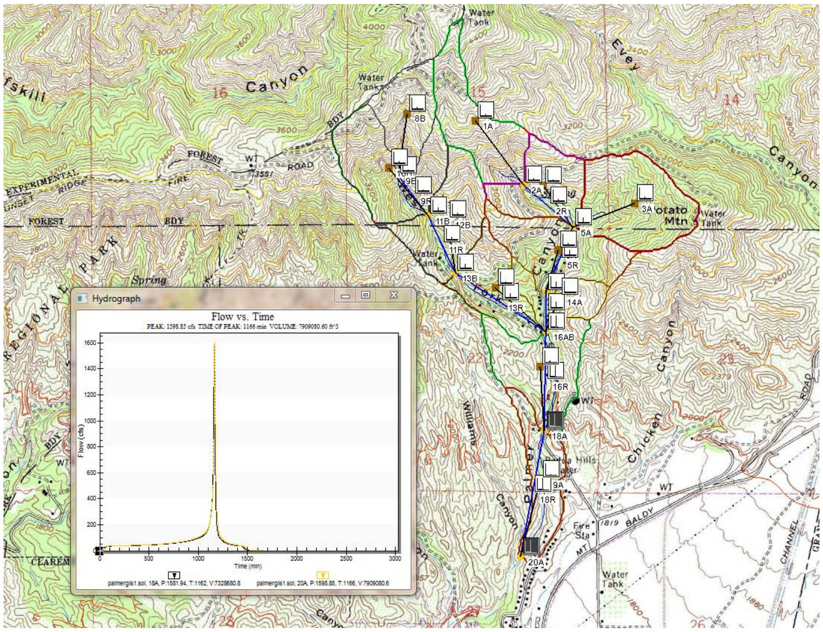
WMS

v. 9.0

WMS 9.0 Tutorial

Watershed Modeling – MODRAT Interface (GIS-based)

Delineate a watershed and build a MODRAT model



Objectives

Delineate a watershed from a DEM and derive many of the MODRAT input parameters from the delineated watershed. Use soil and land use GIS data to derive soil number and impervious values. Compute sub-basin time of concentrations from the LACDPW regression equation and rainfall depths using digital rainfall grids. Define reach and reservoir hydrologic routing and run your MODRAT model.

Prerequisite Tutorials

- Watershed Modeling – DEM Delineation

Required Components

- Data
- Drainage
- Map
- Hydrology
- Hydrologic Models

Time

- 30-60 minutes

AQUAVEO™

1 Contents

2 Introduction

WMS has a graphical interface to the Los Angeles County Dept. of Public Works (LACDPW) Modified Rational (MODRAT) model. Geometric attributes such as areas, lengths, and slopes are computed automatically from the digital watershed. Parameters such as soil numbers, impervious percentages, and routing data are entered through a series of interactive dialog boxes. Once the parameters needed to define an MODRAT model have been entered, an input file with the proper format for MODRAT can be written automatically. Since only parts of the MODRAT input file are defined in this exercise, you are encouraged to explore the different available options of each dialog, being sure to select the given method and values before exiting the dialog.

3 Objectives

As a review you will delineate a watershed from a DEM. You will then develop a simple watershed model using the delineated watershed to derive many of the parameters. Land use and soil shape files will be used to develop a soil number and impervious values. Time of concentration will be computed using the LACDPW regression equation. Rainfall depths will be computed from digital rainfall grids for Los Angeles County. After establishing the initial MODRAT model other variations will be developed, including defining reach routing and including a reservoir with storage routing.

4 Delineating the Watershed

The Palmer Canyon watershed will be delineated from a 10 meter resolution DEM file.

1. Close all instances of WMS.
2. Open WMS.
3. Select ***File* | *Open*** **.**
4. Locate the ***MODRAT*** 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 *"mtbaldy_10m.asc"*.
6. Select *OK* in the Importing ArcInfo Grid window that appears and shows the DEM bounding coordinates.
7. Select ***File* | *Open*** to open the background quad image for this area.
8. Find and open *"mtbaldy.tif"*.

The image of the quad map will appear in the background with the DEM contours on top.

9. Use the *Zoom* tool to view the area shown in the Figure 4-1, below.

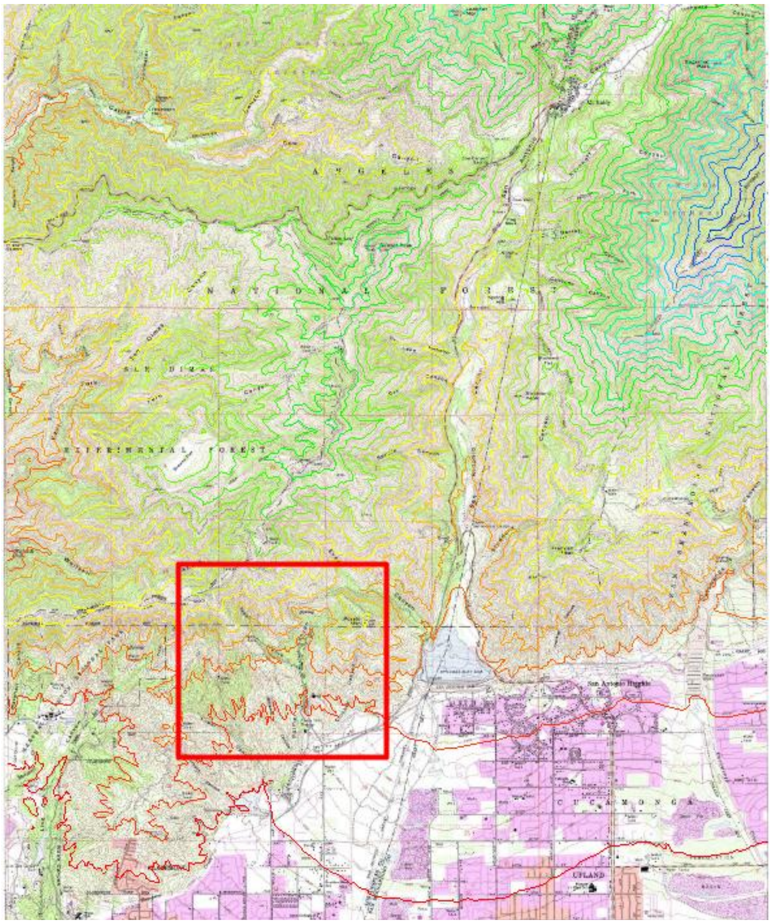


Figure 4-1: Zoom in on the area bounded by the rectangle

4.1 Delineate the Watershed

1. Switch to the *Drainage* module .
2. Select ***DEM | Compute Flow Direction/Accumulation....***
3. Select *OK*.
4. Select the *Current Projection* button in the Units dialog.
5. Select the *Global Projection* button in the Current Projection dialog.
6. Set the Projection to *State Plane Coordinate System.*
7. Set Zone to *California Zone 5 (FIPS 405).*
8. Set the Datum to *NAD83.*
9. Set Planar Units to *Feet (U.S. Survey).*
10. Select *OK* on the Select Projection dialog.
11. Set the Vertical Projection to *NGVD 29 (US)* and Units to *U.S. Survey Feet.*
12. Select *OK* – you have now told WMS what the units and projections are for the raw DEM data you have loaded and are about to process.
13. Set the Basin Areas to *Acres*; leave the Distances in *Feet*. WMS will report computed geometric parameters in these units after you delineate the watershed.
14. Select *OK* in the Units dialog to begin the TOPAZ processing.
15. Select *Close* once TOPAZ finishes running (you may have to wait for the computer to finish processing the information).

Blue lines will appear on the DEM indicating location of flow accumulation channels (streams). You will need to edit the display to match the channel complexity desired.

16. Select ***Display | Display Options*** .
17. Choose *DEM Data* and set Point Display Step to 1 and Min. Accumulation for Display to 40.0.
18. Select *OK* – note the change in the flow accumulation stream network.

You are now ready to define the main watershed outlet location and define the drainage area. You will then define interior outlet points and let WMS subdivide the watershed.

19. Use the *Zoom* tool to view the area shown in Figure 4-2 which shows the confluence of Palmer Canyon and Williams Canyon.

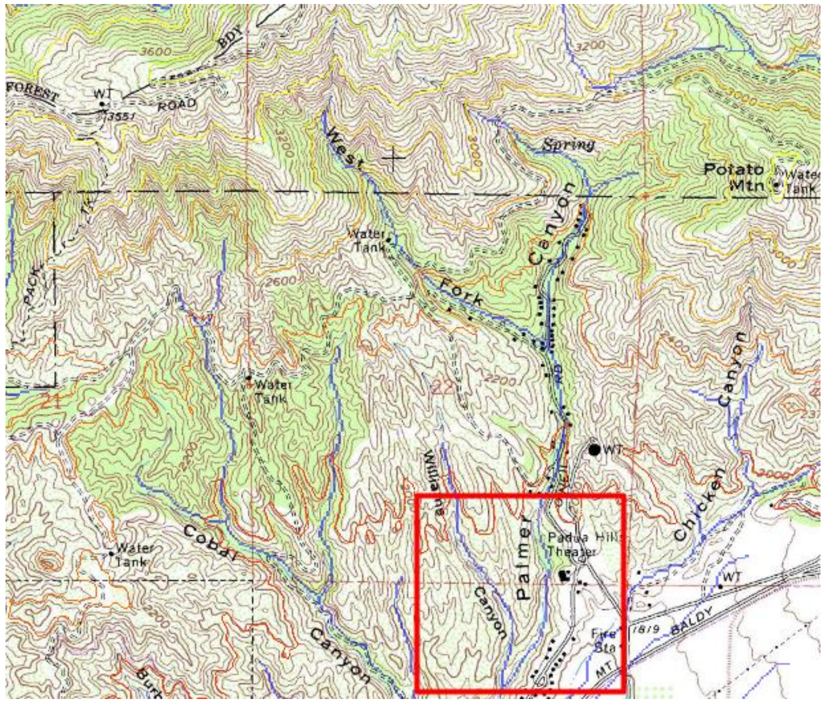


Figure 4-2: Zoom in on the area bounded by the rectangle

20. Select the *Create Outlet Point* tool .
21. Create a new outlet point in Palmer Canyon, slightly upstream of the confluence point of the two canyons. The outlet needs to be inside one of the flow accumulation (blue) cells. WMS will move the outlet to the nearest flow accumulation cell if you do not click right in one of the flow accumulation cells.
22. Select ***Display* | *View* | *Previous View*** .
23. Select ***DEM* | *DEM->Stream Arcs.***
24. (Optional) Turn on the *Display stream feature arc creation* toggle if you would like see the digitizing process of the streams.
25. Select *OK.*
26. Select ***DEM* | *Define Basins*** – you will then see the boundary of the drainage area for the outlet point/streams you created.

The boundary shown is accurate for the single outlet point created. However, you will need to subdivide the watershed to smaller sub-basins to meet Los Angeles County Department of Public Works standards for hydrologic modeling. Follow the steps below to subdivide and finalize the watershed delineation.

27. Select the *Create Outlet Point* tool .
28. Point and click to create an outlet point at each of the 6 locations on the stream arcs indicated in Figure 4-3.

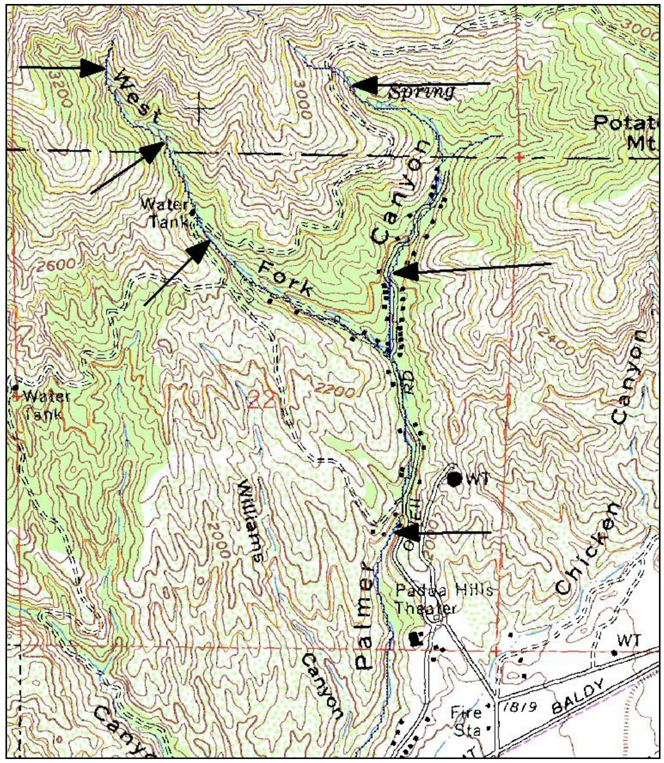


Figure 4-3: Create outlets at each location indicated

29. Select the *Select Feature Point/Node* tool.
30. Double-click on each of the two nodes located at the stream branches indicated in Figure 4-4 and change the attribute to *Drainage outlet*, then click *OK*.

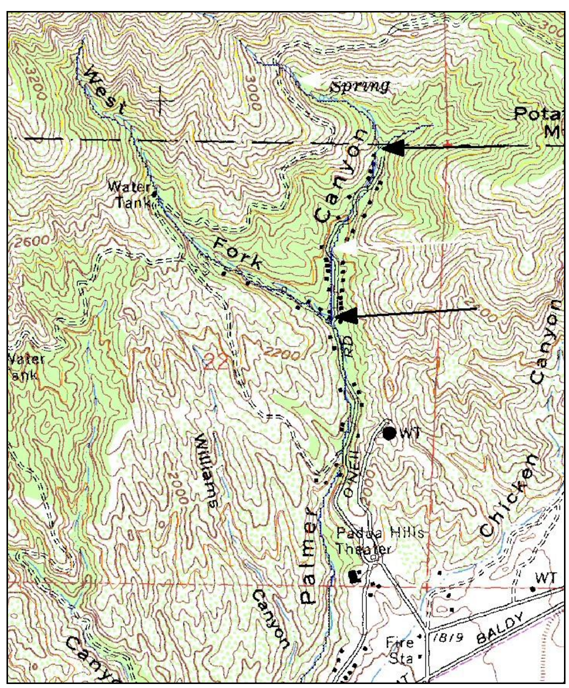


Figure 4-4: Double-click nodes at each location indicated

31. Select ***DEM | Define Basins*** – this will show you the subdivided watershed incorporating your defined outlet points.
32. Select ***DEM | Basins->Polygons.***
33. Select ***DEM | Compute Basin Data.***
34. Select *OK.*

The Palmer Canyon watershed is now delineated with acceptable sub-areas for MODRAT modeling. The next steps will clean up the display and smooth basin boundaries of the watershed before completing the MODRAT simulation.

35. Select ***Display | Display Options*** .
36. Choose *DEM Data* and toggle off the display for *Watershed*, *Stream*, *Flow Accumulation*, *Color Fill Drainage Basins*, and *Fill Basin Boundary Only.*
37. Choose *Drainage Data* and toggle *Basin Areas* off.
38. Select *OK.*
39. Switch to the *Map* module .
40. Choose the *Select Feature Arc* tool .
41. Select ***Edit | Select All.***
42. Select ***Feature Objects | Redistribute.***
43. Enter 90.0 in the Spacing box.
44. Select *OK.*

Note that the watershed boundaries and streams are now smoothed and the display is generally cleaner. You may want to zoom in and pan to visually inspect the watershed delineation. Once finished, you should zoom out/in until the Palmer Canyon watershed is framed in your screen – see Figure 4-5. You are now ready to begin setting up the MODRAT simulation.

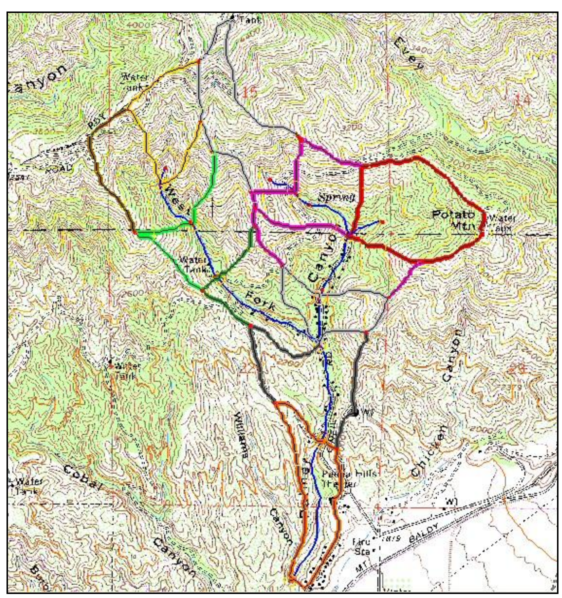


Figure 4-5: Completed watershed delineation of Palmer Canyon

This is a good place to save your project to the hard drive before continuing. Save this data to a WMS project file:

45. Select ***File* | *Save As***.
46. Enter "PalmerCyn25_GIS.wms" and click *Save*.
47. Select *No* if asked to save image files in project directory.

WMS will save your project to a set of WMS Project files. The *.wms file is an index file and contains other information that instructs WMS to load all the files associated with the project when you open your project at a later time.

5 MODRAT Global Setup

The MODRAT analysis setup requires you to enter Job Control data, basin data for each subarea, reach data for each channel, and elevation-storage-discharge relationships for each storage facility. The following sections will guide you in entering data and using GIS data layers to acquire input data for MODRAT.

5.1 Job Control

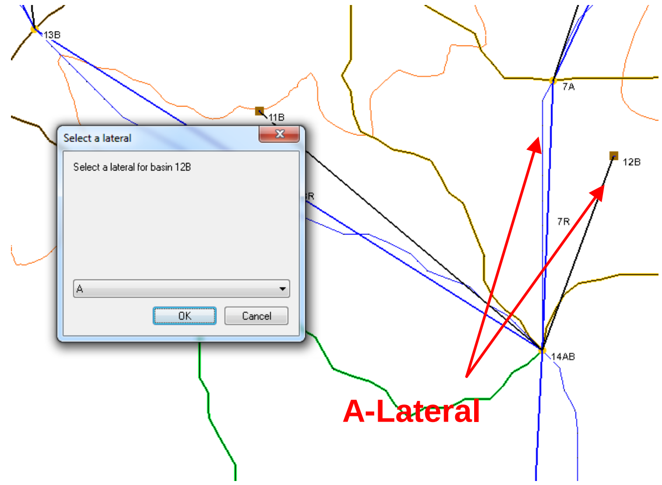
Most of the parameters required for a MODRAT model are defined for basins, outlets, and reaches. However, there are a few "global" parameters that control the overall simulation. These parameters are not specific to any basin or reach in the model. These parameters are defined in the WMS interface using the Job Control dialog.

1. Switch to the *Hydrologic Modeling* module.
2. Select *MODRAT* from the drop down list of models found in the Models Window – a MODRAT menu item will appear in the Menu Bar.
3. Select ***MODRAT* | *Job Control.***
4. Choose *MODRAT 2.0* at the top of the dialog.
5. Select *2* days in the Run time drop down list.
6. Select *25 year* in the Storm Frequency drop down list.
7. Enter "palmergis1" in the Prefix box, then click *Update*. Note that the default prefix for output files is now updated.
8. Enter *"palmergis_rain.dat"* in the Rain file box.
9. Click the browse button next to the Soil file and open the file "*sgr_soilx_71.dat"*.
10. Select *OK.*

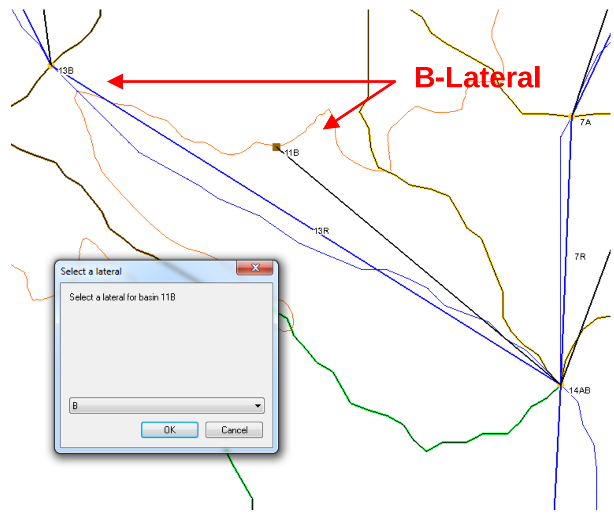
5.2 Tree Numbering

Each basin or reach is assigned a default name when it is created by WMS. However, these must be named and numbered in sequential order from upstream to downstream using a MODRAT naming convention so that MODRAT analyzes the model in the proper order.

1. Select the *Select Basin* tool.
2. Click on the brown square sub-basin icon at the most upstream end of Palmer Canyon (the canyon forking to the right). Selecting this basin defines the upstream end of the main line of the watershed.
3. Select ***MODRAT* | *Number Tree.***
4. Select *OK* to start numbering with location/lateral of 1A.
5. As the numbering process proceeds you will be prompted to "Select a lateral" for each of the basins at a confluence. Notice that WMS first zooms into a basin along the "A" lateral and its surrounding outlet points. Since the outlet point upstream from this basin is located on the "A" lateral (you can determine this because the upstream outlet name ends with "A"), assign the first basin to the "A" lateral of the watershed and select *OK*.



6. Now notice that WMS zooms into a basin on the "B" lateral and its surrounding outlet points. Since the outlet point upstream from this basin is located on the "B" lateral (you can determine this because the upstream outlet name ends with "B"), assign this basin to the "B" lateral of the watershed and select *OK*.
7. Right-click on the ***Drainage*** coverage in the **Project Explorer** and select ***Zoom To Layer***.



The numbering is now complete. Note that the selected basin is now 1A. The main line is met by Line B at the 16AB confluence (outlet) point. The numbers now indicate the order in which the units will be processed by MODRAT.

6 MODRAT Basin Data Setup

Each basin in the watershed requires a number of input parameters. Many of these can be computed using tools in WMS.

6.1 Basin Data Parameters

1. Double-click on the basin icon labeled 1A. You may need to toggle the *mtbaldy* image off in the Project Explorer to better see the basin labels.

Double-clicking on a basin or outlet icon brings up the parameter editor dialog for the current model (in this case MODRAT)

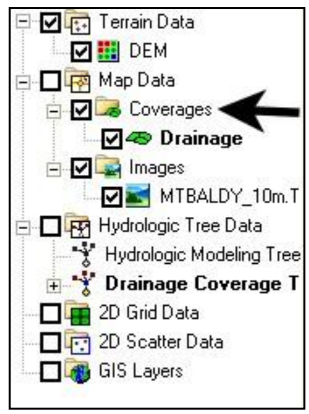
2. Notice that the area has been calculated but all other parameters are empty.
3. Click *OK* to exit the window.

Many of the parameters can be computed by WMS using GIS data layers. The following sections will compute Soil Number, Percent Impervious, and Rainfall Depth for each basin.

6.2 Soil Number Computation

You will load soil data for Los Angeles County and let WMS compute the dominant soil type for each basin.

1. Right-click on the Coverages folder in the Map Data section of the Project Explorer.
2. Choose ***New Coverage*** from the pop-up menu.
3. Select *Soil Type* as the Coverage Type in the Properties window.
4. Select *OK.*
5. Switch to the *GIS* module.
6. Select ***Data | Add Shapefile Data.***
7. Open *"soils_2004.shp"* from the ***MODRAT\SoilType*** folder – the soil map for all of L.A. County will be loaded.



8. Right-click on the ***Drainage*** coverage in the **Project Explorer** and select ***Zoom To Layer***.
9. Select the Soil Type coverage.
10. Switch back to the *GIS* module.
11. Choose the *Select Shapes* tool.
12. Drag a selection box around the watershed extents – the soil polygons covering the watershed will be selected.
13. Select ***Mapping | Shapes -> Feature Objects.***
14. Select *Next.*
15. Make sure the CLASS field is mapped to the *LA County soil type* attribute.
16. Select *Next.*
17. Select *Finish.*

18. Hide the *soils_2004.shp* file by deselecting its check box in the Project Explorer.

Now that the soil data is loaded, do the following to compute and assign the soil numbers to MODRAT:

19. Select the Drainage coverage in the Project Explorer to designate it as the active coverage.
20. Select the *Hydrologic Modeling* module .
21. Select ***MODRAT | Map Attributes***.
22. Select *LA County soil numbers* as the Computation type.
23. Select *OK*.
24. Once the computation is finished, double-click on any sub-basin icon to bring up the MODRAT Parameters window and view the Soil number assigned.
25. Click *OK* to exit the MODRAT Parameters window.

6.3 Percent Impervious Computation

You will now load land use data for Los Angeles County and let WMS compute the average percent impervious for each basin.

1. Right-click on the Coverages folder in the Map Data section of the Project Explorer.
2. Choose ***New Coverage*** in the pop-up menu.
3. Select *Land Use* as the Coverage Type in the Properties window.
4. Select *OK*.
5. Switch to the *GIS* module .
6. Select ***Data | Add Shapefile Data.***
7. Open *"landuse_imp_2004.shp"* in the ***MODRAT\Landuse*** folder – the land use map for all of L.A. County will be loaded.
8. Right-click on the ***Drainage*** coverage in the **Project Explorer** and select ***Zoom To Layer***.
9. Select the ***Land Use*** coverage.
10. Switch back to the *GIS* module .
11. Choose the *Select Shapes* tool .
12. Drag a selection box around the watershed extents – the land use polygons covering the watershed will be selected.
13. Select ***Mapping | Shapes -> Feature Objects.***
14. Select *Next*.
15. Make sure the IMPERV_ field is mapped to the *Percent Impervious* attribute.

16. Select *Next*.
17. Select *Finish*.
18. Hide the *landuse_imp_2004.shp* file by toggling off its check box in the Project Explorer.

Now that the land use data is loaded, do the following to compute and assign the percent impervious to MODRAT.

19. Select the Drainage coverage in the Project Explorer to designate it as the active coverage.
20. Select the *Hydrologic Modeling* module.
21. Select ***MODRAT | Map Attributes***.
22. Select *LA County land use* as the Computation type.
23. Select *OK*.
24. Once the computation is finished, double-click on any sub-basin icon to bring up the MODRAT Parameters window and view the Impervious % assigned.
25. Click *OK* to exit the MODRAT Parameters window.

6.4 Rainfall Depth and Distribution Assignment

You will now load a rainfall depth grid for the 25-year storm frequency for Los Angeles County and let WMS compute the average rainfall depth for each sub-basin. Then you will assign a rainfall mass curve to the model to provide the temporal distribution of the storm depth.

1. Select the *Drainage* module.
2. Select ***File | Open***.
3. Change the File Type to *Rainfall Depth Grid (*.*)*.
4. Open the file named *"lac25yr24hr.asc"* in the ***MODRAT*** folder – the rainfall grid will be opened and displayed.
5. Right-click on the ***Drainage*** coverage in the **Project Explorer** and select ***Zoom To Layer***.
6. Select the *Hydrologic Modeling* module.
7. Select ***Calculators | Compute GIS Attributes.***
8. Select *Rainfall Depth* as the Computation.
9. Select *OK*.
10. Choose the *Select Basin* tool and double-click on basin 1A to bring up the MODRAT Parameters window and view the rainfall depth assigned.
11. Choose Show: *All* in the upper left part of the dialog.
12. In the Temporal distribution column click on the *Define…* button in the All row (colored yellow) of the spreadsheet. This will bring up a window

where you will specify the rainfall temporal distribution (time vs. cumulative rainfall percentage).

13. Select the *Import* button in the XY Series Editor.
14. Open the file named *"LACDPWStorm-4thday.xys"*.
15. The Selected Curve in the XY Series Editor should now read *LACDPWStorm-4thday* and the rainfall mass curve is displayed.
16. Select *OK.*
17. Select *OK.*

The process above has assigned a rainfall depth to each basin and also assigned the LACDPW storm distribution curve to all basins.

Clean up the display of your model by turning off several layers now that they have been used and are not needed:

18. Hide the Soil Type coverage by toggling off its check box in the Project Explorer.
19. Hide the Land Use coverage by toggling off its check box in the Project Explorer.
20. Hide the Rain Fall grid file by toggling off its check box in the Project Explorer.
21. Hide the *mtbaldy* image by toggling off its check box in the Project Explorer.

6.5 Time of Concentration

The final parameter needed for each basin in the model is the Time of Concentration (Tc). WMS has the LACDPW Tc equation method built in and linked to GIS data capabilities. Do the following to compute Tc for all basins.

1. Select the *Drainage* module .
2. Select ***DEM | Compute Basin Data***.
3. In the Units dialog, select the *Drain Data Compute Opts...* button. Toggle on the checkbox to *Create Tc Coverage.*
4. Select *OK*, and then *OK* again on the Units dialog.

Note the basin data is recomputed (basin area is displayed) and there is a new coverage named Time Computation and containing the longest flow path of each basin.

5. Select ***Display | Display Options*** .
6. Choose Drainage Data and toggle *Basin Areas* off.
7. Select *OK.*
8. Ensure that the Drainage coverage is the active coverage by selecting it in the Project Explorer.
9. Select the *Hydrologic Modeling* module .
10. Select ***MODRAT | Compute Tc.***

11. Note that a check of required input for Tc computations has been performed. Select *Next* in the Compute MODRAT Tc Wizard.
12. Review the Time of Concentration (Tc) computed for each basin.
13. Select *Done.*
14. Once the computation is finished, double-click on any sub-basin icon to bring up the MODRAT Parameters window and view the Tc assigned.
15. Click *OK* to exit the MODRAT Parameters window.

The input parameters for all basins should now be entered for the simulation. Save this data to your working project file.

16. Select ***File* | *Save*** .
17. Select *No* if asked to save image files in the project directory.

7 MODRAT Reach/Outlet Data Setup

Each reach must have data associated with it to be successfully simulated by MODRAT. Reaches are selected in WMS by clicking on an outlet (confluence) point. The parameters for that point, and the channel downstream from that point to the next, can be edited.

1. Double-click the outlet labeled 2A at the upper end of the Palmer Canyon main channel – this will load the parameters for that reach into the MODRAT Parameters window for review/editing.

Note that Length and Slope have been computed and are entered in the MODRAT Parameters window.

2. Select *Mountain* as the Routing Type.

Note that the Manning's n box is inactive and no additional variables need to be entered for this channel type.

3. Choose *Hydrograph (*.HYF) and WMS plot file (*.SOL)* for the Hydrograph Output.

You have now completed the input for one of the reaches in the Palmer Canyon watershed. You will need to define data for all reaches in a similar fashion:

4. Choose Show: *All* from the drop-down box in the upper-right corner.
5. Use the table below to fill in values:

Reach Name	Routing type	n	Output
2A	Mountain	-	HYF/SOL
5A	Valley	-	HYF/SOL
7A	Valley	-	HYF/SOL
10B	Mountain	-	HYF/SOL
12B	Mountain	-	HYF/SOL
14B	Valley	-	HYF/SOL
16AB	Valley	-	HYF/SOL
18A	Valley	-	HYF/SOL
20A	Variable	0.014	HYF/SOL

6. Note that Reach 20A, the most downstream outlet, does not need a length or slope defined since it is at the downstream end of the watershed. When you are finished entering the parameters choose *OK* on the MODRAT Parameters dialog.

The input parameters for all reaches should now be entered for the simulation. Save this data to your working project file.

7. Select ***File | Save***.
8. Select *No* if asked to save image files to the project directory.

8 Running a MODRAT 2.0 Simulation

All the data required to run a simulation is now ready. To make sure there are no omissions in the data, WMS will perform a model check. Follow the steps below:

1. Select ***MODRAT | Check Simulation***
2. Review the model check report noting that there are 2 possible errors in the MODRAT model.

Notice the line stating "No reach length is defined for outlet 20A". The outlet 20A is the watershed outlet; therefore, there is no reach downstream that you need to define.

3. Select *Done* to exit the MODRAT Model Check.

The model checker is a simple way to verify that you have not left out any needed data. It does not verify that the model is correct, but that all the data needed to run the simulation is in place. To execute the MODRAT simulation, do the following:

4. Select ***MODRAT | Run Simulation.***
5. The Input File should be named *"Palmergis1.lac"*.
6. Ensure that the *Save file before run* toggle is checked.
7. Ensure that the Prefix for output files box contains "Palmergis1".
8. Click *OK* to start the simulation.

A window will appear and report the progress of the MODRAT simulation.

9. Select *Close* once MODRAT finishes running (you may have to wait a few seconds to a minute or so).

The resulting hydrographs will be read in and a small hydrograph plot will appear next to each basin and outlet.

10. Double-click on the hydrograph icon next to outlet 20A.
11. Review the hydrograph plot that appears in a new plot window. It should look similar to Figure 8-1.

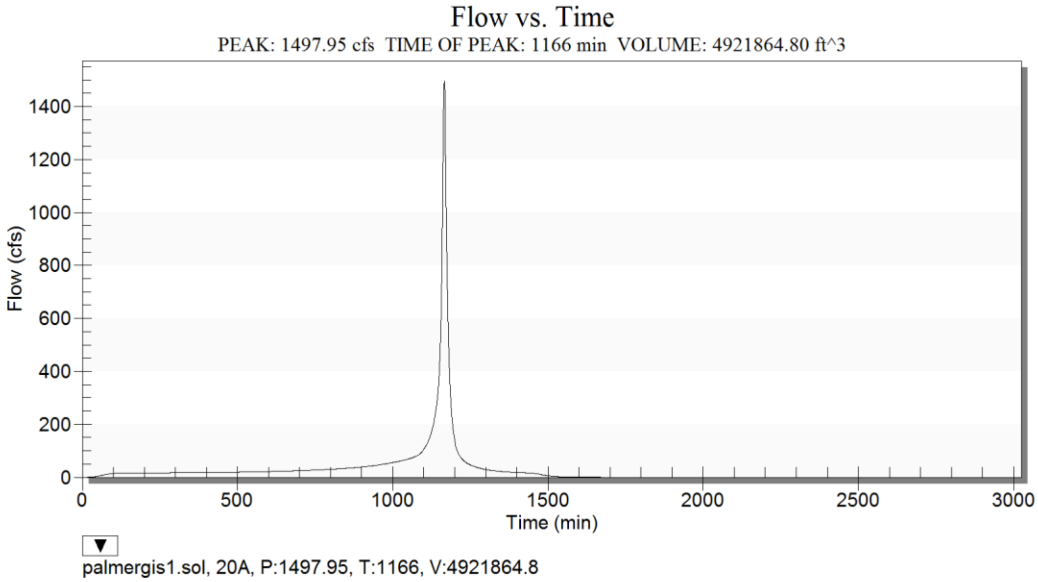


Figure 8-1: Solution hydrograph for MODRAT simulation

12. Close the plot window by clicking on the X in the upper right corner.
13. Select ***File | Edit File.***
14. Open the file named *"Palmergis1.out"*.
15. Confirm that you want to open the file with Notepad, if prompted.
16. Review the text summary output of the simulation.
17. Close the file by clicking on the X in the upper right corner of the Notepad window.

You have successfully completed a simulation with MODRAT. There are many other options in the MODRAT model that were not included in this simple model. The following section will show how to quickly create a burned watershed simulation. Other options such as detention basins and diversions (flow splits) can be reviewed in the MODRAT Interface (Schematic-based) chapter. Those options work the same whether a simulation is GIS-based or schematic-based.

9 Burned Watershed Simulation

To create a burned simulation from a regular MODRAT watershed model, you only need to change some Job Control options and soil number data. Tools in WMS can automated these changes:

1. Select ***MODRAT | Job Control***.
2. Enter "Palmergis_burn" in the Prefix box.
3. Click the *Update* button.
4. Ensure that "C:\Program Files\WMS90\modrat\sgr_soilx_71.dat" is entered for the Soil file.
5. Select *OK.*

The Soil file designated in Job Control must contain soil data for burned conditions. The sgr_soilx_71.dat file contains regular soil data for typical LA County soil numbers (2-180) and for burned soil numbers (202-380). Now you will update all the soil numbers in the watershed to reflect burned conditions.

6. Select ***MODRAT | Create Burned Simulation.***
7. Ensure the default of 200 is entered for Burned soils increment – this is the increment to be added to normal soils to get the corresponding burned soil.
8. Ensure that the impervious limit is set at *15%* – basins with higher impervious values will not be affected by the burn.
9. Select *OK.*
10. Once the computation is finished, double-click on any sub-basin icon to bring up the MODRAT Parameters window and view the new Soil type number assigned.
11. Click *OK* to exit the MODRAT Parameters window.
12. Select ***MODRAT | Run Simulation.***
13. The Input File should be named *"Palmergis_burn.lac"*.
14. Ensure that the *Save file before run* toggle is checked.
15. Ensure that the Prefix for output files box contains "Palmergis_burn".
16. Click *OK* to start the simulation.
17. Select *Close* once MODRAT finishes running (you may have to wait a few seconds to a minute or so).

The resulting hydrographs will be read in and a small hydrograph plot will appear next to each basin and outlet.

18. Double-click on the hydrograph icon next to outlet 20A.
19. The plot will contain the original hydrograph (from the previous run) and the new hydrograph from the burned simulation (read in from the new *.sol file). It should look similar Figure 9-1.

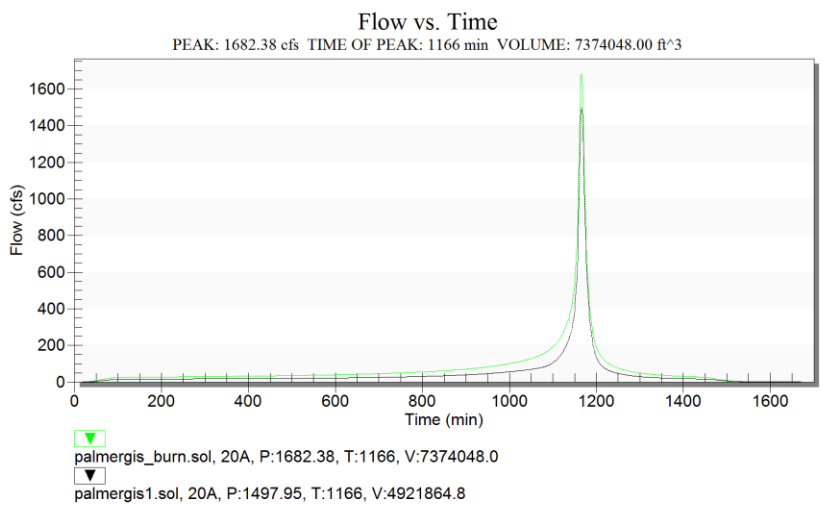


Figure 9-1: Burned/original hydrographs for MODRAT simulation

20. Close the plot window by clicking on the X in the upper right corner.
21. Clear the results by selecting ***Hydrographs | Delete All.***

10 Debris Production

WMS has a tool for computing the amount of debris produced as part of the runoff from mountainous and burned drainage basins. The following sections demonstrate how to compute debris production, account for debris control structures, and generate reports detailing the calculations.

10.1 DPA Zones

The debris production calculator in WMS requires GIS data that defines debris production area (DPA) zones. Drainage basin polygons are overlaid with the DPA zone polygons in order to determine debris production rates, which are applied to the area contributing to any outlet point of interest.

1. Right-click on the Coverages folder in the Map Data section of the Project Explorer.
2. Choose ***New Coverage.***
3. Select *MODRAT DPA Zone* as the Coverage Type in the Properties window.
4. Select *OK.*
5. Switch to the *GIS* module .
6. Select ***Data | Add Shapefile Data.***
7. Open *"dpazones.shp"* – the Debris Production Area (DPA) zones for all of L.A. County will be loaded.
8. Right-click on the ***Drainage*** coverage in the **Project Explorer** and select ***Zoom To Layer***.
9. Select the ***MODRAT DPA Zone*** coverage.
10. Switch back to the *GIS* module .
11. Choose the *Select Shapes* tool .
12. Drag a selection box around the watershed extents – the DPA zone polygons covering the watershed will be selected.
13. Select ***Mapping | Shapes -> Feature Objects.***
14. Select *Next.*
15. Make sure the DPA_ZONES field is mapped to the *DPA Zone* attribute.
16. Select *Next.*
17. Select *Finish.*
18. Hide the *dpazones.shp* file by toggling off its check box in the Project Explorer.

19. Switch to the *Hydrologic Modeling* module .
20. Select ***MODRAT|Debris Production\Bulking….***
21. Click on the *Compute GIS Data…* button on the Basin data tab.
22. Verify that the MODRAT DPA Zone and Land Use coverages are selected for this calculation.
23. Select *OK.*
24. Activate the Debris production tab in the dialog.

Total debris produced is reported for each outlet shown in the spreadsheet by summing the debris produced within each DPA zone for all undeveloped drainage areas that contribute runoff to the outlet.

10.2 Debris Control Structures

Adequate or undersized debris control structures can be defined at any outlet point and included in debris production calculations for all downstream outlets.

1. In the 5A column of the spreadsheet click on the Control structures *View…* button.

All of the outlets located upstream of outlet 5A are listed in this dialog.

2. Toggle on the checkbox next to location 2A.
3. Ensure that the Size is set to *Adequate.*
4. Select *OK.*

Notice that the debris calculations are automatically updated. An adequately sized control structure will remove all debris produced upstream of the location of the control structure from the results reported at all downstream locations.

5. In the 5A column of the spreadsheet click on the Control structures *View…* button.
6. Change the Size to *Undersized.*
7. Enter 10,000.00 for Capacity.
8. Set the Units to *yd^3.*
9. Select *OK.*

Values now show up in the Excess controlled debris row of the spreadsheet and are included in the total debris reported for all outlets downstream of the control structure.

10. In the 5A column of the spreadsheet click on the *View…* Control structures button.
11. Set the Units to *(%).*
12. Enter 0.5 for Capacity (only decimal percentages are valid).
13. Select *OK.*

In this case 50% of the debris produced upstream of the control structure is contained and the other 50% of the debris in included in the total debris reported for all outlets downstream of the control structure.

10.3 Reports

It is important to understand the difference between using report nodes and exporting a report that details the debris production computations.

1. Click on the *All On* button.

This marks all of the outlets displayed in the spreadsheet as report nodes. Report node locations are saved with the WMS project file and are used to indicate which outlets are of interest within a specific project. For example:

2. Click on the *All Off* button.
3. Toggle on the checkbox for outlet 5A in the spreadsheet to mark it as a report node.
4. Set the Show option to *Report nodes* (upper left corner of dialog).

Now only the data for outlet 5A is shown in the spreadsheet. There is also an option for showing outlets selected in the graphics window.

5. Click on the *Export…* button.
6. Click *Save* to save the report using the default location and file name.
7. Click *OK* in the Debris Production dialog.
8. Select ***File | Edit File…***
9. Open "*ladebris.txt*".

Reports will include both summary and detailed data for all outlets that appear in the spreadsheet when exported. In this case only data for outlet 5A is included in the report because that was the only data shown in the spreadsheet at the time that the report was exported.

10. Close the file by clicking on the X in the upper right corner of the Notepad window.

11 Bulking Flows

Burned flow rates are bulked in order to account for the debris/sediment in the runoff flow rate. Bulking factors are determined using DPA zones. The Bulked Flow calculator operates in the same fashion as the Debris Production calculator, including accounting for the effects of debris control structures. Burned flow values can be directly entered into the spreadsheet or read from a MODRAT solution.

1. Select ***MODRAT | Debris Production/Bulking….***
2. Activate the Bulked flow tab.
3. Click on the *Get Burned Flows From MODRAT Solution…* button.
4. Open "Palmergis_burn.out".

The bulked flow values in the spreadsheet are automatically updated. Click on the Control structures *View…* button in column 5A of the spreadsheet to experiment with combining debris control structures with flow bulking.

12 Conclusion

This concludes the exercise on defining MODRAT files and displaying hydrographs. The concepts learned include the following:

- Entering job control parameters
- Defining basin parameters from GIS data
- Defining routing parameters
- Saving MODRAT input files
- Reading hydrograph results
- Creating a burned simulation
- Debris production and flow bulking
- Comparing results from two runs

There are many other options in the MODRAT that were not included in this simple model. Other options such as detention basins and diversions (flow splits) can be reviewed in the MODRAT Interface (Schematic-based) chapter. Those options work the same whether a simulation is GIS-based or schematic-based.

