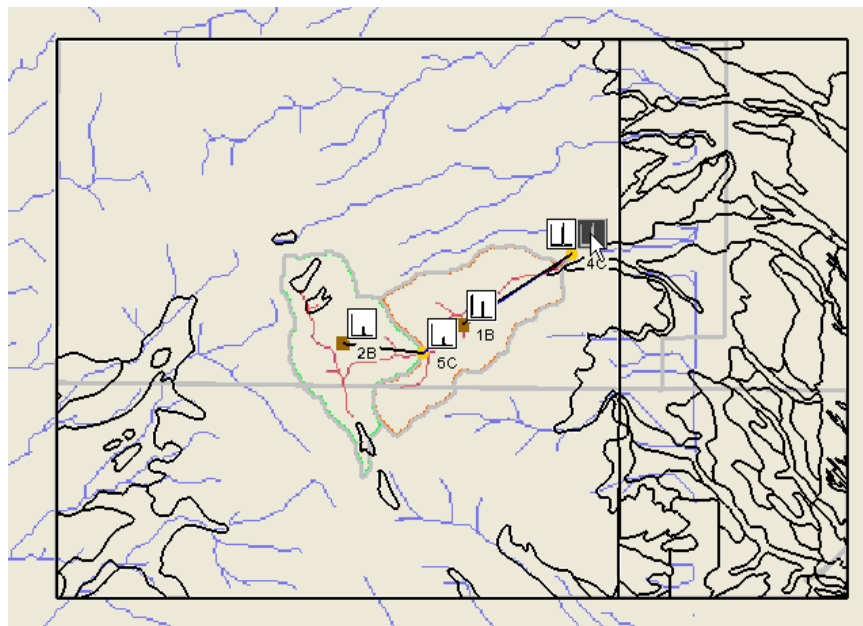


## WMS 10.1 Tutorial

# Watershed Modeling – Maricopa County: NSS and HEC-1

Build a basic watershed model for a location in Maricopa County, Arizona, USA



## Objectives

Learn to delineate a basin for an area of interest, run NSS to estimate peak flows at different recurrence intervals, set up a basic HEC-1 model, and run the HEC-1 model to compute a hydrograph.

## Prerequisite Tutorials

- Watershed Modeling – National Streamflow Statistics Program (NSS) Interface

## Required Components

- Data
- Drainage
- Map
- Hydrology

## Time

- 30–60 minutes

**AQUAVEO™**



<b>1</b>	<b>Introduction .....</b>	<b>2</b>
<b>2</b>	<b>Getting Started .....</b>	<b>2</b>
<b>3</b>	<b>Delineating the Watershed .....</b>	<b>2</b>
3.1	Using TOPAZ .....	3
3.2	Defining the Basins .....	5
<b>4</b>	<b>Building the NSS Simulation .....</b>	<b>8</b>
4.1	Creating the NSS Regions Coverage.....	8
4.2	Running NSS and Viewing Results.....	10
<b>5</b>	<b>Building the HEC-1 Simulation .....</b>	<b>11</b>
5.1	Defining Precipitation Data .....	11
5.2	Defining Land Use .....	12
5.3	Defining Soil Type .....	14
5.4	Computing Losses .....	15
5.5	Setting the Unit Hydrograph Method .....	16
5.6	Specifying the Routing Method .....	16
<b>6</b>	<b>Saving and Running the HEC-1 Simulation .....</b>	<b>17</b>
<b>7</b>	<b>Conclusion.....</b>	<b>18</b>

## 1 Introduction

---

This tutorial demonstrates how to use WMS to set up and run HEC-1 and National Streamflow Statistics (NSS) simulations. The WMS customizations for drainage studies in Maricopa County will be highlighted. This tutorial will discuss and demonstrate delineating a watershed using DEM data, building a Maricopa County NSS simulation, running NSS and viewing the results in WMS, building a Maricopa County HEC-1 simulation, and running HEC-1 and viewing the results in WMS

## 2 Getting Started

---

Starting WMS new at the beginning of each tutorial is recommended. This resets the data, display options, and other WMS settings to their defaults. To do this:


1. If necessary, launch WMS.
2. If WMS is already running, press *Ctrl-N* or select *File | New...* to ensure that the program settings are restored to their default state.
3. A dialog may appear asking to save changes. Click **No** to clear all data.


The graphics window of WMS should refresh to show an empty space.

## 3 Delineating the Watershed

---

Before building a hydrologic model such as HEC-1, it is necessary to define the watershed boundaries and calculate required parameters (such as basin area, surface slope, etc). This is done by importing a DEM and using the automatic basin delineation tools in WMS.

1. Select *File | Open*  to bring up the *Open* dialog.
2. Select “Arc/Info Grid (\*.asc;\*.grd)” from the *Files of type* drop-down.

3. Browse to the *MARICOPA\MARICOPA\TUTI\* directory and select “elevations.asc”.
4. Click **Open** to exit the *Open* dialog and bring up the *Importing ArcInfo Grid* dialog.
5. Click **OK** accept the defaults, close the *Importing ArcInfo Grid* dialog, and import the DEM.
6. Click **Display Options**  to bring up the *Display Options* dialog.
7. Select “DEM Data” from the list on the left.
8. On the *DEM* tab, turn on *DEM Contours* and click **OK** to close the *Display Options* dialog.

The DEM contours will be generated and displayed (Figure 1). In order to delineate watershed basins, compute flow directions and flow accumulations for each DEM cell.

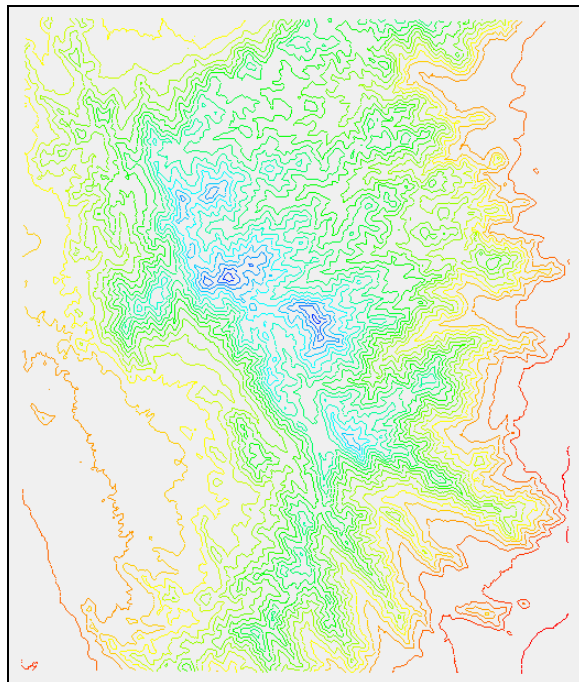




Figure 1 Imported DEM with contours visible

### 3.1 Using TOPAZ

1. Switch to the **Drainage**  module.
2. Select *DEM / Compute Flow Direction/Accumulation...* to bring up the *Flow Direction/Accumulation Run Options* dialog.
3. In the *Input/output file directory* section, select *Write files to a specified directory* and click **Browse**  to bring up the *Browse for Folder* dialog.
4. Browse to the *MARICOPA\MARICOPA\TUTI\* directory used for the tutorial and click **OK** to close the *Browse for Folder* dialog.

5. Click **OK** to close the *Flow Direction/Accumulation Run Options* dialog and open the *Units* dialog.
6. In the *Model units* section, click **Current Projection...** to bring up the *Display Projection* dialog.
7. In the *Horizontal* section, select *No Projection* and choose “Feet (U.S. Survey)” from the *Units* drop-down.
8. In the *Vertical* section, select “Feet (U.S. Survey)” from the *Units* drop-down.
9. Click **OK** to close the *Display Projection* dialog.
10. In the *Parameter units* section, select “Square miles” from the *Basin Areas* drop-down and “Feet” from the *Distances* drop-down.
11. Click **OK** to close the *Units* dialog and bring up the *Model Wrapper* dialog.
12. Once TOPAZ finishes running, turn on *Read solution on exit*.
13. Click **Close** to exit the *Model Wrapper* dialog and import the solution from TOPAZ.

Flow directions and accumulations are now automatically imported into WMS. The blue lines that appear on the screen represent anticipated stream locations (Figure 2).

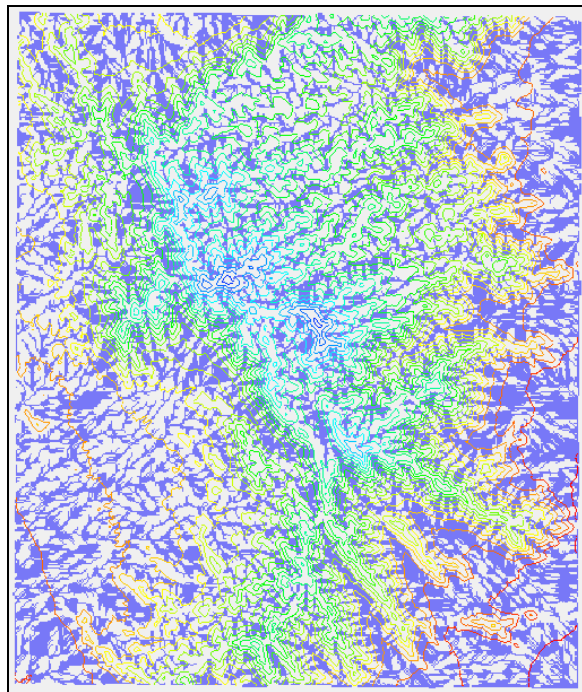


Figure 2 After initial TOPAZ run

11. Select *Display / Display Options...* to open the *Display Options* dialog.
12. Select “DEM Data” from the list on the left.
13. On the *DEM* tab, enter “0.09” as the *Min Accumulation for Display*.
14. Click **OK** to close the *Display Options* dialog.

The display should now be similar to Figure 3. Notice how adjusting the minimum accumulation changed the display. Adjusting this setting allows more or less precision in

the amount of accumulation displayed in the Main Graphics Window. It does not alter the calculated amount of accumulation in the simulation.

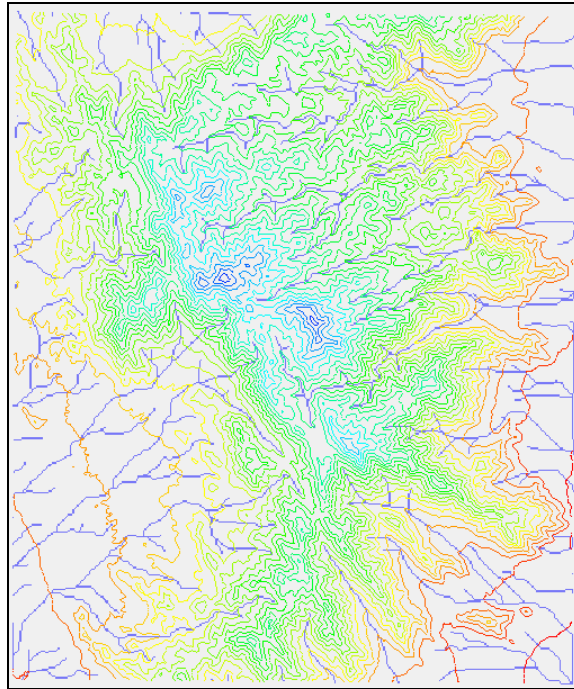



Figure 3 A higher minimum accumulation for display

### 3.2 Defining the Basins

The first step in defining basins and sub-basins is to place outlet points at the desired locations along the stream(s). WMS then creates stream arcs based on the outlet locations. Basin boundaries are then delineated based on the stream network and areas contributing to these streams.

1. Switch to the **GIS**  module.
2. Select *Data / Add Shapefile Data...* to bring up the *Select shapefile* dialog.
3. Select “Shapefiles (\*.shp)” from the *Files of type* drop-down.
4. Select “basins.shp” and click **Open** to import the shapefile and close the *Select shapefile* dialog.

This shapefile acts as a background image to help with placing the outlets (Figure 4). Notice that the shapefile contains data which extends outside the DEM contour area.

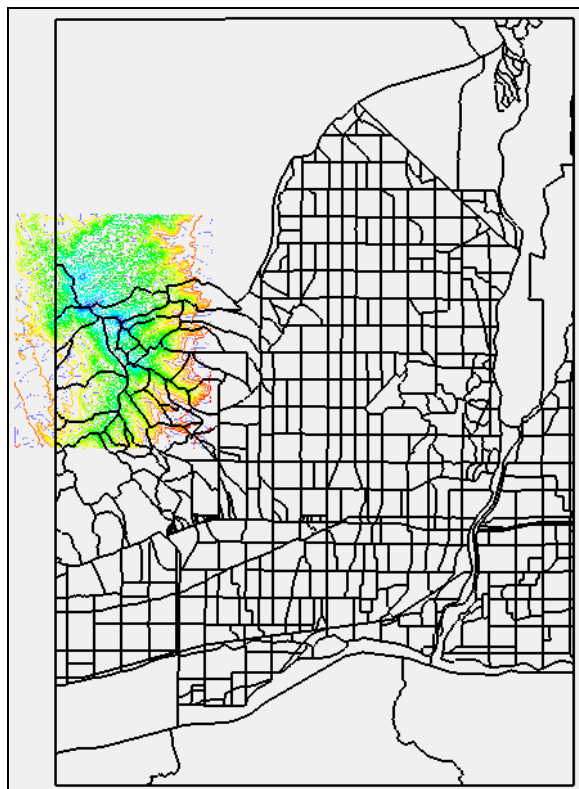



Figure 4 The shapefile contains data outside the DEM

5. Switch to the **Drainage**  module.

To aid in placing the outlet points, zoom in on a portion of the DEM. Then import a shapefile containing all of the watershed basins in Maricopa County:

6. **Zoom**  in to the area shown in Figure 5 by clicking and dragging a box.

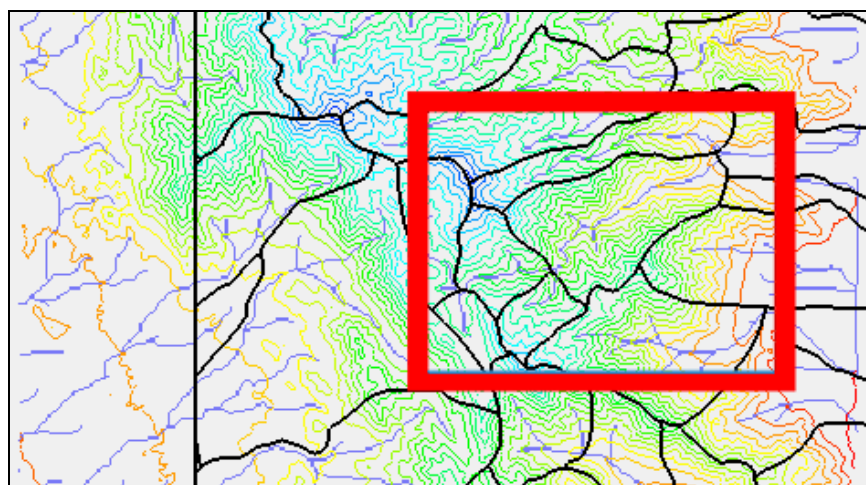





Figure 5 Zoom in to this area

In order to make it easier to place the outlets, hide the DEM contours:

7. Click **Display Options**  to open the *Display Options* dialog.
8. Select “DEM Data” from the list on the left.

9. On the *DEM* tab, turn off *DEM Contours* and click **OK** to close the *Display Options* dialog.
10. Using the **Create Outlet Point**  tool, create two outlet points as indicated in Figure 6.

Make sure the outlets are located where the stream arcs cross the drainage basin boundary arcs. **Zoom**  in if needed for accuracy.

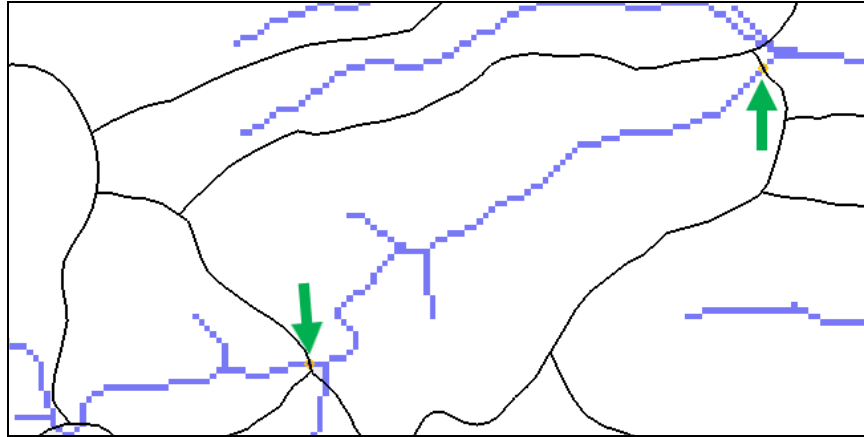


Figure 6 Locations for the two outlets

11. Select *DEM / DEM* → **Stream Arcs...** to bring up the *Stream Feature Arc Options* dialog.
12. Click **OK** to accept the default threshold value and close the *Stream Feature Arc Options* dialog.

This value is the minimum accumulation (in units of area) for creating stream arcs. In other words, for a DEM cell to be considered a stream cell, the upstream area contributing to that cell must be greater than or equal to the threshold value.

13. Select *DEM / Define Basins*.

Colored basin boundaries will be created (Figure 7).

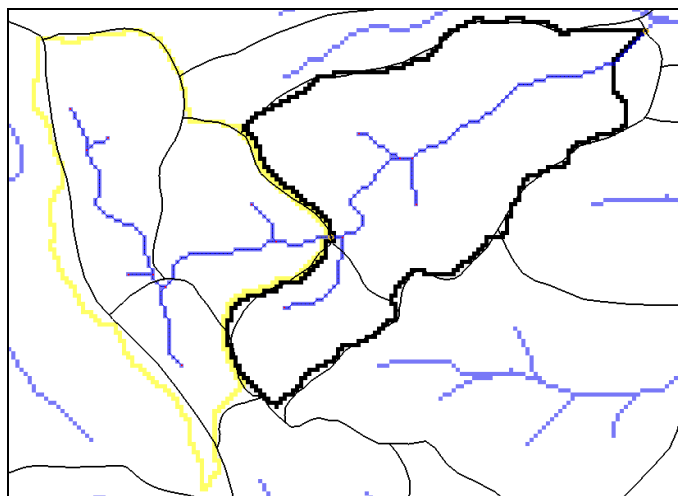


Figure 7 The two drainage basins

If only one or no basins are delineated, delete the two outlets and redo steps 11-14, making sure the outlets are located directly on stream cells.

14. Select *DEM / Basins* → **Polygons**.

15. Select *DEM / Compute Basin Data* to bring up the *Units* dialog.

16. Click **OK** to close the *Units* dialog.

A watershed has now been delineated with two sub-basins (corresponding to the two outlets) and computed basin parameters to be used in a hydrologic model (Figure 8). Notice that in some areas, the DEM generated basin boundaries diverge from those of the imported shapefile. Such variations might be attributed to different dates of production for the DEM and shapefile, and/or limitations of gridded elevation data.

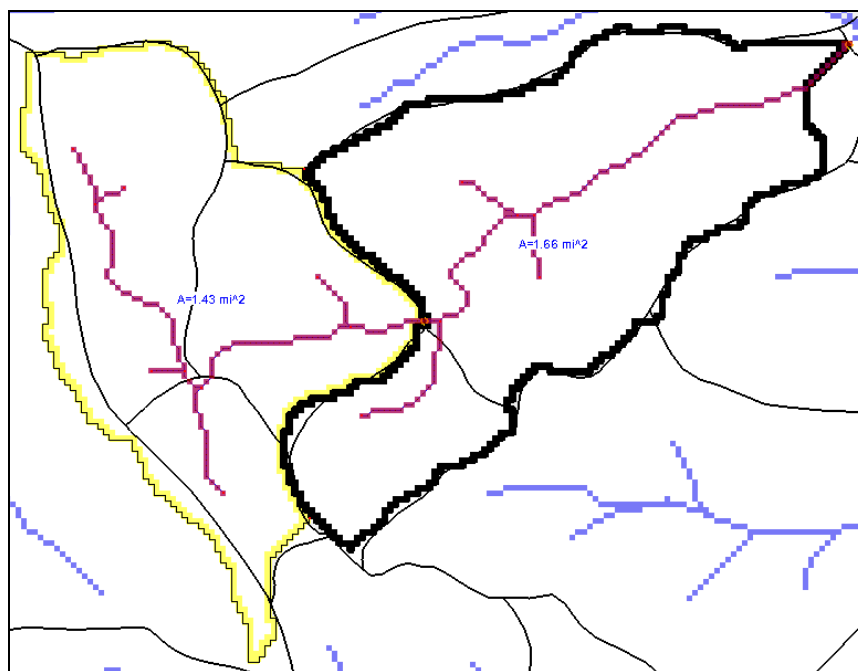


Figure 8 The two sub-basins


## 4 Building the NSS Simulation




WMS includes an interface to the National Streamflow Statistics Program (NSS). The NSS program is a compilation of all the current statewide and metropolitan area regression equations, including equations specific to Maricopa County, Arizona.

The watershed developed in the previous section will be used to run the NSS model. To begin, import a shapefile containing the NSS Regions of Arizona and map the regions to feature objects. These region polygons will be used by WMS to automatically set up the needed regression equations used by NSS.

### 4.1 Creating the NSS Regions Coverage

Before adding the new coverage, hide the shapefile to make the screen less cluttered:

1. Turn off “ basins.shp” in the Project Explorer.

2. Switch to the **Map**  module.
3. Right-click “ Coverages” in the Project Explorer and select **New Coverage** to bring up the *Properties* dialog.
4. Select “NSS Region” from the *Coverage type* drop-down and click **OK** to close the *Properties* dialog.
5. Click **Open**  to bring up the *Open* dialog.
6. Select “Shapefiles (\*.shp)” from the *Files of type* drop-down.
7. Select “arizusgs.shp” and click **Open** to exit the *Open* dialog.

The Graphics Window should appear similar to Figure 9. The DEM area should appear as a very small gray spot near the bottom border of the large center region of the map.

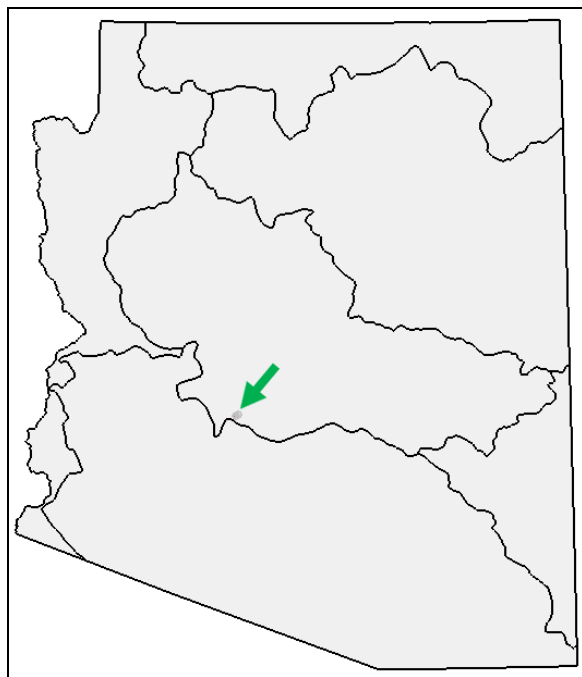





Figure 9 Location of DEM

8. Switch to the **GIS**  module.
9. Select *Mapping / Shapes* → **Feature Objects** to bring up the *GIS to Feature Objects Wizard* dialog, clicking **Yes** if asked to use all shapes.
10. Click **Next** to go to the *Step 1 of 2* page of the *GIS to Feature Objects Wizard* dialog.



Notice that the STATE and NSS\_REGION fields are automatically mapped to the correct attributes.

11. Click **Next** to go to the *Step 2 of 2* page of the *GIS to Feature Objects Wizard* dialog.
12. Click **Finish** to close the *GIS to Feature Objects Wizard* dialog.
13. Switch to the **Map**  module.

14. Using the **Select Feature Polygon**  tool, double-click on the central polygon containing the DEM (Figure 9) to bring up the NSS Region Polygon Attributes dialog.
15. In the *Select NSS Region* section, verify that “Arizona” is selected from the *State* drop-down and “Central Arizona Region 12” is selected from the *NSS Region* drop-down.
16. Click **OK** to close the *NSS Region Polygon Attributes* dialog.

The NSS Regions coverage has now been created so the NSS computations can be run.

## 4.2 Running NSS and Viewing Results

1. **Zoom**  in on the area indicated by the arrow in Figure 9 until the two sub-basins are clearly visible.
2. Switch to the **Hydrologic Modeling**  module.
3. Select “NSS” from the Model drop-down (Figure 10).

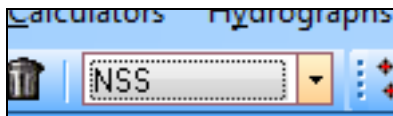




Figure 10 Model drop-down


4. Using the **Select Basins**  tool, double-click the left basin icon to bring up the *National Streamflow Statistics Method* dialog.
5. Click **Yes** when asked to use the polygons to compute region areas.
6. Click **Yes** when asked to compute mean annual precipitation values.
7. In the *Results* section, click **Compute Results**.

The peak flows for the various return periods should appear in the output spreadsheet below the button.

8. Click **Done** to close the *National Streamflow Statistics Method* dialog.
9. Using the **Select Basins**  tool, double-click the right basin icon to bring up the *National Streamflow Statistics Method* dialog.
10. Click **Yes** when asked to use the polygons to compute region areas.
11. In the *Results* section, click **Compute Results**.
12. In the output spreadsheet below the button, scroll down to and select the row with “50” in the *Recurrence [years]* column. Select the entire row, not just an individual cell.
13. Click **Compute Hydrograph...** to bring up the *NSS Hydrograph Data* dialog.
14. In the *Compute lag time* section, click **Compute Lag Time – Basin Data...** to bring up the *Basin Time Computation* dialog.
15. Select “Tulsa 100% Urban Method” from the *Method* drop-down.
16. Click **OK** to close the *Basin Time Computation* dialog.

17. Click **OK** to close the *NSS Hydrograph Data* dialog.
18. Click **Done** to exit the *National Streamflow Statistics Method* dialog.
19. Using the **Select hydrograph** tool, double-click on the hydrograph icon next to the right basin icon to bring up the *Hydrograph* dialog.

The hydrograph for the right basin should be similar to Figure 11. The peak discharge values have now been computed for both basins. Similar steps are used to create hydrographs for any basin.

20. When done reviewing the hydrograph, click the  to close the *Hydrograph* dialog.

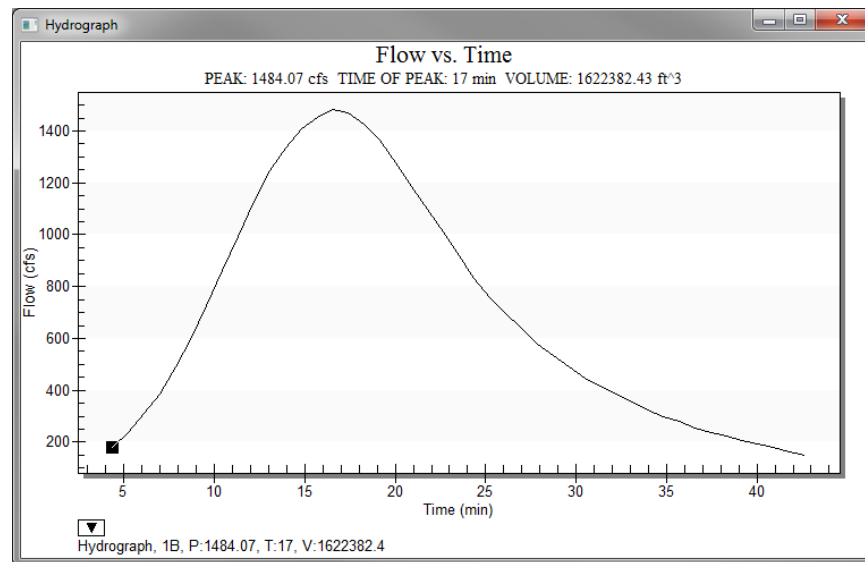


Figure 11 Hydrograph for right basin


## 5 Building the HEC-1 Simulation

Now proceed to set up the HEC-1 / Maricopa County parameters. Before running HEC-1, several required parameters need to be defined (precipitation, losses, and unit hydrograph method), plus any desired optional parameters (such as routing).

### 5.1 Defining Precipitation Data

To input the Maricopa County precipitation data, use the *HEC-1 Job Control* dialog:

1. Select “HEC-1” from the *Models* drop-down. This is drop-down previously set to “NSS”.
2. Select *HEC-1 / Job Control...* to bring up the *HEC-1 Job Control* dialog.
3. Near the bottom of the dialog, click **Initialize Maricopa County Precipitation Data** to bring up the *Maricopa County Precipitation Data Initialization* dialog.
4. Select *Basin Average (PB/PC)* and select “24-hour” from the dropdown. This is the storm duration.

5. Click **Browse**  to bring up the *Open* dialog.
6. Select “noaa10y24h” and click **Open** to exit the *Open* dialog and bring up the *Compute Rainfall* dialog.

This is a NOAA rainfall grid corresponding to a 10 year, 24-hour storm. It will be used to compute precipitation.

7. Click **OK** to accept the default *Rainfall computation cell size* and close the *Compute Rainfall* dialog.

Because the rainfall grid is approximately a 1 km grid size, no additional accuracy is achieved if the computational cell size is smaller than the default.







8. Click **OK** to close the *Maricopa County Precipitation Data Initialization* dialog.
9. Click **OK** to close the *HEC-1 Job Control* dialog.

The computed precipitation (centroid) is copied to the HEC-1 basin average parameter.

## 5.2 Defining Land Use

Along with land use and soil type data, WMS can compute Loss Data based on the Maricopa County methodology. First import land use and soil type shapefiles and convert them to feature objects. Then, using land use soil type look-up tables, compute the required Loss parameters.

To add land use data, do the following:

1. Right-click on “ Coverages” in the Project Explorer and select **New Coverage** to bring up the *Properties* dialog.
2. Select “Land Use” from the *Coverage type* drop-down.
3. Click **OK** to close the *Properties* dialog.
4. Switch to the **GIS**  module.
5. Select *Data / Add Shapefile Data...* to bring up the *Select shapefile* dialog.
6. Select “Shapefiles (\*.shp)” from the *Files of type* drop-down.
7. Select “landusewhitetanks.shp” and click **Open** to import the shapefile and close the *Select shapefile* dialog.
8. Select “ Land Use” to make it active.
9. Turn off “ NSS Region” and “ arizusgs.shp” in the Project Explorer.
10. **Frame**  the project to view the extents of the land use coverage.

The land use polygons should appear similar to Figure 12.

If there are multiple shapefiles present in the GIS Module in the Project Explorer, it is important to hide all shapefiles that should not be mapped to feature objects. Only visible shapefiles can be selected with the Select Shapes tool, so only selected visible shapefiles will be mapped to feature objects. Note that if no shapefiles are selected, WMS will map all of the shapes from all of the shapefiles, whether visible and hidden.

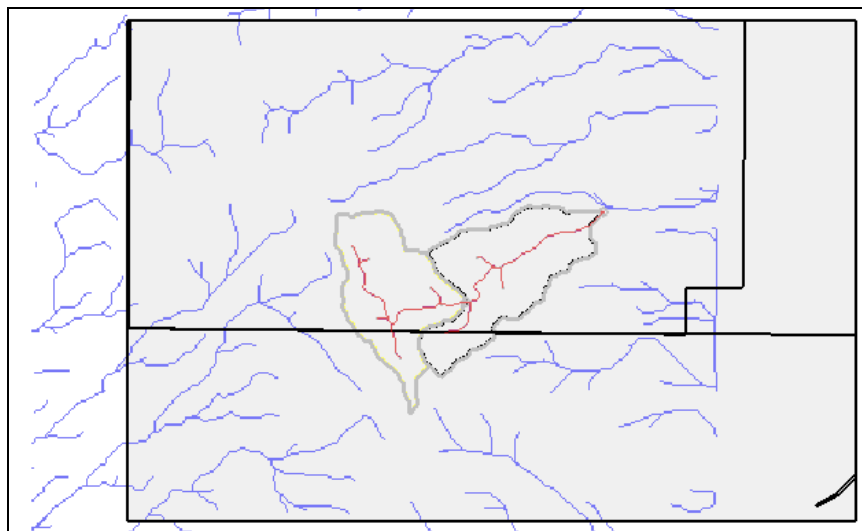









Figure 12 Land use polygons encompassing the watershed

11. Using the **Select Shapes**  tool, drag a selection box around the two sub-basins to select the land use polygons encompassing the watershed.
12. Select *Mapping / Shapes* → **Feature Objects** to bring up the *GIS to Feature Objects Wizard* dialog.
13. Click **Next** to go to the *Step 1 of 2* page of the *GIS to Feature Objects Wizard* dialog.
14. In the *Mapping* row of the *LDUSE\_LID* column, select “Land use” from the drop-down.
15. Click **Next** to go to the *Step 2 of 2* page of the *GIS to Feature Objects Wizard* dialog.
16. Click **Finish** to close the *GIS to Feature Objects Wizard* dialog.
17. Switch to the **Map**  module.
18. Select the **Select Feature Polygon**  tool and select *Feature Objects / Attributes...* to bring up the *Land use mapping* dialog.
19. In the *Import land use attribute file* section, select “Green-Ampt Parameter file” from the *Import file type* drop-down.
20. Click the **Import file** button to bring up the *Open* dialog.
21. Click **OK** when advised the new table will overwrite the current land use table.
22. Select “Land/Soil Table File (\*.txt)” from the *Files of type* drop-down.
23. Select “landusemagtable.txt” and click **Open** to import the file and exit the *Open* dialog.
24. In the *Display parameters* section, turn off *SCS CN's* and turn on *Green-Ampt*.
25. Click **Apply** to close the *Land use mapping* dialog.

### 5.3 Defining Soil Type

To add soil type data, do the following:

1. Right-click on “ Coverages” in the Project Explorer and select **New Coverage** to bring up the *Properties* dialog.
2. Select “Soil Type” from the *Coverage type* drop-down.
3. Click **OK** to close the *Properties* dialog.
4. Switch to the **GIS**  module.
5. Select *Data / Add Shapefile Data...* to open the *Select shapefile* dialog.
6. Select “Shapefiles (\*.shp)” from the *Files of type* drop-down.
7. Select “soilwhitetanks.shp” and click **Open** to import the shapefile and close the *Select shapefile* dialog.
8. Turn off “ landusewhitetanks.shp” in the Project Explorer.
9. Using the **Select Shapes**  tool, drag a selection box around the two sub-basins (not the entire soil type coverage).

This selects some of the soil type polygons, as shown in Figure 13.

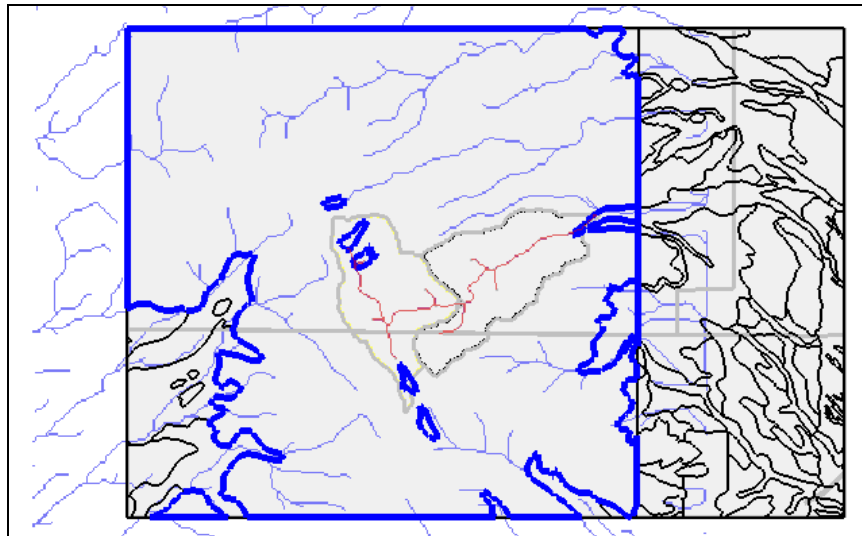




Figure 13 Selected soil type polygons highlighted


10. Select *Mapping / Shapes* → **Feature Objects** to bring up the *GIS to Feature Objects Wizard* dialog.
11. Click **Next** to go to the *Step 1 of 2* page of the *GIS to Feature Objects Wizard* dialog.
12. On the *Mapping* row in the *SLTYP\_LID* column, select “SCS soil type” from the drop-down.
13. Click **Next** to go to the *Step 2 of 2* page of the *GIS to Feature Objects Wizard* dialog.
14. Click **Finish** to close the *GIS to Feature Objects Wizard* dialog.

15. Click anywhere outside of the soil type shapefile's boundaries to deselect the polygons
16. Switch to the **Map**  module.
17. Using the **Select Feature Polygon**  tool, double-click inside one of the soil polygons to bring up the *Soil type mapping* dialog.
18. In the *Import soil attribute file* section, select "Green-Ampt Parameter file" from the *Import file type* drop-down and click **Import file** to bring up the *Open* dialog.
19. Click **OK** when advised the new table will overwrite the current soil type table.
20. Select "Land/Soil Table File (\*.txt)" from the *Files of type* drop-down.
21. Select "soiltable.txt" and click **Open** to import the file and exit the *Open* dialog.
22. In the *Display parameters* section, turn off *SCS soil type* and turn on *Green-Ampt*.
23. Click **Apply** to close the *Soil type mapping* dialog.

## 5.4 Computing Losses

---


Once land use and soil type coverages are defined, compute losses by doing the following:

1. Switch to the **Hydrologic Modeling**  module.
2. Select *Calculators / Compute GIS Attributes...* to bring up the *Compute GIS Attributes* dialog.
3. In the *Computation* section, select "Green-Ampt parameters" from the drop-down.
4. In the *Mapping* section, select *Land use mapping* and make sure the mapping table appears in the text area below that.
5. Repeat step 4 for *Soil type mapping*.

If the tables are not visible, the **Import** button can be used to import them.

6. Click **OK** to close the *Compute GIS Attributes* dialog.
7. If a dialog appears advising that some soil type polygons have a value of "0.0", click **OK**.

Based on the land use and soil type data, WMS now computes all the HEC-1 loss parameters. Now verify that the values were copied to HEC-1:

1. Using the **Select basin**  tool, double-click on the left basin icon to bring up the *Edit HEC-1 Parameters* dialog.
2. In the *Basin HEC-1 Cards* section, click **Precipitation...** to bring up the *HEC-1 Precipitation* dialog.
3. Notice that the rainfall has been mapped to the *Average precipitation* field. Click **OK** to close the *HEC-1 Precipitation* dialog.
4. Click **Loss Method...** to bring up the *HEC-1 Loss Methods* dialog.

5. Notice that *Green ampt (LG)* is selected on the left, and that the Green-Ampt values computed from the land use and soil coverages are listed on the right.
6. Click **OK** to close the *HEC-1 Loss Methods* dialog.

## 5.5 Setting the Unit Hydrograph Method

---

For this HEC-1 model, use the Clark Method to develop the unit hydrographs for both sub-basins. To calculate these parameters:

1. Click **Unit Hydrograph Method...** to bring up the *HEC-1 Unit Hydrograph Methods* dialog.
2. Select *Clark (UC)* and click **Compute Tc and R – Maricopa County** to bring up the *Maricopa County Clark Tc and R* dialog.
3. Select *Resistance coefficient roughness type* and select “B-Moderately Low Roughness” from the drop-down.
4. Click **Compute Tc and R** and click **OK** to close the *Maricopa County Clark Tc and R* dialog.

Notice that the values from the *Maricopa County Clark Tc and R* dialog transferred to the corresponding fields in the *Clark (UC)* section (the *Tc* in minutes has been translated into decimal hours).

5. Click **OK** to close the *HEC-1 Unit Hydrograph Methods* dialog.
6. Click **Next Hydrograph Station** → twice to select the sub-basin on the right.
7. Click **Unit Hydrograph Method...** to bring up the *HEC-1 Unit Hydrograph Methods* dialog.
8. Select *Clark (UC)* and click **Compute Tc and R – Maricopa County** to bring up the *Maricopa County Clark Tc and R* dialog.
9. Click **OK** to the message stating that the slope is greater than allowed.
10. Select *Resistance coefficient roughness type* and select “C-Moderately High Roughness” from the drop-down.
11. Click **Compute Tc and R** and click **OK** to close the *Maricopa County Clark Tc and R* dialog.
12. Click **OK** to close the *HEC-1 Unit Hydrograph Methods* dialog.

## 5.6 Specifying the Routing Method

---

To simulate routing from the upstream basin (on the left) to the downstream basin, use the Muskingum routing method.


1. Click ← **Previous Hydrograph Station** until outlet “4C” is selected. The *HEC-1 File Output* field at the bottom of the dialog will show “4C” to the right of “CNAME” (Figure 14).
2. In the *Routing HEC-1 Cards* section, click **Routing Data...** to bring up the *HEC-1 Routing Data* dialog.

HEC-1 File Output - select to edit					
KK	4R	CNAME	4C		
KO	0	0	0.0	0	22
RN	4R				

Figure 14 Outlet 4C selected

3. In the second section, select “Muskingum (RM)” from the *Routing type* drop-down and click **Compute NSTPS** to bring up the *Compute NSTPS* dialog.
4. Select *From Channel Velocity Estimate* and enter “4.0” as the *Channel velocity estimate*.
5. Click **OK** to close the *Compute NSTPS* dialog.
6. Click **OK** to close the *HEC-1 Routing Data* dialog.
7. Click **Done** to close the *Edit HEC-1 Parameters* dialog.

## 6 Saving and Running the HEC-1 Simulation

1. Select *HEC-1 / Run Simulation...* to bring up the *HEC-1 Run Options* dialog.
2. Click **Browse**  to bring up the *Select HEC-1 Input File* dialog.
3. Enter “Maricopa.hc1” as the *File name* and click **Save**.

This specifies the file name to be saved, but does not actually save it.

4. Turn on *Save file before run* and click **OK** to close the *HEC-1 Run Options* dialog and bring up the *Model Wrapper* dialog.
5. Once the model is finished, turn on *Read solution on exit* and click **Close** to import the solution and close the *Model Wrapper* dialog.

Notice that HEC-1 has computed basin and outlet hydrographs (Figure 15).

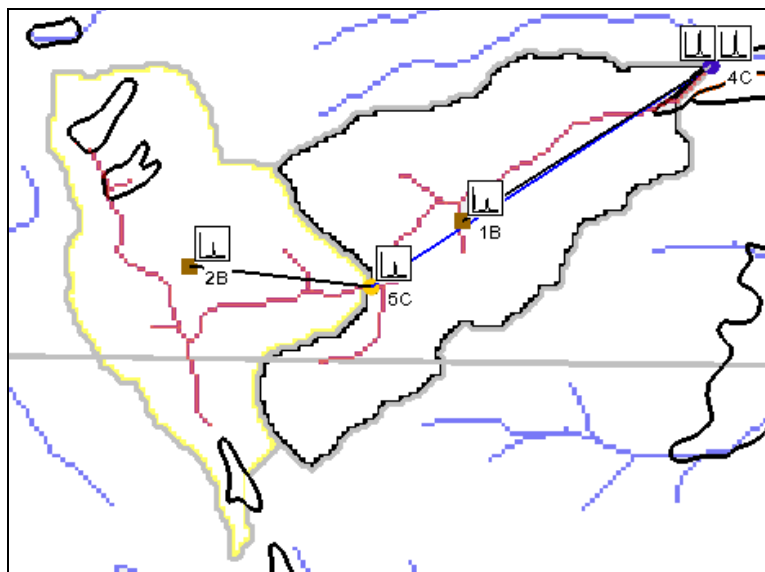




Figure 15 Computed basin and outlet hydrographs

Now view the hydrographs with the visualization tools in WMS:

6. Using the **Select hydrograph**  tool, double-click on the hydrograph icon for the left basin to bring up the *Hydrograph* dialog.
7. When finished reviewing the hydrograph, click  to close the *Hydrograph* dialog.
8. Repeat steps 6–7 to review the hydrographs for the other basin or for either outlet.

---

## 7 Conclusion

---

This concludes “Watershed Modeling – Maricopa County: NSS and HEC-1” tutorial. Key topics discussed and demonstrated include:

- Delineating a watershed using DEM data
- Building a Maricopa County NSS simulation
- Running NSS and viewing the results in WMS
- Building a Maricopa County HEC-1 simulation
- Running HEC-1 and viewing the results in WMS