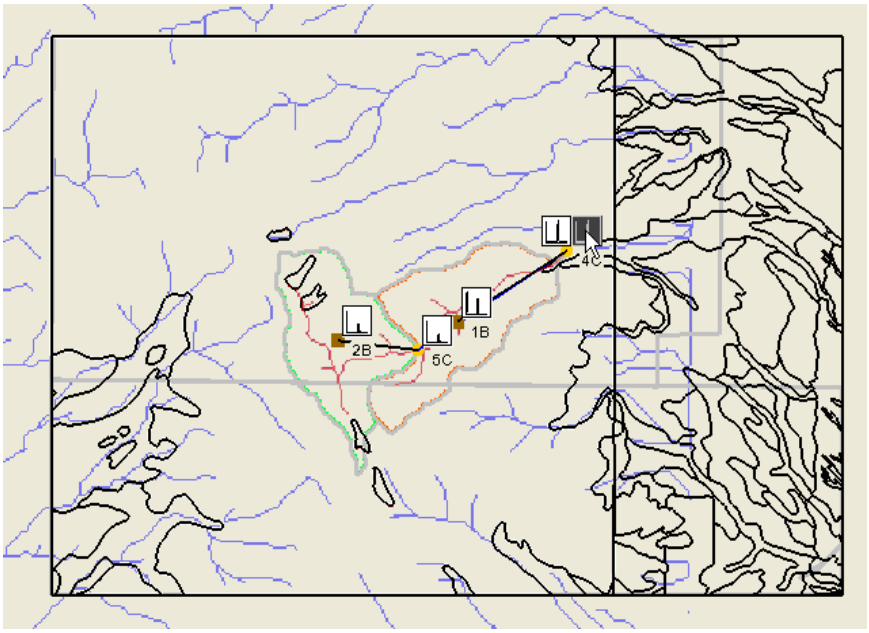
WMS

v. 9.0

WMS 9.0 Tutorial

Watershed Modeling – Maricopa County: NSS and HEC-1

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



Objectives

Delineate a basin for an area of interest and run NSS to estimate peak flows at different recurrence intervals. Also, set up a basic HEC-1 model and run this 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™

1 Contents

2 Objectives

In this exercise, you will learn how to use WMS to set up and run HEC-1 and National Flood Frequency (NFF) simulations. The WMS customizations for drainage studies in Maricopa County will be highlighted. By following these exercises, you will learn how to:

1. Delineate a watershed using DEM data
2. Build a Maricopa County NFF simulation
3. Run NFF and view the results in WMS
4. Build a Maricopa County HEC-1 simulation
5. Run HEC-1 and view the results in WMS

3 Delineating the Watershed

Before building a hydrologic model such as HEC-1, we need 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. Close all instances of WMS
2. Open WMS
3. Select ***File | Open***
4. Locate the ***Maricopa\tut1*** folder in your tutorial files. If you have used default installation settings in WMS, the tutorial files will be located in ***\My documents\WMS 9.0\Tutorials\.***
5. Open *"elevations.asc"*

6. Select *OK* to import the DEM
7. The DEM contours will be generated and displayed. In order to delineate watershed basins, we need to compute flow directions and flow accumulations for each DEM cell.

3.1 Using TOPAZ

1. Switch to the *Drainage* module
2. Select ***DEM | Compute TOPAZ Flow Data***
3. Change the input/output location if desired by clicking the browse button
4. Select *OK*
5. Select the *Current Projection...* button
6. Change both Horizontal and Vertical units to *U.S. Survey Feet*
7. Select *OK*
8. Select *OK*
9. Select *Close* once TOPAZ finishes running (you may have to wait a few seconds to a minute or so)

Flow directions and accumulations are now automatically read into WMS. The blue lines that appear on the screen represent anticipated stream locations.

10. Select ***Display | Display Options***
11. Choose *DEM Data* and set the *Min Accumulation for Display* to 0.09
12. Select *OK*

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). Then, WMS will create stream arcs based on the outlet locations. Finally, basin boundaries are delineated based on the stream network and areas contributing to these streams.

1. Switch to the *GIS* module
2. Select ***Data | Add Shapefile Data***
3. Open *"basins.shp"*. This shapefile will act as a background image and help us to place the outlets.
4. Switch to the *Drainage* module

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

5. Select the *Zoom* tool and draw a box around the area indicated by the rectangle in Figure 3-1

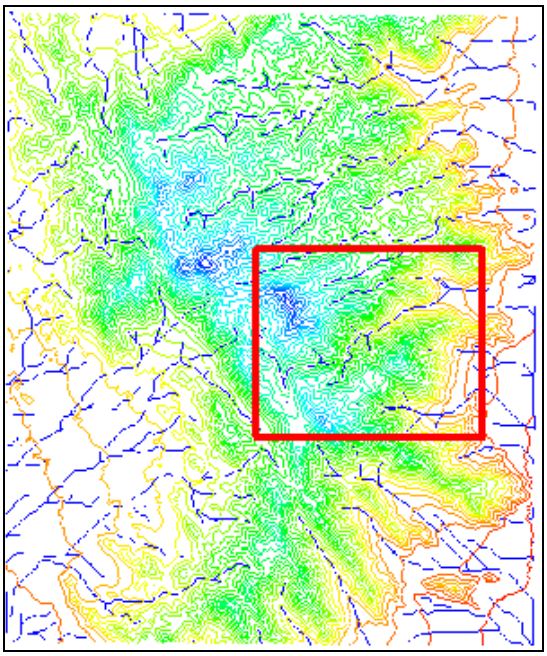


Figure 3-1: Zoom in on the area in rectangle.

In order to make the screen less busy, we will hide the DEM contours. This will make it easier for us to place the outlets:

6. Select ***Display | Display Options*** 
7. Choose *DEM Data* and toggle off the *DEM Contours* toggle box
8. Select *OK*
9. Select the *Create Outlet Point* tool 
10. Create a point in each of the two locations shown in Figure 3-2. Be sure to place each point directly on a stream (zoom in if you need to).

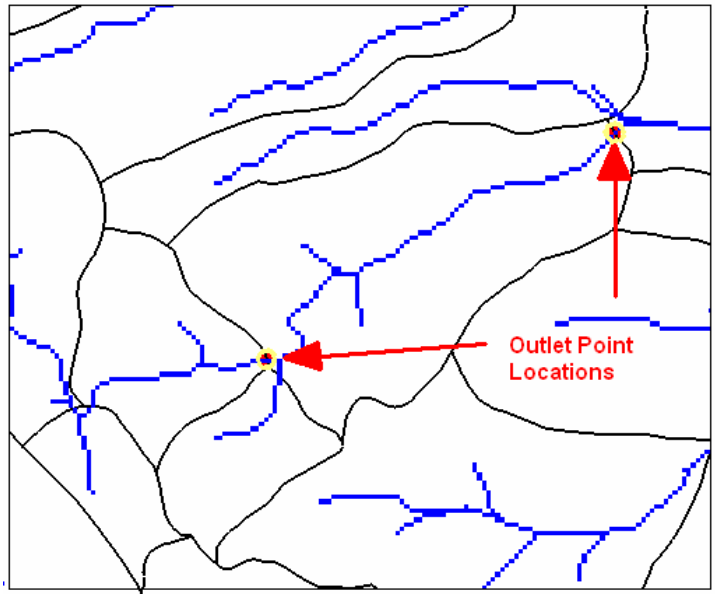


Figure 3-2: Approximate locations for placing the two outlets

11. Select ***DEM | DEM -> Stream Arcs***
12. Select *OK* to accept the default threshold value. 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 *upst*ream area contributing to that cell must be greater than or equal to the threshold value.
13. Select ***DEM | Define Basins***
14. At this point you should see the creation of colored basin boundaries. If you see only one or no basins delineated, go back and verify that both outlet points are located directly on a stream cell (use the *Zoom* tool) and repeat the steps again.
15. Select ***DEM | Basins -> Polygons***
16. Select ***DEM | Compute Basin Data***
17. Select *OK*

You have now delineated a watershed with two sub basins (corresponding to two outlets) and computed basin parameters to be used in a hydrologic model. You'll notice that in some areas, the DEM generated basin boundaries diverge from those of the shapefile. Such variations might be attributed to different dates of production for the DEM and shapefile, and/or limitations of gridded elevation data.

4 Building the NFF Simulation

WMS includes an interface to the National Flood Frequency Program (NFF). The NFF program is a compilation of all the current statewide and metropolitan area regression equations, including equations specific to Maricopa County, AZ.

We will be using the watershed developed in the previous section to run the NFF model. If you haven't already done so, copy the files *"evaporationgrid"* and *"MeanAnnualRainAsciiGrid"* to the same directory as your WMS executable.

To begin, we will import a shapefile containing the NFF 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 NFF.

4.1 Creating the NFF Regions Coverage

Before adding another coverage, we will hide the *basins.shp* file to make the screen less cluttered:

1. Toggle off the check box next to *basins.shp*
2. Switch to the *Map* module 
3. Create a new coverage by right-clicking on the Coverages folder in the Project Explorer and selecting ***New Coverage***
4. Change the Coverage Type to *NFF Region*
5. Select *OK*
6. Select ***File | Open*** 

7. Open *"arizusgs.shp"*
8. Switch to the *GIS* module
9. Select ***Mapping | Shapes -> Feature Objects***
10. Select *Yes*
11. Select *Next >*
12. Notice that the STATE and NFF_REGION fields are automatically mapped to the correct attributes
13. Select *Next >*
14. Select *Finish*
15. Switch to the *Map* module
16. Choose the *Select Feature Polygon* tool
17. Double-click on the polygon in the center, as shown in Figure 4-1, to verify that the State is Arizona and the Region is Central Arizona Region 12

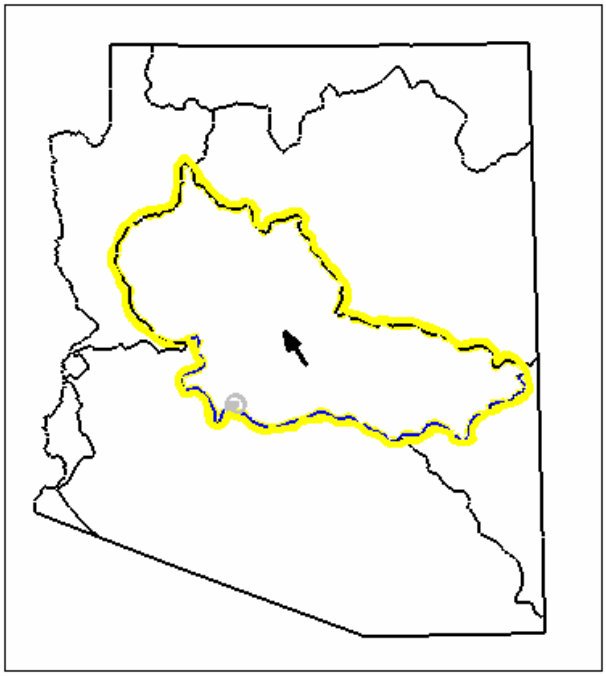


Figure 4-1: Double-click on the highlighted polygon to view its attributes

18. Select *OK*

You have now created the NFF Regions coverage and can proceed to run the NFF computations.

4.2 Running NFF and Viewing Results

1. Select the *Zoom* tool and zoom in on the two sub basins, which currently appear as small gray circles, until they are distinguishable
2. Switch to the *Hydrologic Modeling* module

3. Change the Model list box (at the mid-top of the window) to *NFF*
4. Choose the *Select Basins* tool 
5. Double-click the basin icon on the left, labeled 2B
6. Select *Yes* when asked if you want WMS to compute region areas, annual rainfall, and evaporation
7. Choose the *Compute Results* button

WMS displays the peak flows for the various return periods in the window at the bottom of the dialog.

8. Select *Done*
9. Double-click the basin icon on the right, labeled 1B
10. Select *Yes*
11. Choose the *Compute Results* button
12. In the output window at the bottom, click on the line for Recurrence [years] = 50
13. Choose the *Compute Hydrograph* button
14. Choose the *Compute Lag Time – Basin Data* button
15. Change the method to *Tulsa 100% Urban Method*
16. Select *OK* twice
17. Select *Done* to exit the NFF dialog
18. Double-click on the hydrograph icon 

You have now computed peak discharge values for both basins and learned how to generate a hydrograph for any of the design storms using the NFF interface of WMS

19. Close the hydrograph plot window

5 Building the HEC-1 Simulation

Now, we will proceed to set up the HEC-1 / Maricopa County parameters. Before we can run HEC-1, we need to define several required parameters, plus any optional parameters that we desire. We will define the following parameters in this portion of the exercise:

- Precipitation
- Losses
- Unit Hydrograph Method
- Routing (optional)

5.1 Precipitation Data

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

1. Make sure the Models drop-down field at the top of the interface is set to *HEC-1*
2. Select ***HEC-1 | Job Control***
3. Choose the *Initialize Maricopa County Precipitation Data* button
4. Choose the *Basin A*verage option and select *24-hour* (storm duration) from the drop-down list
5. Click on the Browse button  to select a rainfall grid to read in and use to compute precipitation
6. Open the file named *"noaa10y24h"*. This is a NOAA rainfall grid corresponding to a 10 year, 24-hour storm
7. Select *OK* to accept the default Rainfall computational cell size. 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. Select *OK* to close both dialogs

The computed precipitation (centroid) is copied to the Basin Average parameter of HEC-1

5.2 Computing Losses

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

Adding Land Use Data

1. Right-click on the Coverages folder in the Project Explorer
2. Select ***New Coverage*** from the pop-up menu
3. Change the Coverage Type to *Land Use*
4. Select *OK*
5. Switch to the *GIS* module 
6. Select ***Data | Add Shapefile Data***
7. Open *"landusewhitetanks.shp"*
8. Hide the NFF Region coverage by un-checking its box in the Project Explorer
9. Click the *Frame* macro  in order to view the extents of the land use coverage.
10. Hide *arizusgs.shp* by un-checking its box in the Project Explorer, if you have not already done so

If there are multiple shapefiles present in the Project Explorer of the GIS Module, then it is important to hide any and all shapefiles that you do not wish to map to feature objects. Since only visible shapefiles can be selected with the Select Shapes tool, you can hide all shapefiles but the one you are interested in, and then select the shapes you want to map to

feature objects. Note, however, that if you do not use the Select Shapes tool to select the shapes that you want to map, then WMS will map all of the shapes from all of the shapefiles in memory, visible and hidden.

11. Select the *Select Shapes* tool 
12. Draw a selection box around the two sub-basins, thereby selecting the land use polygons that cover your watershed
13. Select ***Mapping | Shapes -> Feature Objects***
14. Select *Next*
15. Find the column labeled LDUSE_LID and set its Mapping to *Land use*
16. Select *Next*
17. Select *Finish*
18. Switch to the *Map* module 
19. Make sure the Land Use coverage is still the active coverage
20. Choose the *Select Feature Polygon* tool 
21. Select ***Feature Objects | Attributes***
22. Under the Import land use attribute file section, choose *Green-Ampt Parameter file*
23. Click the *Import file* button
24. Select *OK* to accept the warning message
25. Open *"landusemagtable.tbl"*
26. Toggle off the *Display SCS CN's* check box and toggle on the *Display Green-Ampt Parameters* box
27. Select *Apply*

Adding Soil Type Data

1. Right-click on the Coverages folder in the Project Explorer and select ***New Coverage***
2. Change the Coverage type to *Soil Type*
3. Select *OK*
4. Switch to the *GIS* module 
5. Select ***Data | Add Shapefile Data***
6. Open *"soilwhitetanks.shp"*
7. Hide *landusewhitetanks.shp* by un-checking its box in the Project Explorer

If there are multiple shapefiles present in the Project Explorer of the GIS Module, then it is important to hide any and all shapefiles that you do not wish to map to feature objects. Since only visible shapefiles can be selected with the Select Shapes tool, you can hide all shapefiles but the one you are interested in, and then select the shapes you want to map to feature objects. Note, however, that if you do not use the Select Shapes tool to select the

shapes that you want to map, then WMS will map all of the shapes from all of the shapefiles in memory, visible and hidden.

8. Select the *Select Shapes* tool
9. Draw a selection box containing the two sub basins
10. Select ***Mapping | Shapes -> Feature Objects***
11. Select *Next*
12. Find the column labeled SLTYP_LID and change its Mapping field to *SCS soil type*
13. Select *Nex*t
14. Select *Finish*
15. Click anywhere outside of the soil type shapefile's boundaries to clear the selected polygons
16. Switch to the *Map* module
17. Choose the *Select Feature Polygon* tool
18. Double-click inside one of the soil polygons
19. Change the Import file type list box to *Green-Ampt Parameter file*
20. Click the *Import file* button
21. Select *OK* to accept the warning dialog
22. Open *"soiltable.tbl"*
23. Turn off the *Display of SCS soil type* box, and turn on the *Display Green-Ampt parameters* box
24. Select *Apply*

Computing Losses

With the land use and soil type coverages defined, we are ready to compute Losses. To do this:

1. Switch to the *Hydrologic Modeling* module
2. Select ***Calculators | Compute GIS Attributes***
3. Change the Computation list box to Green-Ampt parameters
4. Make sure that you have read in both the Land use mapping and Soil type mapping tables. Choose the Land use mapping and Soil type mapping options to verify this, and click the *Import* button to load them if you have not done so already.
5. Select *OK*

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

6. Double-click on either of the basin icons (brown box)

7. Click on the *Precipitation* button to verify that the rainfall has been properly mapped
8. Select *OK*
9. Click the *Loss Method* button and view the Green-Ampt values computed from the land use and soil coverages.
10. Select *OK*
11. Select *Done*

5.3 Setting the Unit Hydrograph Method

For this HEC-1 model, we will use the Clark Method to develop the Unit Hydrograph for both sub basins. To calculate these parameters:

1. Double-click on the basin icon (brown square) for the basin on the left
2. Click the *Unit Hydrograph Method* button
3. Choose the *Clark (UC)* option
4. Click the *Compute Tc and R – Maricopa County* button
5. Set the Resistance coefficient roughness type to *B-Moderately Low Roughness*
6. Click the *Compute Tc and R* button
7. Select *OK*
8. Select *OK* again to return to the *Edit HEC-1 Parameters* dialog
9. Click the *Next Hydrograph Station* -> button twice to select the sub basin on the right
10. Click the *Unit Hydrograph Method* button
11. Choose the *Clark (UC)* option
12. Click the *Compute Tc and R – Maricopa County* button
13. Select *OK* to the message stating that the slope is greater than allowed, and will be reset to 313 ft/mile
14. Change the Resistance coefficient roughness type to *C-Moderately High Roughness*
15. Click the *Compute Tc and R* button
16. Select *OK*
17. Select *OK*

5.4 Specifying the Routing Method

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

1. While still in the Edit HEC-1 Parameters dialog, click the <- *Previous Hydrograph Station* button to select Outlet 4C

2. Click the *Routing Data* button
3. Choose the *Muskingum (RM)* option
4. Click the *Compute NSTPS* button
5. Choose the *From Channel Velocity Estimat*e option
6. Enter 4 for the velocity estimate
7. Select *OK* two times to return to the main HEC-1 dialog
8. Select *Done* to close the HEC-1 dialog

6 Save and Run the HEC-1 Simulation

1. Select ***HEC-1 | Run Simulation***
2. Click the browse button  next to the Input File
3. For the file name enter "*Maricopa*" and click *Save* (this specifies the file name but does not actually save it)
4. Verify that the Save file before run is toggled on
5. Select OK
6. Select *Close* when the HEC-1 simulation finishes

Now that HEC-1 computed basin and outlet hydrographs, we can view the hydrographs with the visualization tools in WMS. To view the computed hydrographs:

7. Click on any of the hydrograph icons 
8. Select ***Display | Open Hydrograph Plot***. Alternatively, you can double-click on a hydrograph icon to view the hydrograph plot.
9. Close the hydrograph plot window

