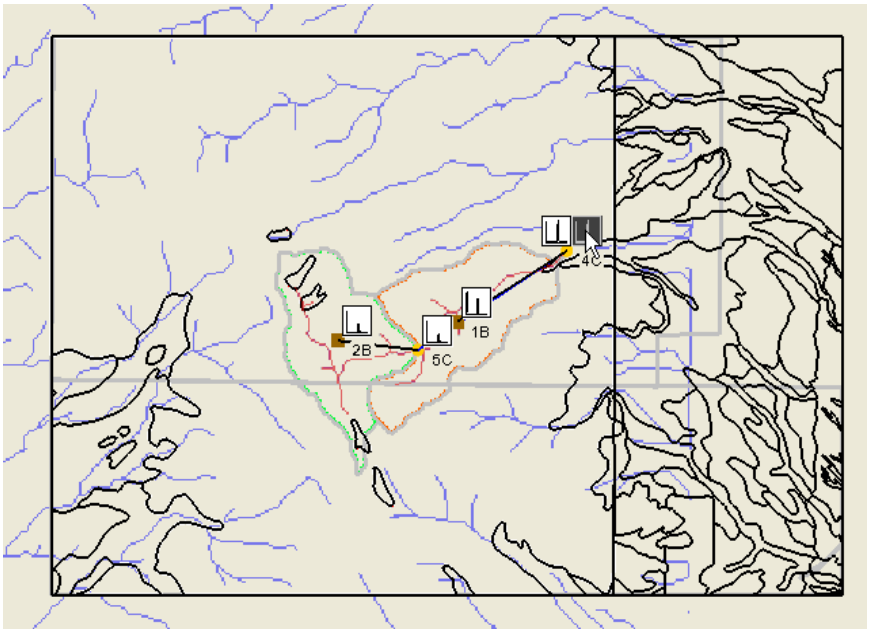
WMS

v. 10.0

WMS 10.0 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 Objectives 2
2 Delineating the Watershed 2
2.1 Using TOPAZ 3
2.2 Defining the Basins 3
3 Building the NSS Simulation 5
3.1 Creating the NSS Regions Coverage 6
3.2 Running NSS and Viewing Results 7
4 Building the HEC-1 Simulation 8
4.1 Precipitation Data 8
4.2 Computing Losses 9
4.3 Setting the Unit Hydrograph Method 12
4.4 Specifying the Routing Method 12
5 Save and Run the HEC-1 Simulation 13
6 Conclusion 13

1 Objectives

In this exercise, users will learn 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. By following these exercises, users will learn how to:

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

2 Delineating the Watershed

Before building a hydrologic model such as HEC-1, users 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. Open WMS. If WMS is already open select *File* | **New**. If asked to save data or changes, select **No**.
2. Select *File* | **Open**  .
3. In the *Open* dialog, locate the "Maricopa\tut1" folder in the files for this tutorial. If needed, download the tutorial files from www.aquaveo.com.
4. Select "elevations.asc" and click **Open**.
5. The *Importing ArcInfo Grid* dialog will open. Leave the default settings and select **OK** to import the DEM.

The DEM contours will be generated and displayed. In order to delineate watershed basins, users need to compute flow directions and flow accumulations for each DEM cell.

2.1 Using TOPAZ

1. Switch to the **Drainage** module.
2. Select *DEM* | **Compute Flow Direction/Accumulation...**
3. The *Flow Direction/Accumulation Run Options* dialog will open. Change the input/output location if desired by clicking the browse button.
4. Select **OK**.
5. The *Units* dialog will appear. Select the **Current Projection...** button.
6. In the *Display Projection* dialog, change both *Horizontal* and *Vertical units* to "U.S. Survey Feet" by selecting that option from the drop down menus.
7. Select **OK** to close the *Display Projection* dialog.
8. Look under *Parameter Units* and make sure *Basin Areas* is set to "Square miles" and *Distances* is set to "Feet".
9. Select **OK** to close the *Units* dialog.
10. The *Model Wrapper* dialog will open and TOPAZ will automatically run. Select **Close** once TOPAZ finishes running (users 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.

1. Select *Display* | **Display Options** to open the *Display Options* dialog.
2. Choose "DEM Data" from the menu on the left and set the *Min Accumulation for Display* to "0.09".
3. Select **OK.**

2.2 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. In the *Select Shapefile* dialog, open "basins.shp". This shapefile will act as a background image and help users to place the outlets.
4. Switch to the **Drainage** module.

To aid users in placing the outlet points, users will zoom in on a portion of the DEM. Then users 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 1.

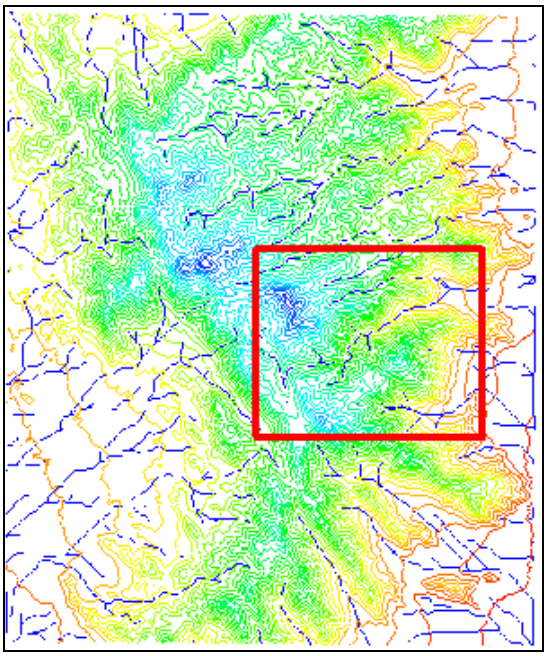


Figure 1 Zoom area

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

6. Select *Display* | **Display Options** to open the *Display Options* dialog.
7. Choose "DEM Data" from the menu on the left and toggle off *DEM Contours*.
8. Select **OK**.
9. Select the **Create Outlet Point** tool.
10. Create a point in each of the two locations shown in Figure 2. Be sure to place each point directly on a stream (zoom in if needed).

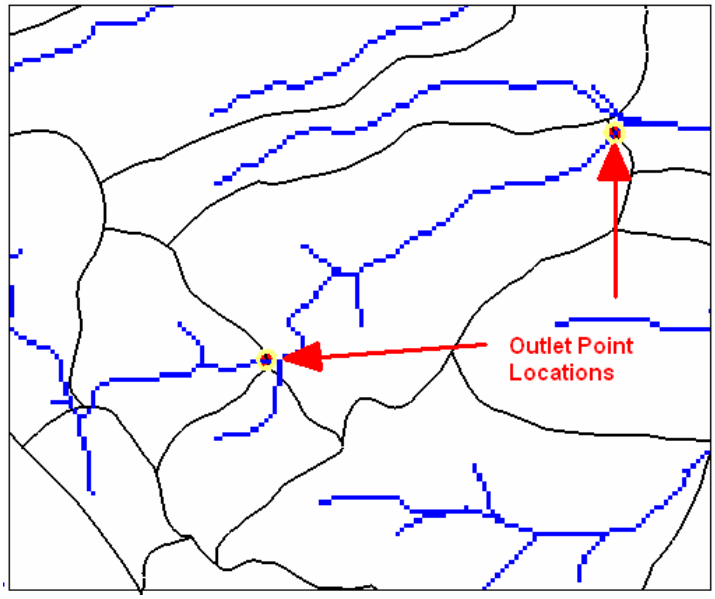


Figure 2 Approximate locations for placing outlets

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 users should see the creation of colored basin boundaries. If only one or no basins are 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. The *Units* dialog will appear. Select **OK**.

Users have now delineated a watershed with two sub basins (corresponding to two outlets) and computed basin parameters to be used in a hydrologic model. Users will 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.

3 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, AZ.

Users will be using the watershed developed in the previous section to run the NSS model.

To begin, users will 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.

3.1 Creating the NSS Regions Coverage

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

1. Toggle off the check box next to "basins.shp" in the Project Explorer.
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. In the *Properties* dialog, change the *Coverage Type* to "NSS Region".
5. Select **OK**.
6. Select *File* **| Open**.
7. In the *Open* dialog, select "arizusgs.shp" and click **Open**.
8. Switch to the **GIS** module.
9. Select *Mapping* **| Shapes → Feature Objects**.
10. If prompted to use all shapes, select **Yes**.
11. The *GIS to Feature Objects Wizard* will appear. Select **Next**.
12. Notice that the STATE and NSS_REGION fields are automatically mapped to the correct attributes.
13. Select **Next**.
14. Select **Finish** to close the *Wizard*.
15. Switch to the **Map** module.
16. Choose the **Select Feature Polygon** tool.
17. Double-click on the polygon in the center, as shown Figure 3, to verify that the State is Arizona and the Region is Central Arizona Region 12.

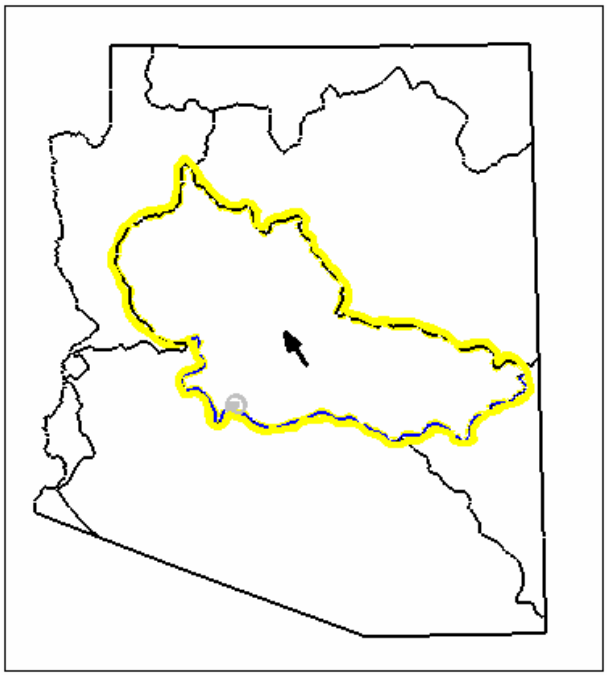


Figure 3 Double-click on highlighted polygon to view its attributes

18. Select **OK**.

Users have now created the NSS Regions coverage and can proceed to run the NSS computations.

3.2 Running NSS and Viewing Results

1. Select the **Zoom** 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, directly to the left of the modules) to "NSS".
4. Choose the **Select Basins** tool.
5. Double-click the basin icon on the left, labeled 2B.
6. Select **Yes** twice when asked to compute region areas, annual rainfall, and evaporation.
7. The *National Streamflow Statistics Method* dialog will appear. 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** when asked to compute.

11. The *National Streamflow Statistics Method* dialog will appear. Choose the **Compute Results** button.
12. In the output window at the bottom, click on the line for Recurrence [years] = 50. This may require scrolling down a bit. Select the whole line, not just one individual cell.
13. Choose the **Compute Hydrograph** button
14. In the *NSS Hydrograph Data* dialog, choose the **Compute Lag Time – Basin Data** button.
15. Change the *method* to "Tulsa 100% Urban Method".
16. Select **OK** twice to close the dialog boxes.
17. Select **Done** to exit the *National Streamflow Statistics Method* dialog.
18. Double-click on the hydrograph icon  .

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

19. Close the hydrograph plot window.

4 Building the HEC-1 Simulation

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

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

4.1 Precipitation Data

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

1. Make sure the Models dropdown field is set to "HEC-1". This is the same model box users previously set to "NSS".
2. Select *HEC-1* **| Job Control**.
3. In the *HEC-1 Job Control* dialog, choose the **Initialize Maricopa County Precipitation Data** button.
4. In the *Maricopa County Precipitation Data Initialization* dialog, choose the *Basin Av*erage option and select "24-hour" (storm duration) from the dropdown list.
5. Click on the **Browse**  button to select a rainfall grid to read in and use to compute precipitation.

6. The *Open* dialog will appear. 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

4.2 Computing Losses

Along with land use and soil type data, WMS can compute Loss Data based on the Maricopa County methodology. Users 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, users 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. In the *Properties* dialog, change the *Coverage Type* to "Land Use".
4. Select **OK**.
5. Switch to the **GIS** module.
6. Select *Data* **|** **Add Shapefile Data**.
7. In the *Select shapefile* dialog, open "landusewhitetanks.shp".
8. Hide the "NSS Region" coverage by un-checking its box in the Project Explorer.
9. Make sure "arizusgs.shp" is hidden by un-checking its box in the Project Explorer.
10. Click the **Frame** macro in order to view the extents of the land use coverage.

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 users do not wish to map to feature objects. Since only visible shapefiles can be selected with the Select Shapes tool, users can hide all shapefiles but the one they are interested in, and then select the shapes they want to map to feature objects. Note, however, that if users do not use the Select Shapes tool to select the shapes that they 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 the user's watershed.
13. Select *Mapping* **|** **Shapes → Feature Objects**.
14. In the *GIS to Feature Objects Wizard* dialog, 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, it should appear bolded and the graphic should be colored.
20. Choose the **Select Feature Polygon** tool.
21. Select *Feature Objects* | **Attributes**.
22. The *Land use mapping* dialog will appear. 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 and allow the land use table to be overwritten.
25. The *Open* dialog will appear. Open "landusemagtable.tbl".
26. Toggle off the *Display SCS CN's* check box and toggle on the *Display Green-Ampt* box.
27. Select **Apply**.

Adding Soil Type Data

1. Right-click on the "Coverages" folder in the Project Explorer and select **New Coverage**.
2. In the *Properties* dialog, change the "Coverage type" to "Soil Type".
3. Select **OK**.
4. Switch to the **GIS** module.
5. Select *Data* | **Add Shapefile Data**.
6. The *Select shapefile* dialog will reappear. 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 users do not wish to map to feature objects. Since only visible shapefiles can be selected with the Select Shapes tool, users can hide all shapefiles but the one they are interested in, and then select the shapes theywant to map to feature objects. Note, however, that if users do not use the Select Shapes tool to select the shapes that they 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. The *GIS to Feature Objects Wizard* will reappear. Select **Next**.
12. Find the column labeled *SLTYP_LID* and change its Mapping field to "SCS soil type".

13. Select **Next**.
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. The *Soil type mapping* dialog will appear.
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. The *Open* dialog will appear. Open "soiltable.tbl".
23. Turn off the *Display of SCS soil type* box, and turn on the *Display Green-Ampt* box.
24. Select **Apply**.

Computing Losses

With the land use and soil type coverages defined, users 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 users 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 users 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. Users 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. The *HEC-1 Precipitation* dialog will open. Select **OK**.
9. Click the **Loss Method** button and view the Green-Ampt values computed from the land use and soil coverages in the *HEC-1 Loss Methods* dialog.
10. Select **OK**.
11. Select **Done**.

4.3 Setting the Unit Hydrograph Method

For this HEC-1 model, users 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. The *HEC-1 Unit Hydrograph Methods* dialog will appear. 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. The *HEC-1 Unit Hydrograph Methods* dialog will appear. 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**.

4.4 Specifying the Routing Method

To simulate routing from the upstream basin (on the left) to the downstream basin, users 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. The *HEC-1 Routing Data* dialog will appear. Choose the "Muskingum (RM)" option under *Routing type*.
4. Click the **Compute NSTPS** button.
5. The *Compute NSTPS* dialog will open. Choose the *From Channel Velocity Estimat*e option.
6. Enter "4" for the *Channel velocity estimate*.

7. Select **OK** two times to return to the main *HEC-1* dialog.
8. Select **Done** to close the *HEC-1* dialog.

5 Save and Run the HEC-1 Simulation

1. Select *HEC-1* | **Run Simulation**.
2. In the *HEC-1 Run Options* dialog, 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* option is toggled on.
5. Select **OK.**
6. Select **Close** when the HEC-1 simulation finishes.

Now that HEC-1 computed basin and outlet hydrographs, users 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, users can double-click on a hydrograph icon to view the hydrograph plot.
9. Close the hydrograph plot window.

6 Conclusion

This concludes this WMS tutorial. Users may continue to experiment with WMS and the tools used in this tutorial or may quit at this time. Users should have learned how to:

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