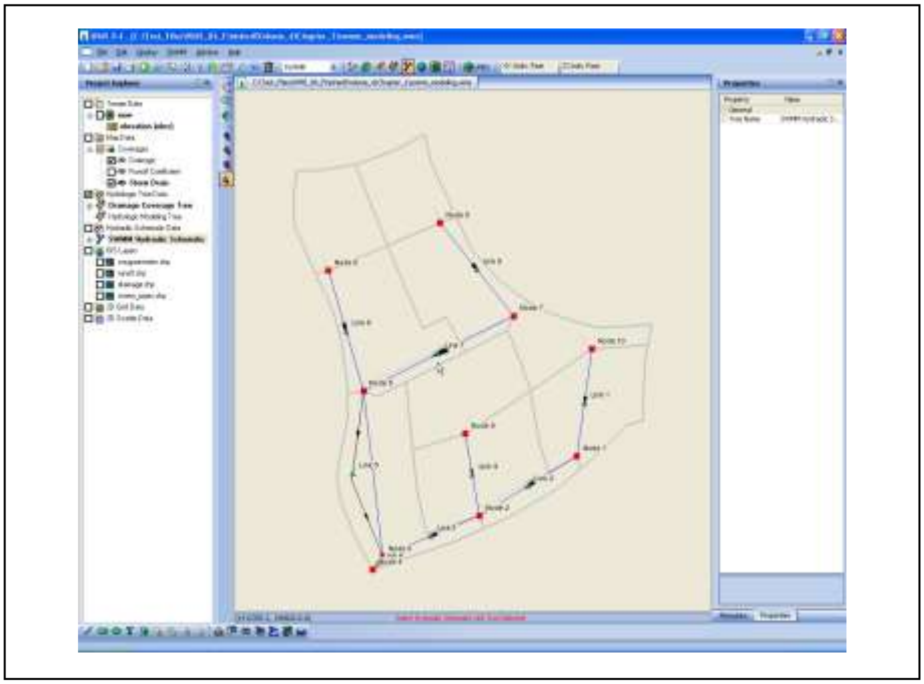
WMS

v. 10.0

WMS 10.0 Tutorial

Storm Drain Modeling – SWMM Modeling

Learn how to link a hydrologic model to the SWMM storm drain model



Objectives

Build a rational method hydrologic model and compute sub-basin flows. Import storm drain network information and link the storm drains to the hydrologic model. Run the flows from the hydrologic model through the storm drain model using either xpswmm or EPA-SWMM.

Prerequisite Tutorials

- Watershed Modeling – Rational Method Interface
- Editing Elevations – Using TINs

Required Components

- Data
- Drainage
- Map
- Hydrology
- Hydrologic Models
- Storm Drain

Time

- 30-60 minutes

AQUAVEO™

1 **Introduction** 2
2 **Objectives** 2
3 **Defining Runoff Coefficients** 3
3.1 Reading in the Subdivision Layout and Elevations 3
3.2 Defining the Runoff Coefficients 3
4 **Compute Flows from the Rational Method** 4
4.1 Creating the Drainage Coverage 4
4.2 Computing Runoff Coefficients 5
4.3 Entering Times of Concentration 5
4.4 Defining Rainfall Data 6
5 **Importing the Pipe Network** 7
5.1 Apply SWMM Attributes 8
6 **Running xpSWMM and Viewing Results** 10
6.1 Running the SWMM Model 10
6.2 Viewing SWMM Output 11
7 **Running EPA SWMM and Viewing Results** 11
7.1 Viewing SWMM Model Inputs 12
7.2 Running the SWMM Model 13
7.3 Viewing SWMM Output 13
8 **Routing Hydrographs in SWMM** 14
8.1 Computing Rational Method Hydrographs 14
8.2 xpSWMM: Running and Viewing Results 15
8.3 EPA-SWMM: Running and Viewing Results 16
9 **Conclusion** 17

1 Introduction

SWMM is a hydrologic and hydraulic analysis tool used primarily designing and analyzing for storm drain systems. Hydrologic parameters are defined to obtain hydrographs and peak flows which can then be fed into the hydraulic component of the model. The hydraulic component lets engineers analyze the capacity of a current storm drain system or design a system to meet certain inflow conditions. WMS supports both EPA SWMM, developed by the Water Supply and Water Resources Division of the US Environmental Protection Agency's National Risk Management Research Laboratory, and xpswmm, developed by XP Software.

2 Objectives

In this exercise, users will set up a drainage simulation based on the Rational Method for a proposed subdivision. The objective of this exercise is to teach users the basic steps for defining a SWMM input file, running the numeric model, and viewing the results. These steps include the following:

- Define runoff coefficients
- Compute flows using the Rational Method
- Import a pipe network
- Run SWMM and view results
- Route hydrographs using SWMM and view results

3 Defining Runoff Coefficients

3.1 Reading in the Subdivision Layout and Elevations

Users will open a shapefile containing the geography of the proposed subdivision. Also, to help us in defining elevations and slopes for the drainage area, users will open a TIN (Triangulated Irregular Network) for the area:

1. Open WMS. If WMS is already open select *File* | **New** and click **No** if asked to save changes.
2. Select *File* | **Open** to access the *Open* dialog.
3. Locate the "stormrat" folder in the files for this tutorial. If needed, download the tutorial files from www.aquaveo.com.
4. Open "cougarestates.tin".
5. Right-click on the "Coverages" folder in the Project Explorer.
6. Select **New Coverage** from the pop-up menu.
7. Change the *Coverage type* to "Runoff Coefficient".
8. Select **OK**.
9. Select *File* | **Open** to access the *Open* dialog.
10. Open "cougarestates.shp".
11. Switch to the **GIS** module.
12. Select *Mapping* | **Shapes** → **Feature Objects**.
13. Select **Yes** to use all visible shapes.
14. Select **Next** in the *GIS to Feature Objects Wizard*.
15. Ensure that the Runoff coefficient is mapped.
16. Select **Next** and then **Finish**.
17. Hide "cougarestates.shp" by deselecting it in the Project Explorer.

3.2 Defining the Runoff Coefficients

Users will turn off the display of the TIN in order to better distinguish the subdivision polygons.

1. Toggle the check box for the "New" TIN off in the Project Explorer.
2. Switch to the **Map** module.
3. Choose the **Select Feature Polygon** tool.
4. Use Figure 1 to assign runoff coefficients by double-clicking each polygon and assigning the appropriate coefficient in the *Runoff Coefficient* dialog, then clicking **OK**.

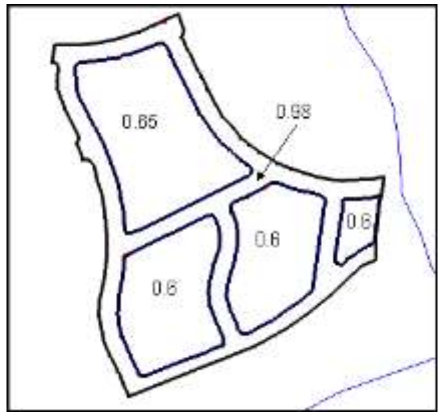


Figure 1 Runoff coefficients

4 Compute Flows from the Rational Method

Now users will set up the Rational Method analysis to compute peak flows for each subbasin. Users will use the computed peak flows in the SWMM storm drain network.

4.1 Creating the Drainage Coverage

1. Select the "Drainage" coverage from the Project Explorer to make it the active coverage.
2. Select *File* | **Open** to access the *Open* dialog.
3. Open "runoff.shp" and "drainage.shp".
4. Switch to the **GIS** module.
5. Select *Mapping* | **Shapes → Feature Objects**.
6. Select **Yes** to use all visible shapes.
7. In the *GIS to Feature Objects Wizard* select **Next**, **Next**, **Next**, and then **Finish**.

Figure 2 shows how the layout should look.

8. Hide "runoff.shp" and "drainage.shp" by deselecting their icons in the Project Explorer.

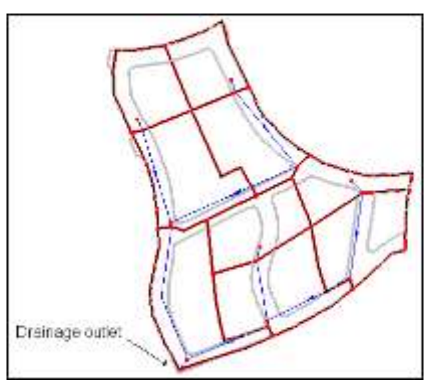


Figure 2 Drainage basins and streams for the subdivision

Before defining nodes as storm drains/drainage outlets, users will hide the Runoff Coefficient coverage in order to simplify the screen display

9. Toggle the visibility check box for the "Runoff Coefficient" coverage off in the Project Explorer.
10. Ensure that Drainage is the active coverage and that the current module is the **Map** module.
11. Choose the **Select Feature Point/Node** tool.
12. Double-click on the node labeled "Drainage Outlet" in Figure 2.
13. In the *Drainage Feature Point Type* dialog, change the *Point Type* to "Drainage Outlet" and select **OK**.
14. Select *Feature Objects* **| Compute Basin Data**.
15. The *Units* dialog will appear. Click the **Current Projection** button.
16. The *Display Projection* dialog will appear. Set the *Horizontal* system to use *no projection* and set the *Horizontal* and *Vertical units* as "U.S. Survey Feet".
17. Select **OK**.
18. Set *Basin Areas* to "Acres" and *Distances* to "Feet".
19. Select **OK**.

4.2 Computing Runoff Coefficients

Composite runoff coefficients must be computed for each drainage area. A weighted average for each drainage area is calculated from data in the Runoff Coefficient coverage. To compute the basin runoff coefficients:

1. Switch to the **Hydrologic Modeling** module.
2. Select *Calculators* **| Compute GIS Attributes**.
3. The *Compute GIS Attributes* dialog will appear. Set the *Computation type* as "Runoff coefficients".
4. Select **OK**.

Composite runoff coefficients for each drainage area are computed and displayed on the screen.

4.3 Entering Times of Concentration

The Time of Concentration for a basin can be calculated using a Time Computation coverage. For this exercise, however, it is assumed that the TC values have already been calculated, and users will input them manually.

1. Change the *Model* drop-down box (located towards the top of the WMS interface) to "Rational".
2. Choose the **Select Basin** tool.
3. Double-click on the basin icon for the basin in the lower left-hand corner of the subdivision to open the *Rational Method* dialog.

4. Enter "14" (minutes) for the *Time of Concentration*. Click **OK**.
5. Select each of the remaining drainage basins and assign Time of Concentration values using Figure 3 as a guide. Double-click each polygon, enter the TC values and click **OK** to close the dialog.

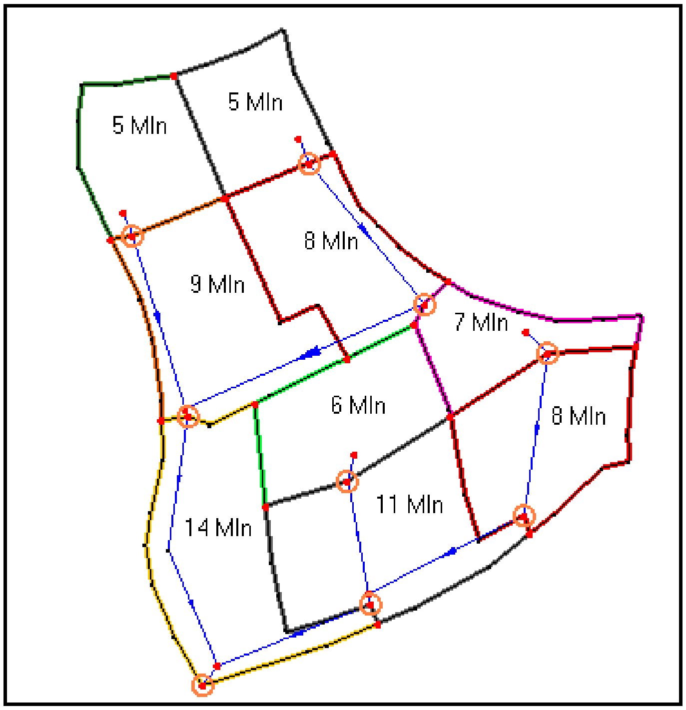


Figure 3 TC values for the drainage areas.

Users have now set up a traditional rational simulation for the Cougar Estates subdivision. The only remaining step is to define the IDF curves.

4.4 Defining Rainfall Data

1. Double click on the basin icon for the basin in the lower left hand corner of the subdivision.
2. Click on the **Compute…** button for the parameter labeled *Compute I – IDF Curves*. The *Rational Method* **- -** *IDF Computation* dialog will open.
3. Choose the *User Supplied Data* option as the *IDF curve computation*.
4. Click the **Define Storm Data** button to open the *Input Variables for IDF Curves* dialog.
5. Change the *Recurrence value* to "25" yr.
6. Enter the precipitation values shown in Figure 4.

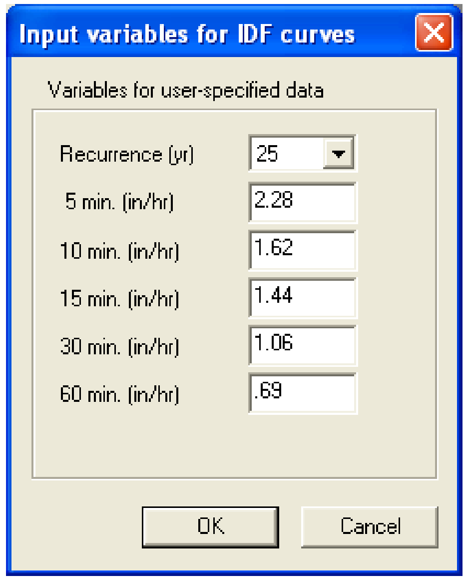


Figure 4 Values for computing the 25 yr IDF curve

7. Select **OK**.
8. Highlight the line corresponding to the 25-yr precipitation values from the window in the upper right-hand corner of the dialog.
9. Click the **Compute Intensity** button.
10. Select **Done**.
11. In the *Display* section, change the *Type* drop-down menu to “Outlets” and change the *Show* drop-down menu to “All”.
12. Click the **Compute…** button next to the *Compute I – IDF Curves* parameter for each basin and outlet. Click the **Compute Intensity** button and click **Done** until each basin and outlet has a rainfall intensity displayed in the Parameters spreadsheet.
13. Notice that a *flow rate* has now been computed for each basin and outlet. Click **OK** to exit the *Rational Method* dialog.

Users will now import a pre-defined storm drain network, assign properties to the network, and link it to the Drainage coverage.

5 Importing the Pipe Network

As mentioned above, users will be importing a pipe network to use in the SWMM model. This network was defined as a shapefile, and will be converted to feature objects in WMS. WMS can also open DXF files and convert them to feature objects. Alternatively, users can manually create a pipe network using the Create Feature Arc tool.

1. Right-click on the “Coverages” folder in the Project Explorer.
2. Select **New Coverage**.
3. Change the *Coverage type* to “Storm Drain” in the *Properties* dialog.
4. Select **OK**.

5. Select *File* | **Open** to access the *Open* dialog.
6. Open “swmm_pipes.shp”.
7. Switch to the **GIS** module.
8. Select *Mapping* | **Shapes → Feature Objects**.
9. Select **Yes** to use all visible shapes.
10. In the *GIS to Feature Objects Wizard*, select **Next**, **Next**, and **Finish**.
11. Hide “swmm_pipes.shp”.

5.1 Apply SWMM Attributes

1. Switch to the **Map** module.
2. Select *Storm Drain* | **Map → 1D Schematic**. In the *Select Model* dialog, click **OK**.

This will convert the storm drain coverage to a network of links and nodes recognized by SWMM. This command also automatically maps the elevations from the TIN to the ground elevations stored at each node. WMS has also computed the length of each storm drain link and applied a default invert elevation of eight feet below the ground elevation. The project should now look similar to Figure 5.

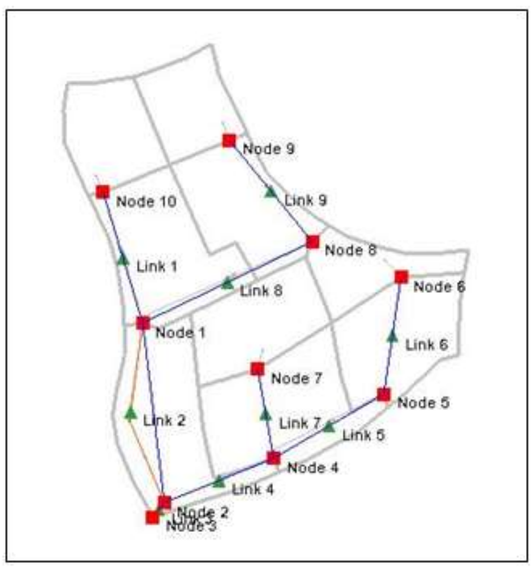


Figure 5 Link-Node Schematic for SWMM Model

3. In the Project Explorer, select the “SWMM Hydraulic Schematic” tree item.
4. Choose the **Select Hydraulic Node** tool.
5. Double-click on any node.

In the node *Properties* dialog, notice the invert elevation of Node 10 and Node 1 (according to Figure 5). Storm water is supposed to flow from Node 10 to Node 1, but the default invert elevation is higher at Node 1 than at Node 10. Users have a similar issue with respect to Node 9 and Node 8. This dialog lets us change the default invert elevations at each node.

Important Note: for the following steps, node numbers may not match node numbers in the model. Use Figure 5 as a reference to determine which nodes correspond to node and link names in the model.

6. Change the *invert elevation* of Node 1 to “4501.8”.

This elevation must be higher than that at Node 2. The elevation at Node 2 is “4501.5”, so users have dealt with this condition.

7. Change the *invert elevation* of Node 9 to “4508.0”.

This elevation is now higher than the elevation at Node 8 (“4507.5”). Now notice the column labeled Linked Outlet Name. The storm drain nodes are not currently linked to the hydrologic analysis. Users want to use the peak flows calculated in the Rational Method Analysis as inflows to the storm drain system.

8. Click **OK** on the node *Properties* dialog.
9. Right click on the “SWMM Hydraulic Schematic” in the Project Explorer and select **Link Outlets to Nodes**.
10. In the *Link Storm Drain and Drainage Nodes* dialog, click the **Auto Link** button.

This will automatically link the storm drain nodes to the drainage outlets that are within a given tolerance of the node. As users can see, Node 2 is not located close enough to any drainage outlet and so has not been linked.

11. Click **OK** to exit the *Link Storm Drain and Drainage Nodes* dialog.

Now that users have changed the elevations at the nodes and linked them to the outlets, users must also modify the upstream and downstream invert elevations of the links.

12. Choose the **Select Hydraulic Link**  tool.
13. Double-click on any link.

The link *Properties* dialog lets users specify the dimensions of the storm drain pipes. Users will assume a circular shape for all pipes and no initial flow or depth. The lengths and invert elevations are shown, but the invert elevations at Link 1 and Link 9 need to be updated to reflect the changes users made to the nodes. Users also need to define the diameter of the pipes. **Remember:** Use Figure 5 as a reference to determine which links correspond to link names in the model.

14. Change the *Downstream Invert Elevation* of Link 1 and the *Upstream Invert Elevation* of Link 2 to “4501.8”.
15. Change the *Upstream Invert Elevation* of Link 9 to “4508.0”.
16. In the *Diameter/Height* column of the link properties spreadsheet, enter a diameter of “4” feet for each link.
17. Click **OK** to exit the link *Properties* dialog.

Now users are ready to export a SWMM project file and run SWMM. Section 7 describes how to export and run an xpswmm model. To export and run an EPA SWMM model, skip to Section 7.

6 Running xpSWMM and Viewing Results

This section describes how to export and run an xpswmm model. The xpswmm model offers many features that can be used to modify the hydrologic analysis, customize the drain system, and analyze the model output. The features covered here represent just a few of the features users might find helpful.

1. Select *SWMM* **| Run xpswmm…**
2. Name the file *"*swmmtut.xpx" in the *Select an XPX File* dialog and click **OK**.

WMS saves an xpswmm project file and as long as xpswmm is loaded on the computer, it launches xpswmm and automatically opens the project users just created.

6.1 Running the SWMM Model

1. When xpswmm opens, if prompted click **OK** to convert the version 10.0 database and close the window displaying errors and warnings.
2. The xpswmm window should now display an image similar to that shown in Figure 6.

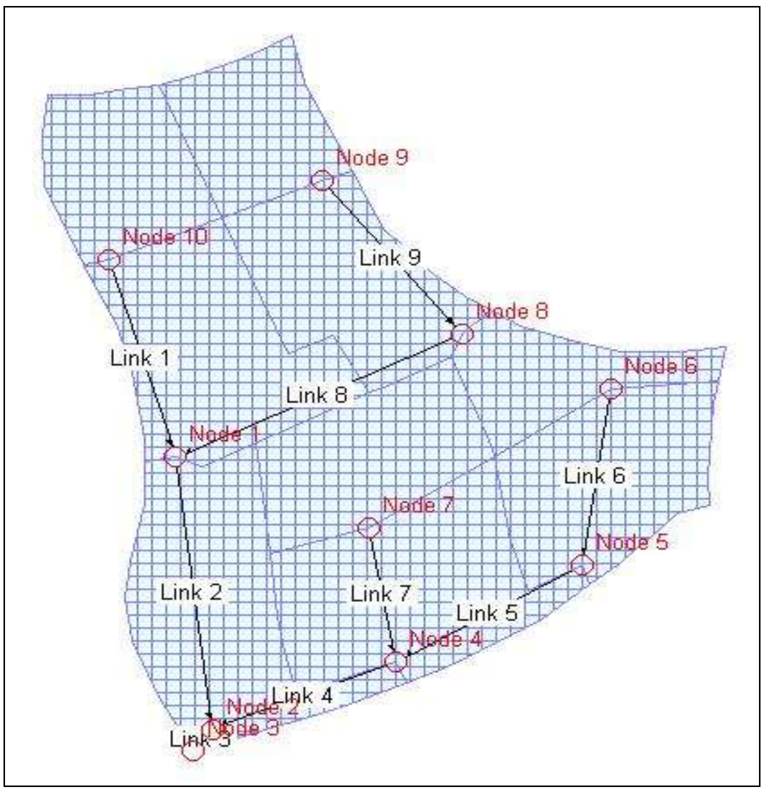


Figure 6 Cougar Estates Storm Drain Schematic from xpswmm

3. Double click on the node labeled "Node 9" in Figure 6.

The *Node Data* dialog displays the ground elevation, invert elevation, and inflow data as well as the ponding type and initial depth if specified in WMS.

4. Click **OK** to close the *Node Data* dialog.

5. Double-click on the link labeled “Link 9” in Figure 6.

The *Conduit Data* dialog displays the pipe data users imported from WMS. On the main window, notice the edit fields for initial flow and initial depth.

6. Click on the button labeled **Conduit Profile**.

Here users find information about the pipe dimensions and material as well as the invert and ground elevations for the upstream and downstream ends of the pipe.

7. Click **OK**, then **OK** again to close both dialogs.

Now before users can run SWMM, they need to specify an outfall condition at the downstream outlet of the storm drain system and set up the run time parameters.

8. Double-click on the most downstream link in the network (Link 3 in Figure 6)
9. Click on the **Outfall** button.

For this exercise, users will assume a free outfall at the downstream outlet.

10. Click on the Type 1, **Free Outfall** button.
11. In the **Outfall Control** dialog, select *Use normal depth* (Yn).
12. Click **OK**, **OK**, then **OK** to exit all the dialogs.
13. Select *Configuration* **|** *Job Control* **| Hydraulics**.
14. In the *Job Control* dialog, change the *ending day* to “1”, and the *ending hour* to “3” and click **OK** to exit the dialog.
15. Select *Analyze* **| Solve** or click the **Solve** button.
16. Name the file “swmmtut.out” and click **Save**.

6.2 Viewing SWMM Output

1. After the model is done running, select any link and select *Results* **| Review Results**.

This option will open a plot showing the upstream and downstream water surface elevations at any point throughout the model run and the computed flows and velocities.

2. Click **Close** and with the same link selected, select *Results* **| Dynamic Section Views**.
3. Click the **Play** button.

This option will play an animation of the storm flow as it moves through the selected link. It displays both a profile view and cross sectional view of the flow through the pipe and the flow graph users saw in the previous plot. Feel free to explore the rest of the options under the Results menu or close xpswmm. Leave the WMS project open for the next example.

7 Running EPA SWMM and Viewing Results

This section describes how to export and run an EPA SWMM model. The EPA SWMM model offers many features that can be used to modify the hydrologic analysis, customize the drain system, and analyze the model output. The features covered here represent just a few of the features users might find helpful.

1. Select *SWMM* **| Run EPA SWMM…**
2. Name the file "epaswmmtut.inp" and click **Save**.

WMS saves an EPA SWMM project file and as long as EPA SWMM is loaded on the computer, it launches EPA SWMM and automatically opens the project users just created.

7.1 Viewing SWMM Model Inputs

1. The EPA SWMM window should display an image similar to that shown in Figure 7.

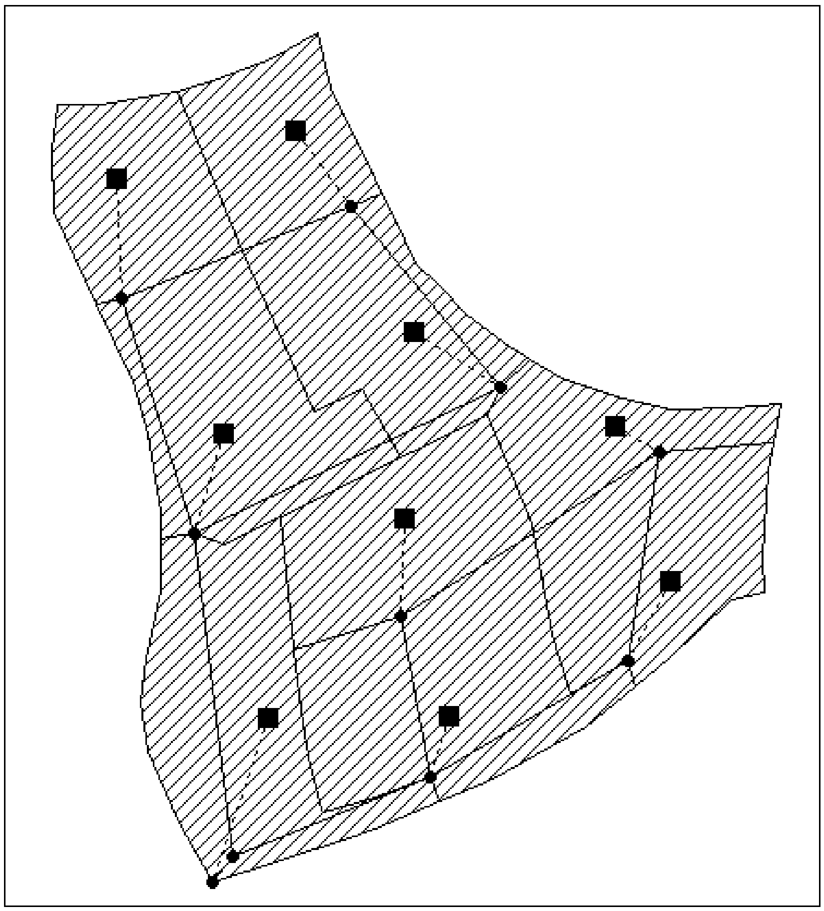


Figure 7 Cougar Estates Storm Drain Schematic from EPA SWMM

2. In the SWMM Data Explorer, expand the *Hydraulics* heading, then the *Links* heading, and select **Conduits**.
3. In the *Conduits* window below, select one of the Links.
4. Double-click the blinking link icon corresponding to the selected link.

5. In the *Conduit Link Properties* window, notice the link shape, maximum depth, and length which were specified and computed in WMS and exported to EPA SWMM.
6. Close the *Conduit Link Properties* window.
7. In the *SWMM Data Explorer*, expand the *Hydraulics* heading, then the *Nodes* heading, and select **Junctions**.
8. In the *Junctions* window below, select a Node.
9. Double-click on the blinking node icon corresponding to the node users selected.
10. In the *Junction Node Properties* window, notice the Invert Elevation of the node which was computed in WMS and exported to EPA SWMM.
11. Close the *Junction Node Properties* window.
12. In the *SWMM Data Explorer*, expand the *Hydrology* heading and select **Subcatchments**.
13. In the *Subcatchments* window below, select a subcatchment.
14. Double-click on the blinking subcatchment icon corresponding to the subcatchment users selected.
15. In the *Subcatchment Properties* window, notice that the subcatchment area has already been computed and imported from WMS.

Feel free to explore the rest of the model inputs within the EPA SWMM interface.

7.2 Running the SWMM Model

1. On the EPA SWMM menu, select *Project* **| Run Simulation**.

In the *Run Status* window, users will notice that the run was successful.

2. On the *Run Status* window, click **OK**.

7.3 Viewing SWMM Output

1. Select *Report* **| Status**.
2. Scroll through the report and view the output. Notice that the size of the pipes could be reduced and the flow would still be contained in the pipes without surcharges. Also notice that the Rainfall/Runoff model included with SWMM was not run because the discharges at the nodes have already been defined. If users want to run the Rainfall/Runoff model, they would need to turn this option on in the project options and they would probably want to remove the inflows at each of the nodes since these would be computed using the hydrologic model.
3. Select the most downstream node in the model.
4. Choose the **Profile Plot**  button.
5. Select the "+" button next to the End Node. Now select one of the most upstream nodes and select the "+" button next to the Start Node.

6. Select the "**Find Path**" button, and then select **OK**. A plot will appear showing the results of the steady state analysis with the rational flowrates. The plot should look similar to Figure 8.

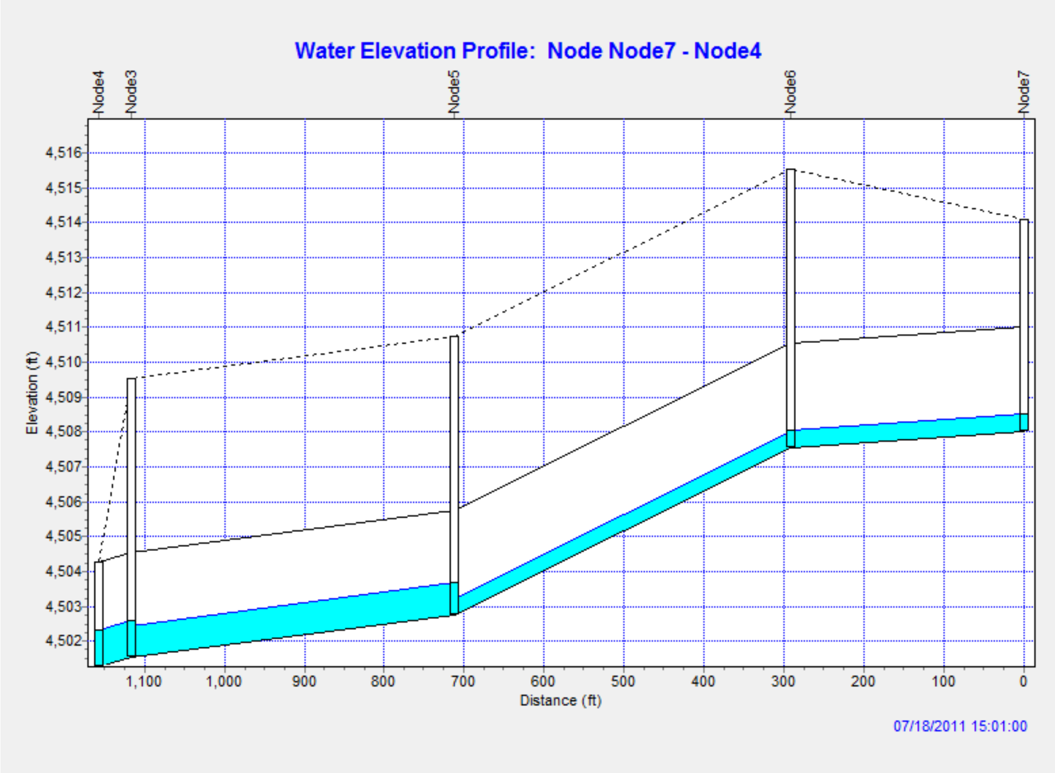


Figure 8 Elevation v. Distance Plot from an upstream to a downstream node in the model

Feel free to explore the rest of the EPA SWMM post processing tools to view the results from the Cougar Estates model run. When users are ready to proceed to the next section, close EPA SWMM (users may want to save the current simulation results if prompted) and leave the WMS window running.

8 Routing Hydrographs in SWMM

In the previous example, users used the Rational Method to compute peak flows which they then fed into a SWMM model. This time, users will use the Rational Method to compute hydrographs at each outlet. Then users will route these hydrographs through the SWMM model.

8.1 Computing Rational Method Hydrographs

1. In the WMS project, select the "Drainage" coverage to make it active.
2. Switch to the **Hydrologic Modeling** module.
3. Choose the **Select Outlet** tool.
4. Using the **Select Outlet** tool, double-click on the most downstream outlet (it will most likely be hidden behind the SWMM node icon).

5. In *Display*, change the *Show* drop down box to "Selected".
6. Click the **Compute…** button next to the *Compute Hydrographs* parameter for the selected outlet and click **Done** in the *Rational Method Hydrographs* dialog.
7. Click **OK** to exit the *Rational Method* dialog.
8. Notice that each basin and outlet icon should show a hydrograph icon next to it as in Figure 9.

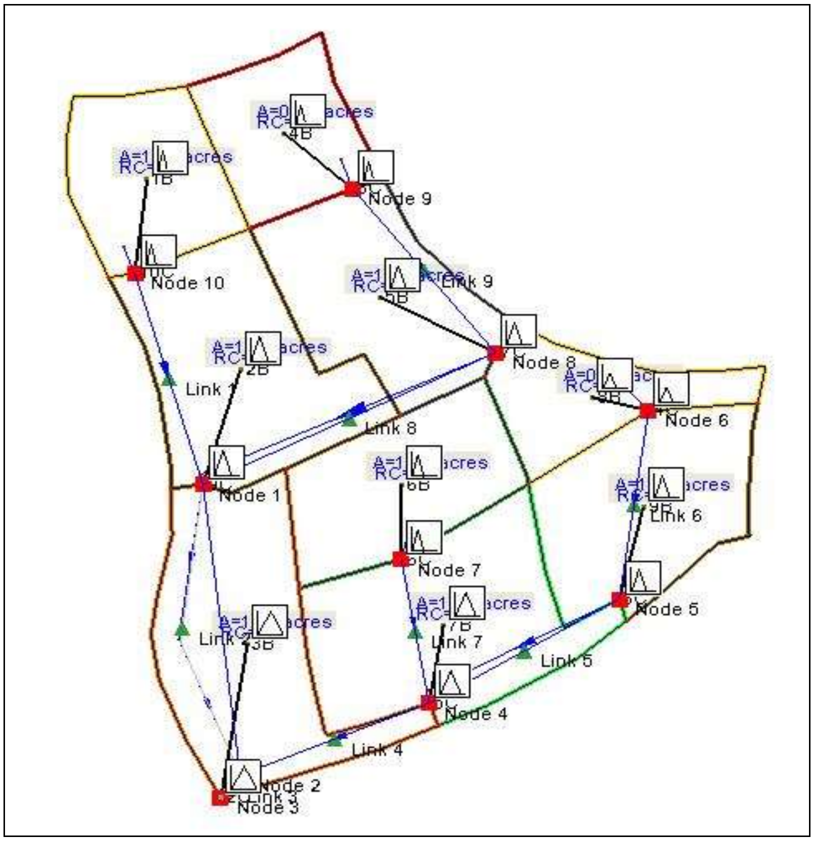


Figure 9 Rational Method hydrographs for SWMM

9. In the Project Explorer, select the "SWMM Hydraulic Schematic" tree item.
10. Depending on the model users wish to run, select *SWMM* **| Run xpswmm…** or *SWMM* **| Run EPA SWMM….**
11. Name the file "*swmm_hyd.xpx"* (or "swmm_hyd.inp") and click **OK**.

8.2 xpSWMM: Running and Viewing Results

1. In the xpswmm interface, double-click on any node.

Notice that the *Inflow* edit field under the *Constant Inflow* category now says "0.0". This is because users have now defined a Time Series Inflow. Notice how the User Inflow button is now checked.

2. Double-click the **User Inflow** button to view the hydrograph users will now be routing through the selected node.

3. Click **OK**, then **OK** to close both dialogs.
4. Double-click on the most downstream link in the network (Link 3 in Figure 6).
5. Click on the **Outfall** button.
6. Click on the Type 1, **Free Outfall** button.
7. In the *Outfall Control* dialog, select *Use normal depth* (Yn).
8. Click **OK**, **OK**, then **OK** to exit all the dialogs.
9. Select *Configuration* **|** *Job Control* **| Hydraulics**.
10. In the *Job Control* dialog, change the *ending day* to “1”, and the *ending hour* to “3” and then click **OK** to close the dialog.
11. Select *Analyze* **| Solve** or click the **Solve** button.
12. Name the file “swmm_hyd.out” and click **Save**.
13. View the results as before and notice the difference between using a constant inflow and a time series inflow.

8.3 EPA-SWMM: Running and Viewing Results

1. On the EPA SWMM menu, select *Project* **| Run Simulation**.

In the *Run Status* window, users will notice that the run was successful.

2. On the *Run Status* window, click **OK**.
3. Select the conduit connecting the most downstream outlet to the junction just upstream.
4. Choose the **Time Series Plot** button.
5. Select any other links where users would like to view output and select the "+" button in the *Time Series Plot* window to add the plot for these links.
6. After all the desired links have been added, click **OK** on the *Time Series Plot* window to view the Flow v. Time plot for the selected storm drains. Users could also view the water depths in the storm drains using this same approach. After the plot is displayed, right-click on the plot to view the Graph Options. Select the *Horizontal Axis* tab and enter a *Maximum axis value* of “3.0” (hours) and an *increment* of “0.25” (hours). Click **OK** to view the final plot.
7. The plot should look similar to Figure 10.

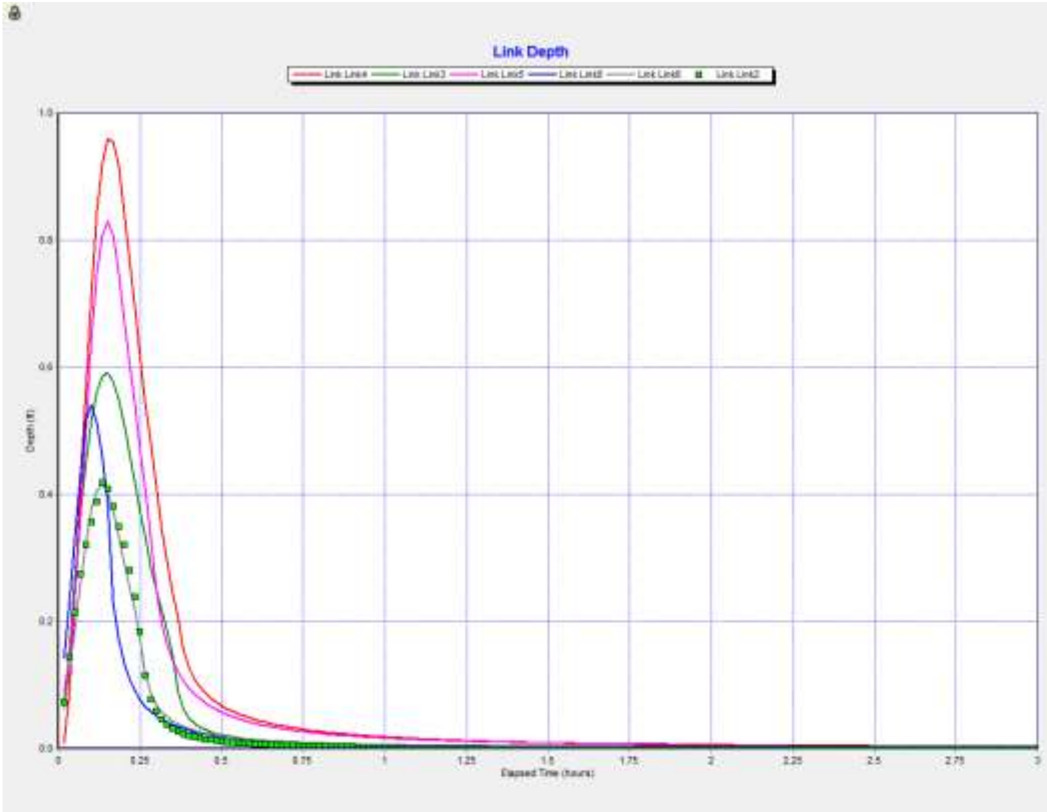


Figure 10 Plot of Depth vs. Time for several links in the model

9 Conclusion

This concludes the SWMM modeling tutorial. Users should feel free to explore the WMS interface or the SWMM interface more thoroughly if they wish.

In this exercise, users should have learned to:

- Define runoff coefficients
- Compute flows using the Rational Method
- Import a pipe network
- Run SWMM and view results
- Route hydrographs using SWMM and view results