. The animation will start playing as soon as the saving process is complete.

1.6.3 Save and Run the Model

We have now imported the NEXRAD rainfall into GSSHA and copied the data into the *Gridded Rainfall Gages* coverage. Next we will save and run the model.

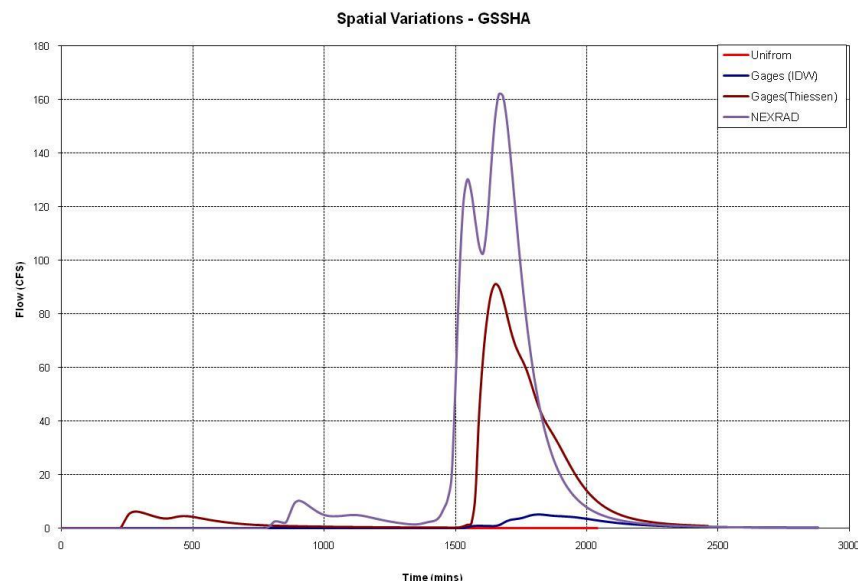
- Select **GSSHA | Save Project file...** Save it as **C:\Training\Personal\Precipitation\nexrad.prj**.
- Select **GSSHA | Run GSSHA....**

1.6.4 Visualization

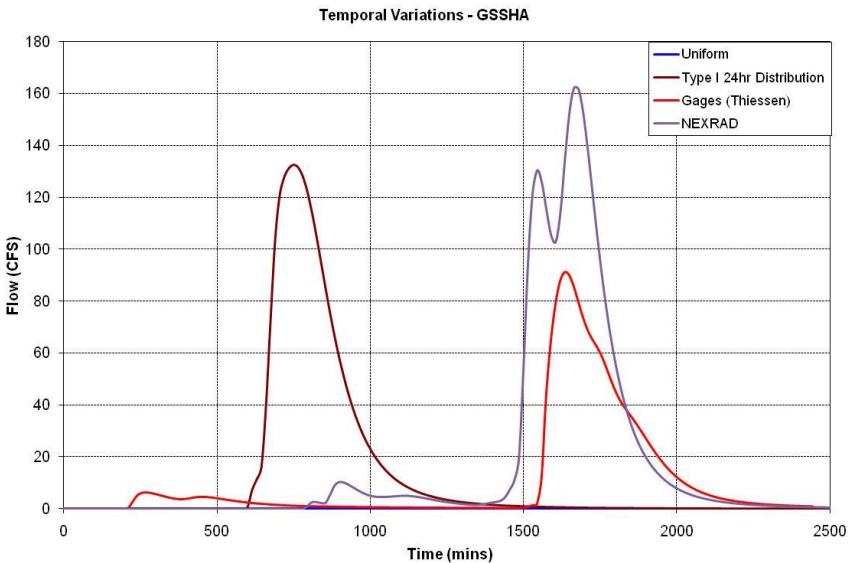
- Once GSSHA has finished running, click *Close*
- With the *Select Hydrographs* tool  selected, double-click on the hydrograph icon to view the runoff hydrograph at the watershed outlet
- For comparison, export the hydrograph ordinates and copy/paste in the spreadsheet **C:\Training\RainfallMethods.xls** under the column *NEXRAD*.
- See the difference in the outlet flow hydrograph in using different precipitation methods.
- You may export the animations to Google Earth (Optional).

1.7 Summary

Open up the comparison spreadsheet and compare the results from different precipitation methods.



1-14 GSSHA Modeling Using WMS – Volume 3



Analyzing the Effects of Land Use Change

2.1 Open an Existing GSSHA Model

In this tutorial we will see how different land use change scenarios can be modeled in GSSHA. These scenarios might be pre-development and post-development scenarios or a wetland restoration scenario.

In this exercise we will first change a portion of land to a residential and an industrial area to simulate a development scenario. We will then use abatement measures such as a detention basin, wetland, infiltration basin and buffer strip. The following are the different scenarios we will develop:

- Change of land use from Evergreen forest land to residential land
- Change this residential land to an industrial area
- Add a detention basin
- Add a wetland
- Add an infiltration basin
- Add a buffer strip

This exercise will illustrate how the same grid and sets of index maps can be used to develop multiple GSSHA models. Throughout we will compare the outcomes from the different scenarios.

1. Switch to 2D Grid Module and open an existing project by selecting **GSSHA / Open Project File**. Browse and open file **C:\Training\Scenarios\BaseModel\base.prj**

Let us first save this project to personal folder so that the original project remains unchanged.

2. Select *GSSHA / Save Project File*. Save the project as *C:\Training\Personal\Scenarios\base.prj*.

2.2 Changing the Land Use to Residential

The land is being developed on one of the hills in the watershed. To simulate this, the forest will be converted to another land use thus changing both infiltration as well as the overland roughness characteristics of the watershed.

2.2.1 Creating a new GSSHA project for the changed land use

There are two steps involved in creating a new GSSHA project for the changed land use:

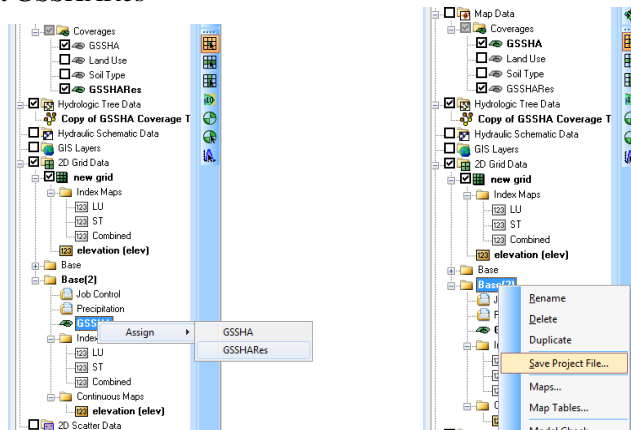
- 1) Create a new GSSHA coverage, and
- 2) Create a new GSSHA project.

2.2.1.1 Creating a new GSSHA Coverage

1. In the project explorer, under *Coverages*, right click the *GSSHA* coverage and select *Duplicate*.
2. Rename the new GSSHA coverage as *GSSHARes*.

2.2.1.2 Creating a new GSSHA Project

1. In the project explorer, under *2D Grid Data*, right click on GSSHA project named *Base* and select *Duplicate*. This will create a new GSSHA project with the name *Base(2)*.
2. Under *2D Grid Data*, expand *Base(2)*. Right click on GSSHA and select *Assign*.
3. Select *GSSHARes*



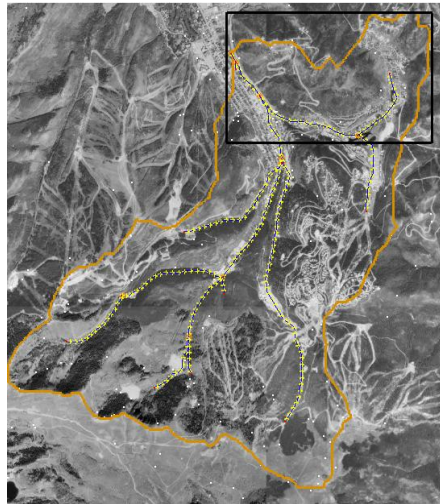
4. Right click on project *Base(2)* and select *Save Project File....* Save it as *C:\Training\Personal\Scenarios\Residential.prj*. This will also rename the project.


2.2.2 Changing the Land Use Map

Let us now modify the land use.

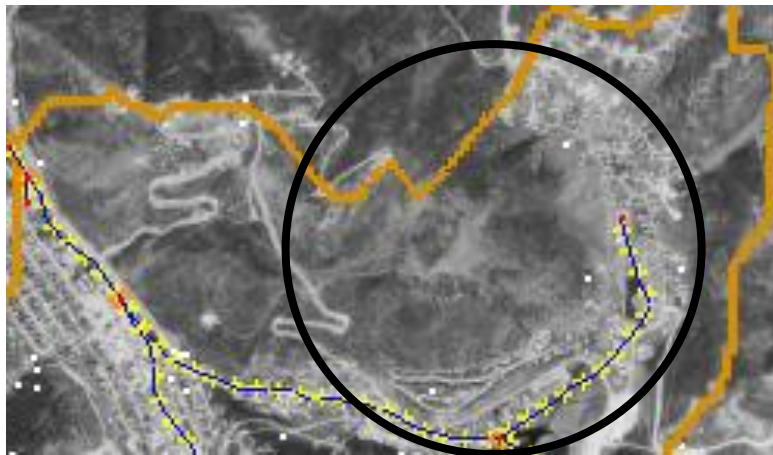
Analyzing the Effects of Land Use Change 2-3

1. Since we are going to change the land use map, let us create a copy of the original land use coverage. Right click on *Land Use* coverage and *Duplicate* it. Rename it to *LUResidential*.
 2. A background image will help to identify the proper location. Select **File / Open**. Browse to *C:\Training\Scenarios\Images\Aerial.jpg* and open it.
 3. Select Yes to build pyramid.
- We will use this image to identify existing land use.
4. Turn off the display of “*GSSHA Coverage*”, “*Soil Type Coverage*”, “*Land Use Coverage*” and “*2D Grid Data*” by checking them off in the data tree. Make sure just “*LUResidential*” Coverage is turned on.
 5. Zoom in around the area as shown in following figure.




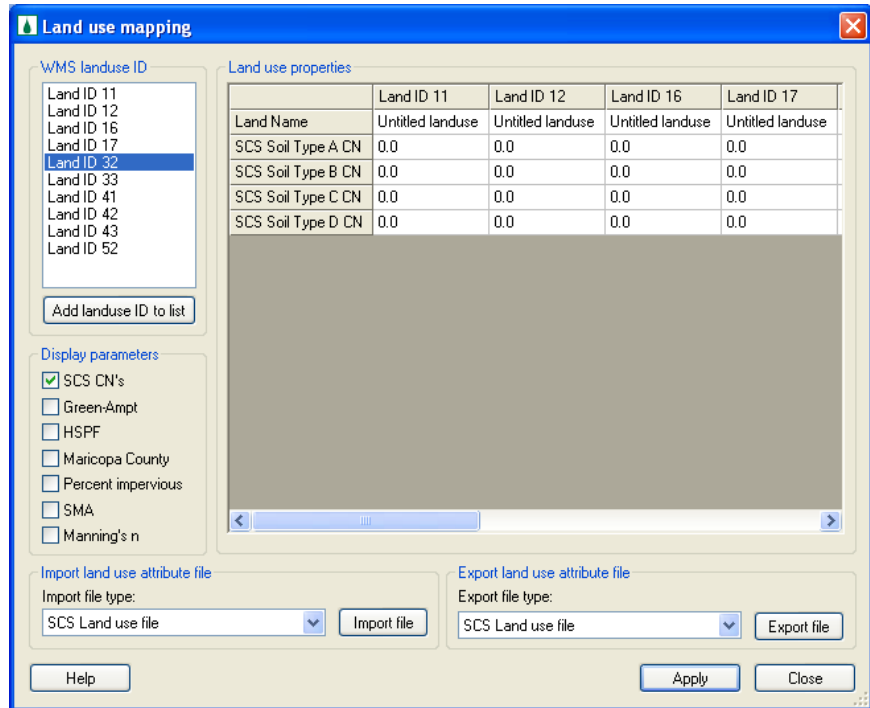
6. Select the **Map Module** of WMS .


Here, we can see that most of the area shown are shrubs and brush rangeland (LUCode 32). After the development, let us assume that the area will be converted to relatively impervious almost similar to a residential area (LUCode 11).

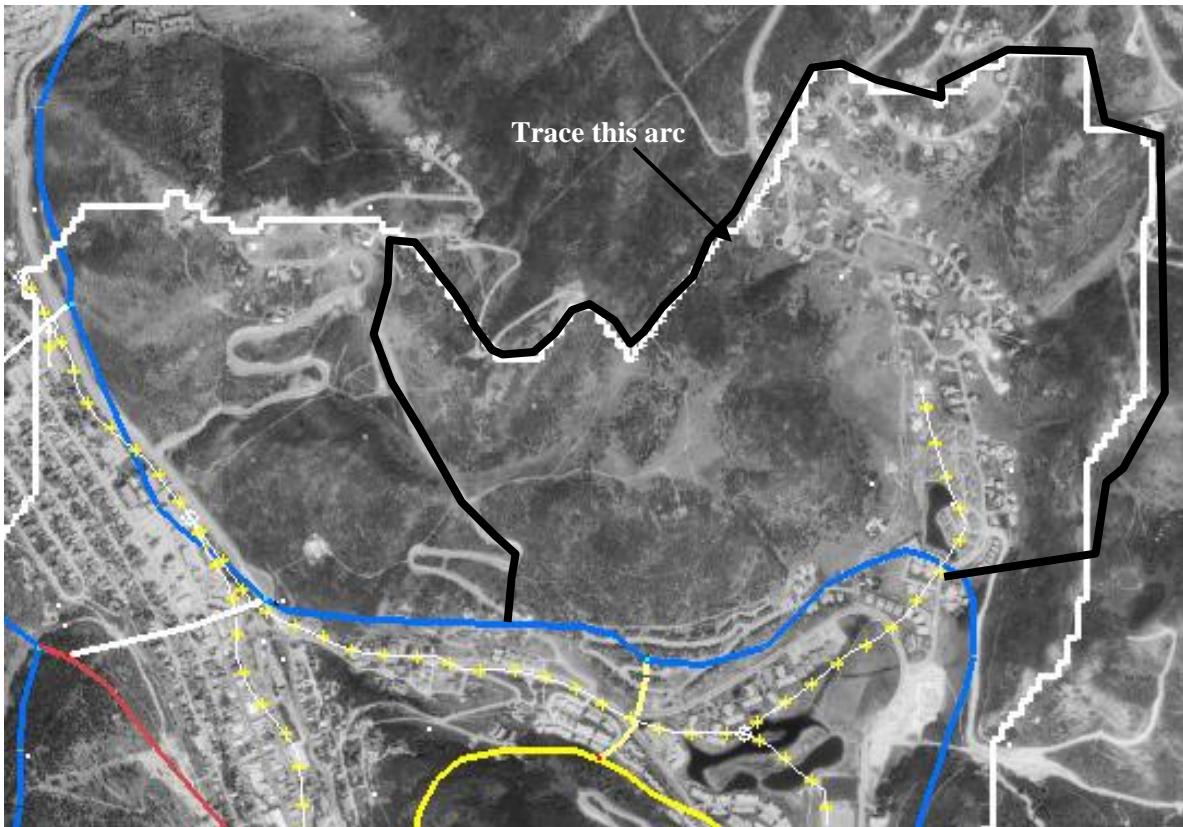



7. Click on the *LUResidential Coverage* to make it an active coverage. .

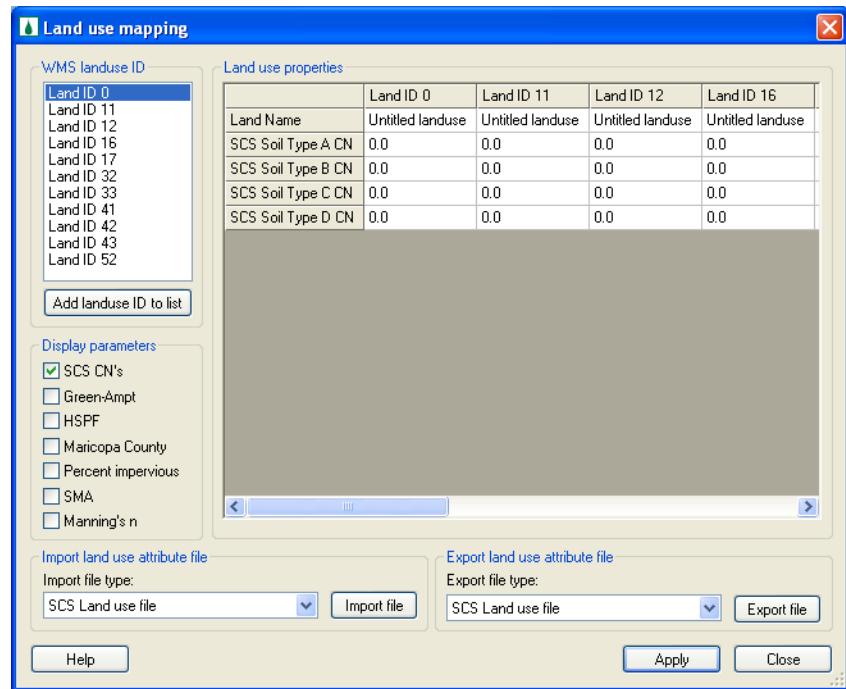
8. Click on “*Select Feature Polygon Tool*” . Then double click on the polygon in the area enclosed in the circle (in the above figure). This will open up the “*Land use Mapping*” dialog.



9. On the left side of the dialog, you can see the current land use ID selected. The Land Use ID selected should be 32. According to the USGS land use classification, Land use ID 32 represents Shrub and Brush Rangeland. We will now change a portion of this land use to “*Residential*”.
10. Click *Close* to close the “*Land use mapping*” dialog.
11. Make sure that *LUResidential Coverage* is active. In the Map Module, Select the “*Create Feature Arc Tool*”  and trace a line as shown in the following Figure. Trace an arc along the line using left mouse click and make sure that the arc you trace will start and end at other existing land use polygons.



12. Select **Feature Object / Build Polygon** and click *OK* to use all arcs.
13. In the Map Module, Click on the “*Select Feature Polygon Tool*” tool . Then double click the area enclosed by the arcs you traced. This will select the new polygon for the area and bring “*Land use Mapping*” dialog up.



14. This time the land use ID is 0, we will assign a new ID for this area. Click on *Add landuse ID to list* button on the left side which will add a new Land ID (*Land ID 53*) to the list. Scroll all the way to the right to find *Land ID 53* and change the name to *New Residential*.
15. Click *Apply*. This will change the land use code for the new polygon.

2.2.3 Creating the Modified Index Maps

With the change in land use coverage, the index maps that use Land use coverage need to be updated as well. To do this we will create new land use and combined index maps.

1. In the project explorer under *2D Grid Data*, right click project *Residential* and select *Maps...*
2. In the “Index-Grid” tab, select the Input coverage to be *LUResidential* with coverage attribute as *ID* and the Index map name *LUResidential*.
3. Click on the *Coverages->Index Map* button.
4. Next we will create a new index map which is a combination of soil type data and the modified land use data. Select the following:

Input coverage (1): *LUResidential*

Coverage attribute: *Id*

Select “input Coverage (2) by checking it ON.

Input coverage (2): *Soil Type*

Coverage attribute: *Texture*

Index map name: *ComboResidential*

5. Click on *Coverages ->Index Map* button.
6. Click *Done* after it completes.

7. Turn off the background image and *LUResidential* coverage. Turn on the display of *2D Grids data*.

You should have two new index maps added to the data tree.

2.2.4 Assigning new index maps and updating the parameters

1. Under *2D Grid Data*, click on the GSSHA project named *Residential*.
2. Right click *Index Maps* folder and select *Assign ST, LUResidential and ComboResidential* index maps.
3. Remove (*Right Click/Remove*) *LU and Combined* index maps which were developed for the original land use.
4. The new GSSHA project uses *ST, LUResidential* and *ComboResidential* as the index maps.
5. We need to update the mapping tables as well. Under *2D grid Data*, right click on GSSHA project *Residential* and select *Map Tables...*
6. In the *Roughness* tab, select "*LUResidential*" for the "*Using index map*" field and click the *Generate IDs* button.
7. Click *No*.
8. There will be a new field added with ID 53 at the end of the list (towards the right). Enter a roughness value of 0.011.
9. Switch to the "*Infiltration*" tab. Select "*ComboResidential*" for the "*Using index map*" field.
10. Click on the *Generate IDs* button and click *NO*.
11. There will be two fields added at the end of the table for the new land use. Find a column that has the same soil type (i.e. *Loam* and *Clay Loam*) and copy the infiltration parameters from that column to the new field(s).
12. Click *Done*
13. On the data tree, right click on *Residential* and select *Save Project File...* Save the project (as *C:\Training\Personal\Scenarios\Residential.prj*)
14. Select Yes if prompted to replace existing file.

2.2.5 Save the projects a GSSHA groups

We will save all these GSSHA projects as GSSHA groups so that they can be opened or run together. If by chance you close any of your GSSHA project, you can open the group and resume the tutorial. As we keep adding new GSSHA model to our project, we will keep saving all of them as GSSHA group. Saving all the models could be done at the end when you will be done creating all scenarios, but it is advisable to keep saving the projects you have created so far as GSSHA groups, so that you can open, modify and close all the projects at any time.

1. In *2D Grid Module*, select *GSSHA/Save Group*.
2. Toggle on all the GSSHA projects listed in *Save GSSHA Groups* dialog, browse and save the group as *C:\Training\Personal\Scenarios\Scenarios.ggp*.

2.3 Changing the Land Use from Residential to Industrial

To simulate a progressive development scenario, let us assume that the portion of land which we changed to a residential in the previous section now changes to an industrial area. So, we will update the GSSHA model to reflect this change and see the differences in the watershed response.

2.3.1 Creating a new GSSHA project for the changed land use

There are two steps involved in creating new GSSHA project.

2.3.1.1 Creating a new GSSHA Coverage

1. In the project explorer, under *Coverages*, right click *GSSHARes* coverage and select *Duplicate*.
2. Rename the new GSSHA coverage as *GSSHAIndus*.

2.3.1.2 Creating a new GSSHA Project

1. In the project explorer, under *2D Grid Data*, right click on GSSHA project named *Residential* and select *Duplicate*. This will create a new GSSHA project with name *Residential (2)*.
2. Under *2D Grid Data*, expand *Residential (2)*. Right click on *GSSHARes* and assign *GSSHAIndus*
3. Right click on project *Residential (2)* and select *Save Project File....* Save it as *C:\Training\Personal\Scenarios\Industrial.prj*. This will also rename the project.

2.3.2 Changing the Roughness and Infiltration Parameters

We are making an assumption here that the portion of the land that was converted to residential now is converted to Industrial. Since we are using the exact same area to be converted, we can just modify the index map used in residential land use (which we have now duplicated to use in this case) instead of creating a new one.

1. Under *2D Grid Data*, right click the GSSHA project named *Industrial*.
2. Right click on the GSSHA project named *Industrial* on the data tree and select *Map Tables...*
3. The roughness will remain the same because the Manning's roughness coefficients for residential and industrial areas are almost the same.
4. Switch to the "*Infiltration*" tab and scroll all the way to the right to find the new land use \soil ID.
5. For the last two columns, reduce the values of hydraulic conductivity and capillary head to simulate the effects of more impervious land cover. Make the following changes:

Parameter	Loam \ Land ID # 33	Clay Loam \ Land ID # 33
Hydraulic Conductivity	0.50	0.09
Capillary Head	4.440	10.44

6. Leave other parameters the same and click *Done*.

7. On the data tree, right click on *Industrial* and select *Save Project File...* Save the project (as *C:\Training\Personal\Scenarios\Industrial.prj*)
8. In *2D Grid module*, select *GSSHA/Save group*. Toggle on all the three GSSHA projects (base, Residential and Industrial) and save as *C:\Training\Personal\Scenarios\Scenarios.ggp*.

2.4 Adding a detention basin

It can be easily inferred that changing the land use in a certain portion of the watershed causes a considerable increase in the peak flow at the watershed outlet. This situation is often undesirable as various hydraulic structures like culverts, channel etc downstream of the outlet become undersized. To mitigate the problems of flooding we will introduce a detention basin in the watershed.

2.4.1 Creating a new GSSHA project for the detention basin

2.4.1.1 Creating a new GSSHA Coverage

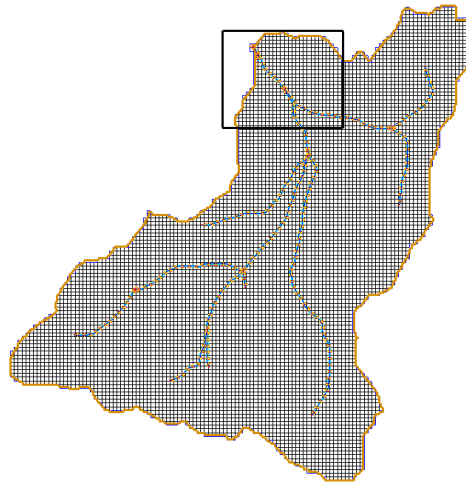
1. In the project explorer, under *Coverages*, right click *GSSHAIndus* coverage and select *Duplicate*.
2. Rename the new GSSHA coverage as *GSSHADB*.





2.4.1.2 Creating a new GSSHA Project

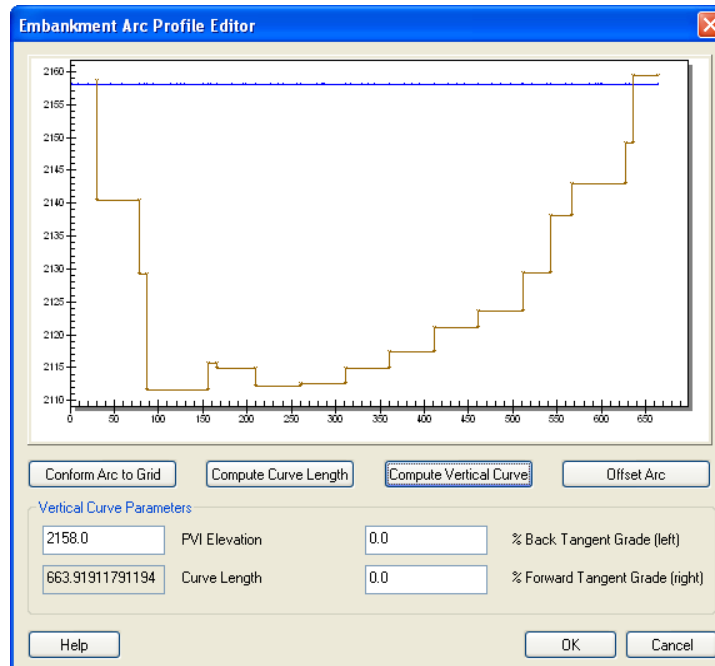
1. In the project explorer, under *2D Grid Data*, right click on GSSHA project named *Industrial* and select *Duplicate*. This will create a new GSSHA project with name *Industrial (2)*.
2. Under *2D Grid Data*, expand *Industrial (2)*. Right click on *GSSHAIndus* and assign *GSSHADB*.
3. Right click on project *Industrial (2)* and select *Save Project File....* Save it as *C:\Training\Personal\Scenarios\GSSHA\DetBasin.prj*. This will also rename the project.
4. Check the display of the background image off (If turned on).





2.4.2 Defining the detention basin parameters

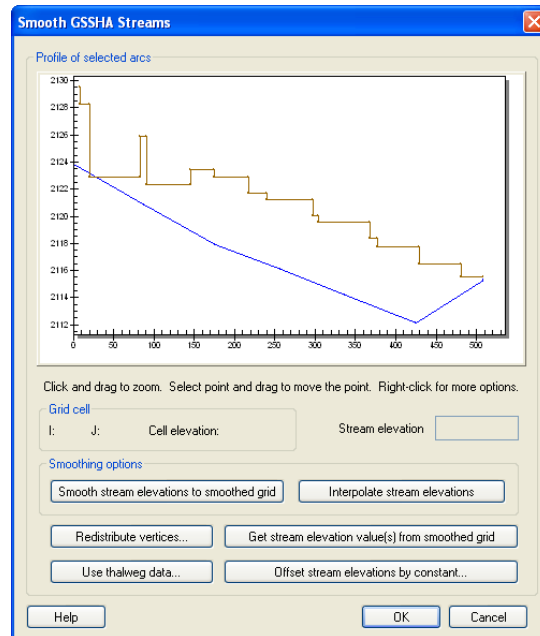
1. Import a background map which will guide you to draw an embankment arc as well as insert a node. Browse and open file *C:\Training\Scenarios\Images\DetBasin.jpg*.
2. Turn on the display of 2D grid data by checking it on. Zoom into the area as shown in the following figure. If you cannot see the background image, it might be because you have selected one of the index maps. In that case, click on *elevation (elev)* under *2D grid Data*.



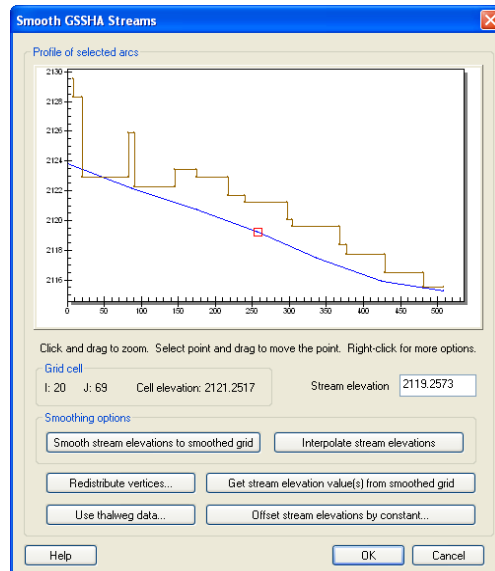
3. To define a detention basin, we need to have an embankment arc that encompasses the detention basin and a culvert at the outlet of the detention basin. We will insert the detention basin on the stream arc.
4. Under Coverages, click on *GSSHADB*, to select it.
5. Switch to *Map Module*  and select the “*Create Feature Point Tool*” .
6. Insert a node on the stream arc at the location as shown in the background map.
7. Draw an embankment arc as shown in the background image as well. **Make sure you do not click on the stream or boundary arcs while drawing the embankment arc which will end the arc at the intersection and we do not want that happen.**
8. Once you have inserted the node and drawn the embankment arc, turn the display of the background image off by checking it off in the project explorer.
9. In the *Map Module*  click on “*Select Feature Arc Tool*”  and double click on the embankment arc. Change the type to *Embankment* and click on the (...) button under the *Embankment* attribute selection.
10. Enter 2158 for a PVI Elevation and click on the *Compute Vertical Curve* button. This will plot the embankment arc over the grid (See the following figure).
11. Click *OK* and *OK* again.




12. In the *2D grid module*, check the elevation of the grid cell at which the node is located. The cell elevation should be 2115.53
13. In the *Map Module* , Click on “*Select Feature Point\Node Tool*”  and click on the node you just created. See the elevation of the node on the properties window to the right side. The node elevation is about 2110 (it might be little different in your case as it depends on the location of the node you have inserted).
14. Edit the node elevation in the properties window so it is 2115.3 such that it is not too far below the grid. Once you change the elevation of the node, the stream arc profile needs to be re-interpolated so that there is no adverse gradient along the flow direction.
15. Still in the *Map Module*  and select the model to be *GSSHA* in the model selection drop down box.
16. Select the stream arc  which is just upstream of the node.
17. Select *GSSHA/ Smooth Stream Arcs..* The profile of the stream arc will be as shown in the following Figure.

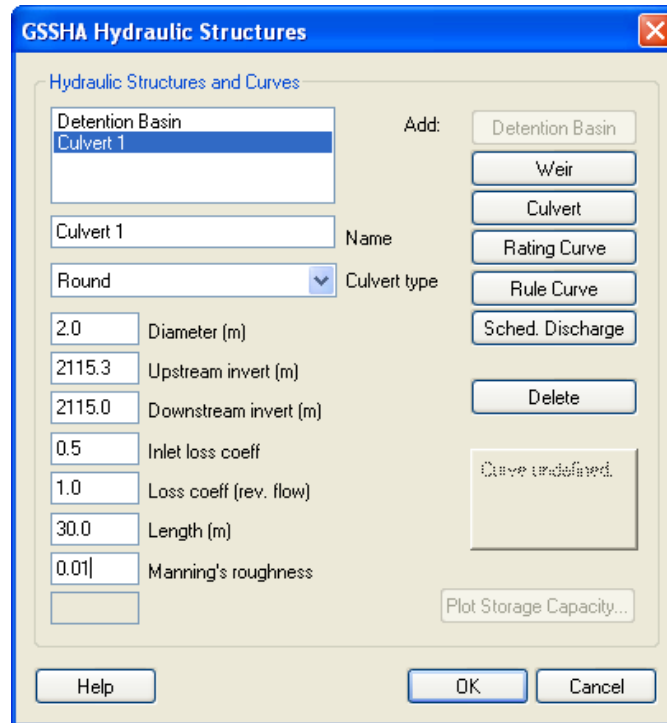


18. Drag the vertices along the stream arc upward (towards the grid) so that there is no adverse gradient and until it similar to the following figure. **Do not click on *Interpolate stream elevations* button which we have been using to smoothen the channel thalweg because doing that will change the node elevation which we do not want.**



19. Click *OK*
20. Click on the “*Select Feature Point/Node Tool*”  and double click on the node you created. This will open the *Properties* dialog. Click on the (...) button under *Hydraulic Structures*.
21. Insert a *Detention basin* by clicking on the *Detention Basin* button. Enter 2115.55 for Minimum, 2115.55 for Initial and 2150 for the Maximum water surface elevations.

22. Then insert a culvert by clicking on *Culvert* button. Enter the parameters for the culvert as shown in the following figure.



The upstream and downstream elevations of the culvert are taken from the node elevations of the stream arc where the culvert is being inserted. The length and diameter of the culvert are governed by the flow rate and site condition.

23. Click *OK* and *OK* again.
24. On the data tree, right click on *DetBasin* and select *Save Project File...*
Save the project file as *C:\Training\Personal\Scenarios\DetBasin.prj*
25. Save the GSSHA group selecting all the 4 GSSHA projects.

This completes another scenario we are going to model.

2.5 Adding a Wet land

A detention basin might not always be the best flood mitigation alternative. As another alternative we will add a wet land in the watershed and compare how it reduces the flood. Restoring a wet land helps maintain an ecological balance in the watershed and this type of solution could be preferable over a structural solutions.

2.5.1 Creating a new GSSHA project for wetland

2.5.1.1 Creating a new GSSHA Coverage

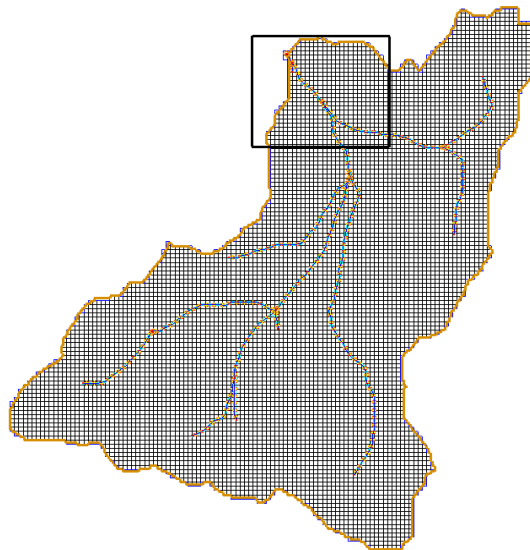
1. In the project explorer, under *Coverages*, right click *GSSHAIndus* coverage and select *Duplicate*.
2. Rename the new GSSHA coverage as *GSSHAWL*.

2.5.1.2 Creating a new GSSHA Project

1. In the project explorer, under **2D Grid Data**, right click on GSSHA project named **Industrial** and select **Duplicate**. This will create a new GSSHA project with name **Industrial**.
2. Under **2D Grid Data**, expand **Industrial**. Right click on **GSSHAIndus** and assign **GSSHAWL**.
3. Right click on project **Industrial (2)** and select **Save Project File....** Save it as **C:\Training\Personal\Scenarios\WetLand.prj**. This will also rename the project.



2.5.2 Defining the wet land parameters

1. Open the background image which will guide you about the location of the wetland. Browse and open file **C:\Training\Scenarios\Images\Wetland.jpg**
2. Zoom into the area as shown in the following Figure.





2-1: Location of Wetland

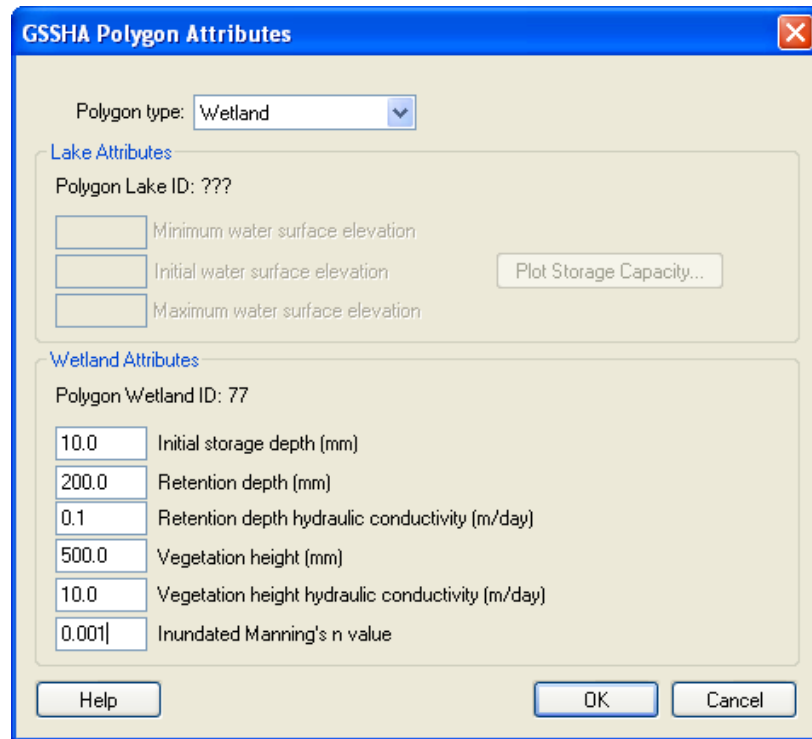
To define a wetland, we need to draw a polygon and define it as the wetland.

3. Make sure **GSSHAWL** is selected as your active GSSHA coverage.
4. Switch to the **Map Module**  and click on the “**Create Feature Arc Tool**” . Then draw a polygon as shown in the background image. While drawing the arc, make sure that you do not click on the stream arc at any location.

In this exercise we have inserted the wetland such that the stream passes through it, but GSSHA can model the wetland both on and away from the stream arc.

5. Once you are done drawing the arc, turn off the display of background image.
6. Select the **Select Feature Arc Tool** 

7. Select the arc you have just created and choose the **Feature Objects / Build Polygon** option.
8. Click on the “*Select Feature Polygon Tool*”  double click inside the polygon you just created. This will open the *GSSHA Polygon Attributes* dialog.
9. Select *Wet Land* for the Polygon Type and define the following attributes for the wetland.



2-2: Polygon Attributes Dialog

10. Click *OK*.
11. On the data tree, right click on *WetLand* and select *Save Project File...*
Save the project file as *C:\Training\Personal\Scenarios\WetLand.prj*
12. Save the GSSHA group selecting all the five GSSHA models.

2.6 Adding an Infiltration basin

This is another abatement measure that can be used as a flood control measure. An infiltration basin is some designated area in the watershed that has a very high infiltration rate. Let us modify one of the existing GSSHA project in order to define the Infiltration basin.

2.6.1 Creating a new GSSHA project for Infiltration basin

2.6.1.1 Creating a new GSSHA Coverage

1. In the project explorer, under *Coverages*, right click *GSSHAIndus* coverage and select *Duplicate*.
2. Rename the new GSSHA coverage as *GSSHAIB*.

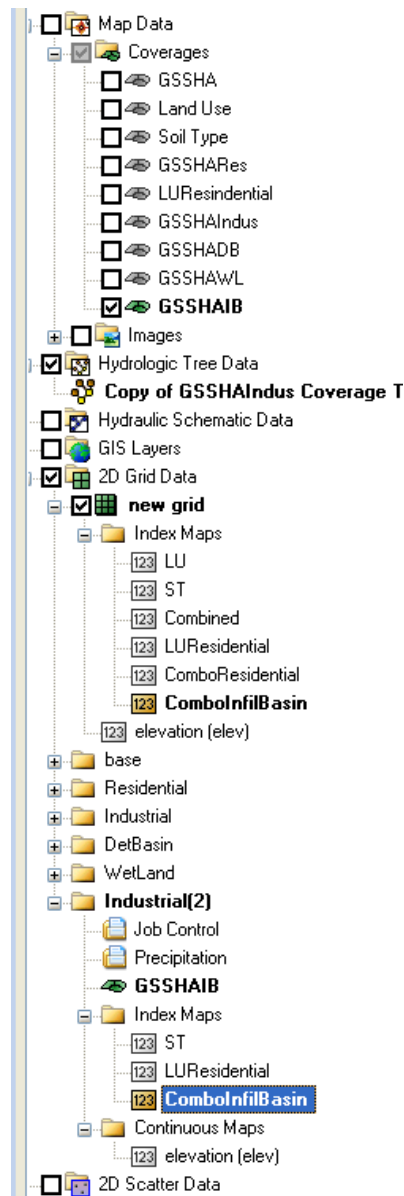
2.6.1.2 Creating a new GSSHA Project

1. In the project explorer, under *2D Grid Data*, right click on GSSHA project named *Industrial* and select *Duplicate*. This will create a new GSSHA project with name *Industrial (2)*.
2. Under *2D Grid Data*, expand *Industrial (2)*. Right click on *GSSHAIndus* and assign *GSSHAIB*.

2.6.1.3 Creating a new Index map

Here instead of modifying the polygons in a coverage and creating a new index map, we will modify an index map. So that the index maps associated with other GSSHA models do not change, let us create a copy of the *ComboResidential index map* and modify it for an Infiltration basin.

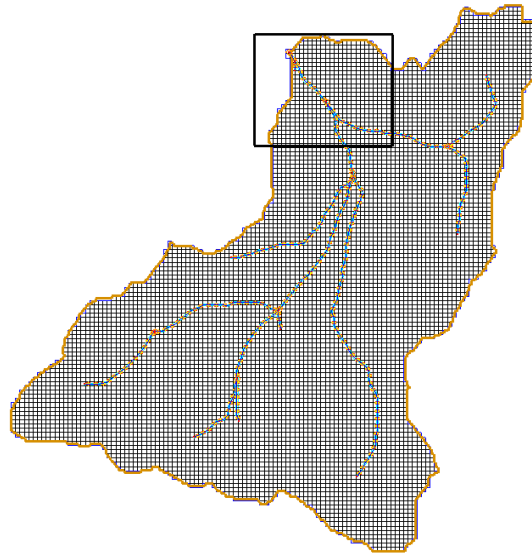
1. Right click on project *Industrial (2)* in the project explorer and select *Maps*.
2. In *GSSHA maps* dialog, click on *Data Calculator* button.
3. In Data calculator, double click on *d6. ComboResidential* dataset which will put a symbol (d6) in the *Expression box*.
4. Enter *ComboInfilBasin* in *Result* field.
5. Toggle *Index Map* option on.
6. Click *Compute*. Click *Done* and *Done* again. This should have added a new index map in the project explorer.
7. Right click *Index Maps* folder under project *Industrial (2)* and assign *ComboInfilBasin* index map. Similarly, remove *ComboResidential* index map from this project. Just to make sure you data tree should look something like the following figure:



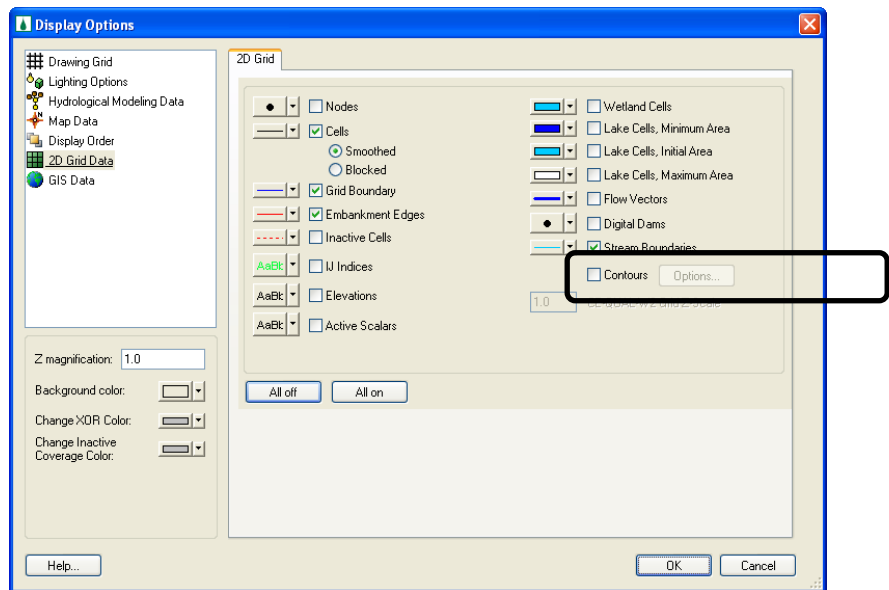
3. Right click on project **Industrial (2)** and select **Save Project File....** Save it as **C:\Training\Personal\Scenarios\InfilBasin.prj**.



2.6.2 Defining the Infiltration basin parameters

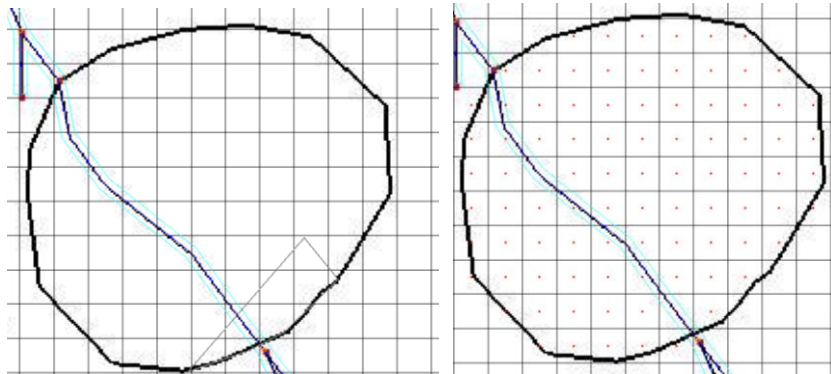
1. Turn on the display of **Wetland** background image (yes! we will create an infiltration basin at the same location as the wetland). If you had deleted the image, you can open it from **C:\Training\Scenarios\Images\Wetland.jpg**
2. Zoom into the area as shown in the following Figure.



3. We will now update the index map properties for the cells that lie within the polygon shown in the background image.
4. In the data tree under **2D Grid Data**, select **InfilBasin** project and click on **ComboInfilBasin** index map to select it. This will make the background image disappear, because the grid contours are color filled. So select **Display/ Display Options**. In the **2D Grid Data** tab, check off the option for grid contour (See the following figure).

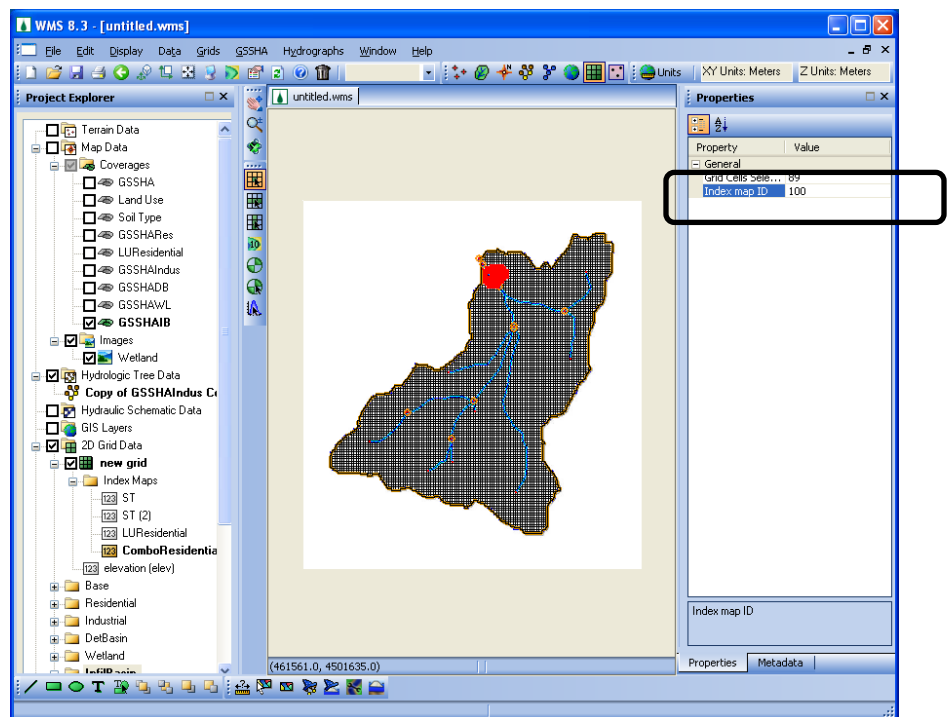


5. In **2D grid Module** , click on **Select Grid Cell** tool . Then select **Edit>Select with Polygon...** Click OK.
6. Now, trace the polygon as seen in the background image and at the end double click. This will select all the grid cells that are within the polygon you traced.



Once the cells are selected, change the *Index Map ID* to **100**. This is done by changing in the properties window on the right hand side of the screen, see the following figure:

We defined a different Index Map ID for these cells so that these will be displayed as different field in the mapping table and we can increase the infiltration rate for these cells.



7. Let us now turn the 'contour color fill' back in. Select **Display/Display Options**. In the **2D Grid Data**, check the option to display contours. Zoom in to the area and see if the cells that are within the infiltration basin are colored differently. If not change the Index Map id for that cell to 100 again.
8. In the project explorer, under **2D Grid Data**, right click **InfilBasin** project and select **Map tables...**

9. Switch to the *Infiltration* tab. Select *ComboInfilBasin* as the index map and Click on *Generate ID* button. Select NO. This will add a new field at the end of the list with ID 100.
10. Enter the following values for this field:

Hydraulic Conductivity	23.56
Capillary Head	4.95
Porosity	0.437
Pore Index	0.694
Residual Saturation	0.02
Field Capacity	0.091
Wilting Point	8.4

11. Click Done.
12. On the project explorer, right click on *InfilBasin* and select *Save Project File*. Save the project file as *C:\Training\Personal\Scenarios\InfilBasin.prj*
13. Save the GSSHA group selecting all six GSSHA models.

2.7 Adding a Buffer Strip

A buffer strip is generally characterized by very high surface roughness because it offers resistance to the overland flow.

2.7.1 Creating a new GSSHA project for buffer strip

2.7.1.1 Creating a new GSSHA Coverage

1. In the project explorer, under *Coverages*, right click *GSSHAIndus* coverage and select *Duplicate*.
2. Rename the new GSSHA coverage as *GSSHABS*.

2.7.1.2 Creating a new GSSHA Project

1. In the project explorer, under *2D Grid Data*, right click on GSSHA project named *Industrial* and select *Duplicate*. This will create a new GSSHA project with name *Industrial (2)*.
2. Under *2D Grid Data*, expand *Industrial (2)*. Right click on *GSSHAIndus* and assign *GSSHABS*.

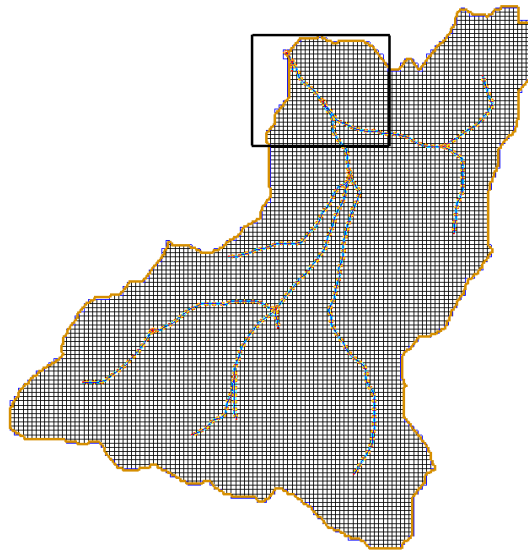
2.7.1.3 Creating a new Index map



We will create a copy of existing index map and modify its IDs to define a buffer strip. So, let us create a copy of the *LUResidential index map* and modify it for the buffer strip.

1. Right click on project **Industrial (2)** in the project explorer and select *Maps*.
2. In *GSSHA maps* dialog, click on *Data Calculator* button.
3. In Data calculator, double click on *d5. LUResidential* dataset which will put a symbol (d5) in the *Expression box*.
4. Enter *LUBufferStrip* in *Result* field.
5. Toggle *Index Map* option on.
6. Click *Compute*. Click *Done* and *Done* again. This should have added a new index map in the project explorer.
3. Right click *Index Maps* folder under project **Industrial (2)** and assign *LUBufferStrip* index map. Similarly, remove *LUResidential* index map from this project.
4. Right click on project **Industrial (2)** and select *Save Project File....* Save it as *C:\Training\Personal\Scenarios\BufferStrip.prj*.

2.7.2 Creating the Buffer Strip

1. Change the display option of the *2D Grid Data* to turn off contour color fill
2. Open the background image which has the outline for the wetland we want to insert here. Browse and open file *C:\Training\Scenarios\Images\BufferStrip.jpg*
3. Zoom into the area as shown in the following Figure.





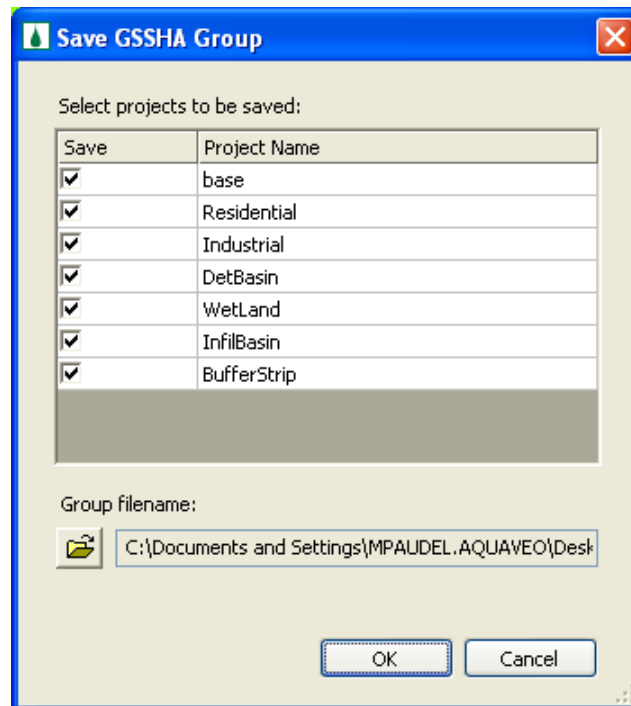
4. You will see an arc embedded in the background image that shows the location of the proposed buffer strip.
5. Since a buffer strip has high roughness value, we will modify the Land Use Index map. In the data tree, under *2d Grid Data*, select project **BufferStrip**.
6. Click on *LUBufferStrip Index map* to select it.
7. In *2D grid Module* , click on *Select Grid Cell* tool .
8. Hold Shift Key in the keyboard and select the grid cells that overlay the buffer strip arc shown in the background image. Once you have selected all the cells release the shift key.

9. Change the *Index Map ID* to **200**. This is done by changing in the properties window on the right hand side of the screen
10. Turn the ‘contour color fill’ option back in. Select **Display/Display Options**. In the *2D Grid Data*, check the option to display contours. Zoom in to the area and see if the cells that are defined as the buffer strip are colored differently. If not change the Index Map id for that cell to 200 again.
11. In the data tree, under *2D Grid Data*, right click **BufferStrip** project and select **Map tables...**
12. In the *Roughness tab* and Click on *Generate ID* button. Select NO. This will add a new field at the end of the list with ID 200.
13. Enter a roughness value of 2.0
14. Click Done.
15. Turn off the display of background image and turn back the contour options on.
16. On the data tree, right click on project **BufferStrip** and select **Save Project File...** Save the project file as **C:\Training\Personal\Scenarios\BufferStrip.prj**

2.8 Saving a GSSHA Group

We are done defining all the different scenarios we wish to compare. You must have noticed that we have not run any of the scenarios yet. Instead of running them individually, we will save all these GSSHA projects as a GSSHA Group. WMS will then run each of the projects and plot the hydrograph from each simulation in the same window so that we can compare the different alternative abatement measures.

1. In the *Grid Module*  select **GSSHA | Save Group...**
2. In the Save GSSHA Group dialog, check on all seven GSSHA models to be saved in the group.
3. Although we have been saving the projects as groups, to make sure they are saved properly, click on the *Browse* button  and save the group as **C:\Training\ Personal\Scenarios\Scenarios.ggp**
4. Select OK.



2.9 Running a GSSHA Group

Now we will run the scenarios so that we can compare the detention basin and wetland alternatives to the modified land use.


1. Select **GSSHA | Run GSSHA group...**
2. Make sure that all seven projects are selected and the path is correct.
3. Click *OK*
4. WMS will run all these seven scenarios and plot the outflow hydrographs on the same plot.
5. Export all seven hydrographs to the spreadsheet named *C:\Training\ScenarioModeling.xls*

Long Term Simulations in GSSHA

In this tutorial you will see how to set up and run a long-term simulation in GSSHA. You will begin with an existing project file. Long-term simulations typically involve running several rainfall events along with the evapotranspiration model for weeks to months. There are two key parts to running a long-term simulation. The first is to set up the precipitation file for multiple events, and the second is to set up the evapotranspiration model with its hydrometeorological (hmet for short) data.

3.1 Open an Existing GSSHA Project

Open a WMS project file for the Judy's Branch watershed. This model has been set up to simulate a single rainfall event. We will modify the project and perform a long term simulation.

1. In the *2D Grid Module*  select *GSSHA / Open Project File...* Browse and open the file *C:\Training\LongTerm\Judys.prj*
2. Save the project with a different name *C:\Training\Perwsonal\LongTerm\longterm.prj*

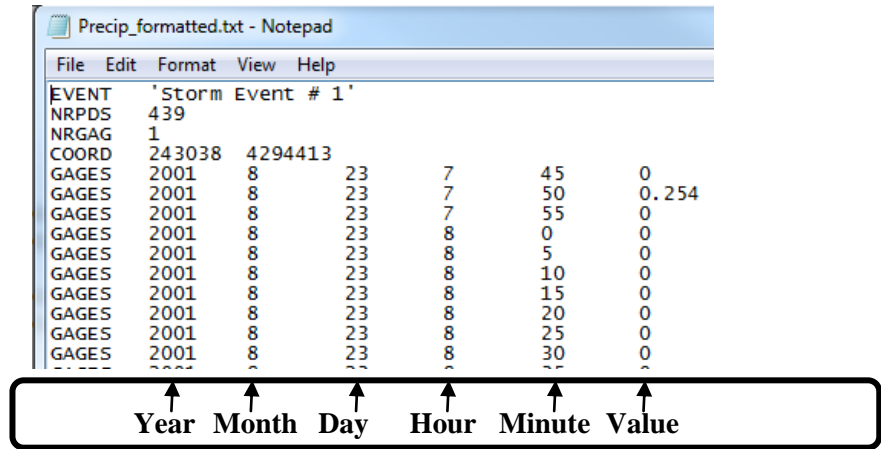
3.2 Format Data for Long-Term Simulation

A long-term event typically consists of multiple rainfall events, often with several rain gages. Multiple gage events can either be setup using WMS or using the *Time Series Data Editor* which is a convenient tool developed for managing such time series data.

Minimize the WMS window and from the *Start* menu of your computer, open the *Time Series Data Editor*.

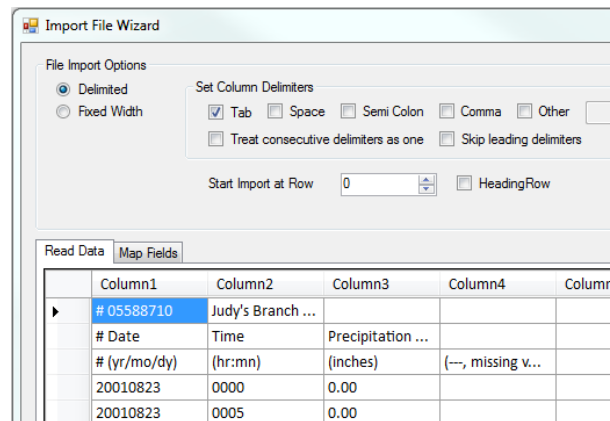
3.2.1 Format the Precipitation Data

Precipitation data preparation for GSSHA involves formatting as shown in the following figure and saving the formatted data as a gage file (*.gag).

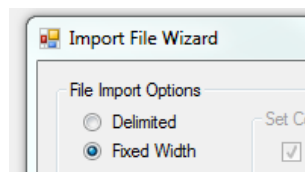


The raw data often does not come in this format and needs some processing before it can be used in a *.gag file. Follow these steps to get the raw data formatted:

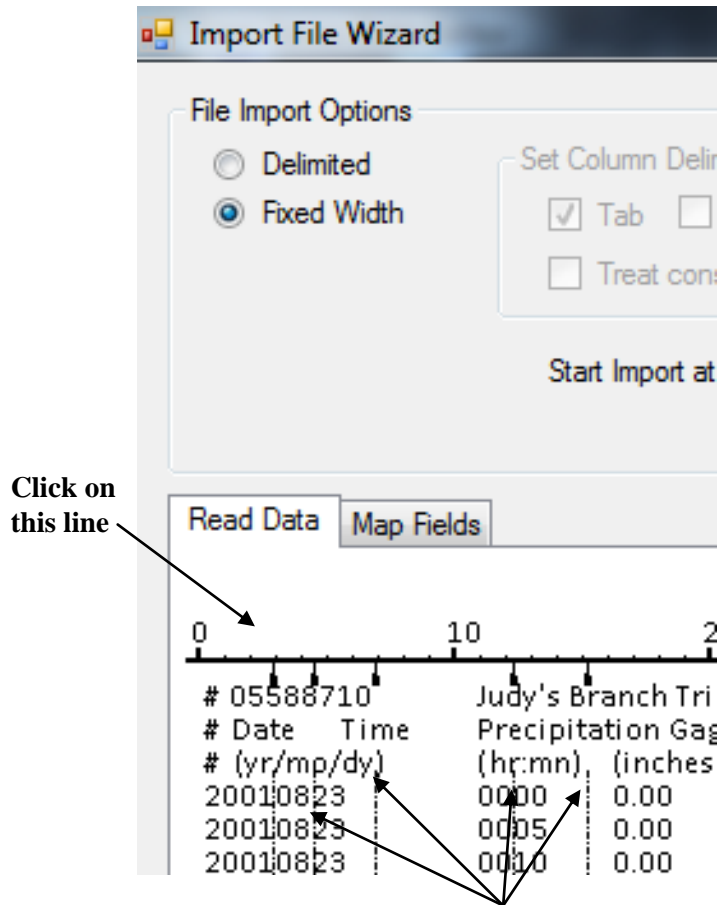
1. In the *Time Series Data Editor* window, select **File/Open**.
2. Open the file **C:\Training\RawData\JudysBranch\Long_Term_Simulation\Precip_data\precip_raw.txt**.
3. This will open the data into the *Import File Wizard*.



4. Select **Fixed Width**



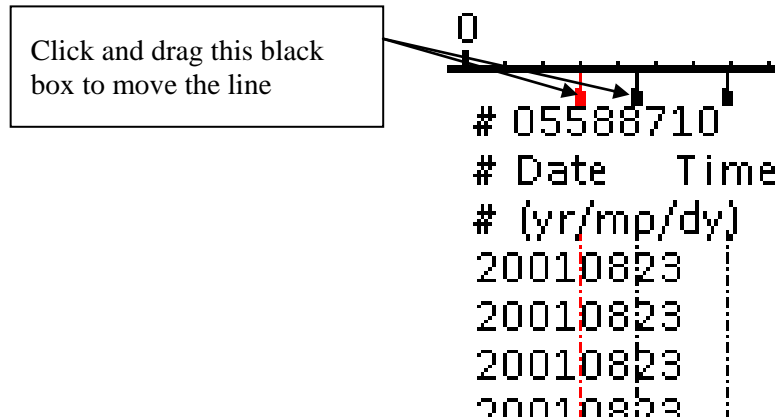
5. Make sure that you are in the *Read Data* tab.
6. Enter 3 for *Start Import at Row* field to clear out three lines of heading text that is in the data. This number represents the row number from which the actual precipitation data begins and means that nothing from the first three lines will be imported. Leave the *Heading Row* option unchecked.
7. Click on the test area or on the spacing bar (Horizontal line with numbers 0, 10, 20 etc) at the top of data which will insert a vertical line representing the location to separate columns.



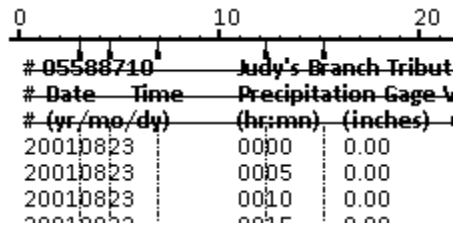
Data Separation lines

8. If you clicked at a wrong location, you can drag the line to the desired location. To do this click on little black dot at the top of the line which will turn to red once selected. Then while holding the mouse down drag the line head and move to the desired location

3-4 GSSHA Modeling Using WMS – Volume 3



- Make sure that you have inserted the column separators such that *Year*, *Month*, *Day*, *Hour*, *Minute* and *Value* are in separate columns, See the following figure.



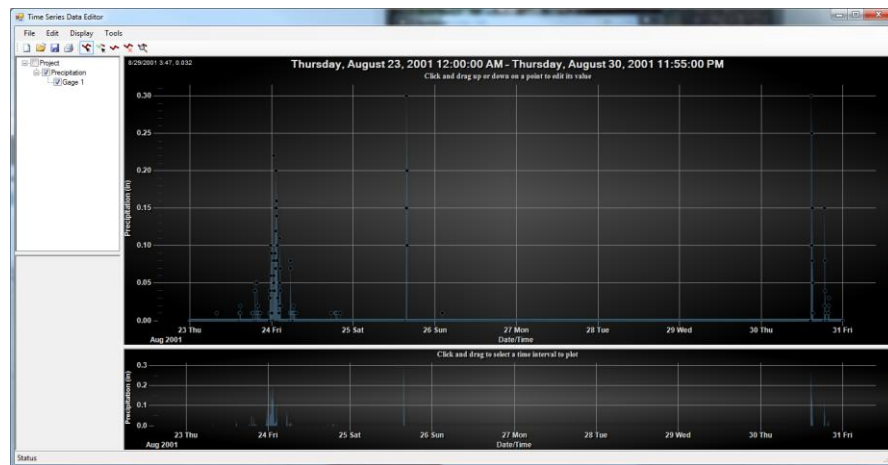
- Then click *Next* which will take you to the *Map Fields* tab.
- Enter the name for *Time Series List* as *Precipitation*.
- Make sure that the data is properly separated into the columns. If they are not separated properly, you can click *Back* button and modify it.
- Now map each column as shown in the following table:

Column	Mapping Data
Column 0	Year
Column 1	Month
Column 2	Day
Column 3	Hour
Column 4	Minute
Column 5	Value

- For the *Value* (Column 5), enter *In* to define the unit and *Gage 1* for the name. See the following figure and compare your mapping:

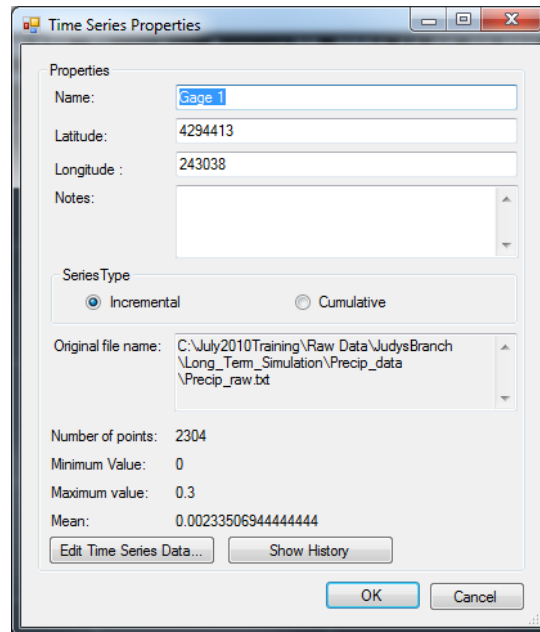
Mapping Values	Year	Month	Day	Hour	Minute	Value
Units						in
Names						Gage 1
*						
	Column 0	Column 1	Column 2	Column 3	Column 4	Column 5
	2001	08	23	00	00	0.00

15. Click OK to close the *Import File Wizard*.
16. You can now see the data plotted in the form of a time series and listed as Time Series on the project explorer.

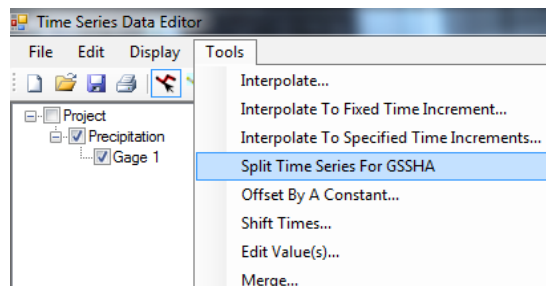



17. The data you just imported is a record of precipitation over about a week's time. Out of this data, we will exclude the times when there was no rainfall for a considerable duration. In other words, we will need to isolate the storms into separate events.
18. On the project explorer, click on *Gage 1* and select **Edit/Properties**. Enter the coordinates for gage location as Latitude = 4294413 and Longitude = 243038.
19. Make sure *Incremental* is selected and click OK.

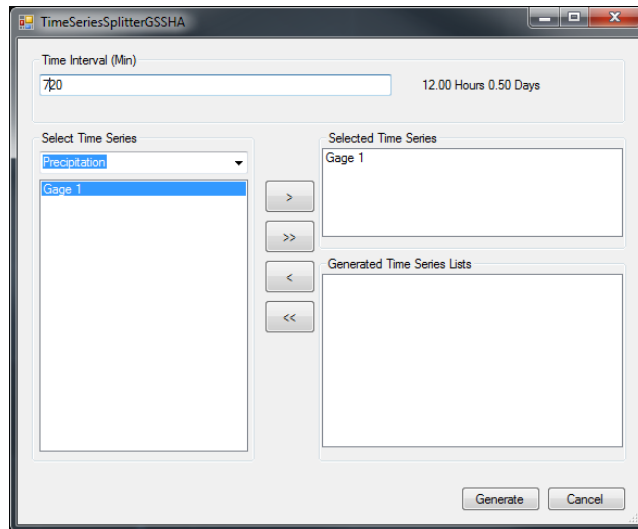
3-6 GSSHA Modeling Using WMS – Volume 3



20. Select *Tools/Split Time Series for GSSHA*.



21. Enter 720 minutes (12 hrs) for the time interval, which is the duration of no rainfall used as a criterion to separate the storms into individual events. In other words, if there is no rainfall for 12 hours after a storm we assume that the storm has ended.
22. Highlight *Gage 1* and click  button to select the data and click *Generate*.



23. This will populate a list of different events on the project explorer. All these events are individual storms each at least 12 hours apart. Now, we are ready to export the data as a gage file to be used in GSSHA.
24. Select **File/Save As**. Save the data as **C:\Training\Personal\LongTerm\precipitation.gag**.
25. Click the **Save** button. This will open *GSSHA File Generator* dialog.
26. Toggle *Convert inch to mm* at the very top of the dialog which will convert the precipitation data to mm since GSSHA requires the precipitation data to be input as mm.
27. Toggle the check boxes adjacent to the three storms named *GSSHA Event 1*, *GSSHA Event 2* and *GSSHA Event 3*
28. Click **OK**.

This data is now formatted properly and ready to use in GSSHA. Do not close the *Time Series Data Editor* yet. We will use it one more time in the following section to format the hydrometeorological data.

In *Time Series Data Editor*, select **File/New**. Select No when prompted to save.

3.2.2 Format the Hydrometeorological Data

Hydrometeorological data are used in GSSHA to determine how the soil moisture is affected by atmospheric conditions. The hydrometeorological data is used to drive the evapotranspiration model which is particularly important during the time between events when it is not raining. In the following steps we will create a file that contains all the hydrometeorological data for the same period as the precipitation data.

1. The HMET data we will be using here is extracted from a spreadsheet **C:\Training\RawData\JudysBranch\Long_Term_Simulation\Hmet_data\Hmet_raw.xls**.
2. Open this file in Excel.

GSSHA needs the following hydrometeorologic information from this data:

3-8 GSSHA Modeling Using WMS – Volume 3

Date	in "YYYYMMDD" format, Example: "20010823" is August 23, 2001
Hour	in "????z" format, Example: "0100z" is 1:00 AM zulu time, the z stands for zulu, and is needed
Barometric Pressure	in decimal value with inches as the unit, Example: "29.92"
Relative humidity	in percent value, Example: "54"
Sky Cover	in 8ths of the sky that is covered, Example: "0" means no clouds, "4" is halfway covered, and "8" is completely overcast
Wind Speed	in knots, with direction and unit label, Example: "21006KT" is wind from 210 degrees at 6 knots
Temperature	in degrees Celsius, Example: "32" is 32 degrees Celsius
Direct Radiation	W h m ⁻²
Global Radiation	W h m ⁻²

The Barometric pressure, relative humidity, Sky cover and Wind speed are in the first sheet (named KBLV_Scott) where as the two radiation parameters are in the second sheet (named Scott_radiation_2001).

- Since the two sets of data in the two sheets have a different time scale (although they are for the same length of time), we will have to import these data individually to the *Time Series Data Editor*.

Note: If your data has the same start/finish time and same temporal resolution (Same time steps), then you can import all the data at once.

- In Excel, create a copy of this data by saving the workbook as **C:\Training\Personal\LongTerm\Hmet_Raw.xls**.
- Switch to tab **KBLV_Scott** sheet. Delete all other column except the ones shown in the following figure (Right click at the column header and select delete)

	A	B	C	D	E	F	G
1	Date	Time	Wind	Sky (8ths)	Temp (C)	RH%	Altimeter (Inches)
2	20010823	0000Z	19011KT	7	32	54	29.90
3	20010823	0100Z	20007KT	4	29	61	29.92
4	20010823	0200Z	20007KT	4	27	73	29.93
5	20010823	0300Z	21006KT	4	27	73	29.92
6	20010823	0400Z	23007KT	4	26	78	29.93
7	20010823	0500Z	22004KT	4	26	78	29.94

- Select **File/Save as** and save this Sheet as a **TEXT** file in **C:\Training\Personal\LongTerm\Input1.txt**. Click OK and Click Yes
- Similarly switch to second tab in the spreadsheet (**Scott_radiation_2001**) and delete all other columns except the ones shown in the following figure.

	A	B	C	D	E	F
1	year	mo	day	hr (Z)	direct rad	total global rad
2	2001	8	23	00	0	41.99586496
3	2001	8	23	01	0	0
4	2001	8	23	02	0	0
5	2001	8	23	03	0	0
6	2001	8	23	04	0	0

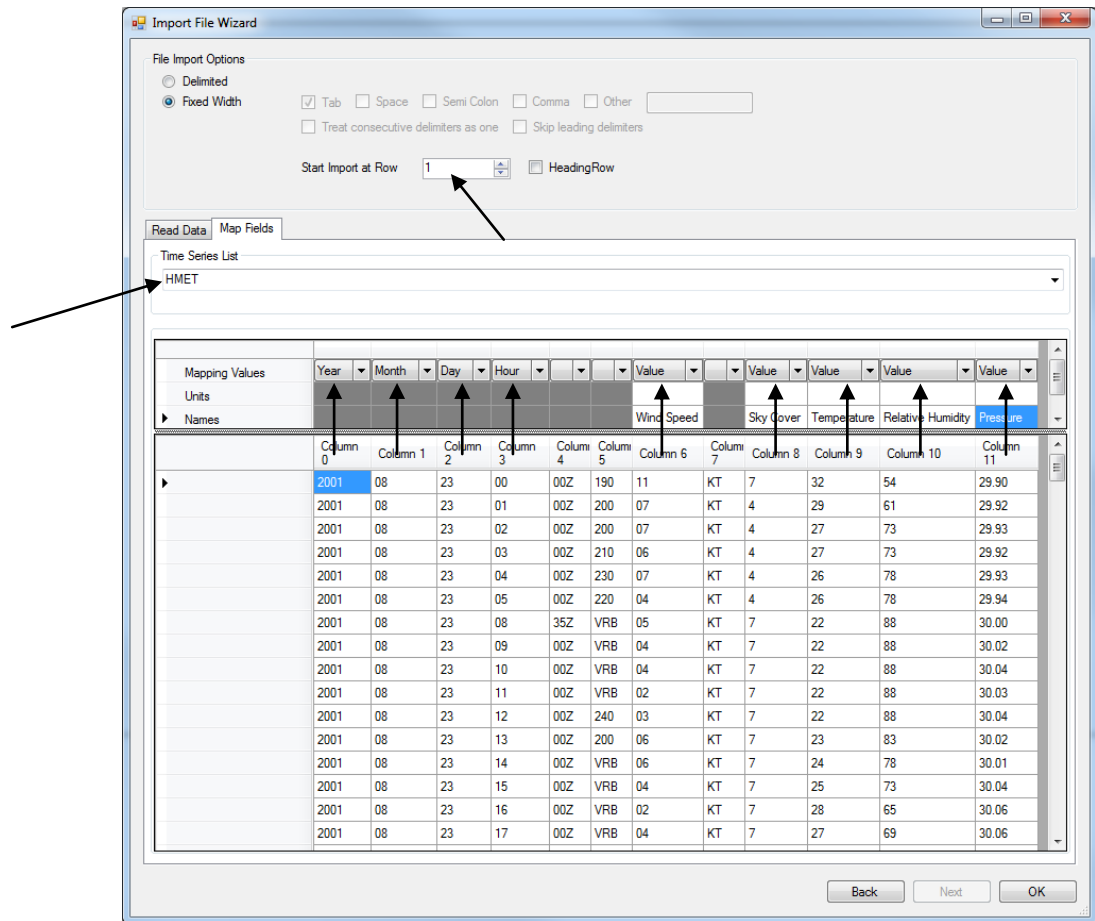
8. Select **File/Save as** and save this Sheet as a **TEXT** file in **C:\Training\Personal\LongTerm\Input2.txt**. Click OK and click Yes.
9. Close Excel and bring the *Time Series Data Editor* back on.
10. Select **File/Open** and open file **C:\Training\Personal\LongTerm\Input1.txt**. This will bring the *File Import Wizard*.
11. Select *Fixed Width*
12. Specify *Start Import at Row* to be 1.
13. Separate the columns as shown below.

Date	Time	Wind	Sky (8ths)	Temp (
20010823	0000Z	19011KT	7	32
20010823	0100Z	20007KT	4	29
20010823	0200Z	20007KT	4	27
20010823	0300Z	21006KT	4	27
20010823	0400Z	23007KT	4	26

Note: The *Date* column should be separated into Year, Month and Day. Initial two digits in the *Time* column represent hours. The fourth and fifth letters on the *Wind* column represent wind velocity, column *Sky(8ths)* is sky cover, column *Temp* is temperature, column *RH%* is % Relative humidity and column *Altimeter(Inches)* is barometric pressure.

14. Click *Next*
15. Enter *HMET* for *Time Series* List name.
16. Map the fields as shown in the following figure and enter the names.

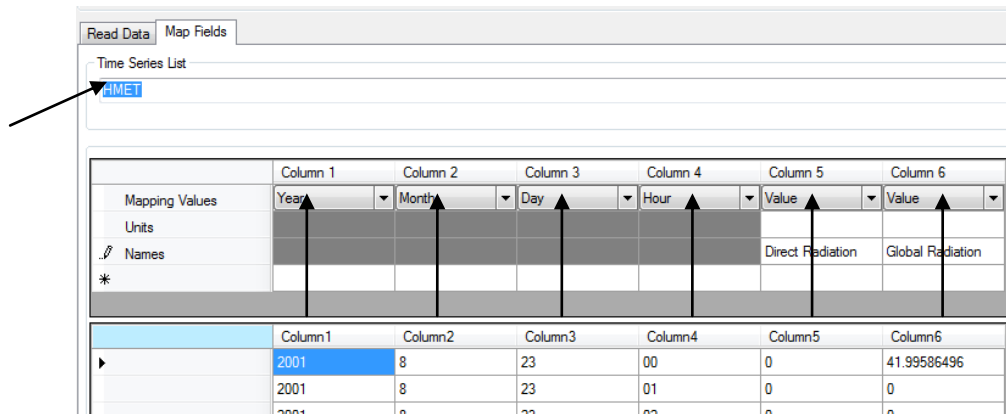
3-10 GSSHA Modeling Using WMS – Volume 3



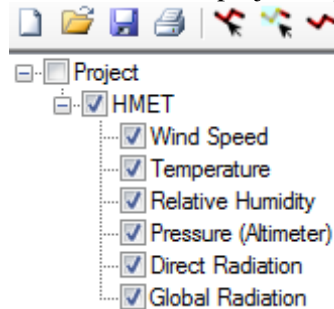
17. Click OK which will bring the data into the *Time Series Data Editor*. You can now see the data plotted as well as listed in the project explorer.

Similarly, let us import *Input2.txt* which has radiation data.

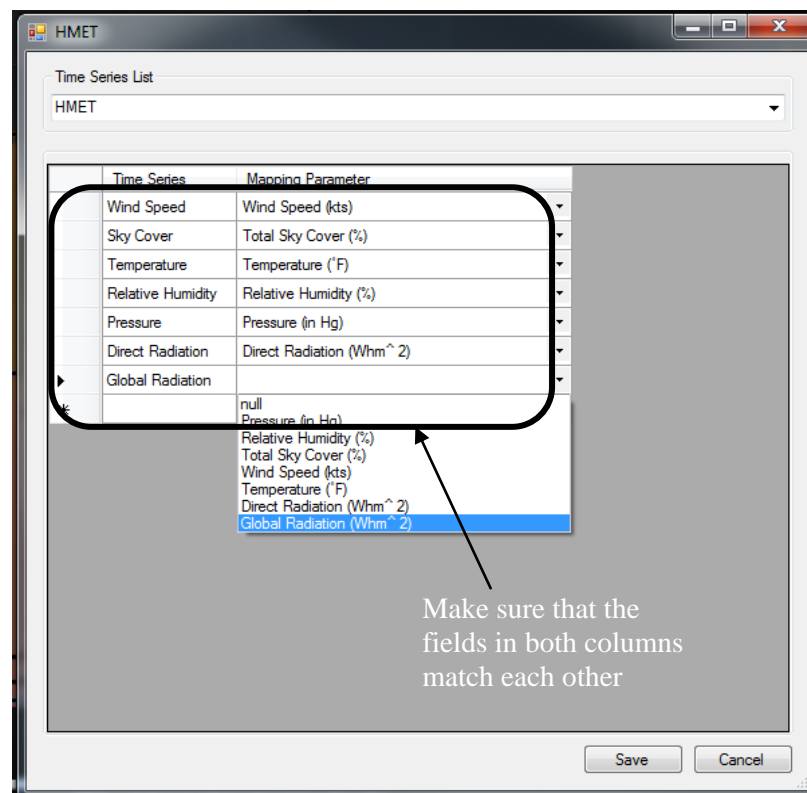
18. Select **File/Open**. Open File located at **C:\Training\Personal\LongTerm\Input2.txt**.
19. All the fields are already split. So you do not have to separate any column.
20. Enter 1 in the *Start Import at Row* field.
21. Click *Next*
22. In the *Time Series Field Select* **HMET** from the drop down menu.
23. Map Columns 5 and 6 to values and enter the names.
24. Select OK



25. Now, you should have seen that the data from second input file has been added to the same time series. Check the project explorer.



26. The data is now ready to export as GSSHA HMET data. Select **File/Save As**.
27. Browse **C:\Training\Personal\LongTerm** and enter **hmet** as the name of the file.
28. In the *Save as Type* field, select (**HMET (*.hmt)**) and click *Save*.
29. In the *HMET* dialog, select the mapping parameters (eg select *Temperature* time series and choose *Temperature* etc) from the drop down box to match with the *Time Series* names. See following figure

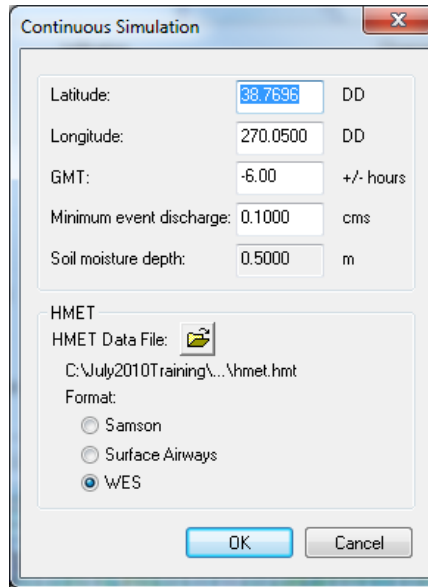


30. Click *Save*.
31. You may now close the *Time Series Data Editor*.

3.3 Setting up Long-Term Simulation

Now we will go back to WMS and set up the Long-term modeling data. First we need to set up the Job control options to turn on long-term mode.

1. Select **GSSHA| Job Control...**
2. In the “Evapotranspiration” section of the window toggle on “Penman Method”.
3. Enter 0.5 for *Soil Moisture depth*
4. Check the box next to "Long term simulation" in the GSSHA Job Control Parameters window.
5. Click the *Edit parameter...* button and enter a value of 38.7696 for “Latitude”.
6. Enter a value of 270.05 for “Longitude”.
7. For “GMT” enter a value of –6.00.
8. Enter 0.10 for “Minimum event discharge”.
9. Make the “Soil moisture depth” equal to 0.5.
10. Click on the folder icon to next to “HMET Data File” to browse for the Hmet text file you created with the *time Series Data Editor*. Navigate to the file and select it. *C:\Training\LongTerm\hmet.hmt*.
11. Under “Format”, toggle on WES.



12. Select *OK*.
13. In the Overland Flow Computation method combo box make sure *ADE* is selected.
14. Select *OK*.

3.3.1 Set up the Evapotranspiration Parameters

1. Select **GSSHA | Map Tables...**
2. Click on the *Evapotranspiration* tab.
3. In the drop down box next to *Using index map* select Land use, then click the *Generate IDs* button.
4. Enter the values required for evapotranspiration using the following table:.

Land use ID	11	14	16	21	41
Land Surface Albedo	0.15	0.22	0.22	0.22	0.2
Vegetation height (m)	0.08	0.1	0.5	1	17
Vegetation Radiation Coeff	0.7	0.5	0.35	0.2	0.15
Canopy stomatal resistance (s\m)	20	20	50	86	100

5. Click *Done*.

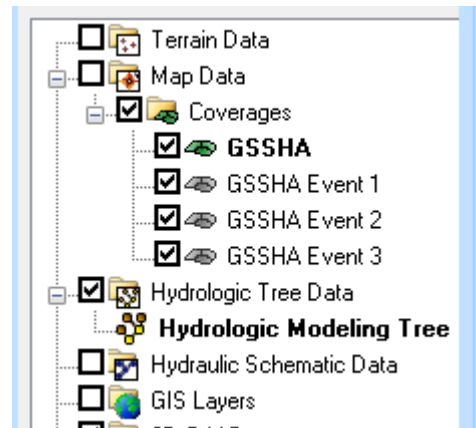
3.3.2 Importing the Gage File

Next we need to tell set up our precipitation file.

1. Select **GSSHA | Precipitation...**
2. Select *Gage* from the drop-down menu.
3. Click the *Import Gage File...* button.
4. Browse to the precipitation file *C:\Training\Personal\LongTerm\precipitation.gag*, select it, and hit *OK*.

Since we only have one gage, the rainfall data is spread out uniformly over the watershed. If we had more than one gage we would pick either Theissen Polygons or Inverse Distance weighted here in this dialog.

5. Select *OK*.
6. You should have seen that three coverages added in the project explorer representing three storms.



3.4 Change the Output Control

We are ready to run now, but first we will want to change some output options. We will not want to output the data sets so frequently.

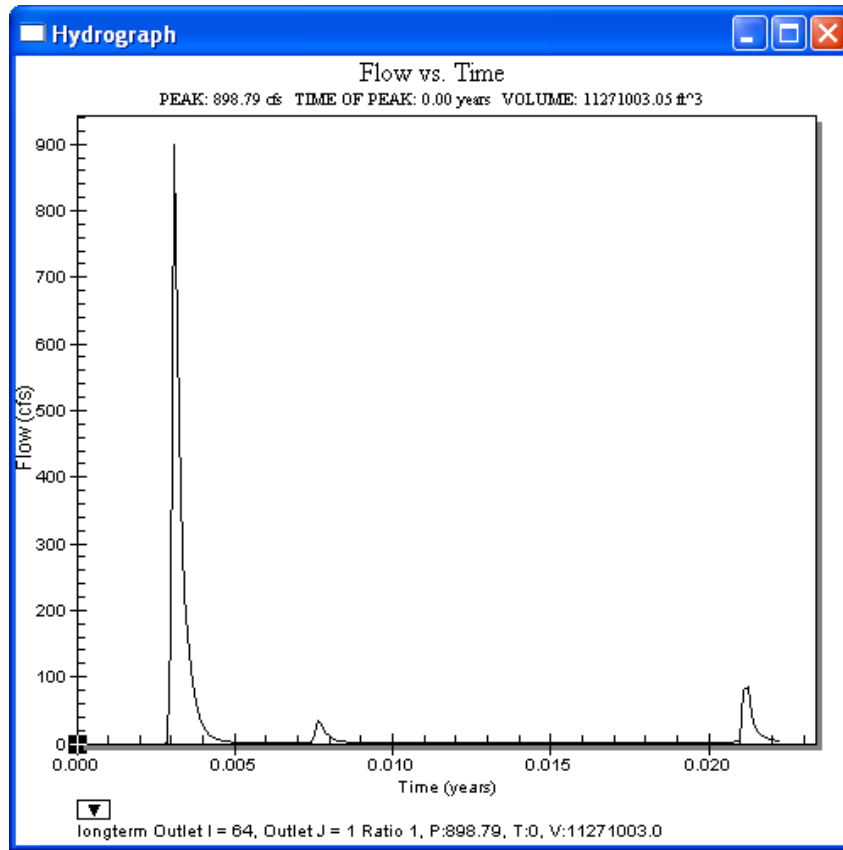
1. Select **GSSHA | Job Control...**
2. Click on the Output Control... button.
3. In the Write frequency section of the dialog, change the Write Frequency to 60 (minutes).
4. Select *OK, OK*.

3.5 Save and Run the model

1. Save the GSSHA project *C:\Training\Personal\LongTerm\ longterm.prj*
2. Select **GSSHA/Run GSSHA** and click *OK*.

3.6 Results

Your results should look something like the figure below.



Simulating Sediment Transport


In this workshop, you will learn how to develop inputs for a sediment transport simulation using GSSHA. You will begin with an existing GSSHA model, the Eight Mile Creek watershed located in Wisconsin, and add the sediment transport features to the model. The model is currently set to run a long term simulation.

4.1 Open an Existing GSSHA Project

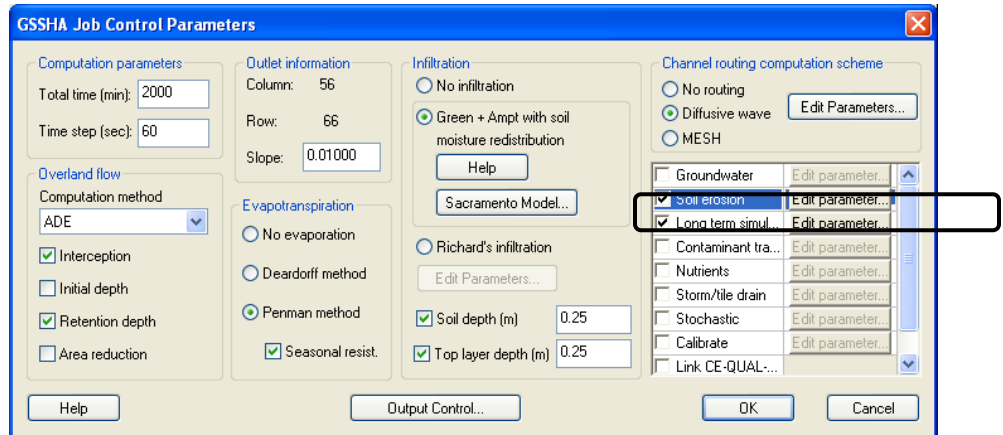
Open the GSSHA model for Eight Mile Creek watershed

1. Start a new instance of WMS.
2. In the 2D Grid Module  select **GSSHA | Open Project File.**
3. Browse and open *C:\Training\Sediment\Base\Base.prj*
4. Save the project as *C:\Training\Personal\Sediment\sed.prj*

4.2 Turning Soil Erosion on and adding sediment

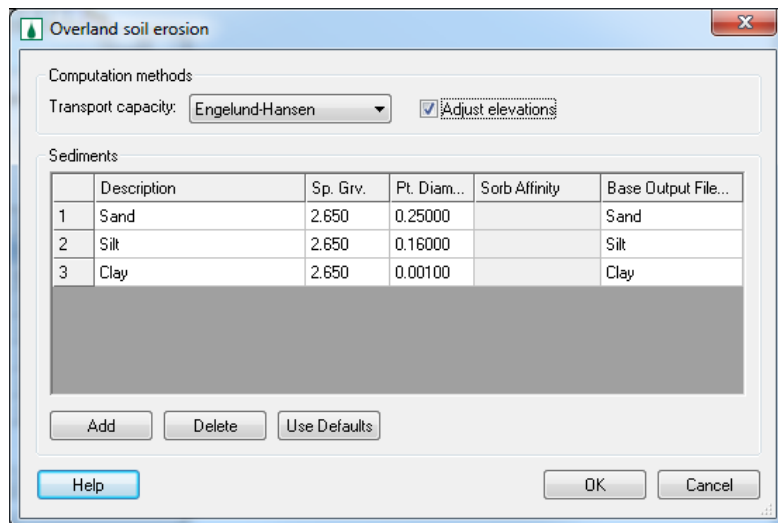
1. In the 2D Grid Module  select *GSSHA/ Job Control...*
2. Click on the *Soil erosion* option to turn it on
3. Click the *Edit parameter...* button next to the soil erosion spreadsheet option

4-2 GSSHA Modeling Using WMS – Volume 3



4. In the *Overland Soil Erosion* dialog, select *Kilinc-Richardson* for Transport capacity.
5. Although you can specify as many sediment particle sizes as you desire, typically three sediment sizes are used to describe the soils – sand, silt, and clay. Click the *Add* button three times to add three sediments
6. Fill in the Sediments spreadsheet with values from the following table:


ID	Description	Sp. Gr.	Particle Diameter (mm)	Base output file name
1	Sand	2.65	0.25	Sand
2	Silt	2.65	0.16	Silt
3	Clay	2.65	0.001	Clay



7. Select the option to adjust the elevation by checking the toggle. If this option is turned on, GSSHA will create an adjusted elevation map based on erosion and deposition which can be compared with the original elevation map to find out the areas of erosion and deposition.
8. Click OK to close the *Overland Soil Erosion* dialog
9. In the *Job Control* dialog, click on *Edit Parameters...* button in the *Channel routing computation scheme* portion (top right corner).
10. Make sure the *Sediment porosity* is set to 0.4, the *Water temperature* at 20.0 deg C, and the *Sand size* at 0.25 mm.
11. Click OK to close the *GSSHA Channel Routing Parameters* dialog
12. Click OK to close the *GSSHA Job Control* dialog

4.3 Specifying Soil Erosion Parameters

Now you will specify the properties of the soils in the watershed related to soil erosion. This is done with the mapping tables.

1. In the 2D Grid Module  select *GSSHA / Map Tables...*
2. Click on the *Soil Erosion* tab.
3. In the *Using index map* combo box, select *LUST* as the index map
4. Click *Generate IDs*
5. Fill in the spreadsheet with values from the following table (Since it is a lot of information, there is an Excel spreadsheet that already has these numbers. Outside of WMS browse and open file *C:\Training\Sediment\ ErosionParams.xls*. Copy the data from this spreadsheet and paste into Erosion tab in the Map table dialog).

Normally you will not change most of these basic parameters. For the Kilinc-Richardson the key parameters are the fractions of each sediment size and the erodibility coefficient.

ID	104	105	106	108	110
Description 1	Sandy Loam	Loam	Silty Loam	Row Crop SL	Commercial
Description 2					
Transport coefficient	0.285	0.285	0.285	0.285	0.285
Transport index	1.3	1.3	1.3	1.3	1.3
Crit. transport capacity	0.0002	0.0002	0.0002	0.0002	0.0002
Rain splash coeff	1200	1200	1200	1200	1200
Runoff detachment coeff	0.0001	0.0001	0.0001	0.0001	0.0001
Runoff detachment index	0.4	0.4	0.4	0.4	0.4
Runoff detachment crit. shear	0	0	0	0	0
Erodibility coeff	0.0048	0.0068	0.0088	0.0688	0.0088

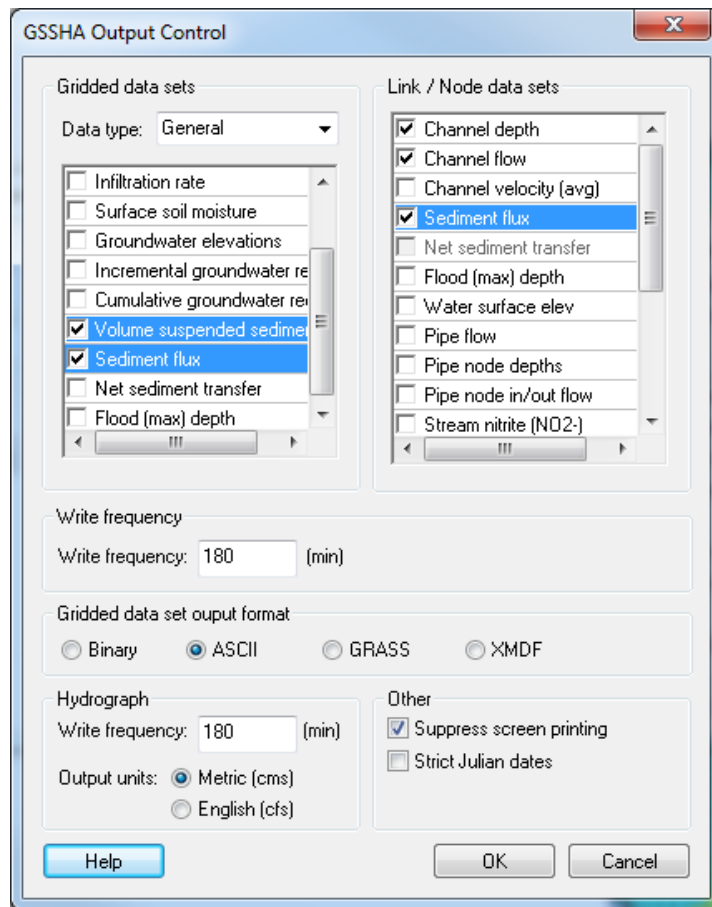
4-4 GSSHA Modeling Using WMS – Volume 3

Sand	0.65	0.45	0.2	0.2	0.2
Silt	0.2	0.35	0.65	0.65	0.65
Clay	0.15	0.2	0.15	0.15	0.15

6. Click *Done* to close the GSSHA Map Table Editor

4.4 Specify the output parameters

1. In *GSSHA/Job Control*, click on *Output Control* button.
2. Under *Gridded Data sets*, turn on *Volume suspended sediment and Sediment Flux*.
3. Similarly, under *Link/Node data sets*, select *Sediment Flux*.
4. Set the *Gridded data set output format* to *ASCII* if it is not selected by default.



5. Click OK and OK again.

4.5 Save and Run the Model

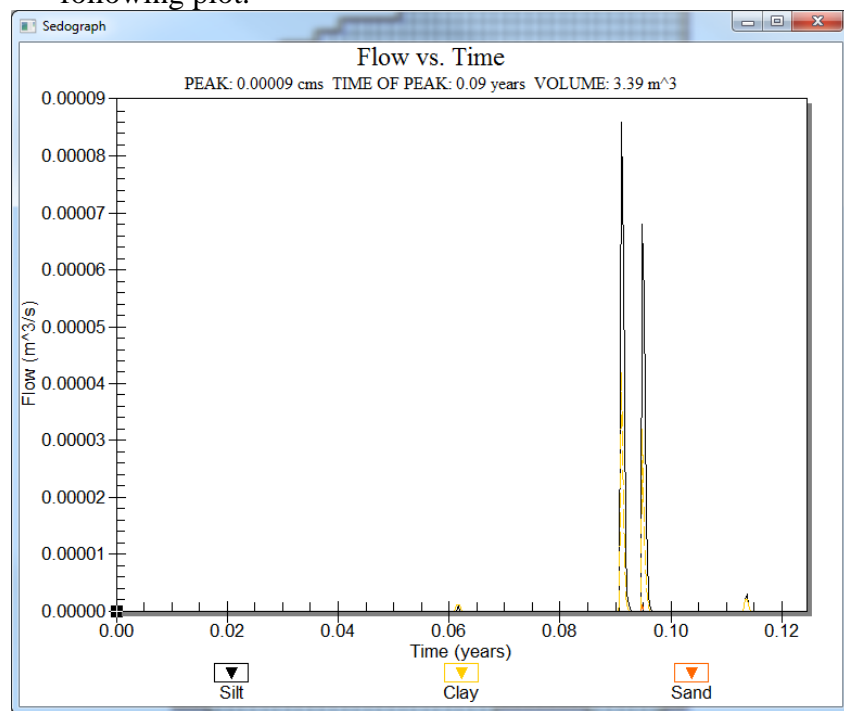
1. In the 2D Grid Module  select *GSSHA / Save Project File...*
2. Save it as *C:\Training\Personal\Sediment\sed.prj*

3. In the 2D Grid Module  select *GSSHA / Run GSSHA...*
4. Once the model is done running, click Close to read in the solution.
5. You may visualize different solution outputs.

4.6 Visualizing the results

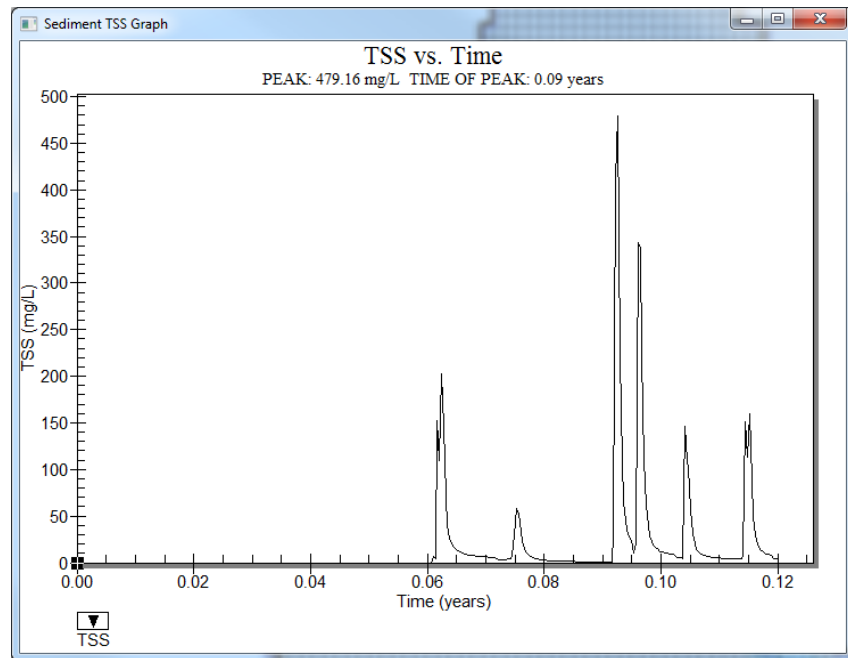
4.6.1 Visualizing Outlet Sediment Graph

1. In the project explorer, find the solution folder for *sed* GSSHA project and right click *Outlet Sediment Graph*. Select *View Graph*.
2. You will see the sedograph plotted something similar to the following plot.



4.6.2 Visualizing the TSS plot

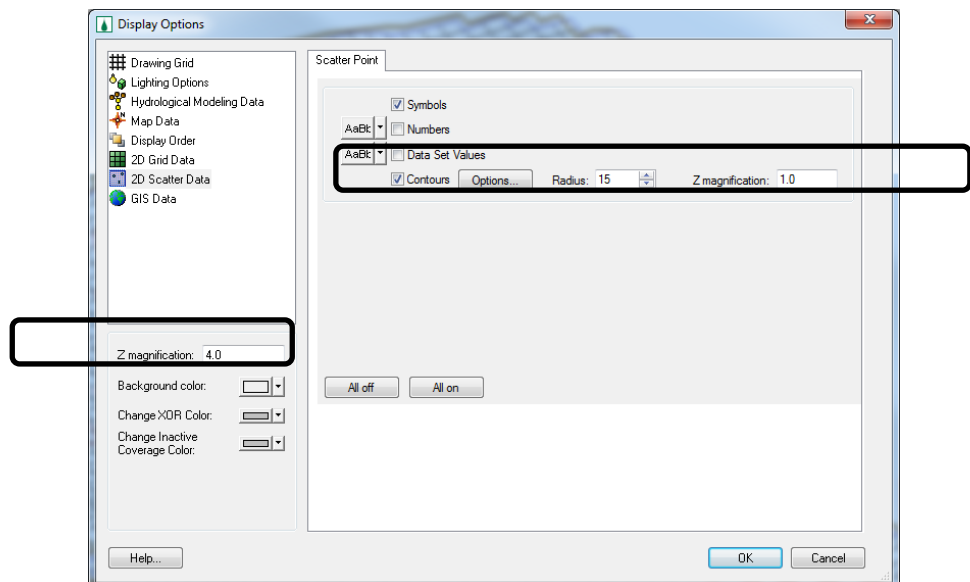
1. GSSHA generates the TSS plot as well. To see the TSS plot at the watershed outlet, right click *Outlet Sediment TSS Graph*. Select *View Graph*.
2. You will see the TSS plotted something similar to the following plot.



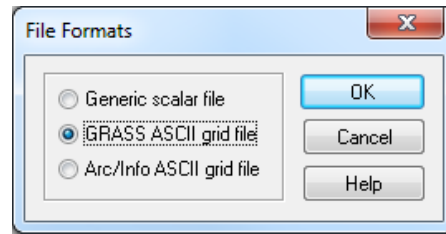
4.6.3 Visualizing Sediment flux in the channel

To visualize the sediment motion in the channel network follow these steps:

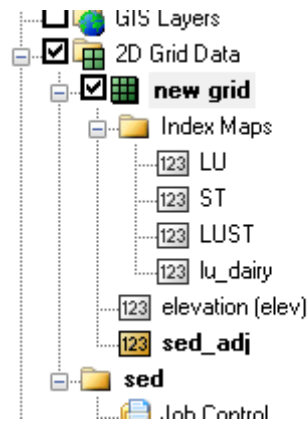
1. Select *Display/Display Options*.
2. Select *2D Scatter Data* and click on the check box for the Contours.
3. Click *Options* button select *Color Fill* for *Contour Method*. Click OK
4. Enter Radius of 15 and Z-magnification of 0.5.
5. Similarly on the left side of the dialog, there is another place to enter Z-magnification. Enter the value to be 4.
6. Click OK.



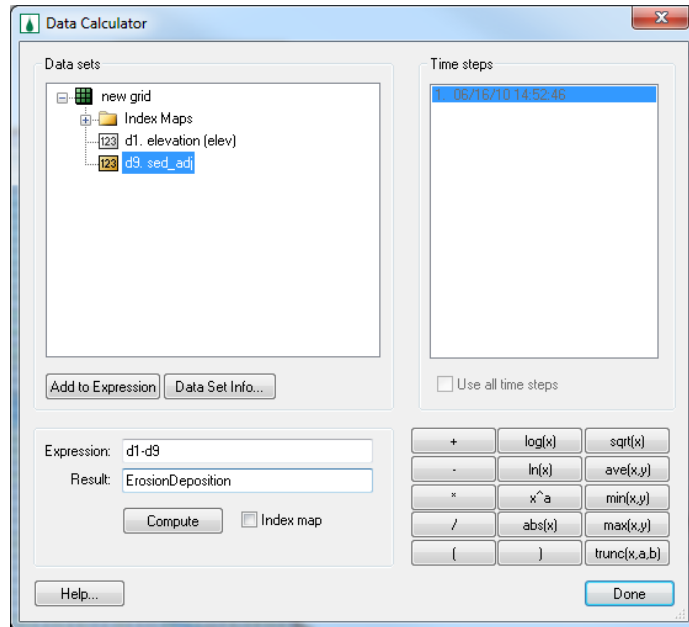
- In the pop up window, select *GRASS ASCII grid file* and click OK.



- Browse and open *C:\Training\Personal\Sediment\sed_adj.ele*. You might need to change the file type filter to *All files (*.*)*
- You can now see Adjusted Elevation data added in the data tree.



- Select *GSSHA/Maps*.
- Select the Data Calculator button.
- Double click *sed_adj* which will put a symbol in the *Expression* field. Type or click '-' (minus) and double click on *elevation (elev)*.
The formula is $(sed_adj - elevation)$
- Enter *ErosionDeposition* in the *Result* field.






9. Click *Compute* (do not select Index map toggle)
10. Click Done and Done.
11. You should have seen a new continuous map under *new grid*.
12. You may now change the contour options to see the areas of erosion and deposition.
13. Cells with negative value are the eroded cells and that with positive value are the ones where deposition occurred.

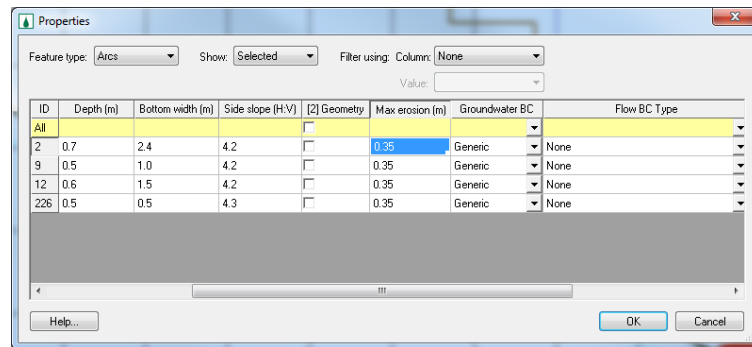
In this exercise we did not allow the channel to erode. In reality the channel beds also erode and the following section shows how you include channel erosion.


4.7 Updating Channel Erodibility Parameters

In the previous exercise, we did not allow the channels to erode. Now you will turn the channel erosion option on and rerun the model.

1. Click on *Plan View* macro  to zoom to the full extent of the watershed in a plan view.
2. In the *Map Module* , click on *Select feature line branch* tool  and double click on the stream arc just upstream of the watershed outlet.
3. In the *Channel Properties* dialog that opens, scroll to the right to find *Max Erosion (m)* column.
4. Enter a value of 0.35 for all the channels which is the maximum erosion depth for the channels.
5. Click OK.

4-10 GSSHA Modeling Using WMS – Volume 3




6. In the *2d Grid Module*  save the project as ***C:\Training\Sediment\SedStrErosion.prj***
7. Run the project.
8. Repeat the visualization you did in section 4.6 of this tutorial and see the differences in stream sediment flux.

Simulating Constituent Transport

Two types of reactive constituent transport are available in GSSHA and these constituents can be simulated as simple first order reactants. The nutrient cycle can also be simulated with the Nutrient Simulation Model (NSM). In either case, the overall simulation methods within the GSSHA model are the same.

We will create a constituent transport model without soil erosion and later add sediment transport to the model to see how the models behave with and without sediment transport associated with constituent transport.

5.1 Open an Existing GSSHA Project

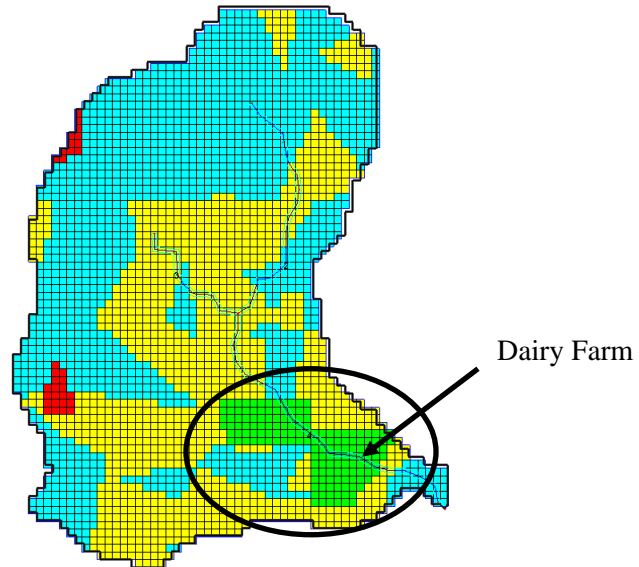
1. In the 2D Grid Module  select **GSSHA | Open Project File**.
2. Browse and open the project *C:\Training\Personal\Sediment\base\base.prj*.
3. Save the project with a different name as *C:\Training\Personal\Sediment\contam\contaminant.prj* so that the original project remains unchanged.
4. Turn off the display of all other coverages except GSSHA coverage.

5.2 Adding Contaminants

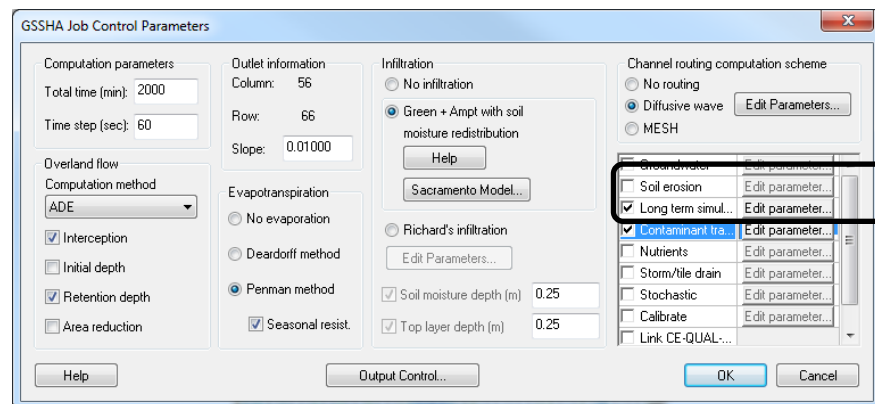
In this exercise we will assume that there is a dairy farm as shown in the following index map which acts as a contaminant source in the watershed.

5-2 GSSHA Modeling Using WMS – Volume 3

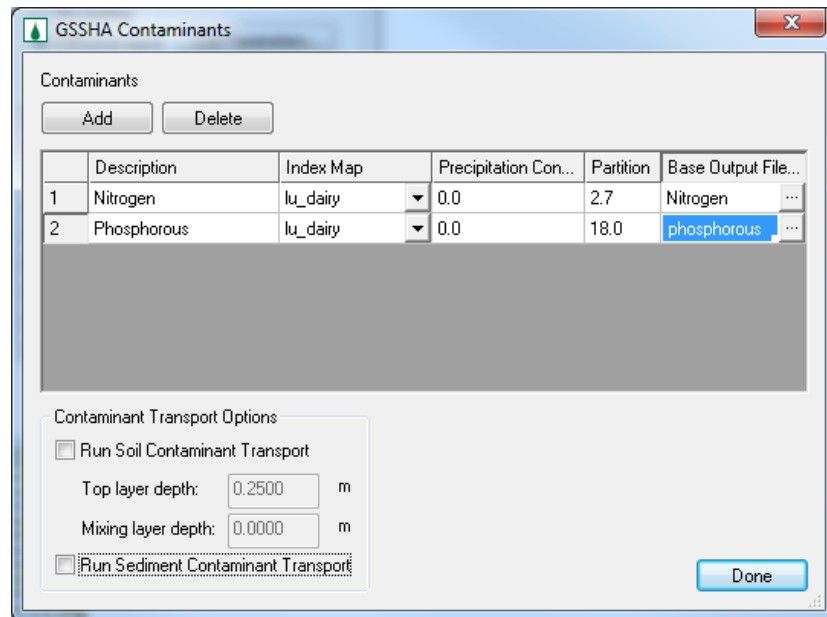
Constituent transport can be simulated on the overland flow plane, in the channel network including reservoirs, and in the soil column. These parameters are defined as shown below.



1. In *2D Grid Module*, select *GSSHA / Job Control*.
2. In the *Job Control dialog*, click on the check box next to *Contaminant Transport* to select it.



3. Click on the *Edit parameter* button for Contaminant Transport.
4. In the *GSSHA Contaminants* dialog that opens, click on the *Add* button twice to add two contaminants and enter the following information.



Note: The base output file column specifies the location of the output files that will be generated for each contaminant. These output files will be written in the same folder where your project files are located (in this case *C:\Training\Personal\Sediment*)

5. Check off both options for *Contaminant Transport Options* at the bottom of the dialog.
6. Click Done.

5.3 Defining Contaminants in the channels

1. In the Job Control dialog, click on Edit Parameters next to Diffusive channel Routing.
2. In *GSSHA Channel Routing Parameters* dialog that shows up, make sure that *Compute Contaminant Transport* is checked on and enter the following values:
 - Decay coefficient: 0.0
 - Dispersion coefficient: 0.0
 - Initial Concentration: 0.0
3. Click OK. This will define the conditions in the channel network.
4. Click OK to close the *Job control* dialog box.

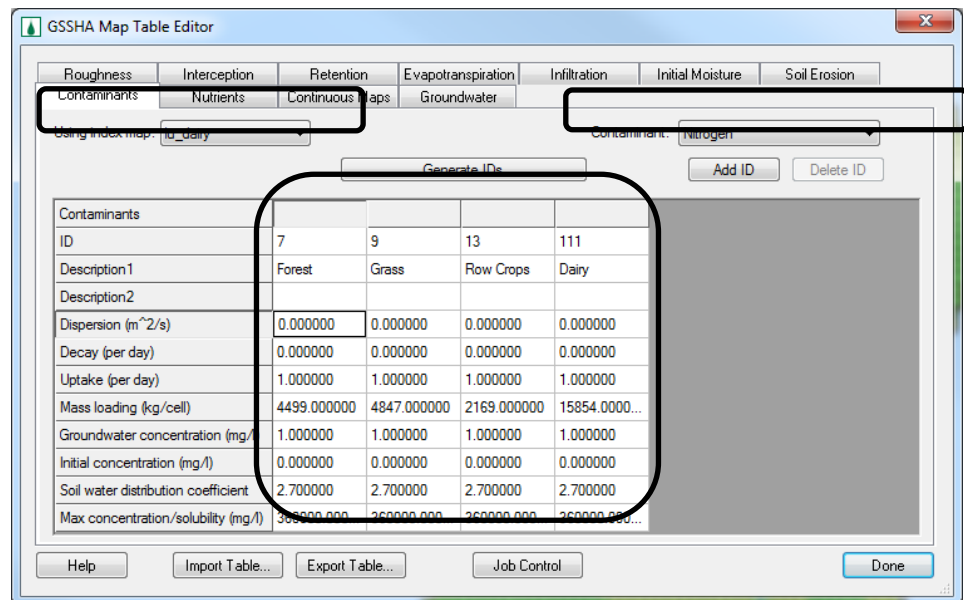
5.4 Defining Contaminants parameters

1. Select *GSSHA / Map Tables*. Values for Roughness, Evapo-transpiration, Infiltration and Initial Moisture are already defined. Here we will define the parameters for the contaminants. If you can recall, we had two contaminants Nitrogen and Phosphorous defined in previous step. We will have to define the parameters for both.

5-4 GSSHA Modeling Using WMS – Volume 3

2. Select the *Contaminants Tab*. In *Using Index Map* drop down box, select “lu_dairy” and click *Generate IDs* button. This will generate 4 IDs.
3. The last column with ID 111 is the Dairy, you may rename the description field if you wish.
4. Enter the values in the spreadsheet for Nitrogen as shown in the following table.

ID	7	9	13	111
Description 1	forest	grass	row crop	dairy
Description 2				
Dispersion (m ² \s)	0.00	0.00	0.00	0.00
Decay (per day)	0.00	0.00	0.00	0.00
Uptake (per day)	1.00	1.00	1.00	1.00
Mass loading (kg\cell)	4499.00	4847.00	2169.00	15854.00
Groundwater concentration (mg\l)	1.00	1.00	1.00	1.00
Initial Concentration (mg\l)	0.00	0.00	0.00	0.00
Soil water distribution coefficient	2.70	2.70	2.70	2.70
Max concentration\solubility (mg\l)	360000.00	360000.00	360000.00	360000.00



5. From the *Contaminant Drop down Menu* select *Phosphorous* and select *lu_dairy* as the index map. Then click *Generate IDs*. Use the following table to fill the spreadsheet for Phosphorous.

ID	7	9	13	111
Description 1	forest	grass	row crop	dairy
Description 2				
Dispersion (m ² \s)	0.00	0.00	0.00	0.00


Simulating Constituent Transport 5-5

Decay (per day)	0.00	0.00	0.00	0.00
Uptake (per day)	1.00	1.00	1.00	1.00
Mass loading (kg\cell)	710.00	901.00	629.00	3115.00
Groundwater concentration (mg\l)	1.00	1.00	1.00	1.00
Initial Concentration (mg\l)	0.00	0.00	0.00	0.00
Soil water distribution coefficient	18.00	18.00	18.00	18.00
Max concentration\solubility (mg\l)	220000.00	220000.00	220000.00	220000.00

Note: More information about these parameters for the contaminants can be found on the GSSHA wiki

6. Click *Done* to close the *Mapping Table* dialog.

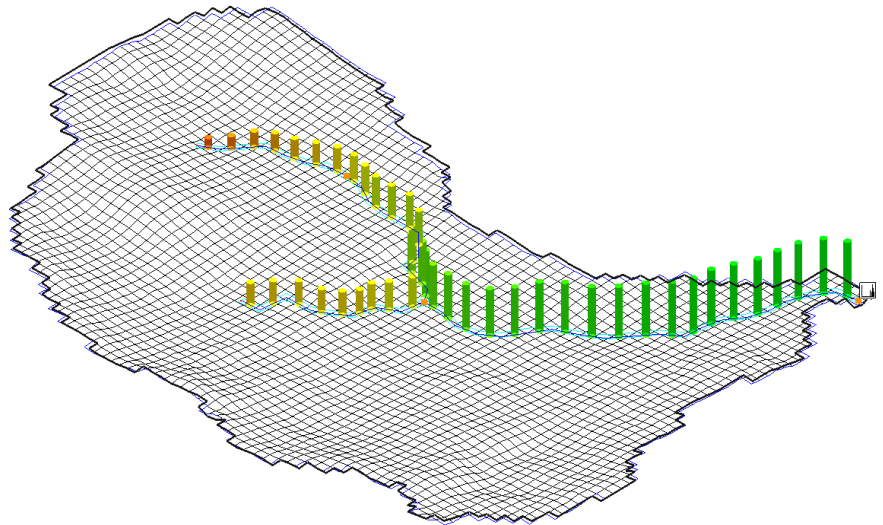
5.5 Save and Run

1. In the 2D Grid Module  select *GSSHA / Save Project File*.
2. Save the file as *C:\Training\Personal\Sediment\contam\contaminant.prj* and click yes to replace the existing file.
3. Select *GSSHA / Run GSSHA*.

5.6 View Results

GSSHA generates contaminant maps for both the channel and the overland flow for all the constituents being modeled.


1. In the Project Explorer, right-click on the *Outlet Constituent Mass graph*, located below the GSSHA solution folder (the folder with the letter "S" on it) and select *View Graph*. In the window that opens, you have the option to select the contaminant you want to view.
2. Close the contaminant plot.
3. You can follow the same steps for the *Constituent Concentration graph*.
4. Visualize the Contaminant concentration and mass for both Nitrogen and Phosphorous by changing grid contour options.
5. Finally, from the display options, set the z-magnification and contour options for 2D scatter data and visualize the contaminant concentration and mass along the channel.



5.7 Adding Sediment Transport

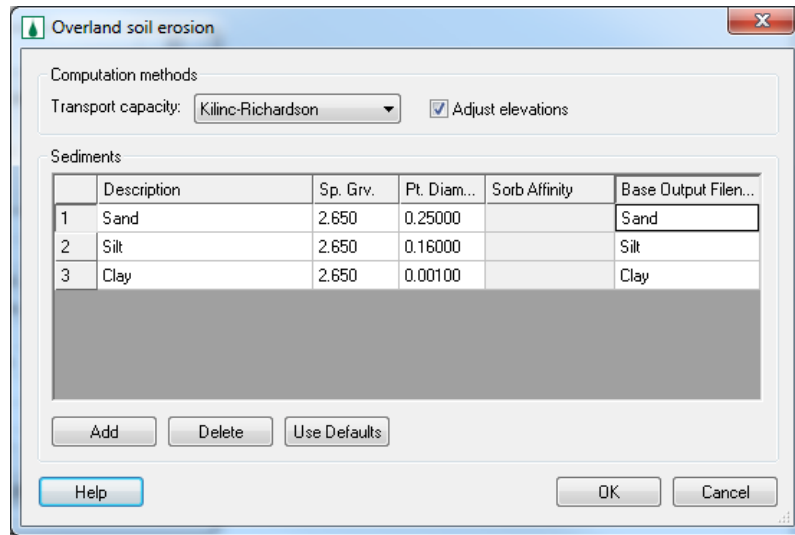
Now, we will turn on the soil erosion and sediment transport in the constituent transport model that we developed in the previous step and see how it affects the constituent transport.

To save some time, a GSSHA project is already created that has sediments turned on and the constituents defined. You will modify this project and run it.

1. Open a new instance of WMS by clicking *New* button or select **File/New**.
2. Open GSSHA project *C:\Training\Sediment\contam\ContaminantWSed.prj*
3. Turn off the display of all other coverages except *GSSHA* coverage.
4. In the 2D Grid Module  select **GSSHA / Save Project File**.
5. Save the file as *C:\Training\Personal\Sediment\contam\ContamWSed.prj*

5.7.1 Sediment Parameters

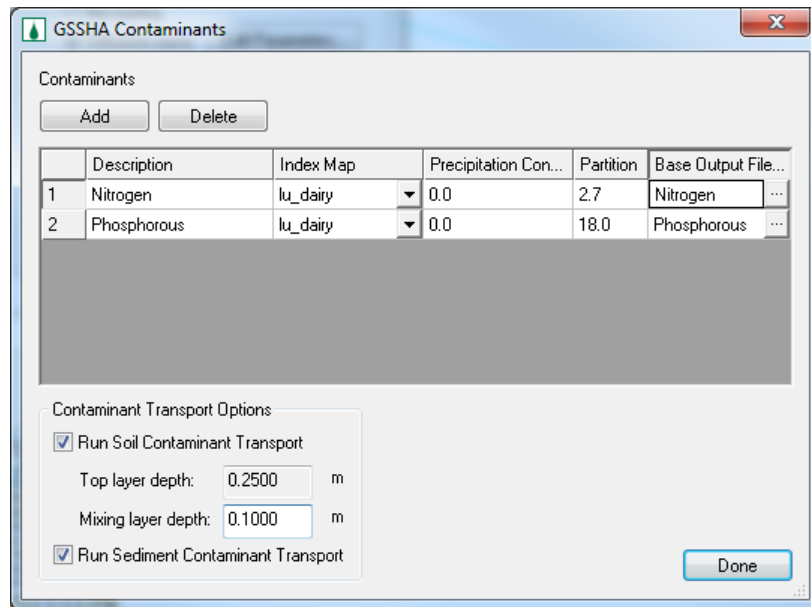
1. In the *GSSHA Job Control*, make sure that *Soil erosion* is turned on and click on *Edit Parameter*
2. Check and see if all the parameters are defined according to the following figure



3. Click OK.


5.7.2 Contaminant Parameters

1. Still in the *GSSHA Job Control* dialog, click on the *Edit Parameters* button next to the *Contaminant Transport* option.
2. Make sure that the parameters are defined as shown in the following figure.




3. Enter 0.1m for *Mixing Layer Depth*
4. Click *Done* and *OK*.

5.7.3 Map Table Parameters

1. In the 2D Grid Module  select **GSSHA/Map Tables**
2. Switch to the *Contaminants Tab* and make sure that the parameters for both *Nitrogen* and *Phosphorous* are defined.
3. Similarly, switch to the *Soil Erosion* tab, make sure the parameters are defined properly.
4. Click *Done*.

5.8 Save and Run

1. In the 2D Grid Module  select **GSSHA / Save Project File**.
2. Save file as **C:\Training\Personal\Sediment\contam\contamWSed.prj** and click yes to replace the existing file.
3. Select **GSSHA / Run GSSHA**.

5.9 View Results

This will take little longer than the model with *Constituents only*. Once done running, GSSHA generates contaminant concentration and mass maps for both the channel and the overland flow for all the constituents being modeled.

1. Toggle through the time steps for both mass and concentration of the contaminants.
2. See the outlet sediment graph

Overland Flow Boundary Conditions in GSSHA

So far the only source of runoff in our GSSHA models has been precipitation. However, there is the possibility of defining sources separate from rainfall with other boundary conditions. Two such types of boundary conditions are hydrographs defined on stream nodes and head boundary conditions defined on overland flow cells.

In this workshop you will learn how to define a *hydrograph boundary* and a *variable stage (water surface elevation) boundary* condition using two separate models. You will begin with an existing project file, modify the boundary conditions and evaluate the differences.

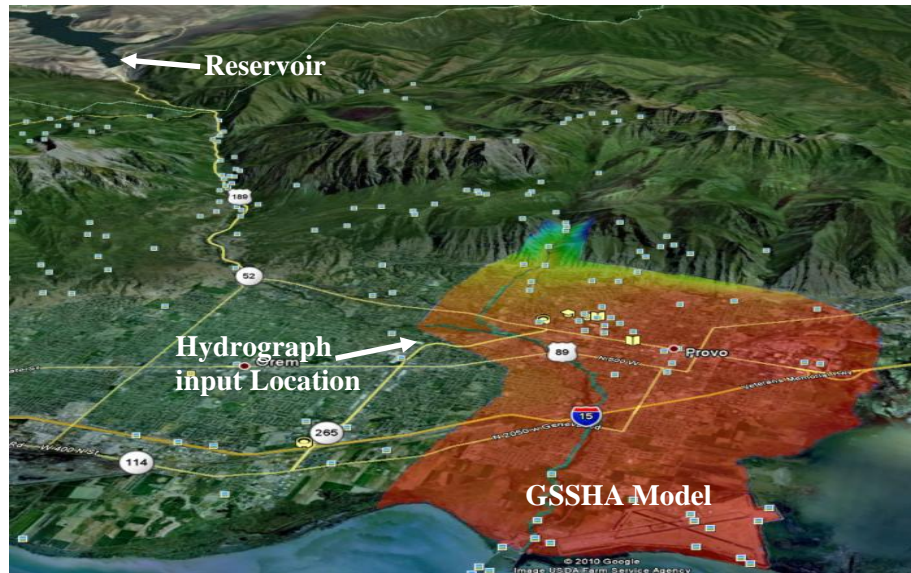
6.1 Hydrograph Boundary Condition

This boundary condition is used if a hydrograph is input from a source upstream of the model, for example using the results from HEC-HMS or some other “regional” model to drive a local model where GSSHA is employed. The hydrograph is defined in GSSHA at a node on a stream arc.

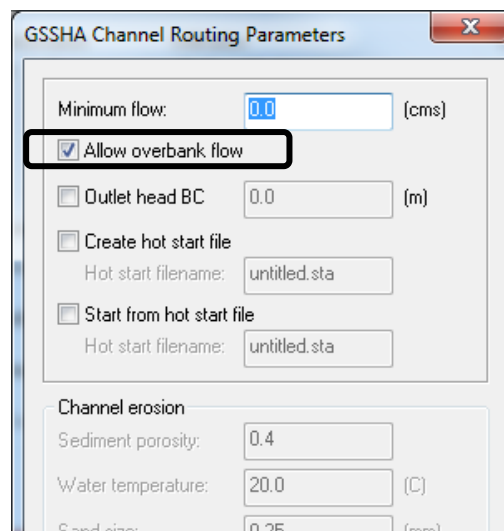
6.1.1 Open Existing Model

In the model you are going to simulate in this workshop, we are assuming a hypothetical scenario where the Deer Creek reservoir upstream of Provo breaches generating a hydrograph input into the GSSHA model, see the following figure. Using the channel overbank option which allows flow from the stream to “spill” onto the overland grid, we will see how the input hydrograph floods the Provo area.


6-2 GSSHA Modeling Using WMS – Volume 3

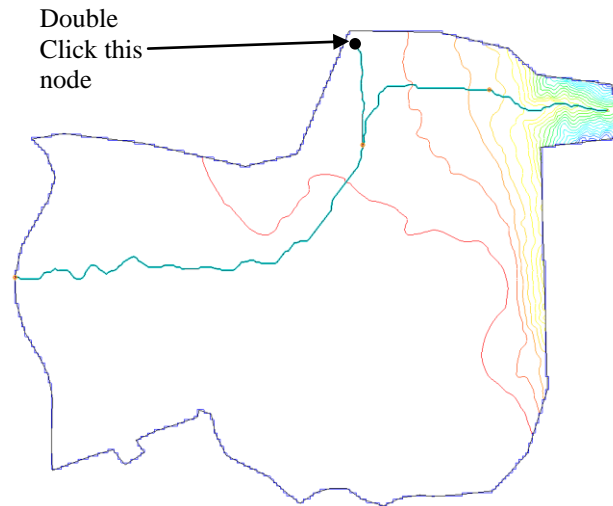


1. Open a WMS project *C:\Training\BoundaryCondition\ProvoBase.prj*. This model has variable overland flow roughness and variable infiltration processes already defined. A negligible amount of precipitation is defined because GSSHA won't run when not performing a long term simulation if no precipitation is defined.
2. Save the project as *C:\Training\Personal\Boundary\Hydrograph.prj* so that the original project remains unchanged.
3. Turn off the display of all other coverages except the GSSHA coverage.
4. In the GSSHA Job Control, select *Diffusive wave* for the *channel routing method* and click the *Edit Parameters* button.
5. Toggle on *Allow overbank flow* option and click OK.

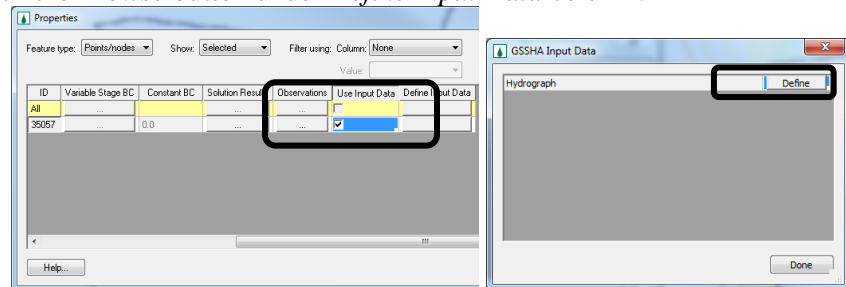


6.1.2 Defining an Input Hydrograph

1. In the *Map Module*, click the *Select Feature Point\Node* tool  and double click on the stream node furthest upstream on the left branch. See the following figure

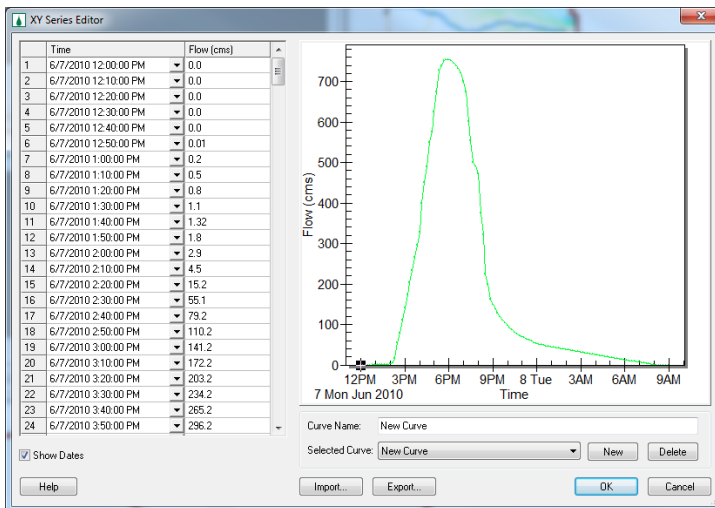


2. This will open the *Properties* dialog. In this dialog, scroll over to the right and select the *Use Input Data* toggle.
3. Click the *Browse* button under *Define Input Data* column.



4. Click the *Define* button. This brings up an XY series editor where the input hydrograph can be defined.
5. Outside of WMS open the file *C:\Training\BoundaryCondition\hydroinput.txt* and copy all the data from it.
6. Return to the WMS, XY Series Editor window and paste the hydrograph ordinates. This is a hydrograph that is being input at the node in the channel. This hydrograph is a hypothetical scenario of the dam failure, but it can come from any source, such as an upstream regional model or an output from dam break simulation, etc.
7. When you paste the data in the XY series editor, the dates are not shown and you can see a time increment of 10 mins. Toggle on *Show dates* at the bottom of XY Series Editor.
8. In the *Reference Time* window, enter 6/7/2010 for data and 12:00:00 PM for time. This is the time when precipitation begins (as does the GSSHA model).

- Click *Select*. You can now see the time column replaced with date/time, see the following figure.



- Click OK, Done and OK to close all the dialogs.

6.1.3 Save and Run the model

- Save the project as *C:\Training\Personal\Boundary\Hydrograph.prj*
- Click Yes if prompted to overwrite existing project
- Select *GSSHA/Run GSSHA*

6.1.4 Results Visualization

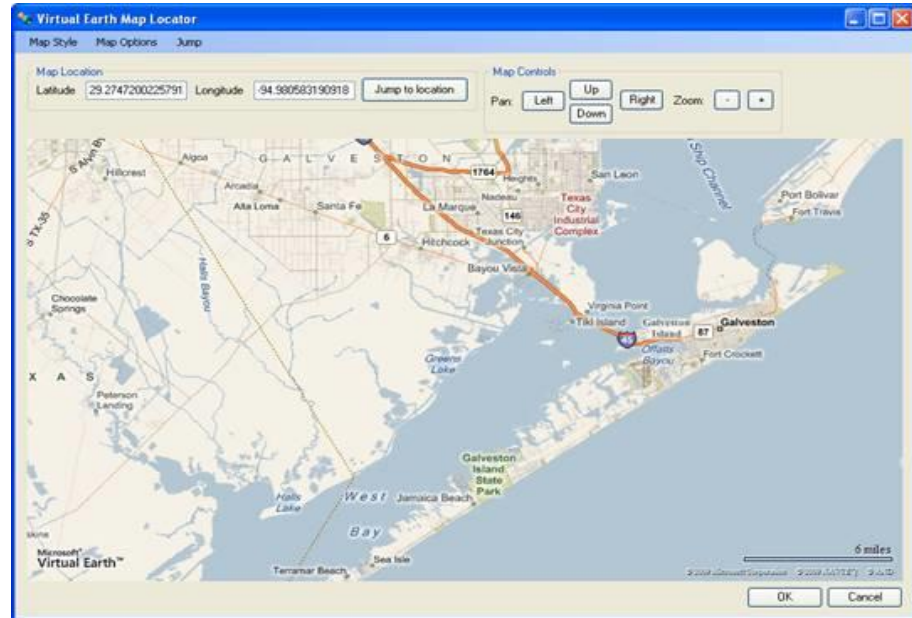
- Select the overland depth datasets in the project explorer and toggle through the time steps in the properties window.
- Turn the stream flow bar plots on from the display options and see how the flood wave propagates downstream.
- You can see as the flood wave proceeds downstream along the stream, it overflows the bank and inundates the flood plain.
- You may wish to export the flood animation into Google Earth.

6.2 Variable Stage Boundary Condition

In this workshop you will create a variable stage overland flow boundary condition in your GSSHA model. For the hydrograph boundary condition, you entered an inflow hydrograph at a stream node and let the GSSHA flood the downstream grid cells. In this case you will put water directly onto the grid cell and let it flow overland on the grid. The condition we are simulating here is a coastal tidal surge.

6.2.1 Open Existing Model

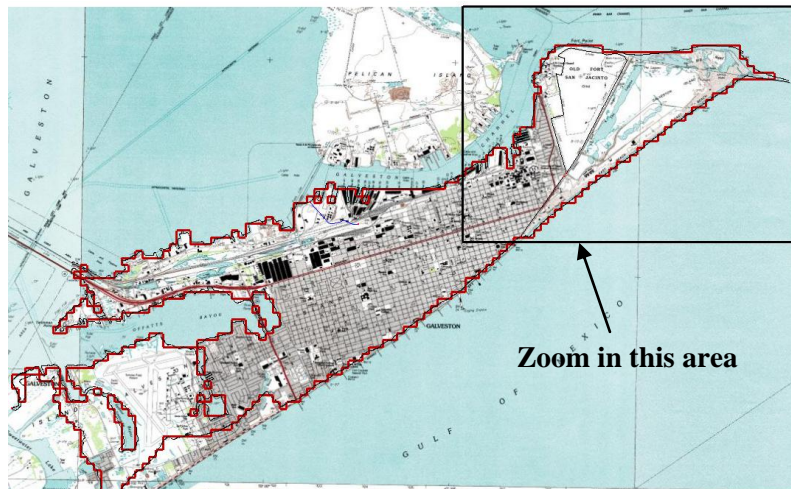
We will simulate the effect of the storm surge from hurricane Ike on Galveston Island, Texas. The tidal surge data were obtained from the *NOAA tides and currents* website. You will begin with a developed GSSHA model in which you will update the boundary conditions and run.



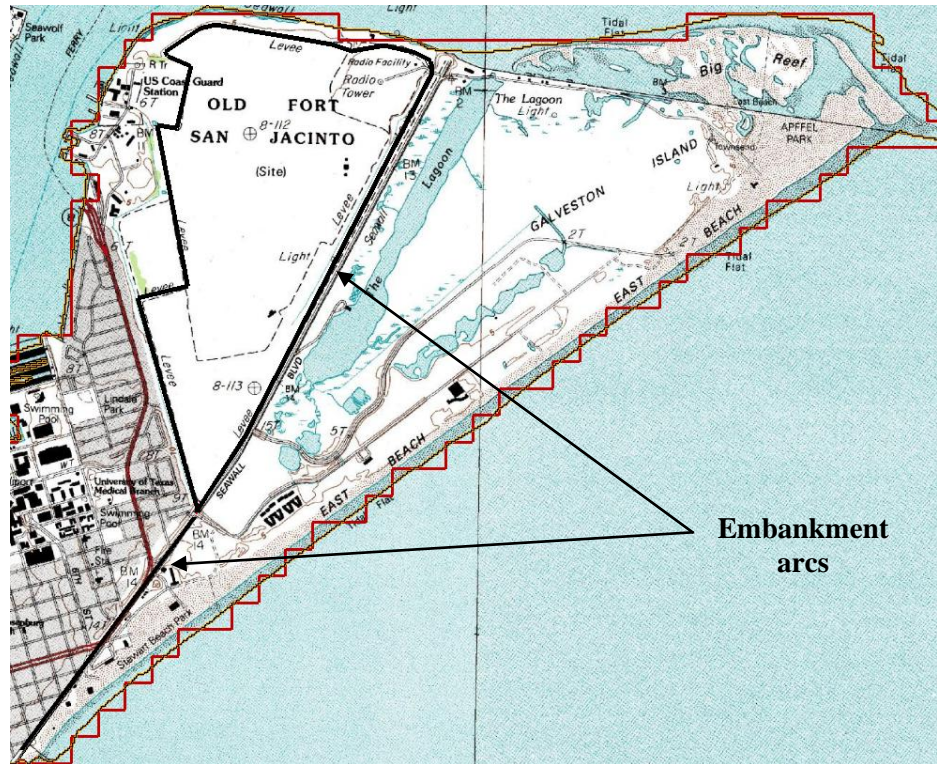
1. Open the GSSHA project ***C:\Training\BoundaryCondition\GalvestonBase.prj***. This model has variable overland flow roughness defined where as the infiltration is turned off. A precipitation of 10.5 mm/hr is defined for one day. We are going to simulate a large overland flow and in such cases the amount of water that infiltrates is negligible by comparison.
2. The model currently does not have a flow boundary condition.
3. Save the project as ***C:\Training\Personal\Boundary\Overland.prj*** so that the original project remains unchanged.
4. Turn off the display of all the coverages except the GSSHA coverage.

6.2.2 Creating an Embankment

1. Open the topographic map for Galveston located at ***C:\Training\BoundaryCondition\Galvestontopo.jpg***.
2. Zoom into the area as shown in the following figure:

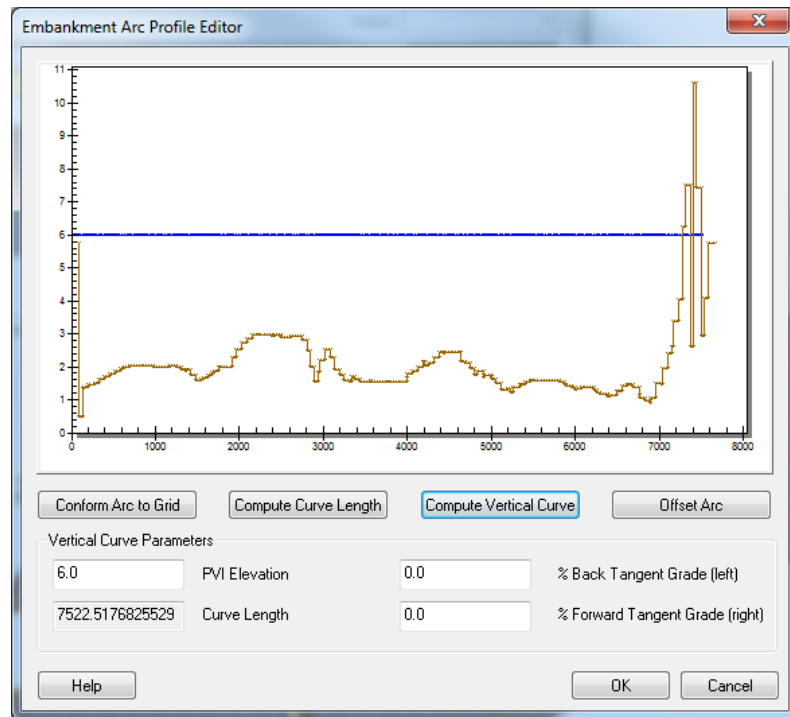


3. Switch to the *Map Module*. With the background image and the following picture as a reference, create an embankment arc for the levee surrounding Old Fort San Jacinto located on the north side of Galveston Island.



4. You should have seen in the background map that there is a dotted line that represents a Levee. Just trace the embankment arc along that line to form a polygon around the protected area.
5. Make sure that you do not intersect the embankment arc with the watershed boundary.
6. The second embankment line that runs to lower left corner (representing a sea wall) should not intersect the embankment arc on the top.

7. For each of the embankment arcs double click to change the type to *Embankment* and click on the *Browse* button under *Embankment*.
8. In the *Embankment Arc Profile Editor* dialog, enter 6 for *PVI elevation* and click *Compute Vertical Curve*. This will set embankment height to 6 meters. Do this for both arcs.



6.2.3 Creating Boundary Arcs

The boundary arcs are used to define the variable surge boundary condition that flows directly into the cells overlapping the arc. In this model, we will use the watershed boundary arc, so the arcs are already there but currently the arc representing the boundary is a single arc. We need to break the arc so that the variable head can be defined only on those locations where the storm surge occurred. You break an arc by inserting a node at the desired location.

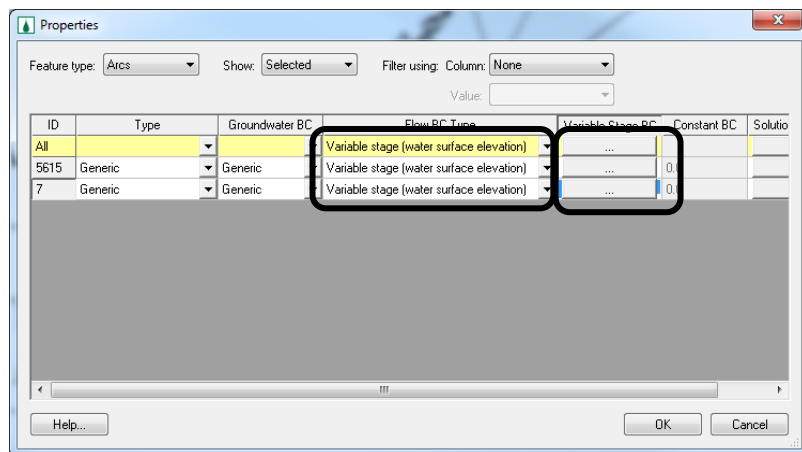


1. Create a node at the start point and end point of the Gulf Boundary arc as shown in above figure.
2. Similarly, insert a node in the start point and end point of the Bay Boundary arc as shown in the above figure. Include a node in the middle where the watershed outlet is located.

NOTE: You do not need to draw an arc, they are already there. The arcs are shown in the above figure to show which arcs are used to define the boundary condition.

6.2.4 Entering Variable stage data

1. Select the arc(s) on the *Gulf Side* and set the arc attributes to *variable stage (water surface elevation)* under the Flow BC type field.



2. Click on the *Browse* button under the *Variable Stage BC* column and enter the Galveston Pleasure Pier storm surge time series into the XY series

editor. The time series can be found in *C:\Training\Boundary\PleasurePier.txt*.

3. Similarly, select the arc(s) on the *Bay Side* and set your arc attributes to *variable stage (water surface elevation)* and enter the Galveston Pier 21 storm surge time series into the XY series data editor. Get the time series data from *C:\Training\Boundary\Pier21.txt*.

6.2.5 Save and Run the model

1. Save the project as *C:\Training\Personal\Boundary\Overland.prj*
2. Click Yes if prompted to overwrite existing project
3. Select *GSSHA/Run GSSHA*

6.2.6 Result Visualization

1. Select the overland depth datasets in the data tree and toggle through the time steps in the properties window.
2. You may export the flooding animation to Google Earth.

