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

GSSHA Using the Watershed Modeling System

Tutorials – Volume 1

WMS BASICS

WMS 8.4 Compatible – Volume 1

The software *WMS* is a product of the Aquaveo, LLC. For more information about this software and related products, contact Aquaveo at:

Aquaveo
3210 N. Canyon Road
Suite 300
Provo, Utah 84604
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

CHAPTER 1

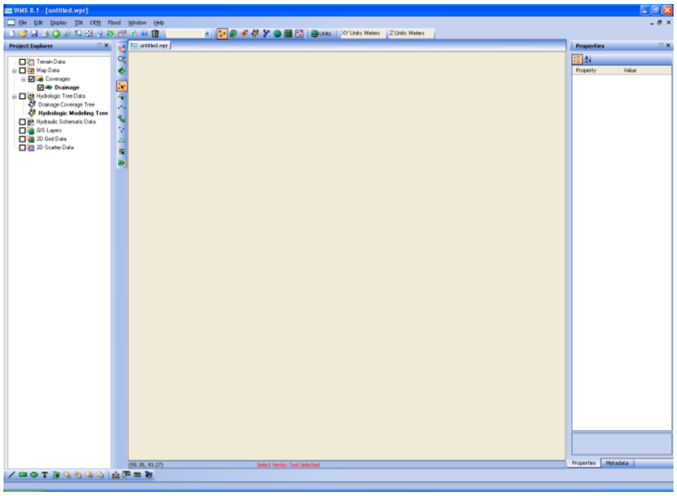
Loading DEMs, Contour Options, Images, and Coordinate Systems

Loading DEMs, Contour Options, Images, and Coordinate Systems

In this first exercise you will get familiar with the WMS interface and the help system using data from the Park City, Utah area.

1.1 Introduction

We will learn the basic layout of WMS and get acquainted with the interface.



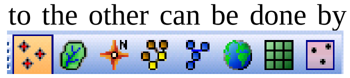
The left most portion of the WMS window is called Project Explorer, the central part is the display window, and the right part is the Properties window. The properties window changes based on the selected item.

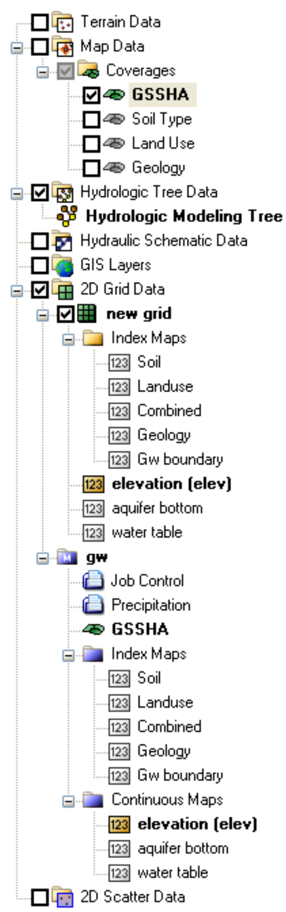
1.2 Getting Around the WMS Interface

The WMS Help file has a section on some of the basic elements of the WMS graphical user interface (GUI).

1.2.1 Self-Guided Tour

The WMS Help file is located on a "wiki" site at http://www.xmswiki.com and documents the important elements of the GUI. In this section you will practice on your own to become familiar with the interface, but you can always refer back to the help file on xmswiki.com as needed.

1. Start WMS
2. Select ***Help | WMS Help***
3. Take a couple of minutes to review the WMS Help wiki and then close the help page
4. Different operations like spatial data reading/editing, hydrologic modeling or grid generation etc. are done using different modules. Switching from one module to the other can be done by clicking the corresponding module icons  in the menu bar. As you do, you will notice menus and tools change according to the available options. Change the module several times and note how the menus and tools change according to the active module. When you are done, be sure the Terrain Data module is active (the first one from the left).
5. If you look at the WMS project explorer, you can see that there is a folder in the project explorer to contain the data associated with each module (the drainage module shares data from the terrain and map modules).
6. The default behavior of the project explorer window is to switch modules as these folders or data contained in them are selected. For example, if you click on the *GIS Layers* folder, the *GIS Module* will be selected and if you click on the *Coverages* folder, the map module will be selected.
7. WMS 8.4 projects are saved as *.wms files. When you save the project, all necessary files are saved and the saved project can be reopened. You should always save your model as a .wms file before creating a 2D grid for a GSSHA model so you have a base project to work with. After building a 2D



grid and initializing your GSSHA model, you should save your project as a GSSHA project file (with a .prj extension).

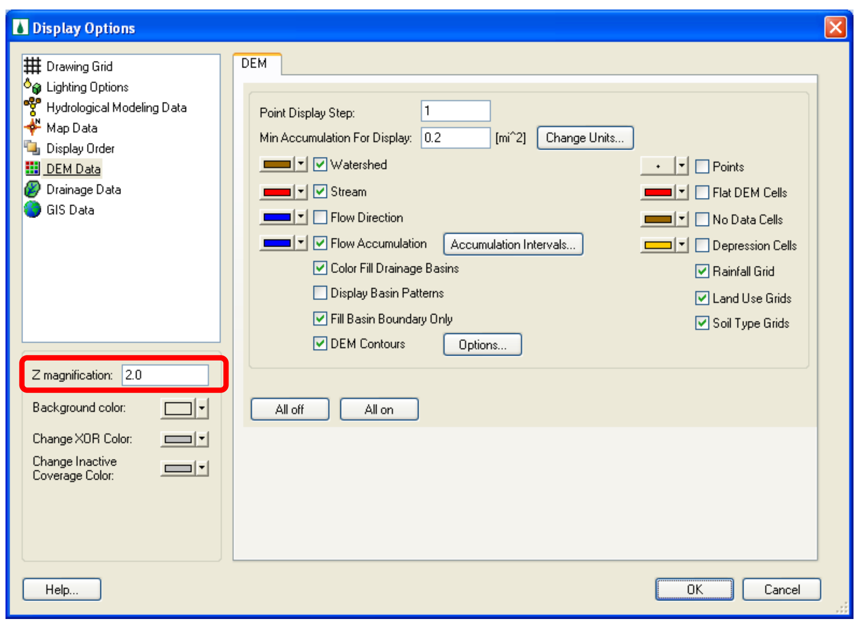
8. You can control which objects are displayed by checking and un-checking the data elements in the Project Explorer window. Set the style, color, and size of elements drawn using the ***Display Options***, ***Contour Options***, and other macros.

9. Select ***File | Open*** Browse and open file ***C:/Training/Images /BaseProj.wms***

10. Select the ***Contour Options*** macro and experiment with changing some of the contouring options such as the ***Contour Methods***, ***Contour Interval*** and ***Color Ramp.***

11. Controlling what and how objects are visually displayed can be set through the display options. To set the display options, select ***Display | Display options*** or simply click the display options button on the menu bar.

12. In the *Display Options* dialog box enter 2.0 for *Z magnification*. You can also access the DEM contour options from the display options by selecting the DEM Data object in the upper left corner of the Display Options window. Click the options button next to DEM Contours. Make sure the DEM Contour Method is set to *Color Fill* and click OK twice.



13. Select the *Rotate Tool* and drag your watershed so you can visualize the watershed relief.

14. Select the ***Contour Options*** macro and set the *Contour Method* to *Normal Linear*. Select OK and then select *Plan View* to reset the drawing of the DEM from the changes you made with the rotate tool.

1.3 Images

Images are an important part of projects developed using WMS. An image is comprised of a number of pixels (picture elements), each with its own color. The resolution, or size, of the pixels determines the area and detail represented in the image. Images may be used in WMS to derive locations for features such as roads, streams, confluences, land use, soils and so forth. Images also provide a base map for your project.

In order to make use of images they must be georeferenced. Georeferencing an image defines appropriate x and y coordinates so that it can be spatially overlaid with other data. Because images are commonly used in Geographic Information System (GIS) programs like WMS, data developers often store the georeferencing information as either part of the image file (a geotiff file for example), or in a separate file commonly referred to as a "world" file.

In this section we will experiment with JPG and TIFF image files. We will also see how an image without spatial information can be georeferenced.

1.4 Geotiff Files

Geotiff images are files that embed the georeferencing information. This means that you do not have to specify coordinates when you read in the image.

1.4.1 Scales

In this section we will see how the scale of an image affects the display. We will use some of the images for Provo, Utah.

1. Select ***File|New*** or click on macro icon to open a new instance of WMS.
2. Select ***File | Open***
3. Browse the folder ***C:\Training\Images*** and open the following images:

    a) provo24K.tif

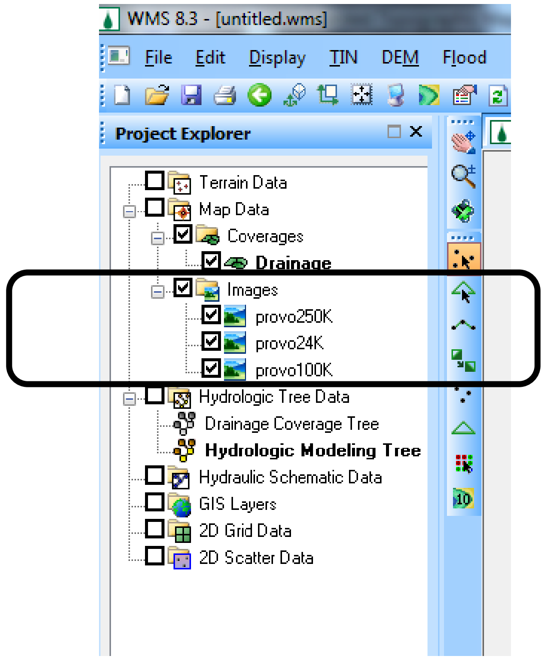
    b) provo100K.tif and

    c) provo250K.tif

Click No if prompted to convert to jpg

Note: These are the topographic maps of the Provo, UT area. The three images are of different scales (1:24000, 1:100000 and 1:250000 respectively).

4. Once these files are open, you can see them listed in the project explorer (see the following figure).



5. Turn the display of the images on and off by checking them on and off in the project explorer. Leave them all on when you are done experimenting.

6. Select the Zoom Tool  and draw a rectangle over the image to zoom into the area of your preference.

Zooming Tips: With the zoom in tool  selected, if you press and hold the Shift key in the keyboard, the zoom in tool changes to zoom out tool

You can also use the scroll wheel of your mouse to zoom in and out with any tool or module selected.

7. Zoom into the same location in the three different images and see the difference in resolution, area covered by the images, details captured etc.

1.5 Overlaying Images

Here we will see how different types of images can be overlaid together.

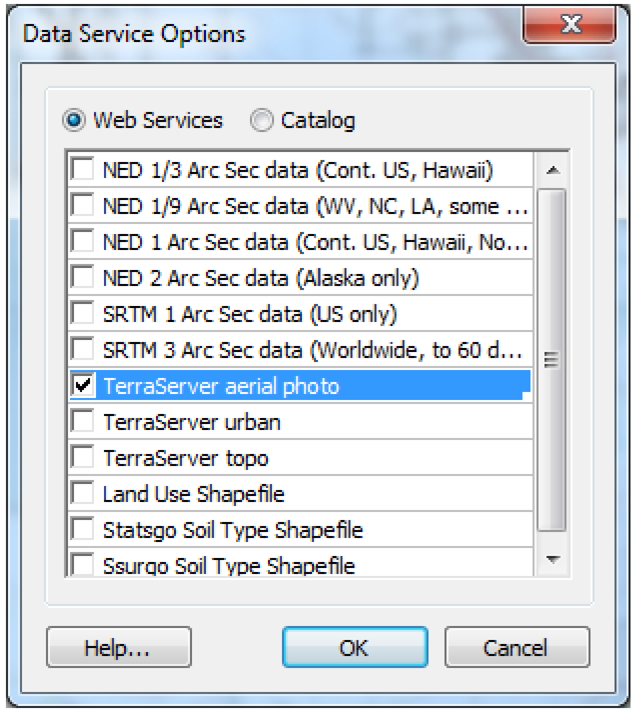
1. Close WMS and open a new instance of WMS. Browse and open image ***C:\Training\ Images\provoTStopo.jpg.*** This is a topographical

map of Provo area obtained from Terraserver ***(http://www.terraserver.com/)***. This image is already georeferenced.

Many of the tools in WMS depend on your currently defined coordinate system. One such feature that uses your current coordinate system (also called a *projection*) is the *get data* tool. Before using the *get data* tool, you must have your current projection defined so WMS can determine the latitude/longitude coordinates of your required data.

2. Select ***Edit | Current Coordinates*** to set your current coordinates
3. Select the *Global Projection* option
4. Select *Set Projection*
5. Set *Projection* to *UTM*, *Datum* to *NAD 83*, *Planar Units* to *Meters*, and *Zone* to *12 (114°W - 108°W – Northern Hemisphere)*
6. Select *OK*
7. Set the vertical projection to *Local* and the vertical units to *Meters*
8. Select *OK*
9. Select the *Get data tool*  located at the lower left corner of the WMS main window.

10. Draw a rectangle with the mouse pointer around the jpg image that you opened previously. This will open the *Data Service options* dialog. Select the *Terraserver aerial photo* option as shown in the following figure.



11. Select OK and browse folder ***C:\Training\Personal\Images*** and enter name ***provoTSaerial*** and click Save.
12. Select *Yes* to create the files.
13. WMS will suggest a suitable resolution for the image based upon the size and coverage of the image. You can change the resolution, but here you should use the suggested resolution. Click OK.
14. This will take a moment for WMS to use the web services and download the data from the server.
15. Once done, you will be prompted to build pyramids, select Yes.
16. You can see both the topo map and the aerial photo listed on the project explorer.
17. Right click on the ***provoTSaerial.tsaerial.web*** item in the project explorer and select *Set Transparency*.
18. Move the slider to 40% and click *OK*. You should be able to see the aerial photo on top of the topo map.

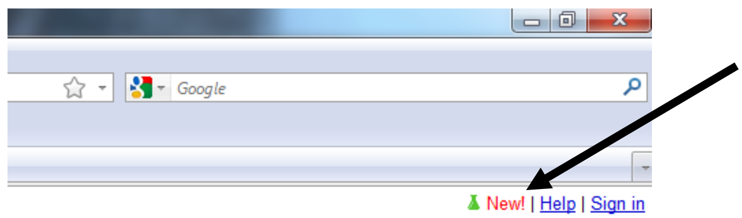
1.6 Registering Images

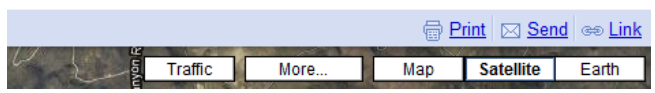
Sometimes you will not be able to obtain a geotiff image or an image with a world file. In this case, you can read a background image from common image sources such as scanned paper maps, Google Maps, or Google Earth and geo-reference the image manually. To do this, you need to know the X and Y coordinates of three points on the image. These coordinates can be in a projected or geographic (latitude-longitude) system. Before you scan your paper image, or download an image from the Internet, you should mark the three points you have selected so you can easily find the points on the image when you register the image in WMS.

1.6.1 Downloading image from Google Maps

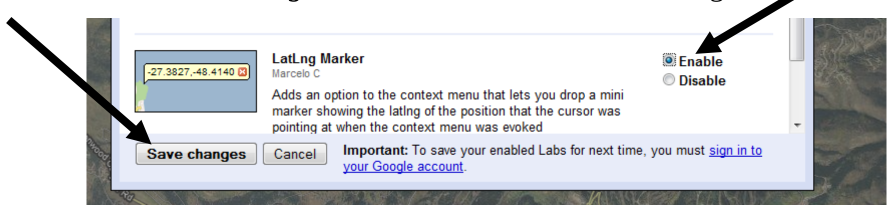
As an example we will see how an image can be imported from Google maps and get it georeferenced so that the image can be used in WMS.

1. Open a web browser and go to http://maps.google.com/ . Turn the Satellite map option on.
2. On the top right corner of the map click *New*.



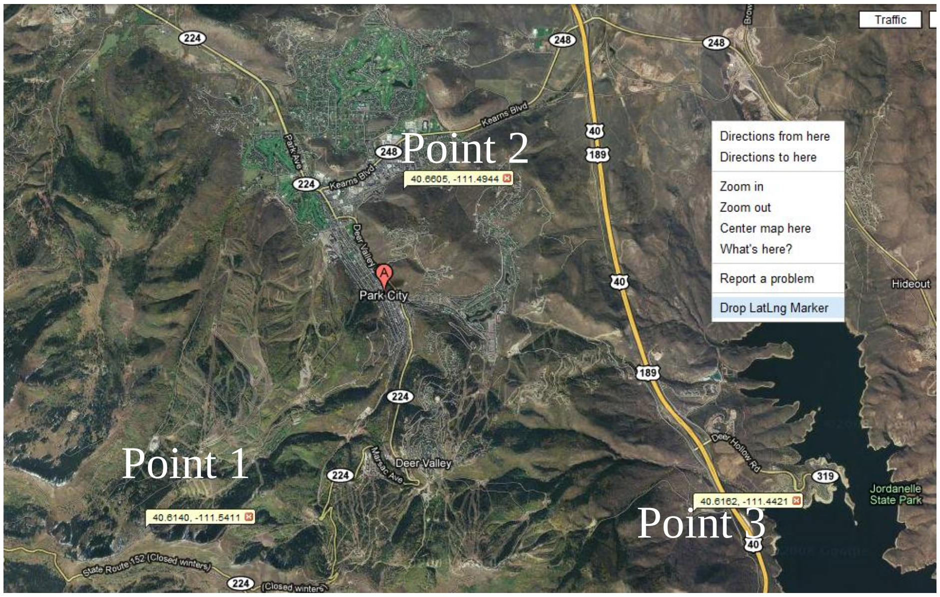


3. Enable the *LatLng Marker tool* and select Save Changes.



4. Enter Park City, UT in the search box and click *Search Maps*. This will zoom into the area where our watershed is located.

5. Right click approximately near the locations shown in the following figure and select *Drop LatLng Marker*. This puts the coordinate marker at each of the locations you specify. See the following figure.



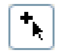
6. You may zoom in more to have a higher resolution image but make sure that all the three markers are within the visible extent.
7. Now press the *Print Screen* key on the keyboard. Open the standard Microsoft *Paint* program from the Start menu on your computer and paste the image into *Paint*. You may want to crop only the portion of the image you need using the Paint program by selecting the area of the image you are interested in, selecting Edit | Copy, selecting File | New, and then selecting Edit | Paste.
8. Once done, save the image as ***C:\Training\Personal\Images\ParkCity.jpg***.
9. If you have trouble downloading the image, there is one already downloaded in ***C:\Training\Images\ ParkCity.jpg***.

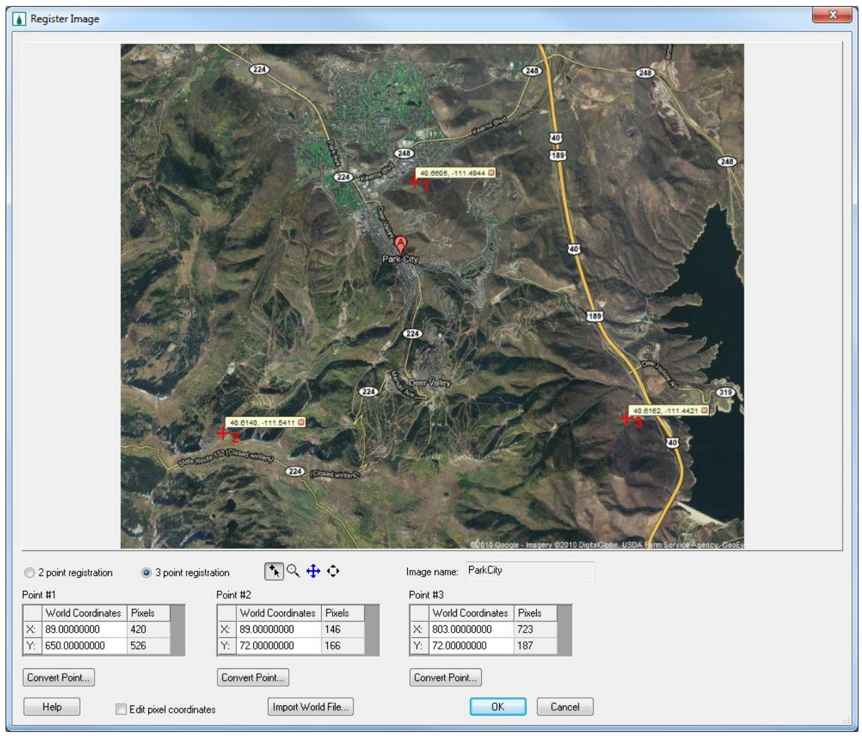
1.6.2 Geo-referencing images in WMS

You will use the Google Maps image that you downloaded for Park City and define the coordinates so it is in the same projection as your hydrologic model.

1. To overlay the image with a hydrologic model, first open the WMS project file for Park City at ***C:\Training\Images\Baseproj.wms***.
2. Open the image you downloaded in a previous step. This image should be located at ***C:\Training\Personal\Images\ParkCity.jpg***. If you had trouble capturing and saving the image, there is an image already saved at ***C:\Training\Images\ParkCity.jpg***.
3. When reading the image, click *Yes* to build pyramids.

The image will appear in the Register Image dialog. The three latitude/longitude markers are also displayed. There are also three "+" symbols with the numbers 1, 2, and 3 by them (much larger and visible). You need to place the numbered plus signs over the tip of the place markers and enter the corresponding coordinates in the fields below to register the image.

4. Use the *Point Selection* tool  to drag each "+" symbol over the closest lat/lon coordinates as shown in the following figure. You may wish to place the symbols close and then zoom  in on the area for more accuracy. Once you zoom in there is a *Frame* tool  that you can use to re-center the image so that you can zoom in on another registration point. Do your best to move each plus mark over the tip of the pins in the image.



The coordinates for the three points are known in geographic (latitude/longitude) coordinates. We will register using these coordinates and then convert to the projected UTM coordinates after reading the image. In order to properly register, the coordinates must be decimal degrees.

5. Using the values shown in the respective *LatLng* markers, enter the appropriate x (longitude) and y (latitude) values for the three points.

Note that longitude values west of the prime meridian should be entered as negative. If you enter the values as specified on the markers (+/-), your coordinates will be correct.

Also notice that Longitude or Easting is the X-coordinate and Latitude or Northing is the Y-Coordinate.

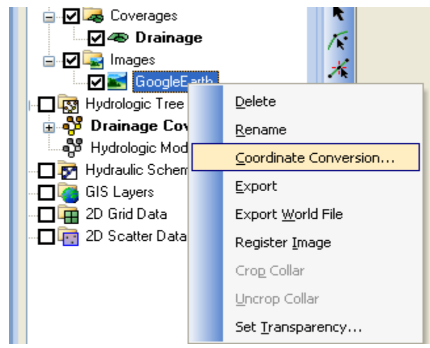
6. When you have correctly entered the three coordinates select *OK* in the Register Image dialog

The data is not showing up because the coordinate system of the delineated watershed does not match with the coordinates of the image you just registered. Do the steps in the following section to learn how to convert the coordinate system of the image so the image overlays the watershed.

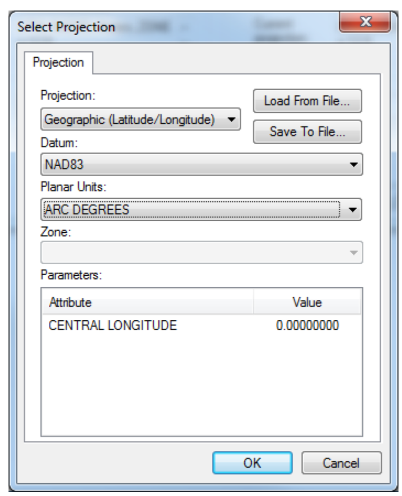
1.6.3 Converting the Coordinates to UTM

Geographic coordinates are commonly used on maps because they provide a global reference for any point in the world. However, for engineering work a planimetric (XY or Cartesian) coordinate system is necessary. The Universal Transverse Mercator (UTM) projection is a commonly used coordinate system. We will convert the coordinate system from geographic to UTM.

1. Right-click on the *ParkCity* image icon in the Project Explorer and choose the ***Coordinate Conversion*** option



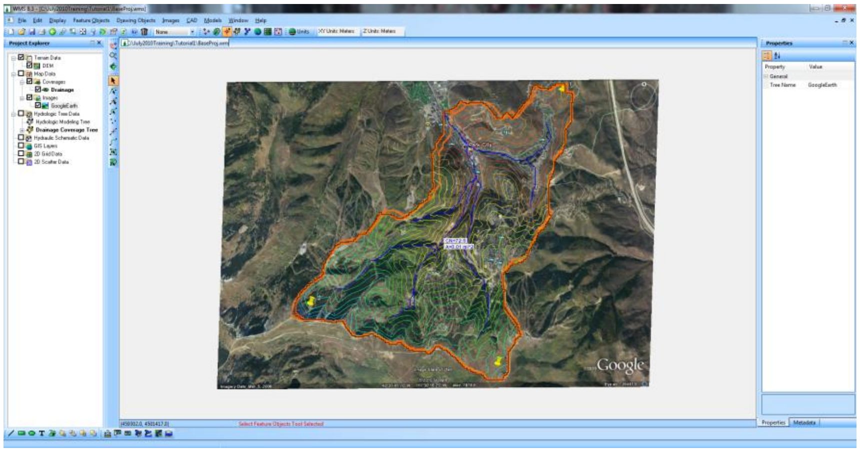
2. Select the *Global Projection* radio button in the *Object Projection* section in the *Reproject Object* dialog
3. Select *Geographic (Latitude/Longitude)* from the *Projection* drop down box
4. Select *NAD 83* from the *Datum* drop down box and *ARC DEGREES* under the *Planar Units*.



5. Select *OK*
6. Set the Vertical System projection to *NAVD 88(US) and* units to *Meters*
7. Toggle on the *Specify* check box in the *Project Projection* in the *Reproject Object* dialog
8. Select the *Global Projection* radio button

9. Set *Projection* to *UTM*, *Datum* to *NAD 83*, *Planar Units* to *Meters*, and *Zone* to *12 (114°W - 108° W – Northern Hemisphere)*
10. Select *OK*
11. Set the Vertical System projection to *NAVD 88(US) and* units to *Meters*
12. Select *OK*

There might be some distortion in the image as it is converted from geographic to UTM because a degree of longitude has a shorter distance the farther north you are, but this is normal. Any measurements taken or data created from the image will have XY distance units of meters.



1.6.4 Exporting a World File

Once you have georeferenced an image, you can then export a world file so that you can use it for future use.

1. Right-click on the *ParkCity* image in the Project Explorer and select *Export World File**.***

   If you save the world file with the same name as the original JPEG image but with a ".jpw" extension. If you export a world file with the same prefix as the JPG image, the image will open be georeferenced next time you open it. However, if a world file is saved with a different name, you will have to import the world file after reading the JPEG image. We will now see how world files work.

2. Save the file as ***C:\Training\Personal\Images\ParkCity.jpw***—the WMS default file name. Click *Save.*
3. In the WMS project explorer, delete the *ParkCity* image by right-clicking on the image and selecting *Delete*.
4. Select ***File | Open*** and select the file ***C:\Training\Personal\Images\ParkCity.jpg.*** If asked, go ahead and build pyramids. Now the image opens as a georeferenced image.

Note: As discussed above, if the image and the world file are at different locations, if they have different names, or if there is no world file, opening an image will display the *Image Registration* dialog.

If you see the ***Image Registration*** dialog but you already have a world file at a different location or with a different name, click the ***Import World File*** button at the bottom of the ***Registration*** dialog. Browse to and open the world file and click ***OK***. The image will show up as a georeferenced image.

1.7 World Files

Many image files do not contain georeferencing information. For example, JPEG files do not have georeferencing "tags" in the file like TIFF images may have. Most organizations that make images available distribute world files containing the georeferencing information along with the image files. These world files usually have the same prefix as the corresponding image file, but with the extension .tfw for TIFF images (for JPEG files, the extension is .jpw, .jgw, or .jpgw). If you download a world file and are asked to supply a name for the file, use the same prefix as the image file with a ".jpw" or a "tfw" extension and WMS will automatically recognize the world file and register the associated image.

Use the following procedure to view the information contained in the world file:

1. Select ***File | Edit File***
2. Browse and open ***C:\Training\Personal\Images\ParkCity.jpw***

Because there is a world file named *ParkCity.jpw*, the image is automatically registered after reading the JPEG image. If WMS does not find a world file for an image, you would have the option of importing the world file from within the registration dialog.

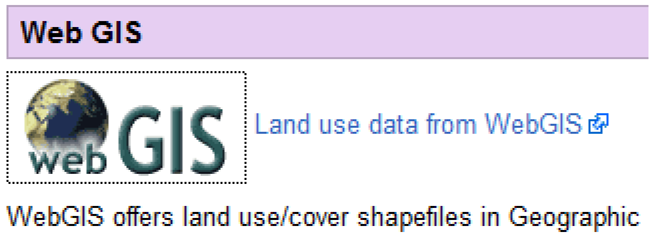
1.8 GIS Shapefiles

GIS shapefiles are another data source frequently used in developing models with WMS. In this section you will see how commonly used shape files for land use and soil data can be obtained and used in WMS.

1.8.1 Geospatial data acquisition

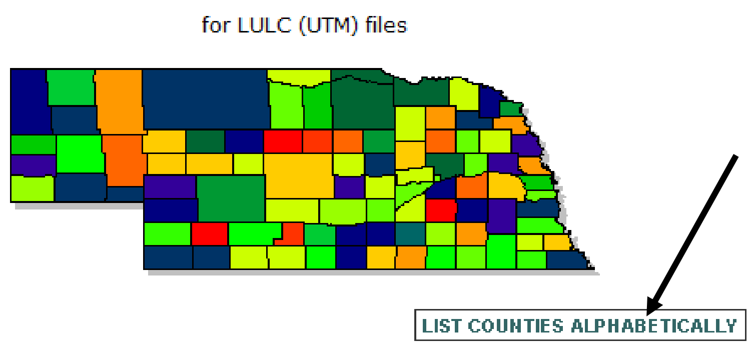
There are several locations where you can obtain various types of geospatial data. The XMS wiki at http://www.xmswiki.com has a special geospatial data acquisition page which has tutorials and tips on downloading these data from the most convenient sources for use in WMS, SMS or GMS. In this section, you will download a shapefile with land use data and open it in WMS.

1. Start a web browser and type the following link into the address bar: http://www.xmswiki.com/xms/GSDA:GSDA. This will take you to the Geo-Spatial Data Acquisition Home page.
2. Under *Surface Characteristics*, select *Land Use.*
3. Select the *WebGIS* link.

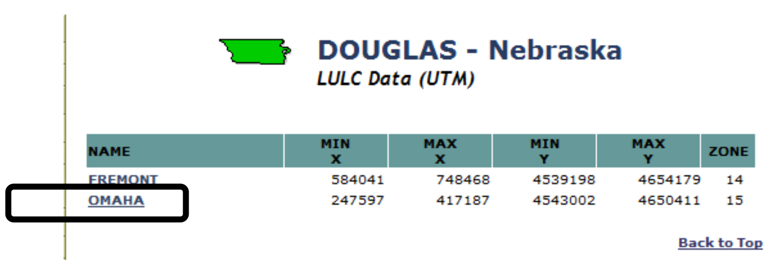


4. Under *Land Use* select *Shapefile (UTM)*

5. For this exercise, you will get the land use data for Omaha. In the US map, select Nebraska.
6. Click on *List Counties Alphabetically*

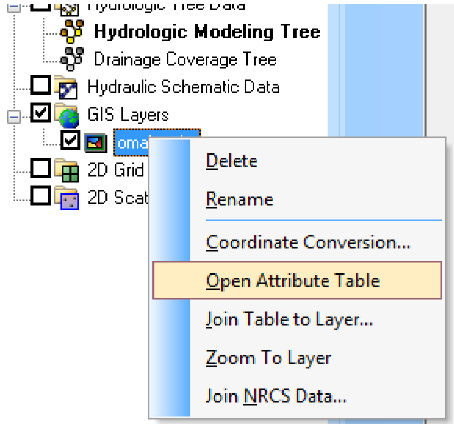


7. In the list that displays, select *Douglas County*
8. Select *Omaha* on the top of the list.



NAME	MIN X	MAX X	MIN Y	MAX Y	ZONE
FREMONT	584041	748468	4539198	4654179	14
OMAHA	247597	417187	4543002	4650411	15

9. Save ZIP file archive on your computer.
10. Outside of WMS, browse to the folder where you saved the Omaha.zip file and extract the files to ***C:\Training\Personal\Images***.
11. In WMS, delete your data by selecting ***File | New***.
12. Before reading any data, you should set your coordinate system by setting your coordinate system. Select ***Edit | Current Coordinates*** to set your current coordinates.
13. Select the *Global Projection* option.
14. Select *Set Projection.*
15. Set *Projection* to *UTM*, *Datum* to *NAD 83*, *Planar Units* to *Meters*, and *Zone* to *15 (114°W - 108°W – Northern Hemisphere)*.
16. Select *OK.*
17. Set the vertical projection to *NAVD 88(US)* and the vertical units to *Meters.*
18. Select *OK.*
19. Open the aerial photo covering the region surrounding Omaha at the following location: ***C:\Training\Images\Omaha\AerialPhoto.jpg***. Select ***Yes*** if asked to generate image pyramids.
20. In the project explorer, right click on *GIS Layers* and select *Add Shapefile Data…*
21. Browse and open the land use shapefile that you downloaded ***(C:|Training\Personal\Images\omaha\omaha.shp)***
22. Now you can see the land use shape file overlaying the aerial photo for the city of Omaha.
23. Right click *omaha.shp* on the project explorer and select *Open Attribute Table.* This will open the attribute table for the land use shape file. You should be able to see *LUCODE* as one of the attributes. *LUCODE* is the USGS land use code. You will be using this attribute to define overland flow roughness values and other surface characteristics in later tutorials.



24. This shapefile is in the GIS module of WMS. To use this shapefile for hydrologic and hydraulic calculations, you need to convert it to data in the map module of WMS. You will do this in a later section.
25. Do not close WMS, we will keep working with this data in the following section.

1.8.2 Geospatial data processing

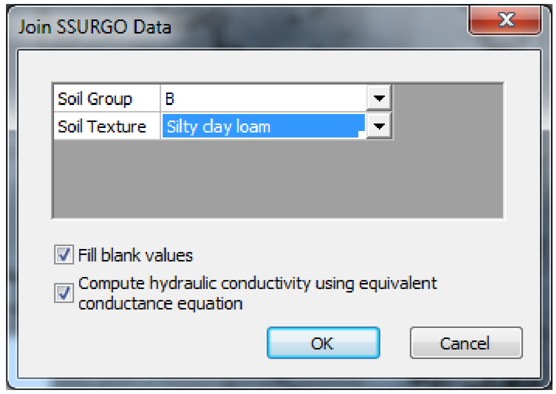
Some of the GIS data are not 'ready to use' like the land use shapefile we used in the previous section. We need to process these data before they can be used. In this section we will see how some of these tools can be used in WMS with a soils shapefile.

We will open a SSURGO soil shapefile for Omaha and process it so that it can be used for defining infiltration and other soil-derived properties required for GSSHA models.

1. Right click *GIS Layers* in the project explorer. Browse to and open the file ***C:\Training\Images\Omaha\Soil\Spatial\soilmu_a_ne055.shp.***
2. When the soil shapefile is read into WMS, the projection file associated with the file is read and the coordinates of the data are converted from Geographic (Lat/Lon) to the current coordinate system (UTM Zone 15).
3. Right click on *soilmu_a_ne055.shp* and open its attribute table. In the attribute table, notice that there are only a few properties associated with each soil polygon, but there are no recognizable soil attributes. To get the soil properties associated with each soil polygon, you will need to read these properties from tables included with the SSURGO data.
4. To read the properties, close the attribute table and right click on *soilmu_a_ne055.shp* in the project explorer. Select *Join NRCS Data.*

The soil attributes associated with the soil polygons are stored as separate tables. Selecting the *Join NRCS Data* command allows you to link these tables to your shapefile so you can use them.

5. In the *Join SSURGO Data* dialog select both check boxes and leave the Soil Group and Soil Texture fields as they are. See the following figure:

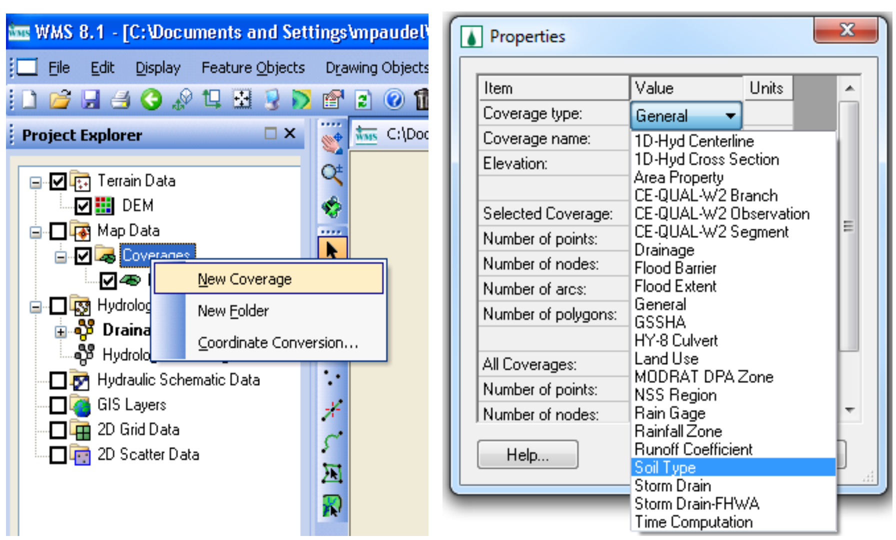


6. Click OK.
7. Right click on the soils shape file under GIS Layers and open the attribute table again. This time you will see that several soils properties have been added as attributes.

1.9 Mapping Shapefiles to WMS coverages

The SSURGO shapefile that we processed in the previous step is still in a GIS format that WMS cannot use directly. In order to use it the polygons overlapping our watershed need to be copied to a WMS Soil Type coverage. This is done from the GIS module.

1. In the project explorer (left side of WMS main window), right click on "*Coverages*" and select "*New Coverage*"



2. Select the coverage type to be *Soil Type* and click *OK*. This will create a new soil type coverage in the project explorer. Note that WMS creates a Drainage coverage automatically as soon as WMS is opened. When you create a GSSHA grid, the drainage coverage is converted to a GSSHA coverage.

3. Click on the new soil type coverage in the Explorer window to make it active.
4. Click on the *soilmu_a_ne055.shp* (under GIS layers) to make it active and change to the GIS module as we are going to map the data from the soil shapefile to the soil type coverage.
5. Turn off the check box next to the *omaha.shp* shapefile you read in a previous section. This is the land use shapefile and you do not want the data in this shapefile included in your soil type coverage.
6. Click on the "*Select shapes tool*"  and drag a rectangle around a watershed area you are interested in to select all land use polygons that overlap your watershed. Do not worry if you select polygons that are outside the watershed boundary, WMS will use the watershed boundary to clip the land use polygons.
7. Select ***Mapping | Shapes –> Feature objects***, click "*Nex*t" and make sure that the fields are mapped properly. For example, the TEXTURE field should be mapped to Texture, HYDGRP should be mapped to SCS soil type, and so on. Click "*Next*" and click "*Finish*".
8. The selected soil polygons are now converted to a coverage in the WMS map module and saved in the WMS map file format. You no longer need the GIS data once it is mapped to a coverage. You may delete the soil shape file under GIS layers (Right click on *soilmu_a_ne055.shp* and select delete).

CHAPTER 2

Watershed Delineation using DEMs and 2D Grid Generation

2.1 Initiating WMS

1. Close WMS and open a new instance of WMS.

2.2 Downloading and Importing DEM Data

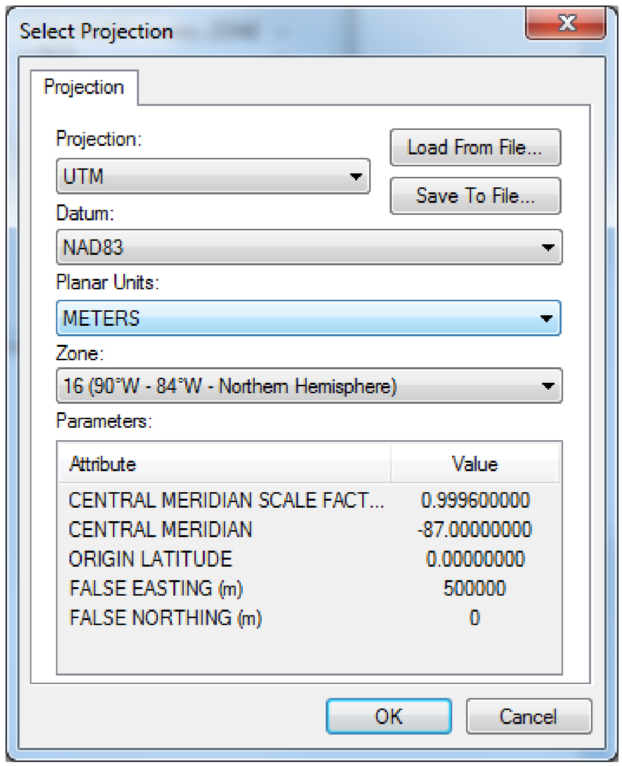
DEM data can be obtained from a variety of sources. If you already have a DEM stored on your computer, you can just open it in WMS using the File | Open command. Alternatively, you can download DEM data from the USGS seamless server (http://seamless.usgs.gov/)

WMS has a web services tool that links directly to the USGS seamless data server. This tool can help you download DEM data. We will use the web services tool in this workshop.

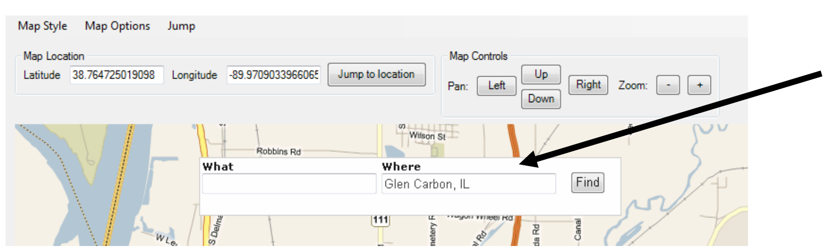
1. Click on the *Hydrologic Modeling Wizard*  button at the lower left corner of the WMS main window.

2. In the *Hydrologic Modeling Wizard* dialog, browse for the folder ***C:\Training\Personal\WatershedDel*** and enter the project name ***JudysBranch.wms***.

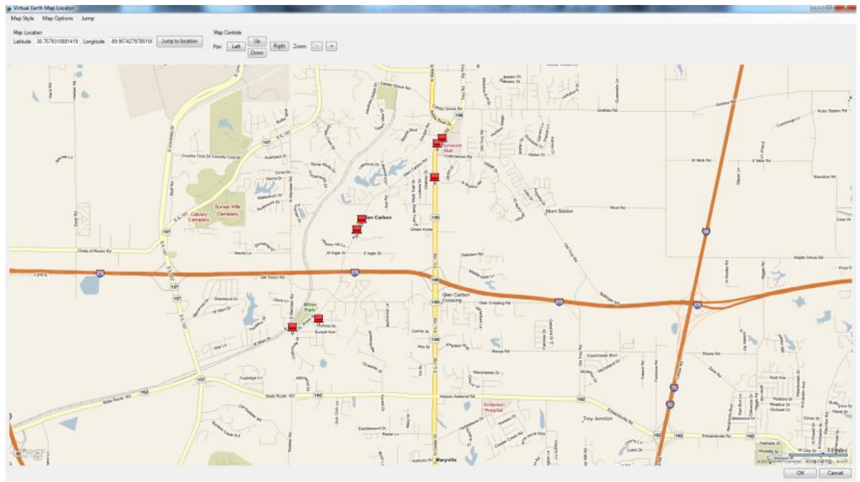
3. Click the *Save* button.
4. Click *Next*
5. Click the *Define* button under the *Project Coordinate System*. Select the *Global Projection* option (*Set Projection* if this is already selected) and enter the following information and click *OK*. Select *NAVD 88(US)* for your vertical projection and *Meters* for your units and then select *OK* again.



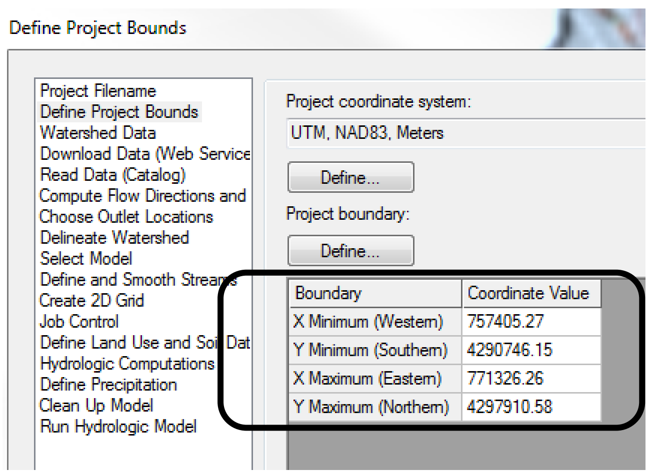
6. Now, select the *Define* button to define the project bounds. This opens a map locator window and lets you navigate to your project area.
7. Maximize the *Virtual Earth Map Locator* window and select ***Map Options | Show Locator Tool.*** This will show a search field in the window.
8. In *Where* field, enter ***Glen Carbon, IL*** and click on *Search* button.



9. The map will show *Glen Carbon* Crossing approximately at the center of the window. Zoom in little more. Compare your display with the following figure.

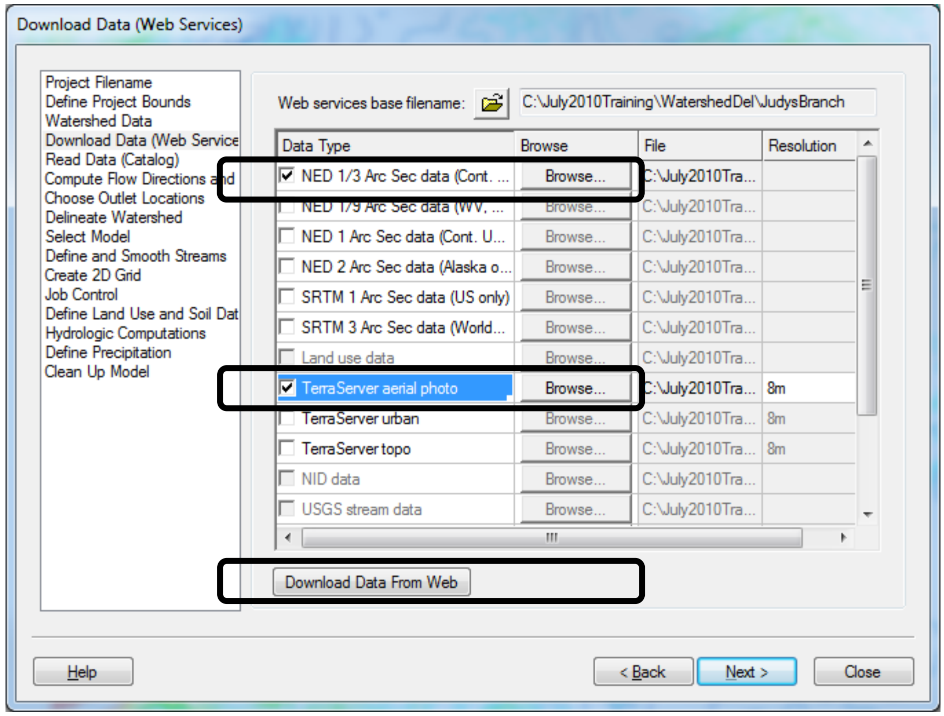


10. Click OK. WMS will now extract the bounding coordinates for the extent of the display of this map. You can see the coordinates listed in the wizard window as shown in the following figure.



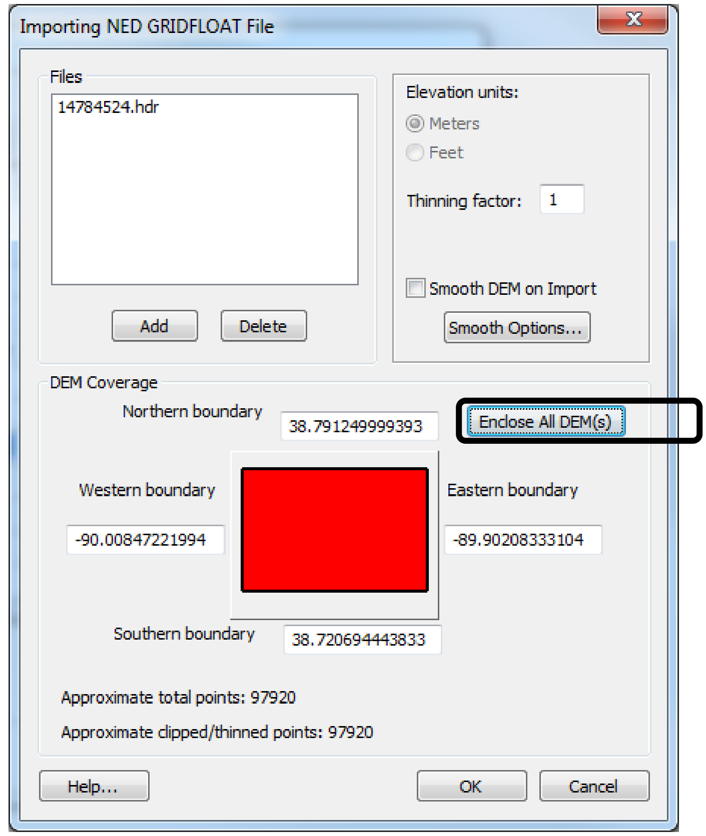
11. Click *Next* and make sure that the *Use web services* option is toggled on.

12. Click *Next* and select *NED 1/3 Arc Sec Data* and *TerraServer aerial photo.* Then click *Download Data From Web.*



13. WMS will download both the DEM and background aerial photo for the watershed (Note: There are times when the web services may be unavailable so if WMS does not download the data directly you could download the data outside of WMS. You can download the DEM data from the USGS at http://seamless.usgs.gov and the aerial photo from MSR maps at http://msrmaps.com. Alternatively, if you were unable to download the DEM and/or image, you can find a copy in *C:\Training\RawData\JudysBranch\DEM*).

14. Once the DEM downloads, the *Importing NED GridFloat File* dialog will open. Select the *Enclose All DEM(s)* button and click OK.



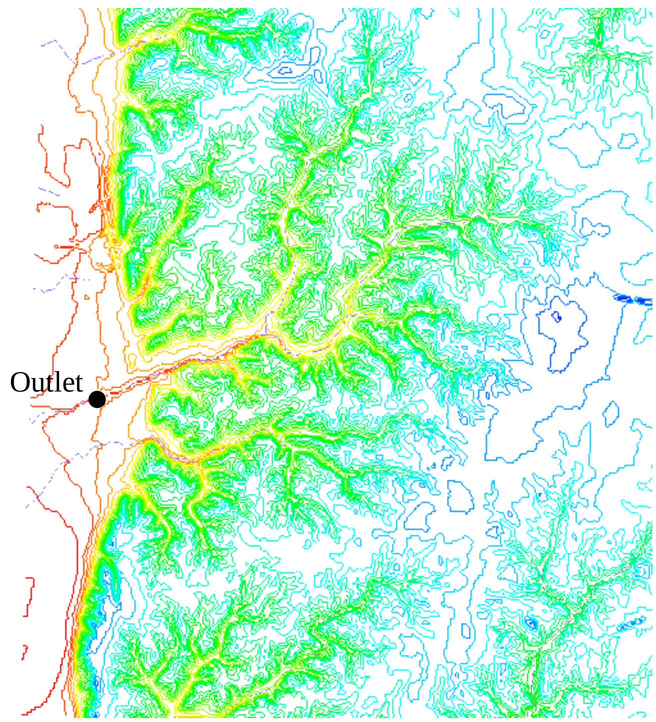
15. WMS will now download the aerial photo from TerraServer. If prompted, select *Yes* to generate pyramids.
16. After the DEM and the image have finished downloading, you can move the Hydrologic Modeling Wizard to the side and you should see both the DEM contours and the aerial photograph in the WMS graphics window.
17. Close the *Hydrologic Modeling Wizard* dialog.

2.3 Computing the Flow Directions and Flow Accumulations

1. To delineate a watershed you should be in the *Drainage Module*. Click  to select the drainage module.
2. Select ***DEM* | *Compute TOPAZ Flow Data*** and Click *OK* twice. TOPAZ will compute the flow direction and accumulation and infer the streams based on the DEM data.
3. Click on “Close” after computations are complete. It will probably take a few seconds to finish, but you can know it is done when the last line of text in the model wrapper reads “Normal Program Termination”.

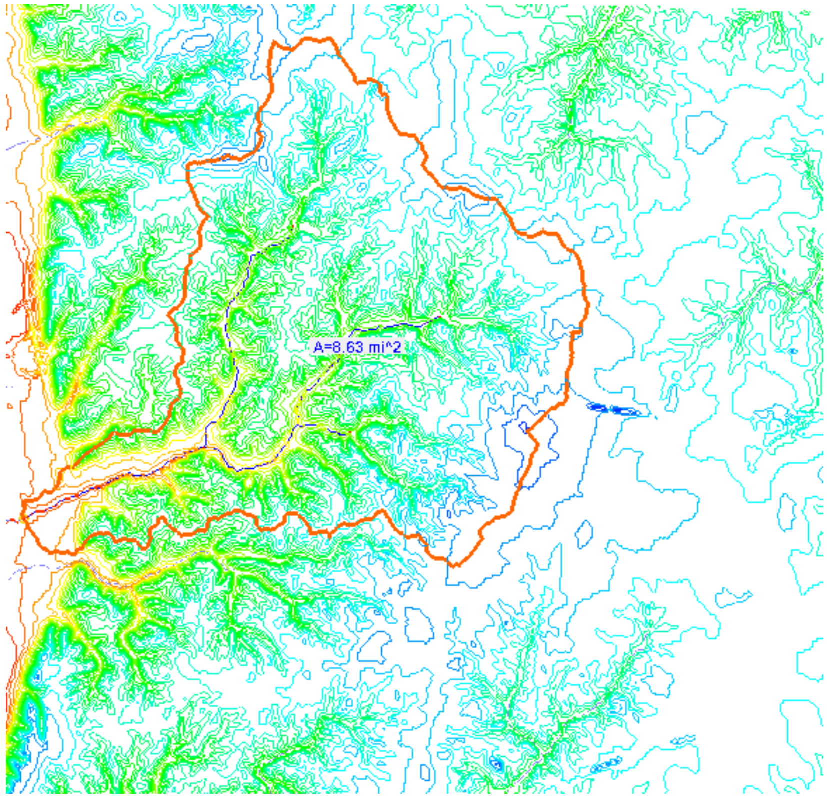
4. You can now see lines representing areas of flow accumulation above a threshold value on the display. These are the areas where flow accumulates on the DEM, and these areas may represent stream channels.

5. You need to create an outlet point to delineate a watershed. Select the *Create Outlet Point Button* . Locate the point where you want the outlet for the watershed to be. See the following figure for the approximate location of the outlet (you can use the middle scroll button of the mouse to zoom in or out).



2.4 Delineating the Watershed

1. Select ***DEM | DEM -> Stream Arcs…***. Make sure the stream threshold value is set to ***1 sq. mile***. Click *OK*.
2. Select ***DEM | Define Basins***
3. Select ***DEM | Basins -> Polygons***
4. Select ***DEM | Compute Basin Data***. Click *OK*.
5. Click on the "*Frame*" button . Your watershed should look somewhat like the following figure.



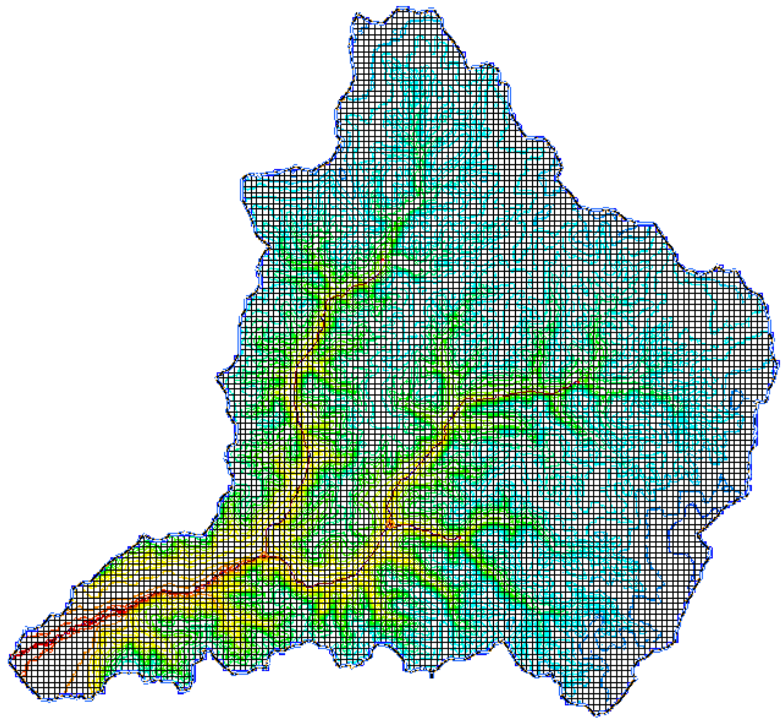
6. Save your WMS project by selecting ***File | Save.*** Save it as ***C:/Training/Personal/ WatershedDel /JudysBase.wms***. Click *Yes* to save the image files in the project directory. Note that at this point you have a completed watershed and you can always open this saved project and start over with the following steps for creating your GSSHA model if you make a mistake.

2.5 2D grid generation

To develop a GSSHA model, you will need to generate a two-dimensional finite difference grid.

1. Switch to the *Map Module* .
2. Click on the *Select Feature Polygon Tool* and right click anywhere within the watershed polygon. Then select *Create Grid* in the popup menu that appears.
3. Select *Yes* to confirm that you are creating a GSSHA grid.
4. Make sure the *Base Cell Size* option is checked on and enter **50m** as the cell size and click *OK.*
5. Click *OK* to interpolate grid cell elevations from the DEM, and select *NO* when prompted if you want to delete the DEM data.

6. You can now see grid cells covering the watershed. Notice that under the *Coverages* in the data tree, the *Drainage* coverage has been now changed to *GSSHA.*
7. Do NOT save the WMS project because the GSSHA grid information and model are saved to a GSSHA project file instead of to a WMS project.
8. Switch to the *2D grid Module* and select ***GSSHA|Save Project file...***
9. Save the project as ***C:/Training/Personal/WatershedDel/ JudysBase.prj***



2.6 Workshop Tasks

1. You can delineate a watershed in your area of interest by following the steps described in this tutorial.
2. If you already have a DEM for your area, open it in WMS (you can skip the section on downloading and importing DEM data).

CHAPTER *Error! Reference source not found.*

Creating Feature Objects and Mapping their Attributes to the 2D Grid

In order to develop a grid into a GSSHA model, several attributes such as surface roughness, infiltration, porosity, moisture content, and others need to be defined for each cell. This tutorial focuses on the development of these hydrologic parameters for GSSHA grids using commonly available land use and soil GIS data.

Since GSSHA uses spatially varying parameters, it would be extremely time consuming to enter such parameters for each grid cell one by one. You will define the parameters for each cell by aggregating cells of similar soil and/or land use properties using Index Maps. If there are only 5-6 different soils in a watershed then you would define infiltration properties for each of these soils in a table. This is much less time consuming than having to define the values separately for each cell. In this tutorial you will learn how these Index Maps are created in WMS.

1. We will continue working with the Judys Branch gridded model that you saved in the previous workshop.

2. To open the GSSHA project, switch to the *2D Grid Module* and select ***GSSHA | Open Project File...***. You can also select ***File | Open*** to open a GSSHA project file.

3. If you have saved the project in the previous workshop, open the project from ***C:/Training/ Personal/WatershedDel/JudysBase.prj***.

   If you were not able to save the project, you can open ***C:/Training/WatershedDel/JudysBase.prj.***

4. Select ***GSSHA | Save Project File***... and save the project with new a name as ***C:/Training/Personal/FeatureObjects/JudysParam.prj*** so the original project is not overwritten.

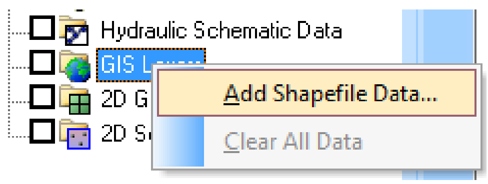
3.1 Mapping GIS shape files to WMS Coverage

A previous workshop described how land use and soil GIS data are obtained and read into WMS. If there are any questions, refer to Section 1.8.1.

In previous workshops, you learned about various WMS data types and their sources. In this section, you will use these data to set up GSSHA modeling parameters.

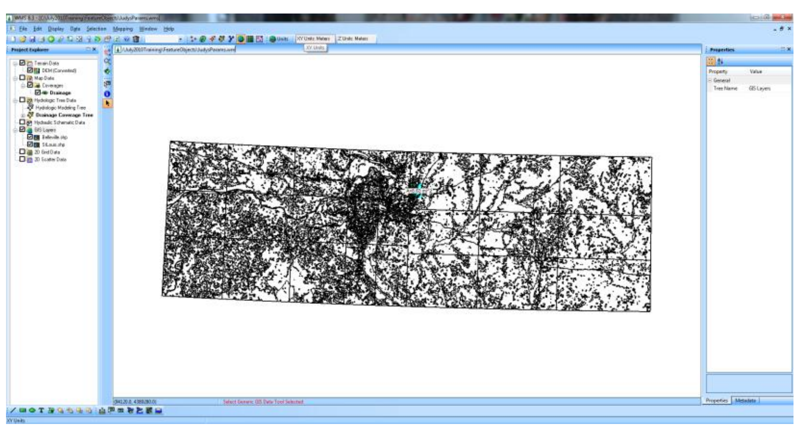
3.1.1 Adding Land Use Coverage

1. In the project explorer, right click on the *GIS Layers* folder and select *Add Shapefile Data.*



2. Browse to ***C:/Training/RawData/JudysBranch/Landuse*** and open ***Belleville.shp and StLouis.shp.*** These are the two land use shapefiles that provide coverage for our watershed.

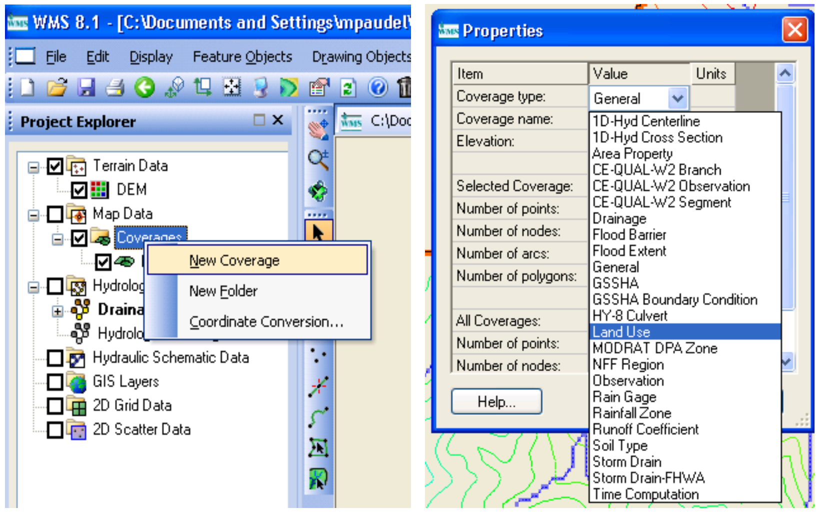
Two shapefile layers are imported and added to the project explorer under *GIS layers*. The land use extent should overlay your watershed. Your project should now look like the following figure.



3. Right click the shapefiles and view the attribute tables. You should be able to see the USGS land use code (LUCODE).

4. In order to use the land use data, the polygons overlapping our watershed need to be copied to a WMS land use coverage. This is done from the GIS module.

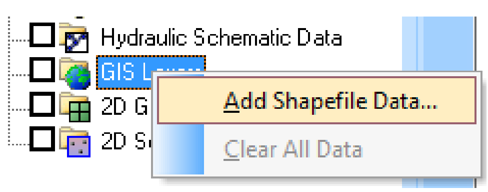
5. In the project explorer, right click on *Coverages* and select *New Coverage.*



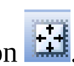
6. Set the coverage type to be *Land use* and click *OK*. This will create a new land use coverage in the project explorer.
7. Click on the new Land use coverage in the Explorer window to make it active.
8. Select either *Belleville.shp* or *StLouis.shp* (under GIS layers) to activate the GIS module.
9. Click on the "*Select shapes tool*"  (should be selected by default) and drag a rectangle around your watershed area to select all land use polygons that overlap this basin.
10. Select ***Mapping*** | ***Shapes*** –> ***Feature objects***, click *Next* and make sure that LUCODE is mapped to Land use. Click *Next* and click *Finish*.
11. WMS no longer needs the GIS data once it is mapped to a coverage in the map module. You may now delete the shape files under GIS layers (Right click on the shapefiles *Belleville.shp* and *Stlouis.shp* and select delete).

3.1.2 Adding Soil Type Coverage

1. In the project explorer, right click on *GIS Layers* folder and select *Add Shapefile Data.*



2. Browse to the following SSURGO soil shapefile and open it: ***C:\Training\RawData\JudysBranch\SSURGOSoil\Raw\Spatial\soilmu_a_il119.shp.***

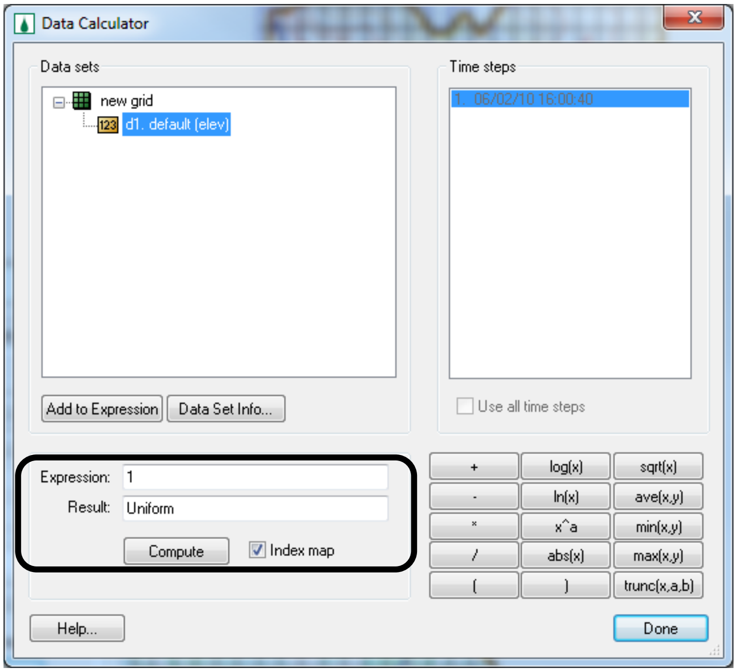
3. This data is as it was downloaded and needs to be processed to join the soil attributes.
4. Join the attributes for the shapefile you added. To do this, right click the shapefile (*soilmu_a_il119.shp*) and select *Join NRCS Data*. Turn on the options to *Fill blank values* and *Compute hydraulic conductivity…*. Refer to section 1.8.2 of this tutorial for details about joining a SSURGO soil shapefile's soil types with other SSURGO attributes.
5. Create a new coverage and change the type to *Soil Type.*
6. Map the polygons from the soil shapefile that overlap the watershed to the newly created soil type coverage. (Similar to how you mapped land use data, see section 3.1.1)
7. Once you have mapped the soil polygons delete the SSURGO shapefile (*soilmu_a_il119.shp*) from the project explorer.
8. You may now turn off the display of the *Land Use* and *Soil Type* coverages by turning off the toggle boxes next to these coverages in the project explorer. Make sure that the display of the *GSSHA* coverage and *2D Grid Data* are turned on.
9. Click the *"Frame"* button .

3.2 Creating Index Maps

WMS has an interface to automatically generate an index map. In this section, you will create uniform, land use, and soil index maps.

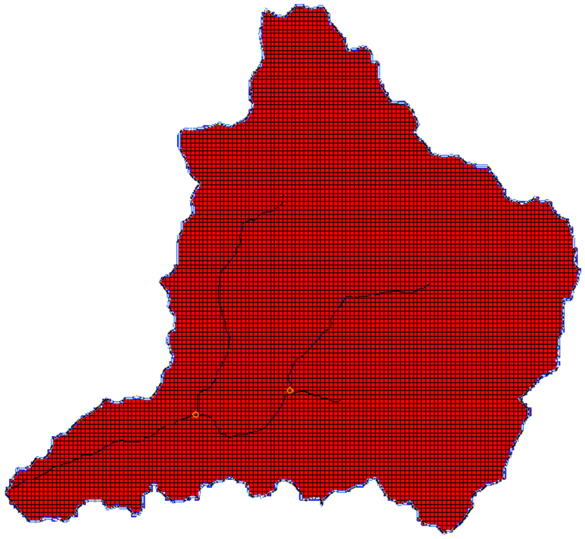
3.2.1 Creating Uniform Index Map

1. Switch to the ***2-D Grid module*** by clicking the grid module  button.
2. Select ***GSSHA | Maps…***
3. Click the *Data Calculator* button.
4. In the *Data Calculator* dialog enter 1 for the *Expression* and enter *Uniform* for the *Result* field. Check on the *Index Map* toggle and click *Compute.*



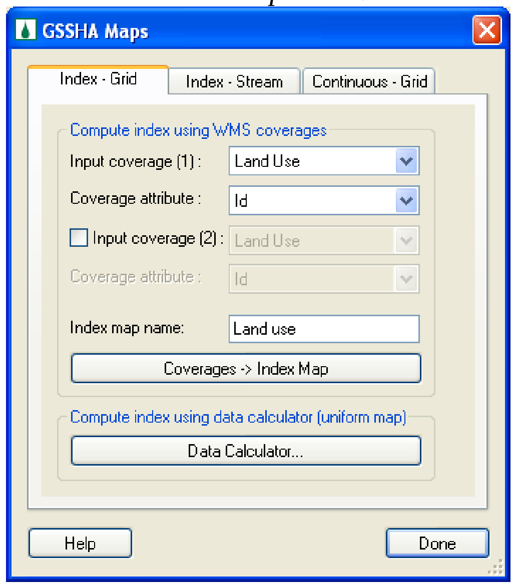
5. Click *Done.*

This will create an index map that has the same ID value (1 in this case) for each grid cell. A uniform index map is used to define parameters which are uniform over the watershed domain. For example, if you assume uniform overland roughness in the watershed, then a uniform index map can be used to define the roughness for the whole watershed.

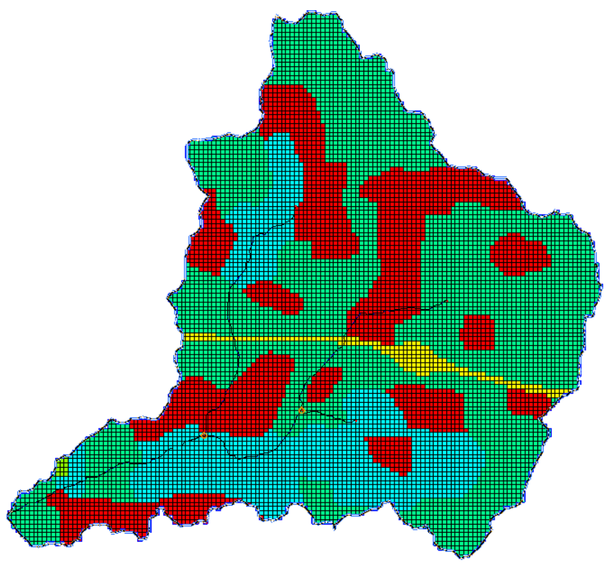


3.2.2 Creating a Land Use Index Map

1. In the *GSSHA Maps* dialog, select *Land Use* for *Input Coverage (1)*. Enter the name Land Use for the *Index Map Name*.

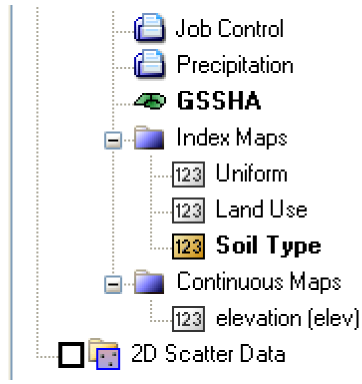


2. Click on the *Coverages -> Index Map* button.
3. You should now see colored grid cells. Each color represents a different land use code.



3.2.3 Creating a Soil Type Index Map

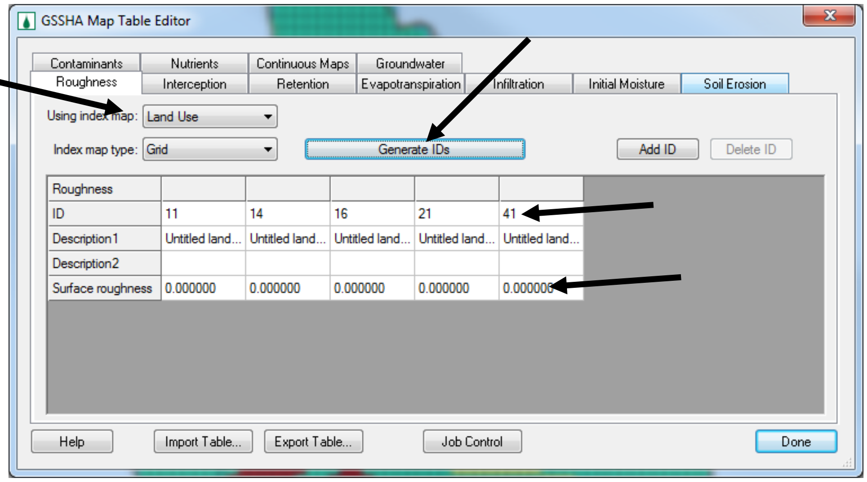
1. In the *GSSHA Maps* dialog, select *Soil Type coverage* for *Input Coverage (1)*. Enter the name Soil Type for *Index Map Name.*
2. Click on the *Coverages -> Index Map* button.
3. Click *Done* to close *GSSHA Maps* dialog.
4. At this point your project should have three index maps and the project explorer will look something like the following figure:



3.3 Exploring Mapping Tables

After index maps are created, land use or soil characteristics can be entered into *GSSHA Map Tables*. Map tables are tables of hydrologic and hydraulic parameters for all possible values in a certain index map.

1. Select ***GSSHA | Map Tables.*** This will bring up the *GSSHA Map Table Editor* dialog.
2. The R*oughness tab* is selected by default. Currently, this tab does not have any information but we can use one the index maps to generate mapping table IDs.
3. Select *Uniform* from the dropdown box for *Using Index Map* field and click *Generate IDs.* This will create one Land Use ID field. If you recall the step when you created *Uniform* index map, you entered a value of 1 for all the grid cells. Since each cell in the index map has a value of 1, you only need to enter the roughness parameters for this single value in the mapping table. Using this index map for your *Roughness* IDs assumes a uniform surface roughness for the entire watershed. In the real world, however, the surface roughness varies over the watershed domain. If you want to create a GSSHA model that uses distributed surface roughness based on land use IDs, you will have to use another map such as the *Land use* index map to create the Roughness mapping table IDs.
4. Select *Land Use* from the dropdown box for the *Using Index Map* field and click *Generate IDs.* Select *Yes* if prompted to delete existing IDs. This will create mapping table IDs for each unique land use ID that existed in the land use shape file (or now exists in the land use index map).



5. You could now enter the descriptions and roughness values for each of these IDs. You will see in the following section how to read a file containing the descriptions and default roughness values, so there is no need to enter the descriptions and roughness values at this time. The IDs seen in the spreadsheet are USGS Standard Land use codes. The USGS classification table helps you identify each land use type based on its ID. A portion of this table is shown below:

Classification Code	Land Use Description
11	Residential
12	Commercial Services
13	Industrial
14	Transportation, Communications
15	Industrial and Commercial
16	Mixed Urban or Built-Up Land
17	Other Urban or Built-Up Land

6. Map tables are created for other hydrologic and hydraulic characteristics of a watershed using different index maps. You may switch through each of the different tabs in the *GSSHA Map Table Editor* dialog. When selecting some tabs, you will be prompted to change job control settings. For example, if you have not turned infiltration on but if you select the *Infiltration* tab in the mapping table editor, WMS prompts you if you want to turn the infiltration on. Select *No* when these messages appear.
7. Select Done in the *GSSHA Map Table Editor* to close the window.

3.3.1 Understanding the Mapping (.cmt) File

The parameter values you entered in the *GSSHA Map Table Editor* dialog are saved in a mapping table file in the GSSHA project directory that has a *.cmt extension. GSSHA uses the .cmt file as a lookup table to find defined values for GSSHA parameters for corresponding index map IDs.

1. Select ***File* | *Edit File*** in WMS. Browse and open ***C:\Training\ gssha.cmt***

2. This will open the sample .cmt file in a text editor. You can go through different sections of this file and see how the parameters for each index map value are stored in a .cmt file. The roughness values for each USGS land use in this file are obtained from standard text books.
   Similarly, the infiltration parameters in this file were obtained from Rawls and Brakenseik's table (Rawls, W. J., Brakensiek, D. L., and Miller, N. (1983). "Green-Ampt infiltration parameters from soils data," ASCE J. Hyd. Engr. 109(1), 62-70.). This table provides Green and Ampt infiltration parameters based on soil textures. You imported soil texture while processing the SSURGO soil data in a previous section so you can use this texture to derive infiltration parameters.
   The USGS land use files and SSURGO soils files use consistent IDs and texture classifications. To avoid defining the roughness and infiltration values each time you create a new GSSHA model, we created a "master" mapping file from the literature that has roughness values defined for each USGS land use code and infiltration parameters for each SSURGO soil texture. This table can be found in ***C:\Training\gssha.cmt***. These values are not intended to be absolute for every project, but the values define a reasonable starting point.
3. It is important to note that WMS creates a .cmt file each time you create a GSSHA model. This .cmt file is saved in the same folder where your GSSHA project is saved. However, this .cmt file does not have all the parameters listed as you saw in the master .cmt file found in ***C:\Training\gssha.cmt***. WMS will save out only those parameters for the IDs that are present in your index maps.
4. Close the .cmt file.
5. Instead of typing the parameters for roughness, infiltration, or other mapping table parameters, you can import a .cmt file that already has these values listed like the one you just viewed. To import the gssha.cmt file, select ***GSSHA | Map Tables***. Click on the *Import Table* button in the *GSSHA Map Table Editor* dialog and browse for the ***C:\Training\gssha.cmt*** file. Open this file and notice the values in the mapping table editor are populated with the values from the .cmt file. Click *Done* to close the *GSSHA Map Table Editor* dialog.

3.3.2 Combined Index Map Concept

The values of infiltration parameters obtained from Rawls and Brakenseik's table are for bare earth conditions where the soils are assumed to be exposed. But in a natural watershed, the land cover will change the infiltration characteristics of the soil. For example, sand with grassland as land use will have significantly different infiltration characteristics from sand covered with crops or asphalt.

Because of this, it is advisable to create a combined index map which generates unique IDs for each different land use-soil type combination within the watershed. Once you have your soil and land use data mapped to respective WMS coverages, a combined index map can be generated.

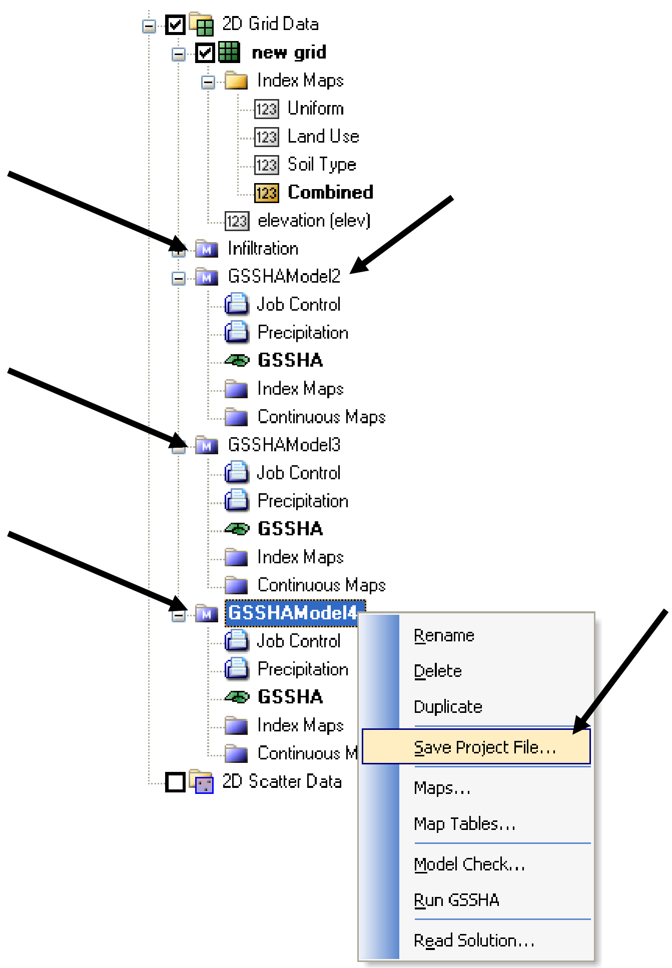
1. In the *2D Grid Module,* select ***GSSHA | Maps*** to open the *GSSHA Maps* dialog. Then check the toggle to use the second input coverage (turn on *Input coverage (2)*).
2. Select *Soil Type* for the first input with *Texture* as the attribute and *Land use* for the second input with *ID* as the attribute.
3. Change the name to *Combined* and create an index map by selecting the *Coverages->Index Map* button.
4. This combined index map can now be used in the *GSSHA Map Table Editor* to generate infiltration parameter IDs.

One thing to note here that when you import a standard .cmt file to fill in the infiltration parameters, the parameter values are obtained from Rawls and Brakenseik's table and, as we already discussed, these values are for the bare earth condition. You will have to manually adjust the parameters during calibration if you are using a combined index map to generate infiltration parameters.

3.4 Saving a GSSHA Project

There are two different commands to save files as you create a GSSHA model in WMS. If you select ***File | Save*** or use the save button you will save a WMS project. This does not save the index map or the GSSHA project. Once you have created a 2D grid, you should save your GSSHA project using the ***Save Project File*** command in the ***GSSHA*** menu. This command saves a GSSHA project and all the data important for running a GSSHA model. It is usually a good idea to save your project using ***File | Save*** *before* creating your 2D grid so you have your pre-GSSHA model saved and then save your project using ***GSSHA | Save Project File*** *after* creating your 2D grid so all the necessary data for running a GSSHA model are saved with the project.

1. To save a GSSHA project, switch to *2D Grid module* and select ***GSSHA | Save Project File.***
2. It is possible that you might have more than one GSSHA model based on the same grid. This is demonstrated in later scenario modeling workshops. In the situation when you have more than one model, when you choose ***GSSHA | Save Project File,*** the GSSHA project which is currently selected in your project explorer will be the project that is saved.
3. As an alternative you can save a GSSHA project by right clicking on the GSSHA model in the project explorer you wish to save and selecting *Save Project File.* See the following figure.



In the above figure, there are four GSSHA projects namely *Infiltration*, *GSSHAModel2*, *GSSHAModel3* and *GSSHAModel4*. The screenshot shows the saving of *GSSHAModel4*.

3.5 Task

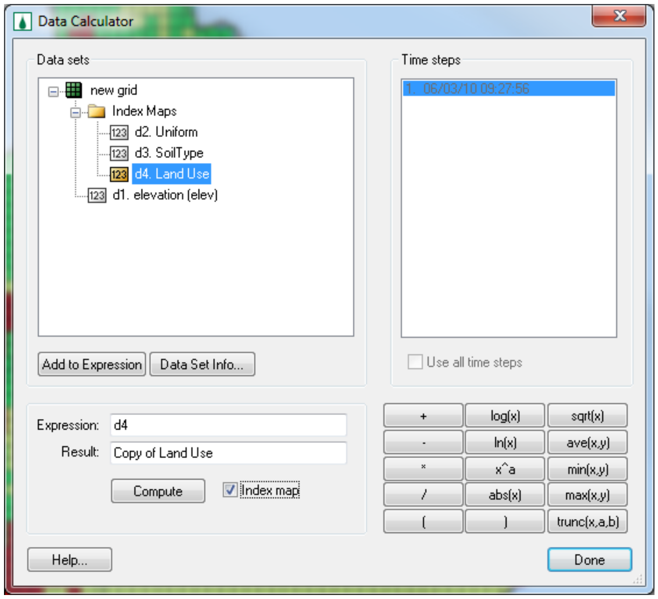
In this workshop you learned how an index map is created using WMS. You learned that an index map can be created using the data calculator (as in the uniform index map example), using land use and soil type data, and using a combination of land use and soil data.

This section gives you a few more tips about editing index maps.

3.5.1 Copying an index map

There might be some situations when you need a copy of an index map, such as when you want to preserve an index map generated from land use but you want to change individual cells in the index map.

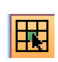
1. In the *2D Grid Module*, open the *GSSHA Maps* dialog (***GSSHA | Maps*** menu item).
2. Open the *Data Calculator* by clicking the *Data Calculator* button.
3. In the data calculator, double click the index map you want to copy. Doing this will put text representing the index map in the *Expression* field.
4. Check the *Index Map* toggle on and enter an appropriate name.
5. Click the Compute button. This will create a copy of the index map you selected.
6. You should see your index map added to the project explorer.



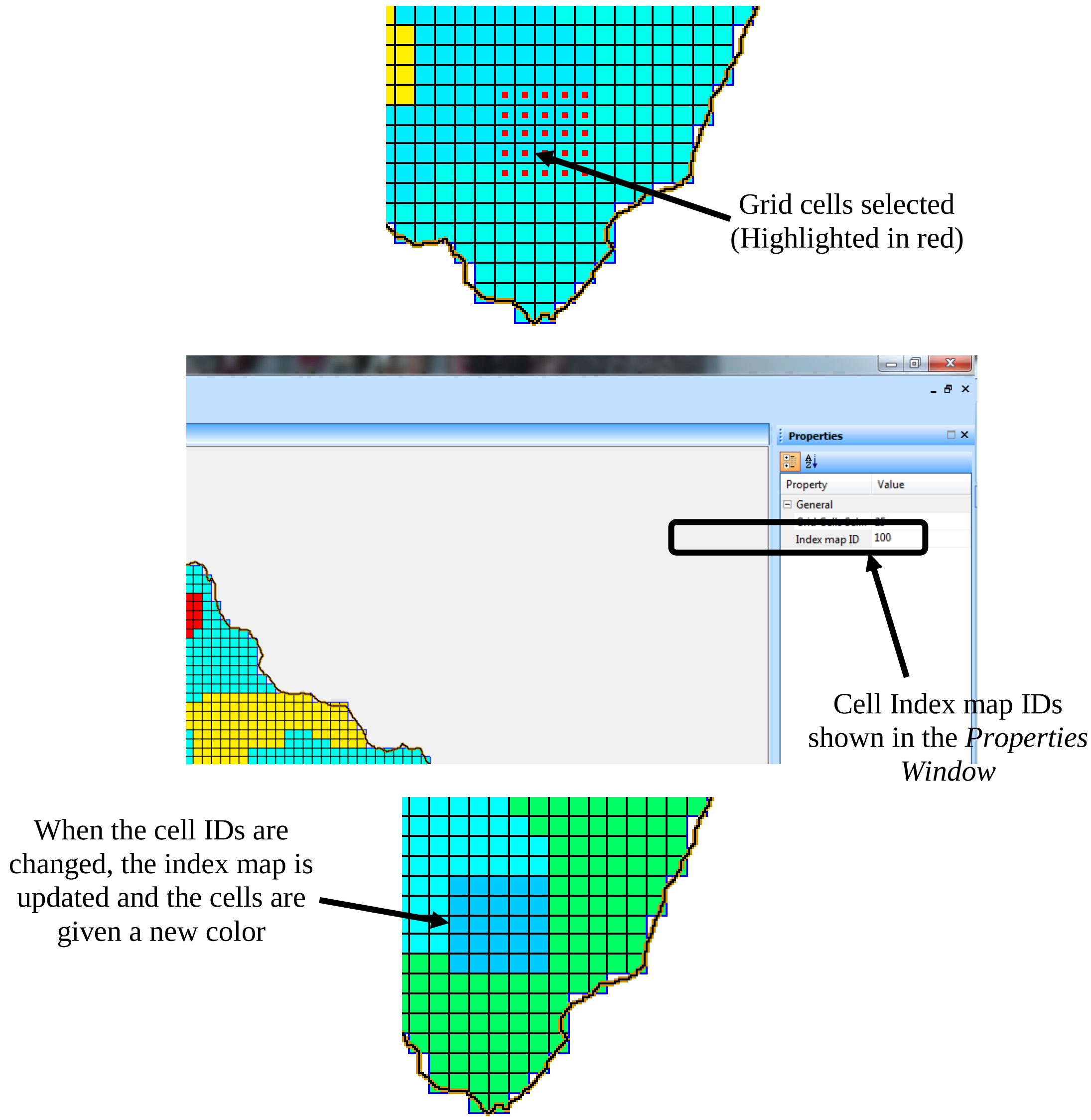
7. In the figure above, the data calculator is set to create a copy of the *Land Use* index map. Close both the *data calculator* and the *GSSHA Maps* windows by clicking on the *Done* buttons in each dialog.

3.5.2 Editing an Index Map

Sometimes you will have to modify an existing index map. Editing an index map cell means assigning it a different ID (overriding what was mapped from a coverage). For example, you have a land use index map but a certain portion of the land has been converted to a built up area. This land use change is not reflected in your land use index map. You can modify the index map for a few cells to incorporate the change.

1. In the project explorer, select the land use index map you copied in the previous step.
2. Select the *Select Grid Cell Tool*  and drag a box to select some grid cells you wish to change.
3. In the WMS properties window you can edit the IDs of the selected cells. Go ahead and assign the *Index Map ID* to be 100.

4. Changing the Index Map IDs for the selected cells will update the index map. Because these cells have new index map IDs, they will now display as a different color.



5. If the area where you want to change IDs is not rectangular, then with the *Select Grid Cell Tool* selected, you can press and hold Shift Key in the Keyboard and click on the cells you want to change.
6. Alternatively, you can select grid cells with a polygon. Select ***Edit | Select with Polygon***. Then draw the polygon to include the cells you want to select.