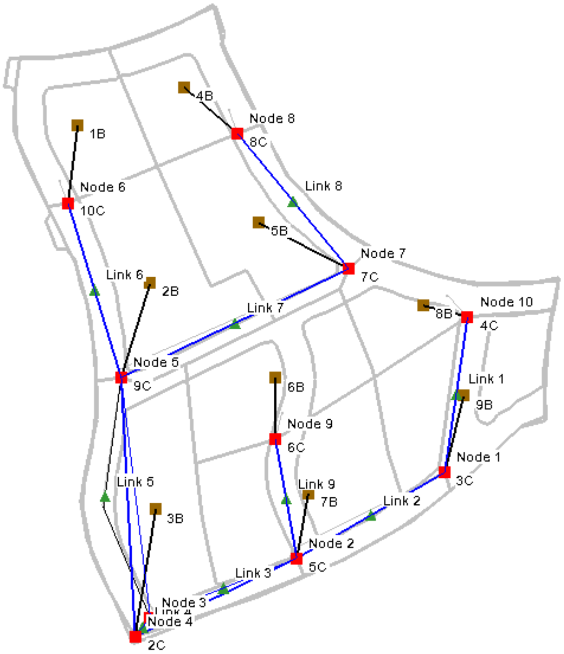
WMS 11.4 Tutorial

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 EPA SWMM or XPSWMM.

Prerequisite Tutorials

- Rational Method Interface
- Using TINs

Required Components

- WMS Core
- EPA SWMM Model
- XPSWMM Support

Time

- 30–60 minutes

1 Introduction

The Storm Water Management Model (SWMM) is a hydrologic and hydraulic analysis tool used primarily for designing and analyzing 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 exporting data file formats for 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.

In this tutorial, a drainage simulation will be set up based on the rational method for a proposed subdivision. The basic steps for defining a SWMM input file, running the numeric model, and viewing the results will be shown. These steps include defining runoff coefficients, computing flows using the rational method, importing a pipe network, running SWMM and viewing the results, and routing hydrographs using SWMM and viewing the results.

2 Getting Started

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

1. If necessary, launch WMS.
2. If WMS is already running, press *Ctrl-N* or select *File* **| New…** to ensure that the program settings are restored to their default state.
3. A dialog may appear asking to save changes. Click **Don't save** to clear all data.

The Graphics Window of WMS should refresh to show an empty space.

3 Defining Runoff Coefficients

3.1 Importing the Subdivision Elevations

A TIN may be used to help define elevations and slopes for the drainage area.

1. Select *File* | **Open…** to bring up the *Open* dialog.
2. Select "All Files (*.*)" from the *Files of type* drop-down.
3. Browse to the *stormrat\stormrat* folder and select "cougarestates.tin".
4. Click **Open** to import the TIN and exit the *Open* dialog.
5. Right-click on " Coverages" in the Project Explorer and select **New Coverage** to bring up the *Properties* dialog.
6. Select "Runoff Coefficient" as the *Coverage type* and click **OK** to close the *Properties* dialog.

The project should appear similar to Figure 1.

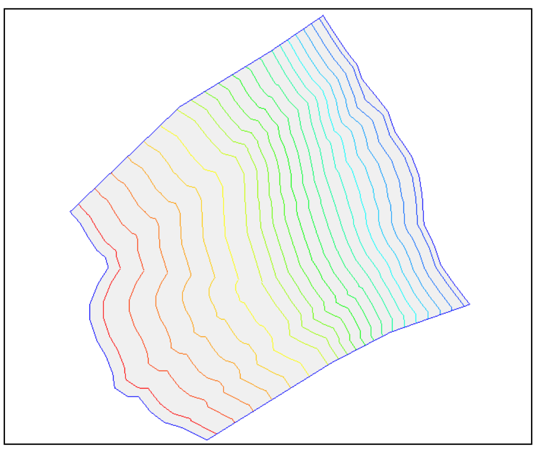


Figure 1 The imported TIN

3.2 Importing the Subdivision Layout

The next step is to open a shapefile containing the geography of the proposed subdivision.

1. Click **Open** to bring up the *Open* dialog.
2. Select "Shapefiles (*.shp)" from the *Files of type* drop-down.
3. Select "cougarestates.shp" and click **Open** to import the shapefile and exit the *Open* dialog.
4. Switch to the **GIS Module** .
5. Select *Mapping* | **Shapes** → **Feature Objects** to bring up the *GIS to Feature Objects Wizard* dialog (*Step 1 of 3*).
6. Click **Yes** if asked to use all visible shapes.

7. Click **Next** to go to the *Step 2 of 3* page of the *GIS to Feature Objects Wizard*.
8. In the *Mapping Preview* section, select “runoff coefficient” from the *Mapping* drop-down in the *RUNOFFC* column.
9. Click **Next** to go to the *Step 3 of 3* page of the *GIS to Feature Objects Wizard* dialog.
10. Click **Finish** to close the *GIS to Feature Objects Wizard* dialog.
11. Turn off “ cougarestates.shp” and “ new” in the Project Explorer.

This allows for better visibility of the subdivision polygons.

3.3 Defining the Runoff Coefficients

1. Switch to the **Map Module** .
2. Using the **Select Feature Polygon** tool, double-click on the upper polygon (labeled “1” in Figure 2) to bring up the *Runoff Coefficient* dialog.
3. Enter “0.65” in the *Enter the coefficient (0.0–1.0) for selected polygon* field and click **OK** to close the *Runoff Coefficient* dialog.
4. Repeat steps 2–3 for each of the other polygons in the subdivision, using the coefficients in the table below and referencing Figure 2:

Polygon	Coefficient
lower left (2)	0.6
lower center (3)	0.6
lower right (4)	0.6
streets (5)	0.98

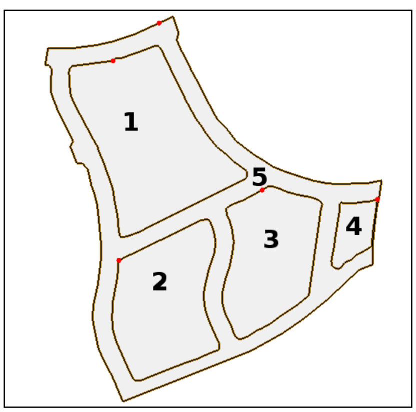


Figure 2 Subdivision polygons

4 Compute Flows from the Rational Method

The next step is to set up the rational method analysis to compute peak flows for each sub-basin. The computed peak flows will then be used in the SWMM storm drain network.

4.1 Creating the Drainage Coverage

1. Select " Drainage" in the Project Explorer to make it active.
2. Click **Open** to bring up the *Open* dialog.
3. Select "Shapefiles (*.shp)" from the *Files of type* drop-down.
4. Holding down the *Ctrl* key, select both "runoff.shp" and "drainage.shp".
5. Click **Open** to import both shapefiles and exit the *Open* dialog.
6. Switch to the **GIS Module** .
7. Select *Mapping* **| Shapes** → **Feature Objects** to bring up the *GIS to Feature Objects Wizard* dialog (*Step 1 of 4*).
8. Make sure only "runoff.shp" and "drainage.shp" are selected for mapping. If any other layers are selected, turn them off.
9. Click **Next** to go to the *Step 2 of 4* page of the *GIS to Feature Objects Wizard* dialog.
10. Click **Next** to go to the *Step 3 of 4* page of the *GIS to Feature Objects Wizard* dialog.
11. Click **Next** to go to the *Step 4 of 4* page of the *GIS to Feature Objects Wizard* dialog.
12. Click **Finish** to close the *GIS to Feature Objects Wizard* dialog.
13. Turn off " runoff.shp", " drainage.shp", and " Runoff Coefficient" in the Project Explorer.

This should simplify the display (Figure 3).

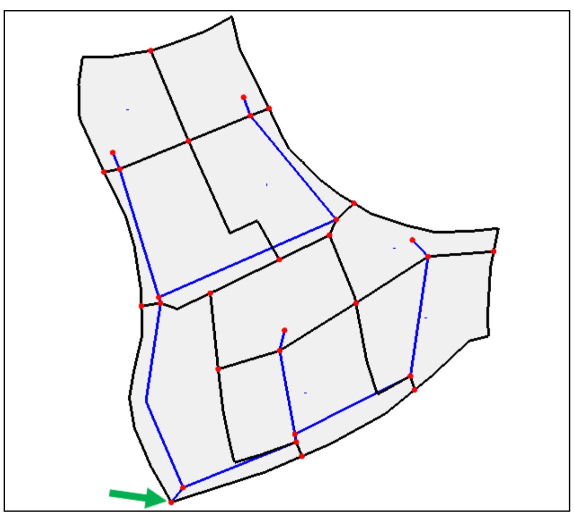


Figure 3 Drainage basins and streams for the subdivision

14. Select " Drainage" to make it active.
15. Switch to the **Map Module** .
16. Using the **Select Feature Point/Node** tool, double-click on the node indicated by the arrow in Figure 3 to bring up the *Drainage Feature Point Type* dialog.

17. In the *Type* section, select *Drainage Outlet* and click **OK** to close the *Drainage Feature Point Type* dialog.
18. Select *Feature Objects* | **Compute Basin Data…** to bring up the *Units* dialog.
19. In the *Model units* section, click **Current Projection…** to bring up the *Display Projection* dialog.
20. In the *Horizontal* section, select *No projection* and select "Feet (U.S. Survey)" from the *Units* drop-down.
21. In the *Vertical* section, select "Feet (U.S. Survey)" from the *Units* drop-down.
22. Click **OK** to close the *Display Projection* dialog.
23. In the *Parameter units* section, select "Acres" from the *Basin Areas* drop-down and "Feet" from the *Distances* drop-down.
24. Click **OK** to close the *Units* dialog.

The area of each basin should now be displayed (Figure 4).

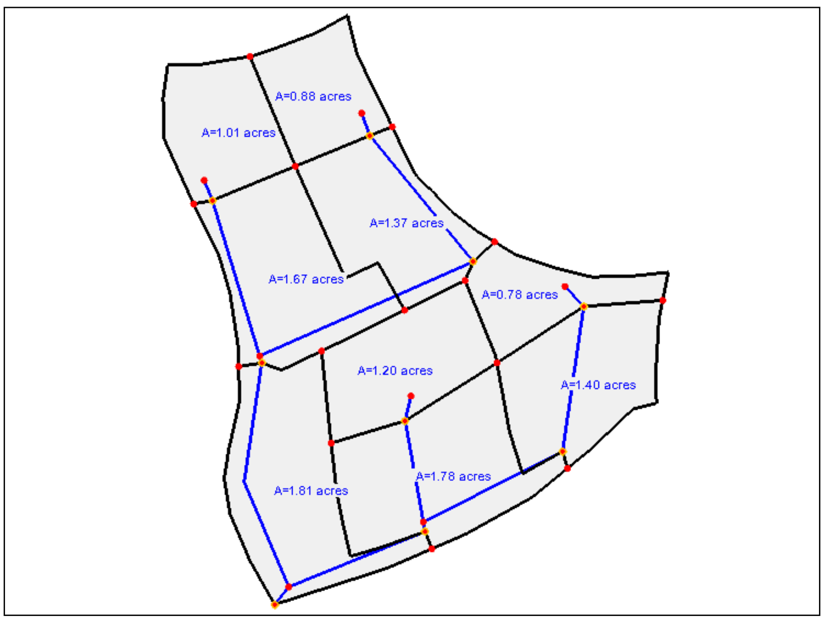


Figure 4 Drainage basins with areas displayed

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…** to bring up the *Compute GIS Attributes* dialog.
3. In the *Computation* section, select "Runoff coefficients" from the drop-down.
4. Click **OK** to close the *Compute GIS Attributes* dialog and compute the basin runoff coefficients.

Composite runoff coefficients for each drainage area are computed and displayed next to the area in each basin (Figure 5).

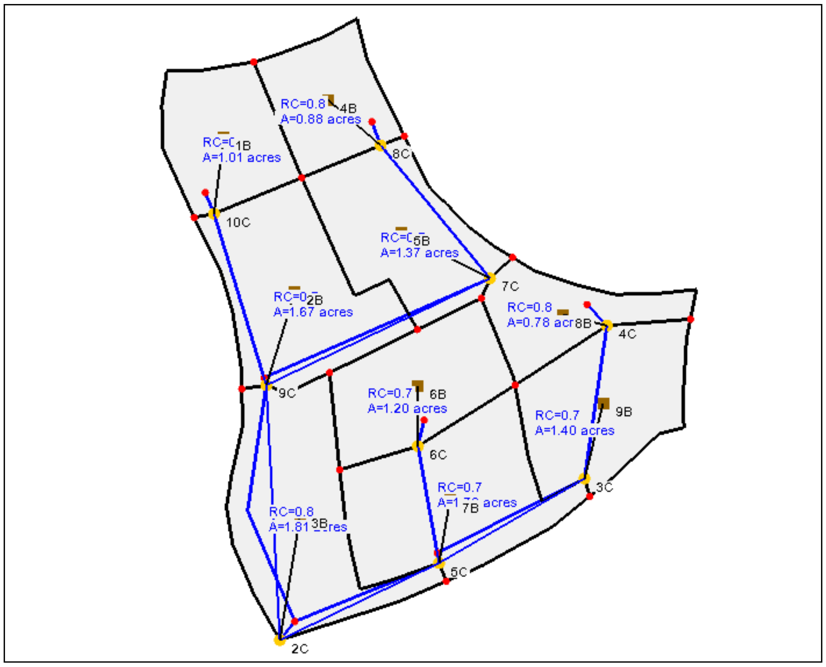


Figure 5 Areas and runoff coefficients

4.3 Entering Times of Concentration

The Time of Concentration (T_C) for a basin can be calculated using a time computation coverage. In this tutorial, the T_C values were pre-calculated and are provided below.

1. Select "Rational" from the *Model* drop-down (Figure 6).

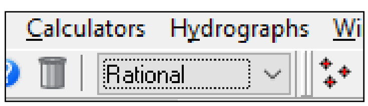


Figure 6 Rational model selected

2. Using the **Select basin** tool, double-click on basin 1B (see Figure 7) to open the *Rational Method* dialog.
3. In the *Parameters* section, enter "5.0" as the *Time of Concentration (Tc)*.
4. Select "All" from the *Show* drop-down in the *Display* section.
5. Enter the correct *Time of Concentration (Tc)* for each basin according to the table below.
6. Click **OK** to close the *Rational Method* dialog.

Basin ID	T_C
2B	9.0
3B	14.0
4B	5.0

5B	8.0
6B	6.0
7B	11.0
8B	7.0
9B	8.0

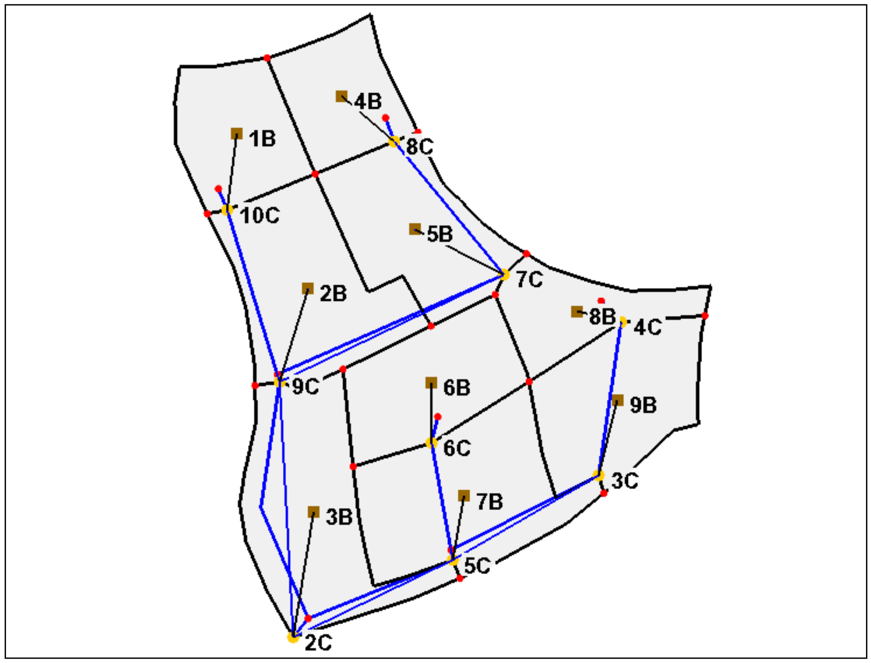


Figure 7   Drainage basins

A traditional rational simulation has now been set up for the Cougar Estates subdivision. The only remaining step is to define the IDF curves.

4.4 Defining Rainfall Data

1. Using the **Select basin** tool, double-click on basin 3B to bring up the *Rational Method* dialog.
2. In the *Display* section, select "Selected" from the *Show* drop-down.
3. Click **Compute…** on the *Compute I – IDF Curves* to bring up the *Rational Method —IDF Computation* dialog.
4. In the *IDF curve computation* section, select *User Supplied Data* and click **Define Storm Data…** to open the *Input variables for IDF curves* dialog.
5. Select "25" from the *Recurrence (yr)* drop-down.
6. Enter the precipitation values from the following table:

5 min. (in/hr)	2.28
10 min. (in/hr)	1.62
15 min. (in/hr)	1.44

30 min. (in/hr)	1.06
60 min. (in/hr)	0.69

7. Click **OK** to close the *Input variables for IDF curves* dialog.
8. In the upper right corner of the dialog, select “25-yr.” from the list.
9. In the *Intensity computation* section, click **Compute Intensity**.
10. Click **Done** to close the *Rational Method — IDF Computation* dialog.

The rainfall intensity has now been computed for basin 3B.

11. In the *Display* section, select “Outlets” from the *Type* drop-down and “All” from the *Show* drop-down.

Notice that all of the outlets and basins are showing in the *Parameters* section.

12. Click the **Compute…** button next to the *Compute I – IDF Curves* parameter for basin 1B to bring up the *Rational Method — IDF Computation* dialog.

Notice that “25-yr.” is already selected in the text area at the upper right.

13. In the *Intensity computation* section, click **Compute Intensity** and click **Done** to close the *Rational Method — IDF Computation* dialog.
14. Repeat steps 12-13 for each remaining basin and outlet until all of them have a value in the *Rainfall Intensity (I)* row in the *Rational Method* dialog.

Notice that the values in the *Flowrate (Q)* row for each basin and outlet have also been computed.

15. Click **OK** to exit the *Rational Method* dialog.

5 Importing the Pipe Network

Now to import a pre-defined storm drain network, assign properties to the network, and link it to the “ Drainage” coverage. As mentioned above, a pipe network will be imported 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 CAD files and convert them to feature objects. Alternatively, it is possible to manually create a pipe network using the **Create Feature Arc** tool.

1. Select then right-click “ Coverages” in the Project Explorer and select **New Coverage** to bring up the *Properties* dialog.
2. Select “Storm Drain” from the *Coverage type* drop-down and click **OK** to close the *Properties* dialog.
3. Click **Open** to bring up the *Open* dialog.
4. Select “Shapefiles (*.shp)” from the *Files of type* drop-down.
5. Select “swmm_pipes.shp” and click **Open** to import the shapefile and exit the *Open* dialog.
6. Switch to the **GIS Module**.
7. Select *Mapping* **| Shapes** → **Feature Objects** to bring up the *GIS to Feature Objects Wizard* dialog (*Step 1 of 3*).

8. Make sure that "swmm_pipes.shp" is the only shapefile selected for mapping. If other shapefiles are displayed, turn them off.
9. Click **Next** to go to the *Step 2 of 3* page of the *GIS to Feature Objects Wizard* dialog.
10. Click **Next** to go to the *Step 3 of 3* page of the *GIS to Feature Objects Wizard* dialog.
11. Click **Finish** to close the *GIS to Feature Objects Wizard* dialog.
12. Turn off " swmm_pipes.shp" in the Project Explorer.

5.1 Mapping SWMM Attributes

1. Turn on " new" in the Project Explorer.

The TIN must be visible in order for the elevations to properly map.

2. Switch to the **Map Module** .
3. Select *Storm Drain* **| Map → 1D Schematic** to bring up the *Select Model* dialog.
4. Select "SWMM" from the drop-down at the bottom of the dialog and click **OK** to close the *Select Model* dialog.
5. Click **OK** when advised the active TIN was used to assign elevations.

This converts the storm drain coverage to a network of links and nodes recognized by SWMM (Figure 8) and automatically maps the elevations from the TIN to the ground elevations stored at each node. WMS then computes the length of each storm drain link and applies a default invert elevation of eight feet below the ground elevation.

5.2 Node Invert Elevations and Outlet Linking

1. Turn off everything in the Project Explorer except for the " SWMM Hydraulic Schematic".
2. Select " SWMM Hydraulic Schematic" in the Project Explorer to make it active.
3. Using the **Select Hydraulic Node Tool** , double-click on any node to bring up the *Hydraulic Properties* dialog.
4. Select "All" from the *Show* drop-down.

Notice the invert elevation of Node 6 and Node 5. Storm water is supposed to flow from Node 6 to Node 5, but the default invert elevation is higher at Node 5 than at Node 6 (about "4502.8" versus about "4502.2", respectively). A similar issue exists with respect to Node 8 and Node 7. This dialog allows the default invert elevations to be changed at each node.

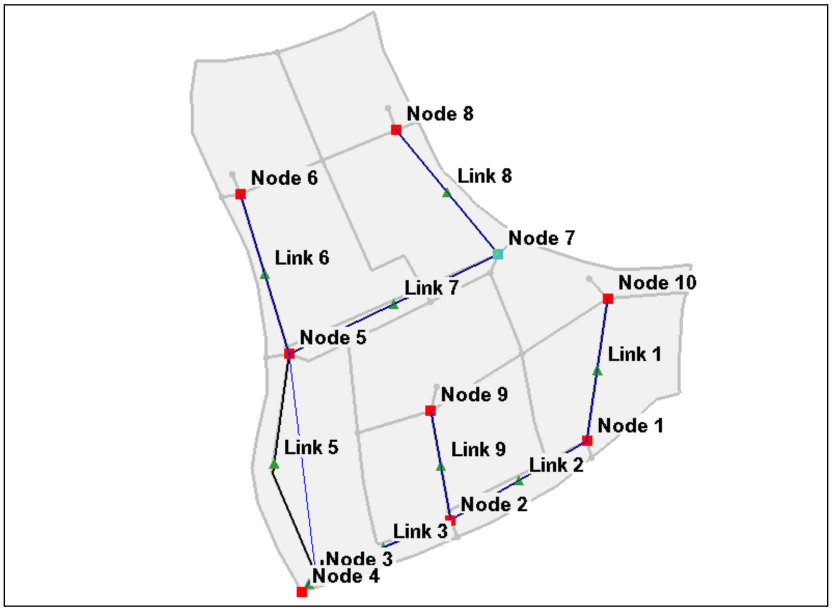


Figure 8 Link-node schematic for SWMM model

Note that node numbers mentioned below may not match node numbers in the model in the Graphics Window. Use Figure 8 as a reference to determine which nodes correspond to node and link names in the model.

5. Enter "4501.8" in the *Invert Elevation* column on the *Node 5* row.

This elevation must be higher than that at Node 3. The elevation at Node 3 is about "4501.5", which is acceptable.

6. Enter "4508.0" in the *Invert Elevation* column on the *Node 8* row.

This elevation is now higher than the elevation at Node 7 ("4507.5").

Now notice the *Linked Outlet Name* column. The storm drain nodes are not currently linked to the hydrologic analysis. The peak flows calculated in the Rational Method Analysis should be used as inflows to the storm drain system.

7. Click **OK** to close the *Hydraulic Properties* dialog.
8. Right-click " SWMM Hydraulic Schematic" and select **Link Outlets to Nodes…** to bring up the *Link Storm Drain and Drainage Nodes* dialog.
9. Click **Auto Link**.

This automatically links the storm drain nodes to the drainage outlets within a given tolerance of the node. Notice that Node 3 is not located close enough to any drainage outlet and so has not been linked.

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

5.3 Link Invert Elevations and Diameters

Now that the node elevations have been changed and linked to the outlets, the upstream and downstream invert elevations of the links must also be modified.

1. Using the **Select Hydraulic Link Tool** , double-click on any link to bring up the *Hydraulic Properties* dialog.
2. Select "All" from the *Show* drop-down.

This dialog allows specifying the dimensions and other parameters of the storm drainpipes. The lengths and invert elevations are shown, but the invert elevations at Link 6 and Link 8 need to be updated to reflect the changes made to the nodes. The diameter of the pipes also needs to be defined. As before, use Figure 8 as a reference.

3. Enter “4501.8” in the *Downstream Invert Elevation* column on the *Link 6* row.
4. Enter “4501.8” in the *Upstream Invert Elevation* column on the *Link 5* row.
5. Enter “4508.0” in the *Upstream Invert Elevation* column on the *Link 8* row.
6. On the empty row at the top, in the *Diameter/Height* column, enter “4.0” to assign to all rows.
7. Click **OK** to exit the *Hydraulic Properties* dialog.

Now everything is ready to export an SWMM project file and run SWMM. Section 6 describes how to export and run an SWMM model using the EPA SWMM program. See the steps in Section 8 to export to xpsmm format.

6 EPA SWMM

EPA SWMM is a version of SWMM developed by the Water Supply and Water Resources Division of the U.S. Environmental Protection Agency’s National Risk Management Research Laboratory. This section requires the EPA SWMM model be installed.

6.1 Exporting and Opening an EPA SWMM File

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. A few of these helpful features are discussed here.

1. Select *SWMM* **| Run EPA SWMM…** to bring up the *Select an EPA SWMM File* dialog.
2. Select “EPA SWMM file (*.inp)” from the *Save as type* drop-down.
3. Enter “epaswmmtut.inp” and click **Save** to export the file, close the *Select an EPA SWMM File* dialog, launch EPA SWMM, and open the exported file in EPA SWMM.

6.2 Viewing the SWMM Inputs

Many of the model inputs were calculated by and imported from WMS. The EPA SWMM window should display a storm drain schematic (Figure 9).

1. On the *Project* tab of the SWMM Data Explorer, expand *Hydraulics* and *Links*, then select “Conduits”.

A “Conduits” section will appear below the *Project* tab section of the SWMM Data Explorer.

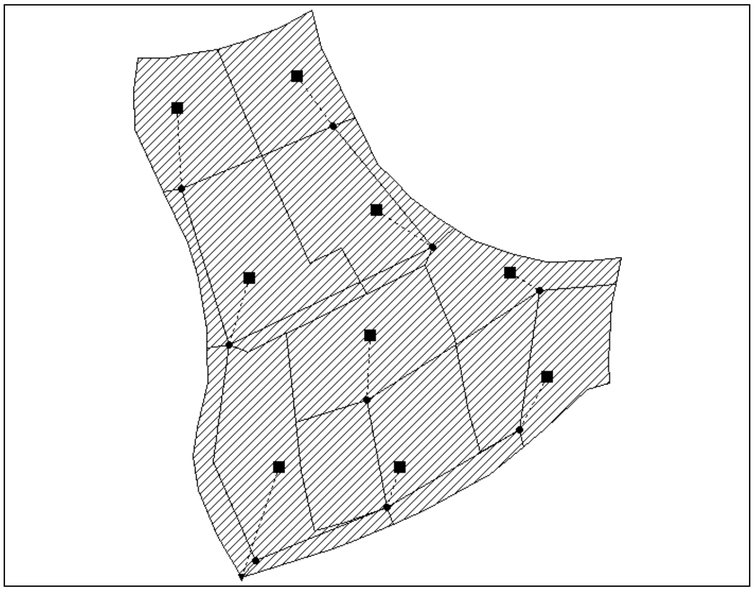


Figure 9 Cougar Estates storm drain schematic in EPA SWMM

2. In the “Conduits” section, select “Link1”.
3. Double-click the blinking link icon corresponding to the selected link to bring up the *Conduit Link1* dialog.

Notice that the link shape, maximum depth, and length specified and computed in WMS were exported to EPA SWMM.

4. Click the ✕ in the top right corner to close the *Conduit Link1* dialog.
5. In the SWMM Data Explorer, expand *Nodes* and select “Junctions”.

A “Junctions” section will appear below the *Project* tab section of the SWMM Data Explorer.

6. In the “Junctions” section, select “Node1”.
7. Double-click on the blinking node icon corresponding to the node selected to bring up the *Junction Node1* dialog.

Notice that the invert elevation of the node that WMS computed was exported to EPA SWMM.

8. Click the ✕ in the top right corner to close the *Junction Node1* dialog.
9. In the SWMM Data Explorer, expand *Hydrology* and select “Subcatchments”.

A “Subcatchments” section will appear below the *Project* tab section of the SWMM Data Explorer.

10. In the “Subcatchments” section, select “1B”.
11. Double-click on the blinking subcatchment icon corresponding to the subcatchment selected to bring up the *Subcatchment 1B* dialog.
12. Click the ✕ in the top-right corner to close the *Subcatchment 1B* dialog.

Notice that the subcatchment area was computed by and imported from WMS. Feel free to explore the rest of the model inputs within the EPA SWMM interface.

6.3 Running SWMM and Viewing Output

1. In EPA SWMM, select *Project* | **Run Simulation** to bring up the *Run Status* dialog.
2. Once the run completes successfully, click **OK** to close the *Run Status* dialog.
3. Select *Report* | **Status** to bring up the *Status Report* screen within the EPA SWMM Main Graphics Window.
4. Review the output. When finished viewing the output, click the in the top-right corner to close the *Status Report* dialog.

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 needing to run the Rainfall/Runoff model, turn this option on in the project options. Then remove the inflows at each of the nodes since these would be computed using the hydrologic model.

5. Select *Window* | **1 Study Area Map** to switch back to the schematic.
6. Select the drainage outlet node at the bottom of the schematic.
7. Click **Create a Profile Plot** to bring up the *Profile Plot Selection* dialog (Figure 10).

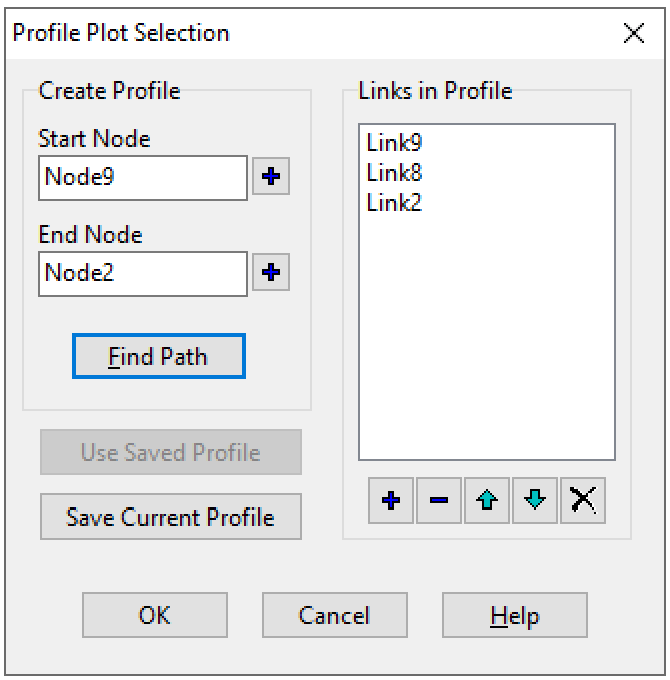


Figure 10 Profile Plot Selection dialog with data already filled in

8. Click the **+** button next to the *End Node* field to add the selected node name to the *End Node* field.
9. Move the *Profile Plot Selection* dialog out of the way, if necessary, and select one of the most upstream nodes in the drainage schematic.
10. Click the **+** button next to the *Start Node* field to add the selected node name to the *Start Node* field.
11. Click **Find Path** to show the path in the *Links in Profile* section.

12. Click **OK** to close the *Profile Plot Selection* dialog and bring up a "Water Elevation Profile" plot (Figure 11).

This plot shows the results of the steady state analysis with the rational flowrates. Feel free to explore the rest of the EPA SWMM post-processing tools to view the results from the Cougar Estates model run.

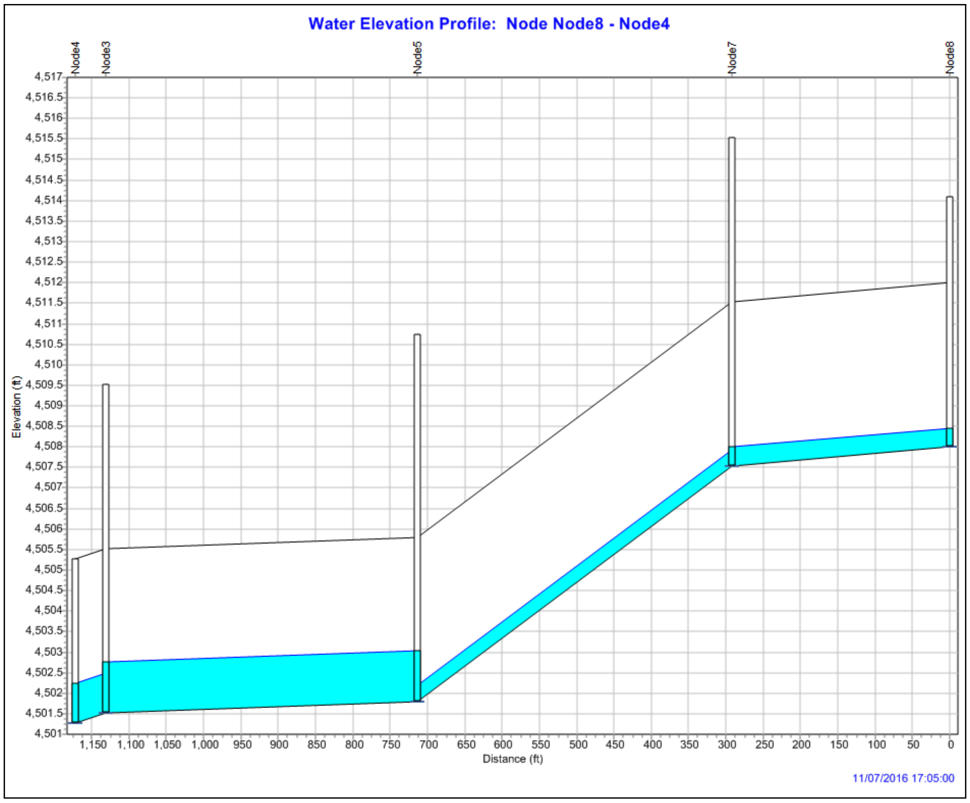


Figure 11   Elevation v. Distance plot from an upstream to a downstream node in the model

13. When done reviewing the results and exploring the EPA SWMM tools, click the ✕ in the upper-right corner to close EPA SWMM.

14. If prompted to save the current simulation, feel free to do so, and then return to WMS.

7 Routing Hydrographs in SWMM

In previous sections, the rational method was used to compute peak flows which were then exported to an SWMM model. Now use the rational method to compute hydrographs at each outlet, then route these hydrographs through the SWMM model.

7.1 Computing Rational Method Hydrographs

1. Select " Drainage" in the Project Explorer to make it active.

2. Switch to the **Hydrologic Modeling Module** .

3. Using the **Select outlet** tool, double-click outlet “2C” (the outlet for the entire drainage) to bring up the *Rational Method* dialog. The SWMM node icon may partially obscure the “2C” icon, so feel free to **Zoom** in if needed.
4. In the *Display* section, select “Selected” from the *Show* drop-down.
5. Click the **Compute…** button on the *Compute Hydrographs* row in the *Outlet* column to bring up the *Rational Method Hydrographs* dialog.
6. Click **Done** to compute the hydrographs and close the *Rational Method Hydrographs* dialog.
7. Click **OK** to exit the *Rational Method* dialog.
8. Notice that each basin and outlet icon shows a hydrograph icon (Figure 12).

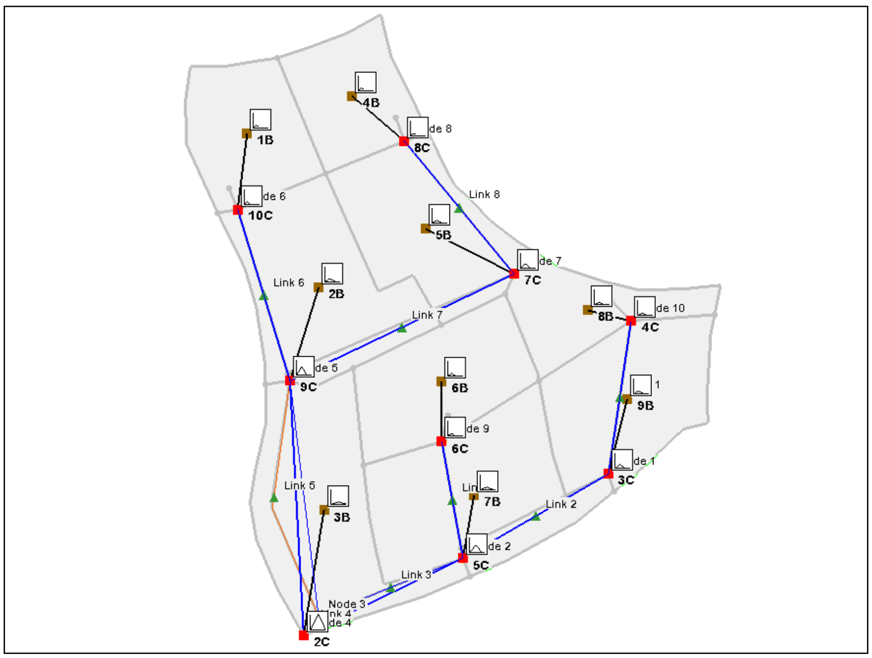


Figure 12 Rational Method hydrographs for SWMM

9. Select “ SWMM Hydraulic Schematic” in the Project Explorer.

If using EPA SWMM, follow the steps below. If using XPSWMM, follow the directions in Section 8.

10. Select *SWMM* | **Run EPA SWMM…** to bring up the *Select an EPA SWMM File* dialog.
11. Enter “swmm_hyd.inp” as the *File name.*
12. Click **Save** to close the *Select an EPA SWMM File* dialog and open the file in EPA SWMM.

7.2 EPA SWMM: Running and Viewing Results

1. In EPA SWMM, select *Project* | **Run Simulation** to bring up the *Run Status* dialog.

Notice that the run was successful.

2. Click **OK** to close the *Run Status* dialog.

3. Select the conduit connecting the most downstream outlet to the junction just upstream.
4. Click the **Create a Time Series Plot** button to bring up the *Time Series Plot Selection* dialog.
5. To add other links, select them in the EPA SWMM Graphics Window and click **Add** in the *Time Series Plot Selection* dialog to add them to the list in the *Data Series* section. Click **Accept** if the dialog asks to specify the object and variable to plot. Move the *Time Series Plot Selection* dialog out of the way, if necessary.
6. After all the desired links (or maximum amount of links) have been added, click **OK** to close the *Time Series Plot Selection* dialog and bring up the *Graph* dialog.

This plot shows the Flow versus Elapsed Time. The water depths, velocity, volume, and capacity in the storm drains can also be viewed by selecting the desired option from the *Variable* drop-down in the *Data Series Selection* dialog which appears each time a new link is added in the *Time Series Plot Selection* dialog.

7. Right-click on the plot to bring up the *Graph Options* dialog.
8. On the *Axes* tab, select *Bottom Axis* and enter “0.5” as the *Minimum*, “5.0” as the *Maximum*, and “0.25” as the *Increment* and click **OK** to close the *Graph Options* dialog and return to the plot (Figure 13).

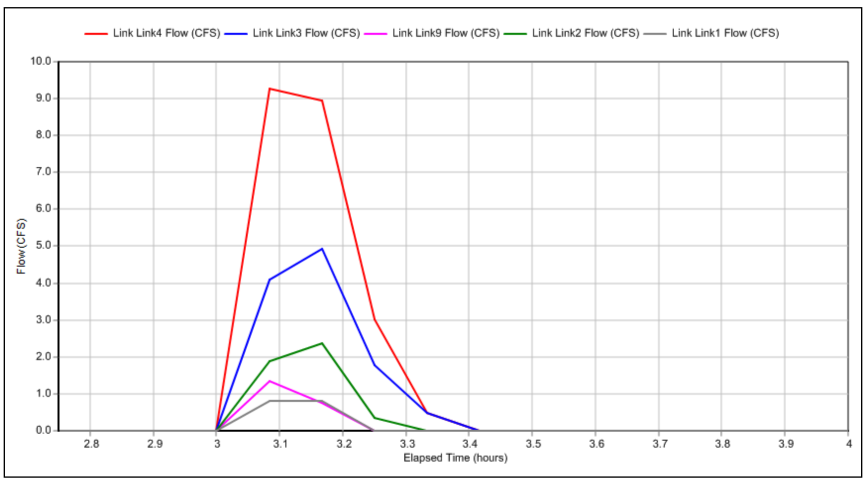


Figure 13 Plot of Flow vs. Time for several links in the model

8 Exporting to XPSWMM

This optional section describes how to export an XPSWMM model. If XPSWMM is not installed, skip this section. The XPX file is an XPSWMM project file. Using XPSWMM requires a separate license purchased directly from XP Solutions.

1. Select *SWMM* | **Run xpswmm…** to open the *Select an XPX File* dialog.
2. Select “XPX file (*.xpx)” from the *Save as type* drop-down.
3. Enter “swmmtut.xpx” as the *File name* and click **OK** to exit the *Select an XPX File* dialog and open the XPX file in XPSWMM.

9 Conclusion

This concludes the “Storm Drain Modeling – SWMM Modeling” tutorial. The following key topics were discussed and demonstrated:

- Defining runoff coefficients
- Computing flows using the rational method
- Importing a pipe network
- Running SWMM and viewing results
- Routing hydrographs using SWMM and viewing results
- Feel free to continue exploring the WMS interface or the EPA SWMM interface more thoroughly, if desired.

