PCSWMM

Workbook (Basic)

Hydrosoft

Think Global, Act Local

CHI, the developer of PCSWMM, has been developing the product for a long time with great care based on the latest information available.
But, No warranties are made, directly or indirectly, regarding this program.
And, No warranty is made as to the validity of the information contained in this user manual or for any particular purpose.

If you have any questions about the product's features and development background, please contact CHI :

Computational Hydraulics, Inc.

e-mail: support@chiwater.com

https://www.pcswmm.com

For the Korean version of PCSWMM or instructions for use, please contact Hydrosoft, CHI's official Korean distributor.

Hydrosoft

e-mail: support@hydrosoft.co.kr

https://www.hydrosoft.co.kr

Contents

1. Creating your first PCSWMM model

This is a basic introduction to modeling with PCSWMM.

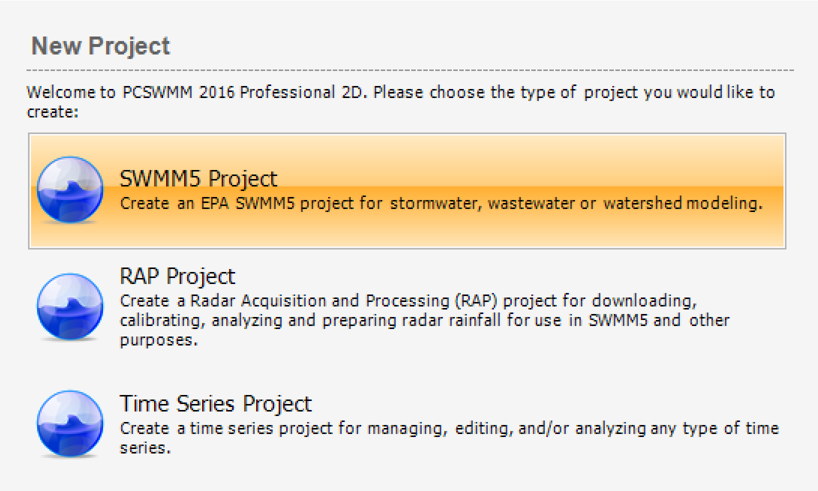
For instructions on how to use the PCSWMM workbook features, please refer to the following article and video. This exercise does not require you to download any files.

1.1 Setting up a new PCSWMM model

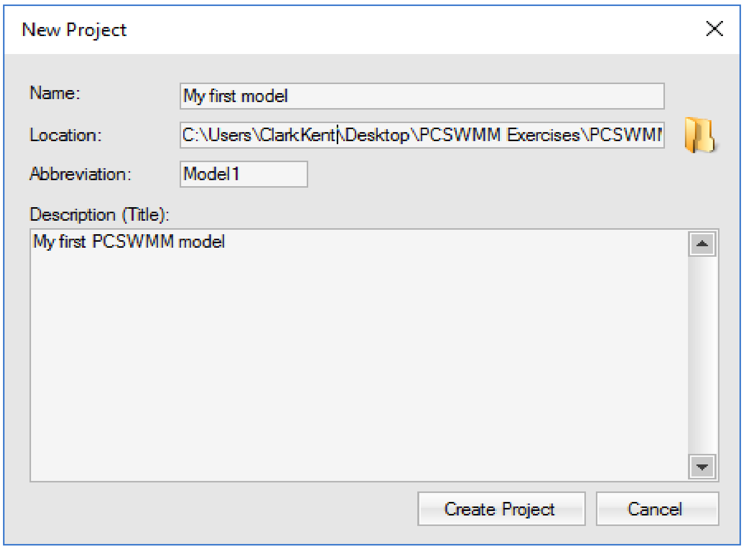
1. To begin with let's create a SWMM5 project:

2. Launch PCSWMM and click the **New** button located on the left side of the opening screen.

3. Select **SWMM5 Project** from the list of new projects.



4. In the **Name** box write **My first model**.



5. In the **Location** box browse to a known location and click **OK**.

6. In the **Abbreviation** box write **Model1**.

7. Click the **Create Project** button.

   Note: You also have the option to add project notes when creating a project. You can then further edit or append the notes with additional information or a list of tasks at any time using the **Project Notes** tab in the **Attributes panel**. Since this example will be located in New York we will begin by setting the project to use US units. If you are working in the United States you can skip this step as your flow units should automatically be set to CFS.

8. Click on **Simulation Options** from the **Project panel**.

9. In the **General** tab in the **Simulation Options editor** change the **Flow units** to **CFS**.

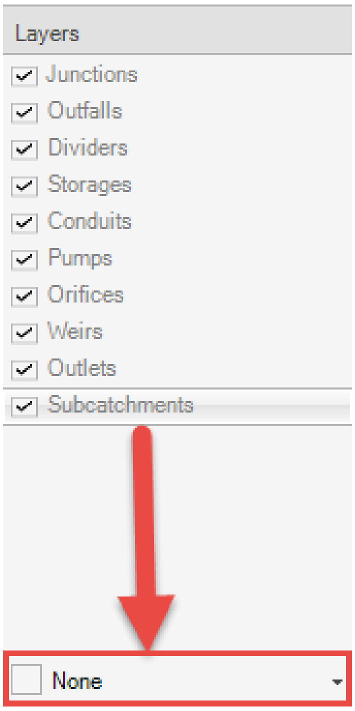
10. Click on the **OK** button.

11. Click on the **Switch** button when asked.

1.2 Opening background layers

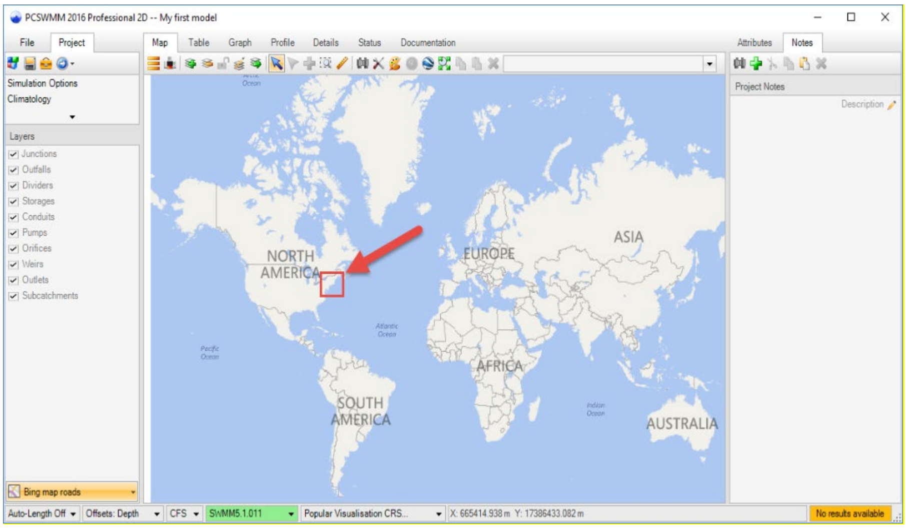
Let's choose a background layer to provide reference for our New York City model. If we had GIS layers for this area we could open them, however for this exercise we are going to use the open source Bing Maps.

1. In the **Layers panel** click on the **Tile Map Service Selector** that says "**None**" (located in the bottom left corner of PCSWMM).

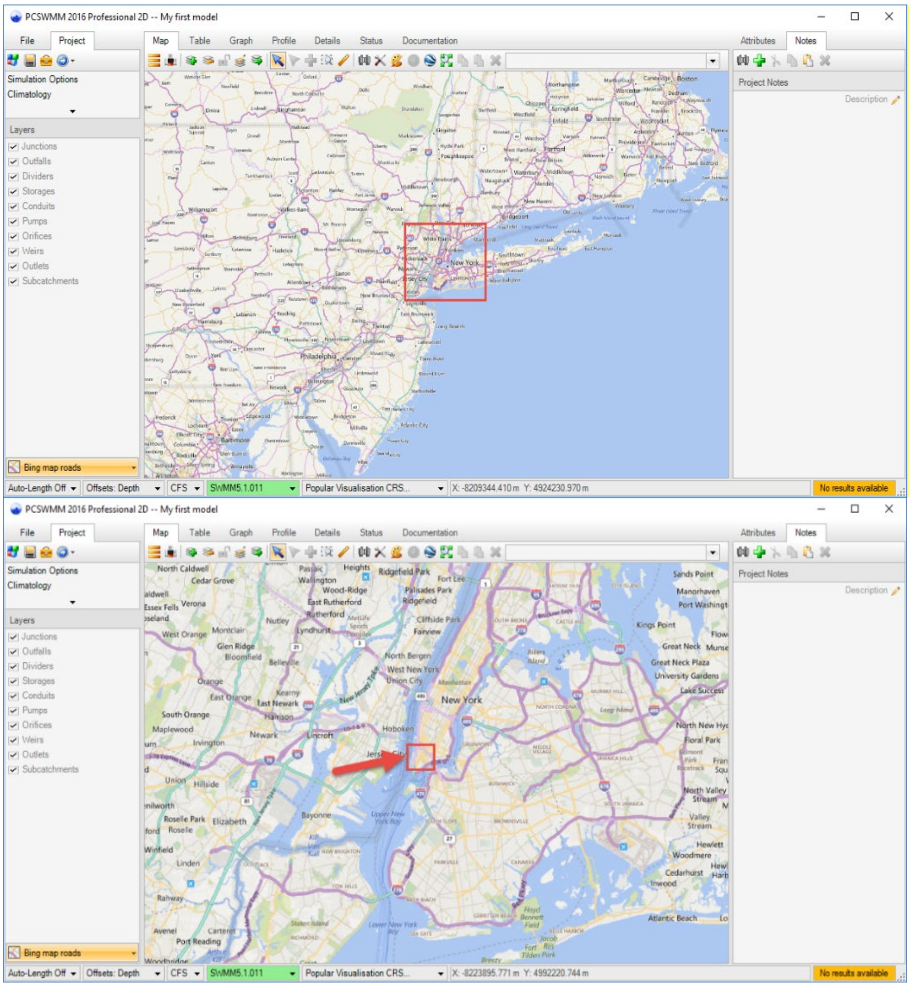


2. Select **Bing map roads** from the list.

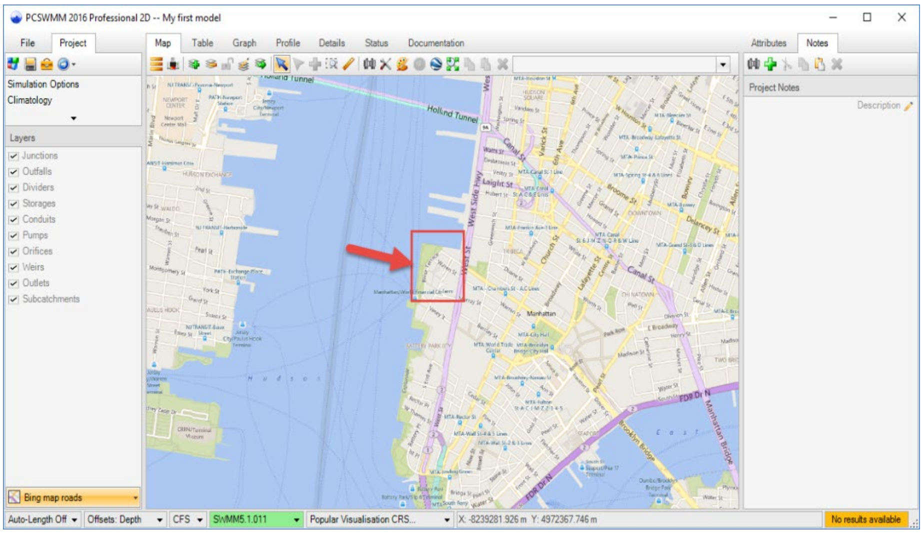
3. Click on the **Zoom** button, or your mouse scroll wheel, and zoom into the approximate location of New York City.

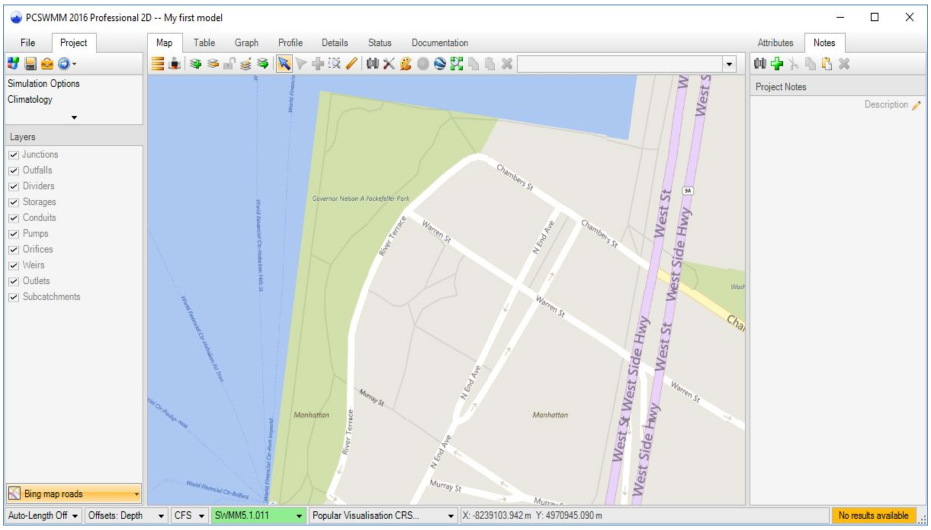


4. Continue to zoom into the location of New York City (Lower Manhattan) as shown in the screenshots.



5. Continue to use the **Zoom** function, or alternately use the mouse scroll wheel, to zoom into the location shown in the screenshots provided: the Governor Nelson A. Rockefeller Park, south of the Holland Tunnel and East of the West Side Hwy (9A).



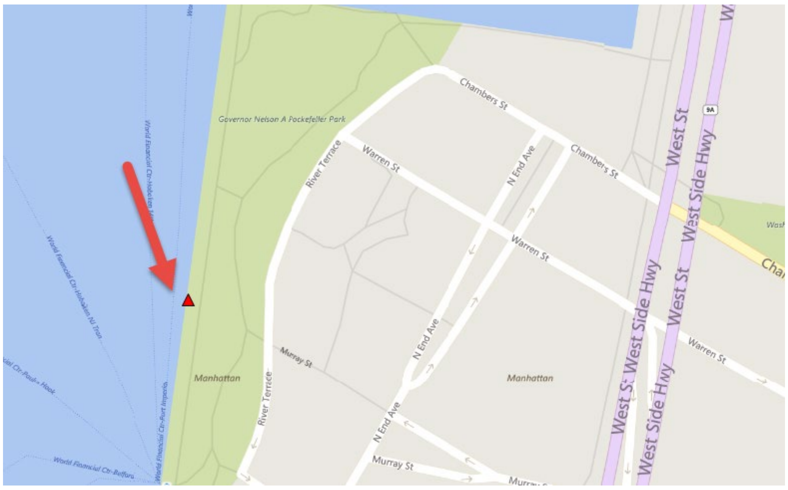


1.3 Creating a SWMM5 model

1. Click on the **Outfalls** layer in the **Layers panel**. It will appear grayed out as there are currently no Outfall entities.

2. Click on the **Add** button in the **Map panel**.

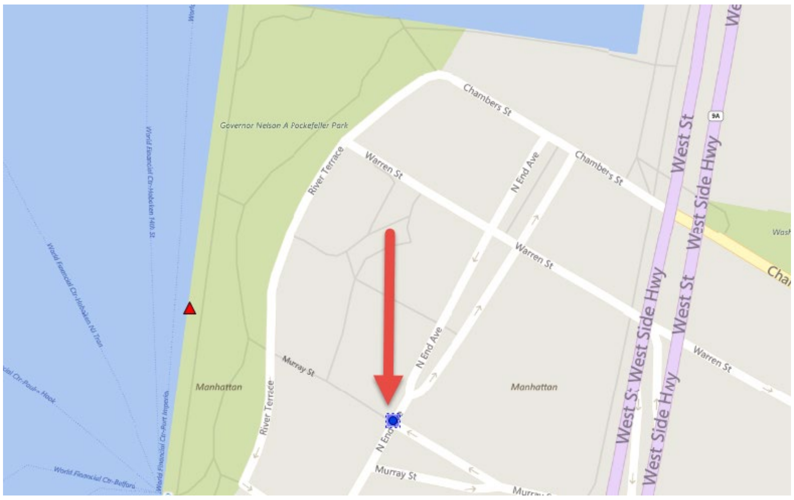
3. Click to add an outfall located on the shore linear to Murray street, as shown in the screenshot provided.



4. We will now add a junction that will act as the outlet for the subcatchment runoff.

5. Click on the junctions layer in the **Layers panel** to select the **Junctions** layer.

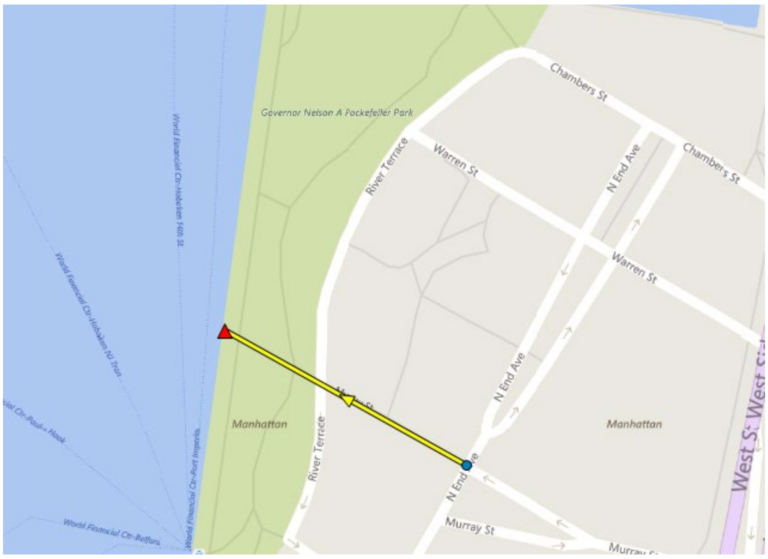
6. Click on the **Add** button and click on the intersection of Murray St and North End Ave, as shown in screenshot provided.



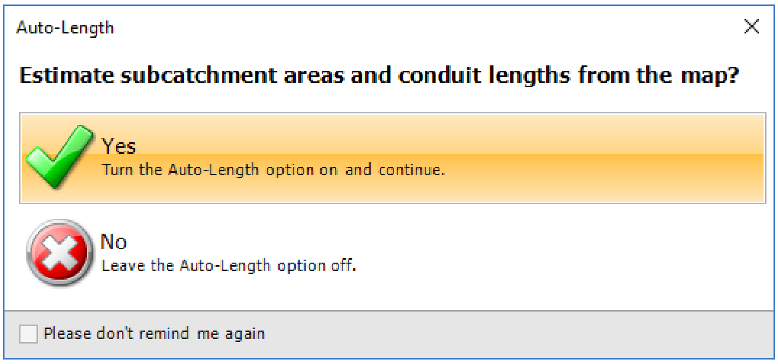
7. We will now connect the two nodes (outfall and junction) with a conduit.

8. Click on the **Conduits** layer in the **Layers panel** and click on the **Add** button.

9. The conduit flow direction is based on the order you draw the link (upstream to downstream). Click on the Junction and then click on the Outfall to draw the conduit connecting these two entities.



10. A message will appear asking if you want to estimate the conduit lengths from the map, click on the **Yes** button.



11. If you look at the **Attributes panel** (on the right of the PCSWMM window) you should see the **Length** attribute of the selected conduit has been assigned the map length.

12. Now we will parameterize the SWMM5 entities.

13. Click on the **Select** button to get out of **Add** mode

14. Click on the **Outfall** to select it and, in the **Attributes panel**, set the **Rim Elev.** to **10** ft. The invert elevation (**Invert El.**) will be left at **0** (sea level).

15. Click on the **Junction** and assign it an **Invert elevation** of **10** ft and a **Depth** of **10** ft.

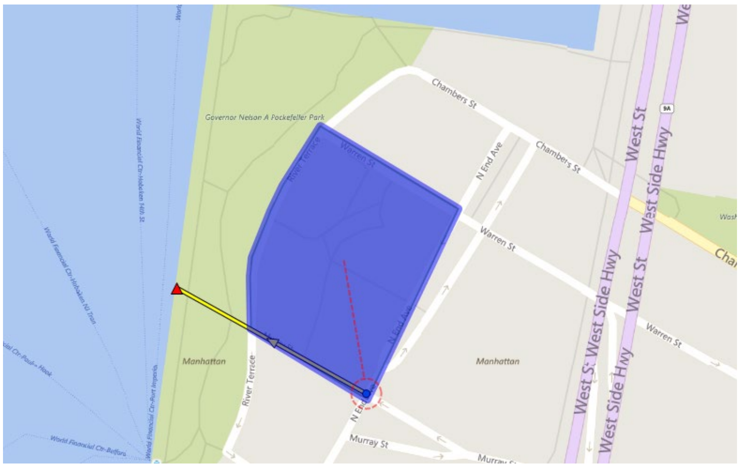
    Note: You can only edit one of **Depth** or **Rim Elev.** (the other will be calculated). If the **Depth** attribute is being calculated (i.e. is disabled), select the **Depth** attribute and click on the **Expression** button that appears in the value field. In the Auto-expression editor, click on the **Calculate Rim Elev. instead** button.

16. Select the **Conduit** and change the **Roughness** to **0.014** and the Diameter or **Geom1** to **2** ft.

17. Now we will add a subcatchment.

18. Click on the **Subcatchments** layer in the **Layers panel**.

19. Click on the **Add** button and draw a subcatchment outlining the Murray St / Warren St block as shown in the screenshot provided. Subcatchments are drawn by clicking at the desired vertex locations in the **Map panel**.



20. Click on the **Commit Changes** button or press the **Enter** key to finish editing the subcatchment.

21. Click on the **Select** button to get out of **Edit** mode.

22. In the **Attributes panel** for the selected subcatchment, change the **Imperv.** attribute (directly connected impervious area) to **50%.**

23. You should notice the **Area** of the subcatchment was automatically assigned as we asked the program to use map measurements. For the sake of brevity, we will skip parameterizing the other Subcatchment attributes (see our other exercises for more details on this).

1.4 Assigning rainfall and running the model

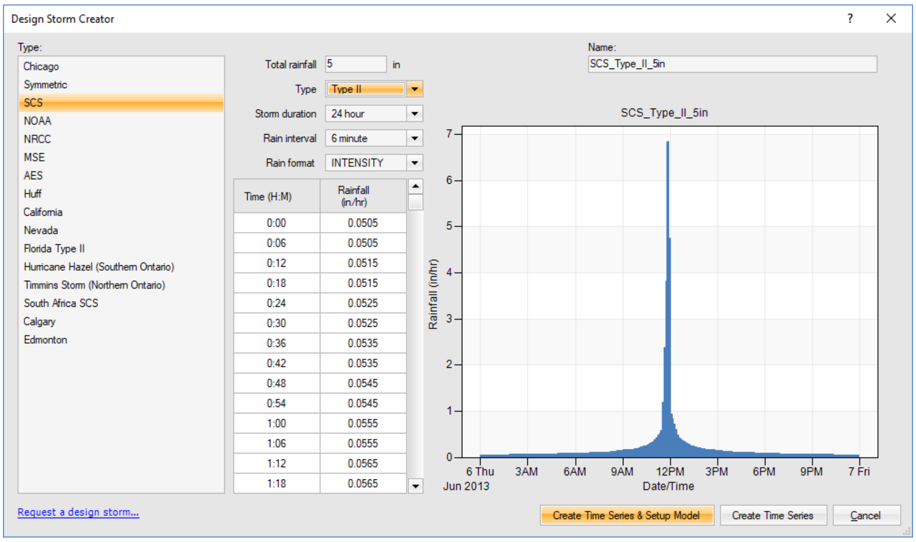
1. The next step is to assign rainfall input to the subcatchment. For this step we will be using a design storm event from the design storm creator tool.

2. Click on the **Graph** tab (along the top of the PCSWMM window) to switch to the **Graph panel**.

3. In the **Graph panel**, on the **Add** button to add a new design storm.

4. Select an **SCS** type design storm**.** Choose **Type II**, set the **Total rainfall** to **5 in.**, set the storm duration to **24 hours** and change the **Rain interval** to be **6 min**.

5. Click on the **Create Time Series & Setup Model** button.



6. PCSWMM created the time series in the Time Series Manager (on the left side of the Graph panel) and setup the model. In setting up the model, PCSWMM inserted the time series as a SWMM5 time series object, created a SWMM5 rain gage object linked to this SWMM5 time series object, assigned the rain gage to the subcatchment, and setup the simulation period to match the time series dates. Now your model is ready to be run.

7. Click on the **Run**  button located in the **Project panel**

1.5 Viewing results

Once your model has successfully ran you will see continuity errors displayed at the bottom right side of the screen. These values will be rendered from Green to Red depending on the size of the error. Generally you want these boxes to be green as that indicates that your runoff and routing errors are below 1%. You may also notice a Flooding warning, which indicates surface flooding (usually from an undersized conduit). We'll discuss how to remedy the flooding later.

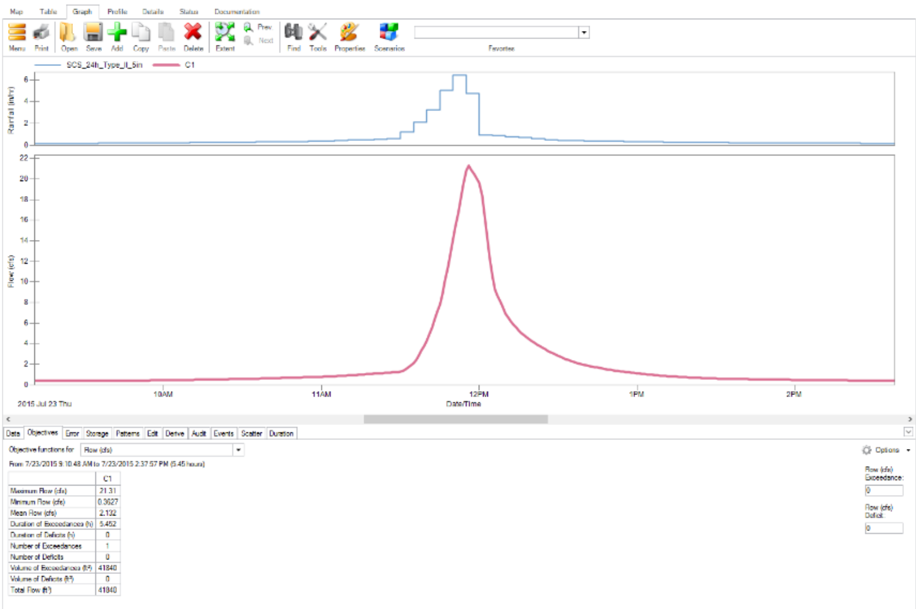
1. Click on the **Runoff** or **Routing** error boxes in the bottom status bar (or click on the **Status** tab and choose the **Continuity Errors** section) to view the mass balance report in the **Status panel**.

2. Review the water balance from the status report to ensure the values seem reasonable.

3. Now let's plot the hydrograph through the conduit.

4. Click on the **Graph** tab to switch to the **Graph panel**.

5. In the Time series manager, select **SWMM5 output > Links > Flow > C1** (put a check beside **C1**). Use your mouse to zoom into the hydrograph by holding down the left mouse button and dragging over the portion of the graph you want to see.



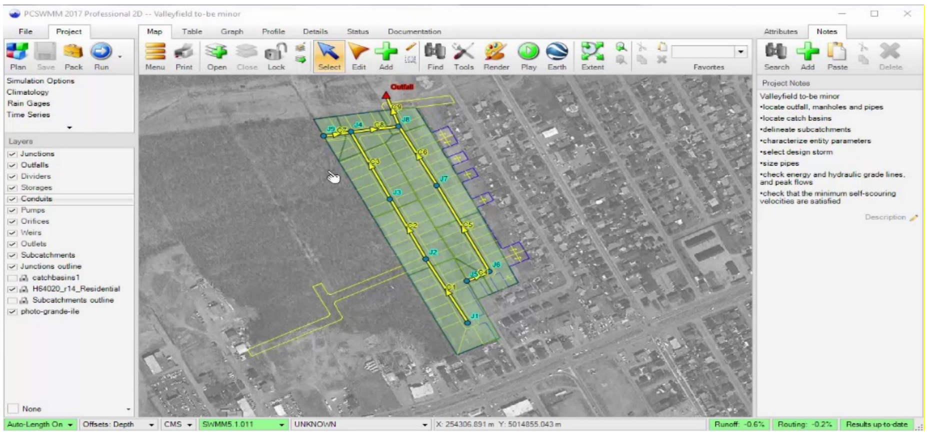
Congratulations, you've successfully completed your first PCSWMM model! To familiarize yourself with SWMM5 model output, we recommend you browse through the **Status panel** result tables, examine the result attributes of the entities in the **Map panel**, and explore the other time series generated by SWMM in the **Graph panel**.

You can also experiment by adding more junctions, conduits and subcatchments, as well as perform sensitivity tests on the various subcatchment attributes by adjusting them and re-running the model.

For more hands-on exercises, please refer to your General Workbook, located under My Training in the Help tab (**File > Help > My Training**).

2. Design of a new stormwater system with pipe-sizing and junction drops and losses (Valleyfield, Quebec)

This exercise illustrates the design of an urban residential stormwater drainage system using PCSWMM. In this example, we develop from scratch a preliminary design for a new stormwater drainage system. The system comprises catch basins, manholes and pipes - capable of handling the 10y design storm - for a new development of approximately 80 single-family detached residences in Valleyfield, Quebec. Pipe-sizing and junction drops and losses are included.

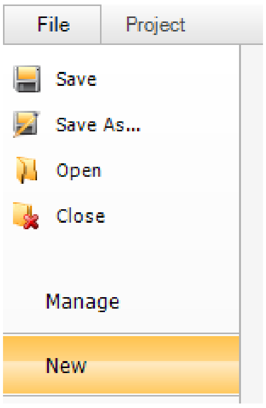


2.1 Setting up a new SWMM project

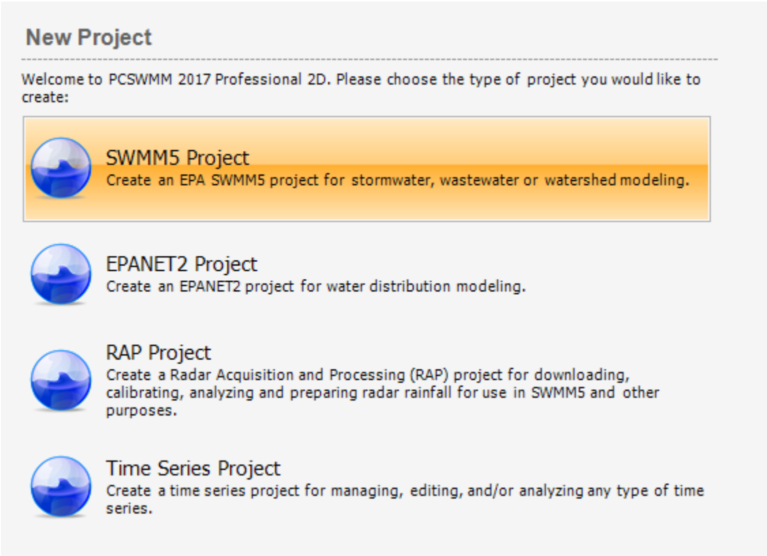
Let’s begin by setting up a new SWMM5 project for the preliminary design:

1. In PCSWMM, create a new SWMM5 project named **Valleyfield to-be minor** in the **PCSWMM Exercises\K018\Initial** folder.

2. Launch **PCSWMM** and click the **New** button located on the left side of the opening screen under **File**.



3. Select **SWMM5 Project** from the list of new projects.



4. In the **Name** box write **Valleyfield to-be minor**.

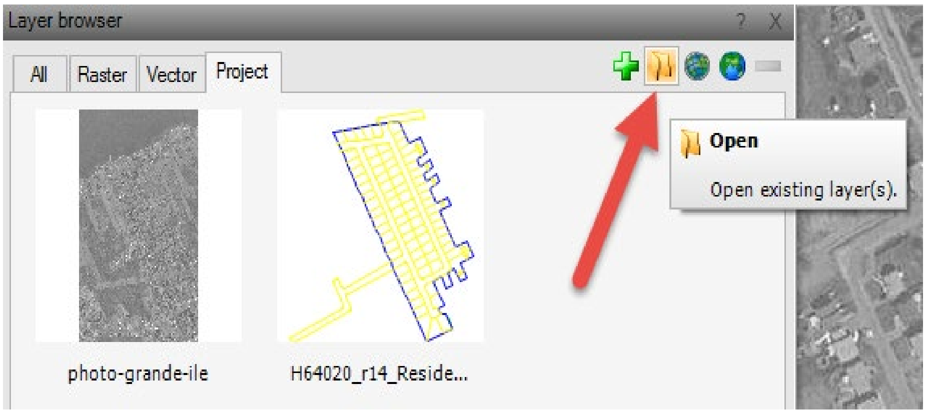
5. In the **Location** box browse to the **PCSWMM Exercises\K018\Initial** folder and click **OK**.

6. Click the **Create Project** button.

7. **Open** up the photo-grand-ile.jpg background layer from the **PCSWMM Exercises\K018\Initial** folder.

8. Click the **Open** button in the **Map panel**.

9. Click on the **Open** button in the top right corner of the **Layer browser**.



10. Navigate to **PCSWMM Exercises\K018\Initial**, select the raster image **photo-grande-ile.jpg** and click **Open**.

11. A message may pop-up asking you to set the coordinate system. If asked, select **Unknown (m)** from the list of options.

    Note: The order of appearance of the layers in the Layer panel dictates their hierarchy in the Map panel. SWMM layers are on top of background layers as default. To arrange the layers, simply drag and drop the layer to its desired position in the Layers panel.

In the satellite photo we see the St. Lawrence River and Seaway to the north. The new development is in the lower portion of the photo, south of the hydro-electric servitude ("hydro-cut" – appearing in the photo as a deforested corridor running east-west), immediately west of the existing residential area and somewhat north-east of the quarry. Zoom into this area:

12. Use the **Zoom** button in the toolbar to drag a selection box over the area you wish to view, or zoom and pan. In this example we will be zooming into the bottom middle of the image (refer to the figure).

13. Click on the **Select** button to exit zoom mode.

14. A CAD file has been prepared showing the location of roads and the proposed residential lots. To overlay this file on top of the satellite image, we will simply open the file.

15. Open the **H64020_r14_Residential.dxf** in the **PCSWMM Exercises\K018\Initial** folder.

16. Click the **Open** button in the **Map panel**.

17. Click on the **Open** button and select the AutoCAD file **H64020_r14_Residential.dxf** (in the folder **PCSWMM Exercises\K018\Initia**l)

18. Click **Open**.

The CAD drawing will appear in the **Map panel** delineating the lots and roadways in yellow.

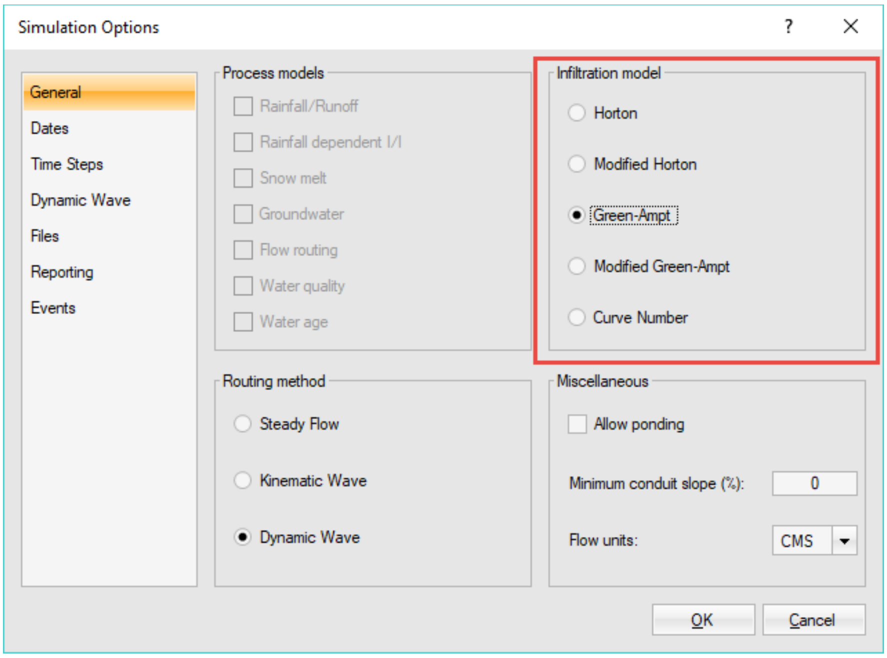
For this exercise we need to develop the minor system drainage network that will convey the 10 year storm water runoff from all lots and roads. The area is relatively flat and must be graded to ensure that minimum slopes are achieved. Thus the invert elevation has been provided for the outfall at the ditch and manhole invert elevations will be calculated in the model from the minimum required slope for all pipes.

2.2 Changing the infiltration method

Change the infiltration method to Green-Ampt in the **Simulation Options**.

In the **Project panel** click on **Simulation Options**.

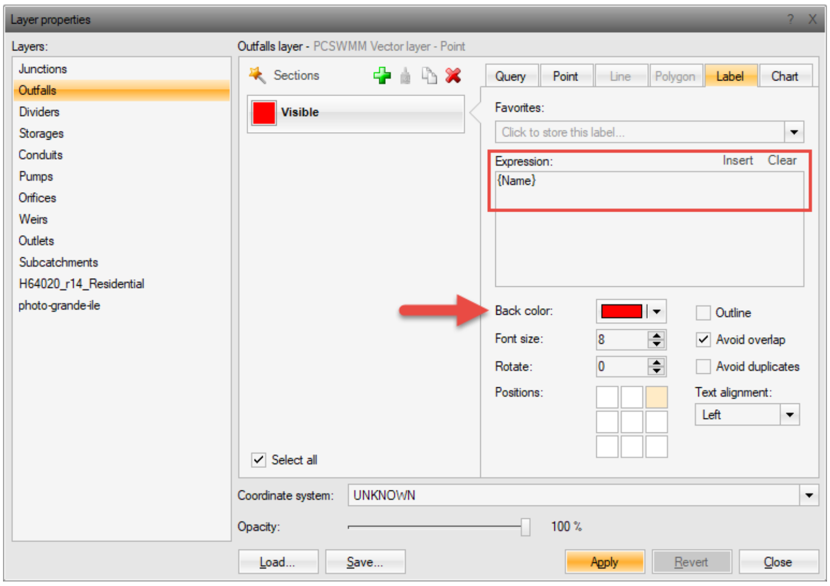
In the **General** tab, under the **Infiltration** model section, select **Green-Ampt** (if it is not already selected) and click **OK**.



2.3 Rendering SWMM5 entity labels and turn on link arrows

In this section we will be setting up the labeling for each of the outfalls, junctions and conduits using the Rendering menu. These labels will not show up in the map panel until after an entity has been added.

1. Set up the **Outfalls** layer to display a label with the **Name** showing and a red back color.
2. Click the **Render**  button in the toolbar of the **Map panel**.
3. Select the **Outfalls** layer from the list of layers.
4. Click on the **Label** button on the right side of the **Layers properties manager**.
5. Click on the **Insert** button and select **Name** from the list of attributes and click **Insert**.
6. Change the **Back color** to red, located under the **Expression** box.
7. Click on the **Apply** button.



Now we are going to do the same thing for the Junctions and Conduits layers.

8. Set up the **Junctions** layer to display a label with the **Name** showing and a blue back color.

9. With the **Layer properties manager** still open, click on the **Junctions** layer in the list of layers.

10. Click on the **Label** button.

11. Click on the **Insert** button and select **Name** from the list of attributes and click **Insert.**

12. Change the **Back color** to light blue, located under the **Expression box**.

13. Click on the **Apply** button.

14. Set up the **Conduits** layer to display a label with the **Name** showing and a yellow back color.

15. Click on the **Conduits** layer in the list of layers.

16. Click on the **Label** button.

17. Click on the **Insert** button and select **Name** from the list of attributes and click **Insert**.

18. The **Back color** should be set to yellow by default so we will leave it.

19. Click on the **Apply** button and click on the **Close** button.

Note that you will not see any changes in the **Map panel** at this point as we have not added any entities. Once we start drawing entities, they will have a name label and the back colors we defined above. Now we are going to turn on the link arrows in order to see the direction of flow in the Conduits.

20. In the **Preferences** dialog, turn on link arrows in the **Map panel** and choose to **Calculate node depth**.

21. Click on the **Menu**  button in the **Map panel** and select **Preferences.**

22. In the **Map** tab put a check beside **Show link arrows**.

23. In the **General** tab make sure there is a check beside **Calculate node depth**. This will allow us to later edit the rim elevations.

24. Click on the **OK** button to close the **Preferences**.

2.4 Locating the outfall and manholes (junctions)

To begin with, we will locate the outfall and the manholes in the network, using a maximum distance of 500 ft | 160 m between manholes, and ensuring the placement of a manhole at every intersection and change in pipe direction. Generally always start by placing the outfall.

1. Add an Outfall in the location shown in the screenshot attached.

2. Click the **Outfalls** item in the **Layers panel** to make this layer active in the **Map panel**.

3. Click the **Add**  button in the toolbar.

4. Now click at the location you wish to place the outfall (see screenshot below).

5. Click the **Select**  button in the toolbar to exit add mode.

6. Once the outfall is added and selected (highlighted in blue), we can edit the attributes.

7. In the **Attributes panel** (on the right side), change the **Name** to **Outfall** and the **Invert El.** to **149.61 (ft) | 45.6 (m)**.

We will be positioning the junctions according to a GIS background layer provided.

8. Open the **Junctions outline.SHP** file in the **PCSWMM Exercises\K018\Initial** folder.

9. Click the **Open**  button in the **Map panel**.

10. Click on the **Open**  button in the top right corner of the **Layer browser**.

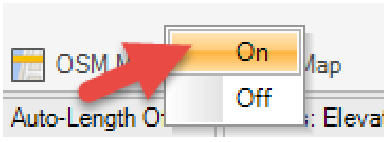
Navigate to the folder **PCSWMM Exercises\K018\Initial**, select **Junctions outline.SHP** and click **Open**.

A pop-up may appear asking you to **Set coordinate system**, if you received this message, select **Unknown (m)** from the list of options.

Keep in mind the junctions outline layer is just a background layer and does not represent actual SWMM5 junctions.

We will now turn on **Auto-Length**. This option will use the GIS map lengths to estimate the length and areas of conduits and subcatchments as opposed to using the default length.

11. Click on the **Auto-Length** box located in the status bar at the bottom of the PCSWMM window and select **On**.



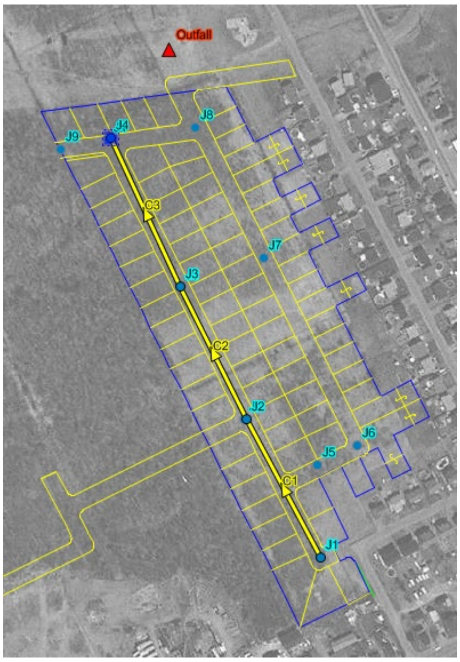
12. Add junctions and conduits based on the **Junctions outline.shp** file. Use the attached screenshot as a guide.

13. Select the **Junctions** layer from the **Layers panel**. The **Junctions** layer will appear gray as there are no junctions in the model yet. Be careful not to select the Junctions outline layer.

14. Click the **Add**  button in the toolbar.

15. Click on **J1** on the background layer and then, holding down the **Shift** key, click on **J2**. Notice how by holding down the **Shift** key, a conduit is automatically created connecting the two junctions.

16. While still holding the **Shift** key click on **J3** and **J4**.



17. Now release the **Shift** key and click on **J5**.

18. Now hold down the **Shift** key and click **J6**, **J7**, and **J8**.

19. Once again release the **Shift** key and click on **J9**.

20. Now hold down the **Shift** key and click on **J4**, **J8** and finally the **Outfall**. PCSWMM will recognize there is already a junction at the **J4** and **J8** locations and connect the network automatically.

Note: If you don't hold down the Shift key a conduit won't be automatically drawn to connect the junctions. If this happens you can add a conduit by clicking on the Conduits layer, clicking on the Add button and clicking the upstream and downstream nodes. If you make a mistake, switch back to the Select button, select the entity and click the Delete button in the toolbar or press the Delete key on the keyboard to delete the junction. Then you can return to the Add button and continue adding manholes. It will be easier later if you rename some of the nodes to reflect the figure below.

21. Click the **Select** button in the toolbar to exit **Add** mode.

22. Check that the conduit flow directions match the screenshot above. If you need to change the direction of a conduit click on the conduit, right click and select **Reverse link**.

23. Since we no longer need the junctions outline template, it can be closed.

24. Select the **Junctions outline** layer from the **Layers panel** and click the **Close** button.

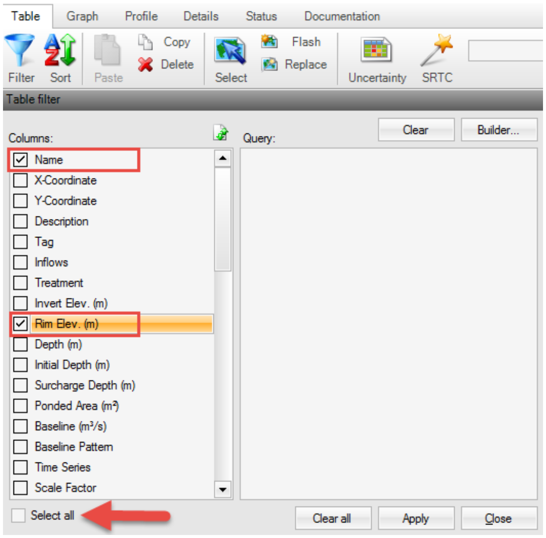
25. In the **Table panel**, filter the **Junctions** layer to display only **Name** and **Rim Elevation**.

26. Click the **Table** tab to open the **Table panel**.

27. Ensure that the **Junctions** layer is selected in the **Layers** list.

28. Click on the **Filter** button and check then un-check the **Select all** box at the bottom.

29. Place a check beside **Name** and **Rim Elev**. and click on the **Apply** button.



The **Table panel** should only be displaying the name and rim elevation attributes for junctions.

Enter the **Rim Elev.** values from the image below (click on the image icon to see the table). Please note this image has been altered to show both metric and US units; this is not automatically shown in PCSWMM.

Note: In PCSWMM, the Junction depth can be defined either by setting the Depth of the junction or by setting the Rim Elevation. Often when creating a model of a subdivision the Junction depths are unknown, however by defining the elevation of the rim, the depth can be automatically calculated by subtracting the Invert Elevation from the Rim Elevation. If the Rim Elev. attribute is being calculated (i.e. is disabled), select the Depth attribute and click on the Expression button that appears in the value field. In the Auto-expression editor, click on the Calculate Depth instead button and vise versa.

Junctions

Name	Rim Elev. (m)	Rim Elev. (ft)
J1	49.6	162.73
J2	48.93	160.53
J3	48.54	159.25
J4	48.18	158.07
J5	49.5	162.4
J6	49.2	161.42
J7	48.38	158.73
J8	47.95	157.32
J9	48.3	158.46

30. Click the **Save** button.

2.5 Setting conduit attributes

1. Return to the **Map panel** by clicking on the **Map** tab.
2. Select all the conduits.
3. Click the **Conduits** layer in the **Layers panel**.
4. Select all of the conduits by pressing the **Ctrl + A** buttons on the keyboard.
5. Change the attributes for all the conduits as follows:

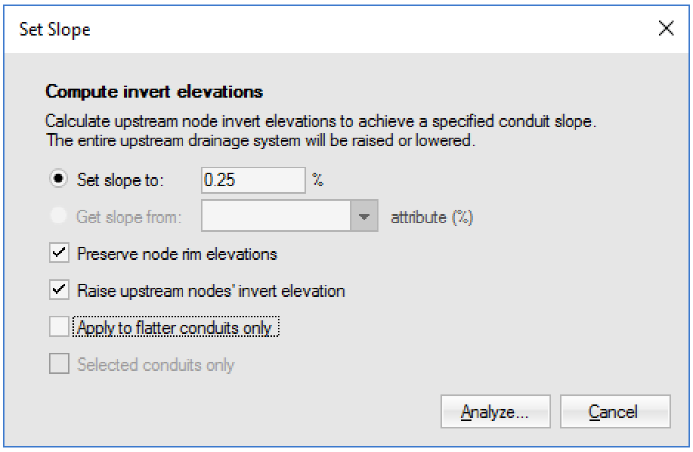
Roughness = 0.013

Cross-Section = CIRCULAR

Geom1 = 3.5 (ft) | 1 (m)

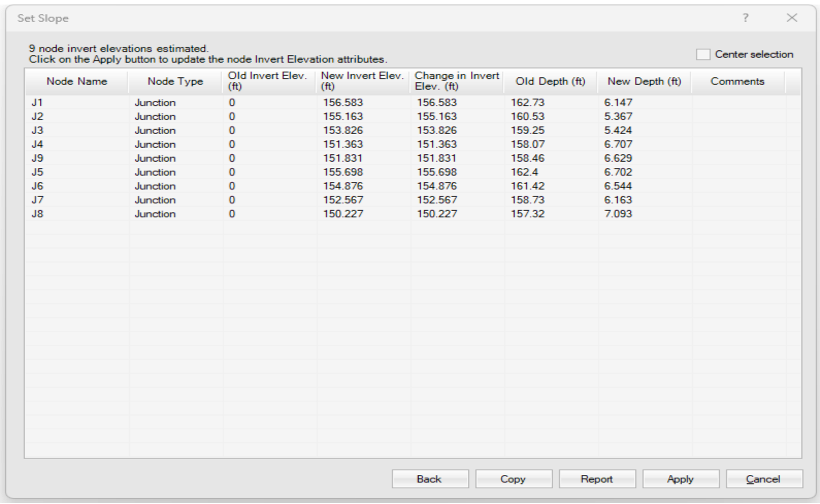
Now we need to assign invert elevations for all manholes in the system. As we need to meet a pipe slope criteria of at least 0.25% and the outfall invert elevation is 149.61 (ft) | 45.6 (m), PCSWMM can calculate the required invert elevations of all the other manholes in the drainage network.

6. Using the **Set Slope** tool, set the slope to 0.25%. Preserve node rim elevations and raise upstream nodes' invert elevation.
7. Click on the **Tools**  button.
8. In the **Conduits** section, click on the **Set Slope** tool.
9. In the **Set Slope tool**, in the **Set slope** box, enter in **0.25** (%)
10. Check the box to **Preserve node rim elevations**.
11. Check the box to **Raise upstream nodes' invert elevation**.
12. Uncheck the box to **Apply to flatter conduits only**.
13. Click on the **Analyze…** button.



A table of calculated changes will appear for review.

14. Click **Apply** to implement them, and then on the **Close** button. (Please note the screenshot shown is in US units. In addition, the values presented below may differ based on how exact the conduits were drawn).



Note: If all 9 nodes are not listed in the Set Slope tool, it may be because the flow direction is incorrect. Under the **Menu** button in the **Map panel** select **Preferences**. In the **Map** tab, click the box next to **Show link arrows** to turn on the feature if it is not already turned on. Click **OK** to close. From there select the conduits that are not in the upstream direction and right-click and select **Reverse link**.

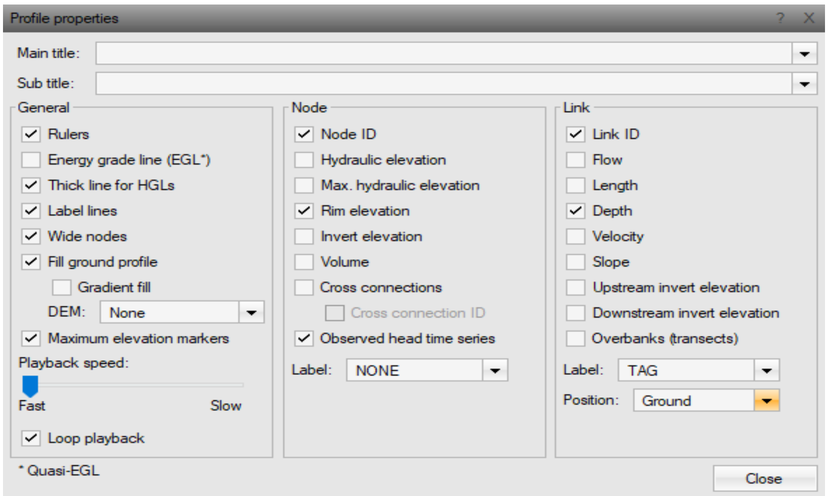
15. Select a pathway from **J1** to the **Outfall** and open the **Profile panel**.

16. In the **Map panel**, click on junction **J1.**

17. Hold the **Shift** key down and click on the **Outfall**. A path of connecting entities should be selected.

18. Click the **Profile** tab view the profile.

19. Display the rim elevations and pipe diameters in the **Profile.**



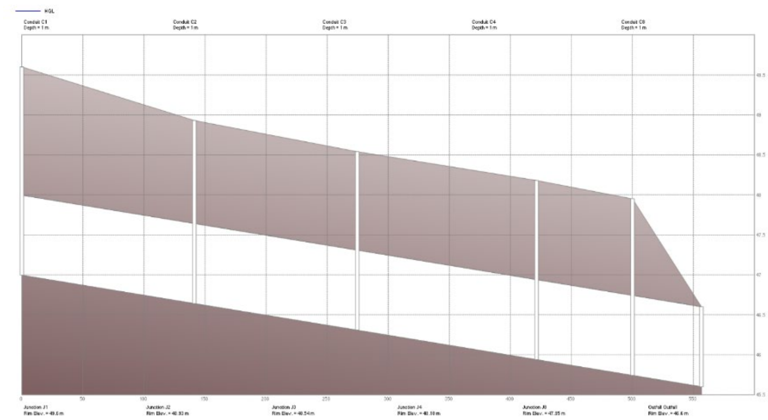
20. Click the **Properties** button in the **Profile panel**.

21. In the **Profile Properties** editor, under the **Node** section, ensure **Rim elevation** is checked.

22. In the **Link** section, ensure **Depth** is checked.

23. Click the **Close** button to exit the **Profile properties**.

The profile should appear similar to the screenshot shown.



2.6 Importing subcatchments

1. We are now going to import subcatchments from an external GIS layer.

   Subcatchments can also be manually added using the **Add** button in the **Map panel**; the way the junctions and conduits were added.

2. Import subcatchments from the **Subcatchments outline.shp** file in the **PCSWMM Exercises\K018\Initial** folder using the **Import GIS/CAD** wizard.

3. Click on the **File** tab.

4. Click on **Import** and select **GIS/CAD**.

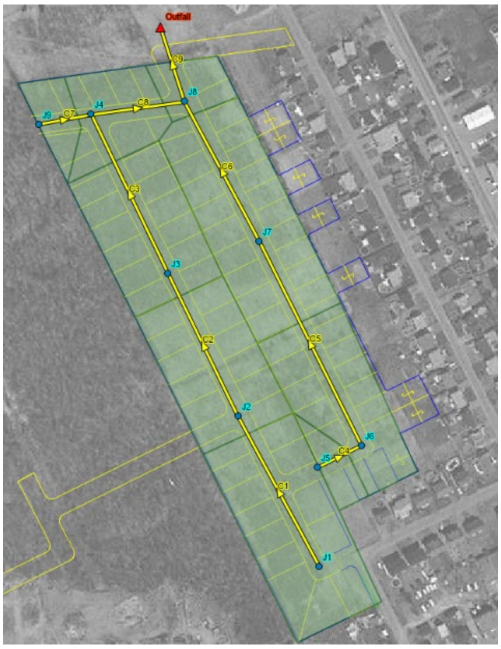
5. Click on the **Subcatchments** item.

6. Click on the **Browse…** button and navigate to **PCSWMM Exercises\K018\Initial** and select **Subcatchments outline.SHP**. Since it was a previously created subcatchment layer, the importer will recognize the attributes and automatically match them up.

7. Click on the **Finish** button. An import report will appear.

8. Click the **Close** button.

9. The subcatchments will appear in the **Map panel**.

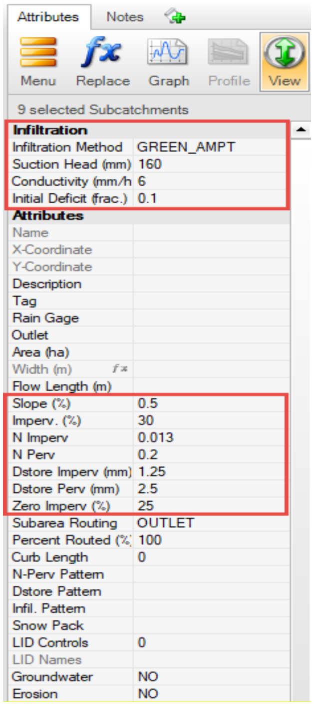


2.7 Assigning subcatchment parameters

Now the subcatchment parameters can be assigned. Subcatchment parameters are given in this example, however values for many of the parameters listed below can be found on the PCSWMM support site in the Reference Tables section. In addition, land use and soil layers can be used to area weight infiltration and imperviousness values from GIS polygons using the **Spatial Weighting** tool in PCSWMM, however this is covered in another exercise (Estimating subcatchment attributes based on land-use and soils layers).

1. Select all the subcatchments.
2. Click on the **Subcatchments** layer from the **Layers panel**.
3. Press the **Ctrl + A** keys on the keyboard to select all subcatchments.
4. Set the following attributes for all subcatchments in the **Attributes panel** (screenshot is also attached).

Attribute	Value	Reference
Slope (%)	0.5	Estimate
Imperv (%)	30	Estimate
N Imperv	0.013	Manning's N - Overland Flow
N Perv	0.2	Manning's N - Overland Flow
Dstore Imperv	0.05 in | 1.25 mm	Depression storage
Dstore Perv	0.1 in | 2.5 mm	Depression storage
Suction Head	6.3 in | 160 mm	Soil characteristics
Conductivity	0.24 in/h | 6 mm/h	Soil characteristics
Initial Deficit	0.1 (fraction)	Soil characteristics



We will now compute the **Width** attribute for the subcatchments by dividing each subcatchment area by an approximate maximum length of overland flow (flow length).

This is not the real width of the subcatchment.

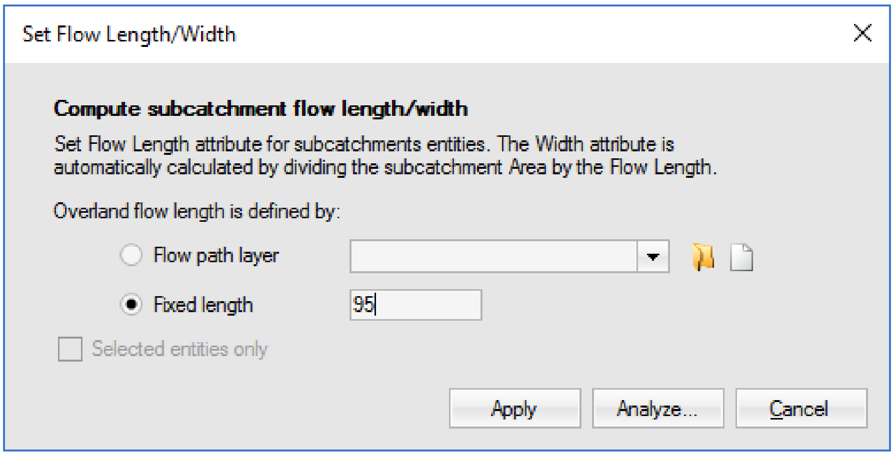
We'll assume the average maximum length of overland flow is the sum of the average lot depth (115 ft | 35 m) plus the curb length between catch basins (197 ft | 60 m), which equals 312 ft | 95 m.

5. Using the **Set Flow Length/Width** tool, set a fixed length of **312 ft | 95 m** for the subcatchments.

6. Click on the **Tools**  button.

7. In the **Subcatchments** section, click on the **Set Flow Length/Width** tool.

8. Select **Fixed length** in the dialog and enter **312 ft | 95 m** in the text box (please note the screenshot shown is in SI units, US units will differ).



9. Click **Apply** to execute it.

10. Examine the log of changes and click **Close**.

11. Switch to the **Table panel** by clicking on the **Table** tab.

12. Select the **Subcatchments** layer if the subcatchments layer is not already selected. You should see the new width values for each subcatchment (please note the screenshot shown is in SI units, US units will differ).

Subcatchments

Name	Outlet	Area (ha)	Width (m)
S1	J1	1.4121	148.642
S2	J2	1.0525	110.79
S3	J3	1.0334	108.779
S4	J4	0.5506	57.958
S6	J6	0.9726	102.379
S7	J7	1.4916	157.01
S8	J8	0.2636	27.747
S9	J9	0.2277	23.968
S5	J5	0.1809	19.042

2.8 Adding a design storm as a rainfall time-series

Now for this simple, straight-forward stormwater minor system design, the next step is to specify a minor design storm suitable for pipe sizing. For this exercise we will use the design storm creator tool and automate the project time series setup. To generate a design storm time series:

1. Click on the **Graph** tab to open the **Graph panel**.

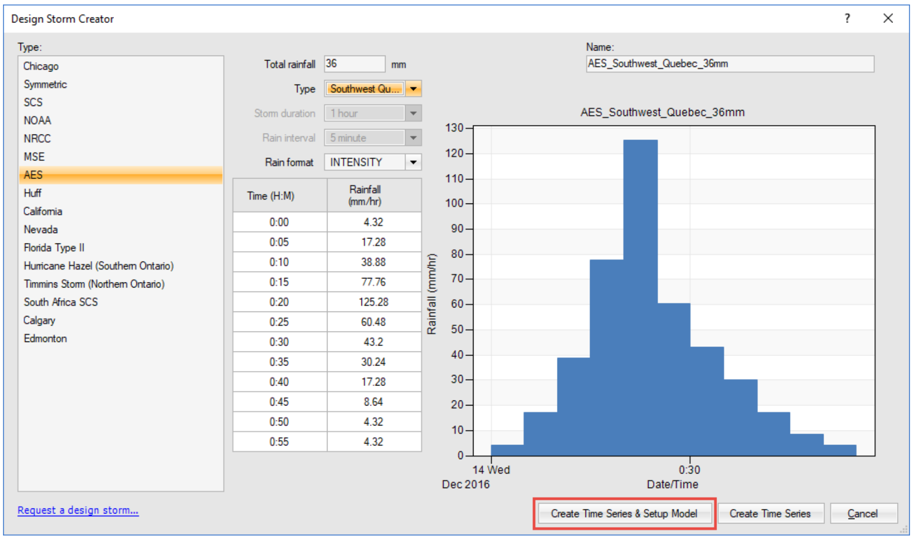
2. Click on the **Add** button to open the **Design Storm Creator**.

3. Select **AES.**

4. Change the **Type** to **Southwest Quebec** from the list.

5. In the **Total rainfall** box type in **1.5 in| 36 mm**

6. Ensure the **Rain format** is set to **Intensity**.



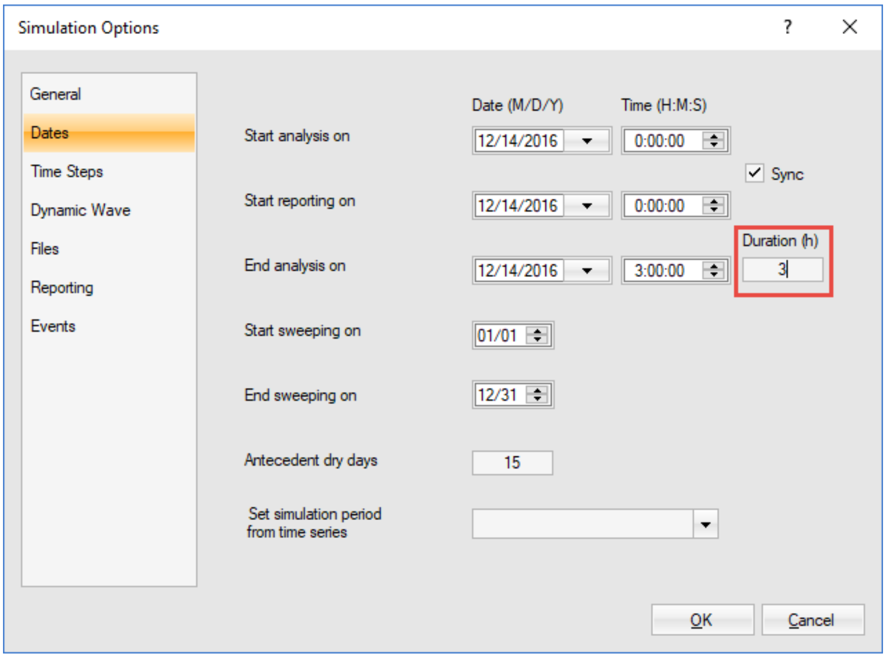
7. Click on the **Create Time Series & Setup Model**. This will automatically create your rain gage, assign it to all of the subcatchments and change the duration of your model to match the rainfall time series.

8. Click on the **Map** tab to return to the **Map panel**.

2.9 Running the model

1. In the **Map panel**, click on **Simulation Options** in the **Project panel**.

2. Click on the **Dates** tab, and change the **Duration (h)** to **3** and click **OK**.



3. Click the **Run** button in the toolbar to save the project and run a simulation. If the run is successful, a pop-up window will show the message, 'Run was successful', with Continuity Error information.

4. Check that the continuity errors are reasonable (say less than 5%, depending on the design accuracy required).

It may seem counter-intuitive, but drops and losses are assigned after the pipes are sized. This is because pipe sizing uses the Manning's formula which does not take into account manhole drops. In addition, minimum pipe drops need to be assigned after the pipes are sized so this suggested sequence saves repeating steps. Once the drops are added and the model rerun, the pipe capacity results should be checked, along with the profiles, and pipe sizes tweaked as necessary. Although the sequence of steps suggested here is more efficient, there is, however, no harm in assigning drops and losses before the pipes are sized, and then executing the drops and losses tool after the pipes are sized.

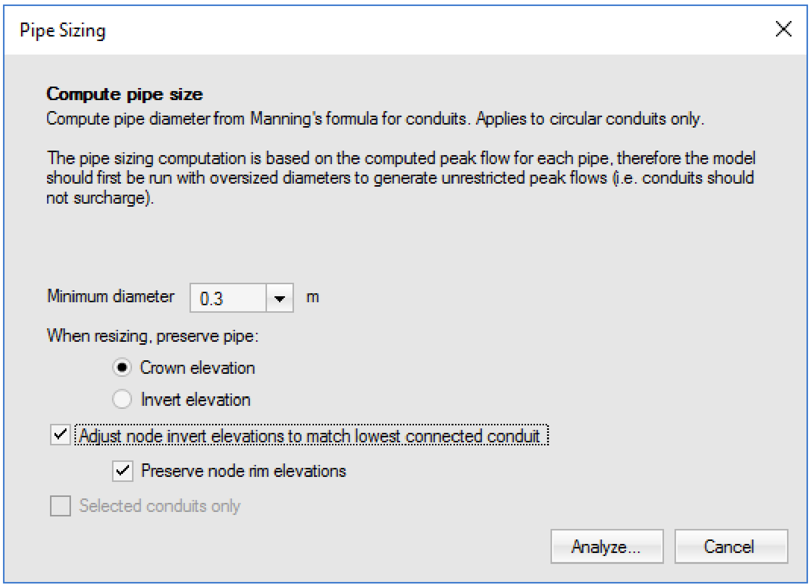
2.10 Sizing the pipe diameters

PCSWMM will compute the minimum circular pipe diameter that will not flow full, provide the nearest standard pipe diameter, and apply them to the model (i.e. update the input data file accordingly). Note that this capability only applies to circular pipes. The program uses the Manning's formula to compute the diameter:

$$D = \left(\frac{Q \cdot n}{0.312 \cdot S^{1/2}}\right)^{3/8} \quad D = \left(\frac{Q \cdot n}{0.464 \cdot S^{1/2}}\right)^{3/8}$$

Manning's n and slope S are conduit attributes entered by the user, but flow Q is computed by the model. If the initial pipe size is too small, the required free flow cannot be computed. Therefore, users must initially oversize the pipes in order for PCSWMM to compute the best standard pipe diameters.

1. Select all the conduits.
2. While still in the **Map panel**, select the **Conduits** layer in the **Layers panel.**
3. Press the **Ctrl + A** keys to select all of the entities.
4. Use the **Pipe sizing** tool to set a minimum diameter of **1 ft | 0.3 m.** Choose to preserve crown elevations, adjust node invert elevations and preserve rim elevations.
5. Click on the **Tools** button.
6. In the **Conduits** section, click on the **Pipe Sizing** tool.
7. In the **Pipe Sizing** tool, set the **Minimum diameter** to **1 ft | 0.3 m.**
8. Select the option to **Preserve pipe: Crown elevations.**
9. Place a check in the **Adjust node invert elevations to match lowest connected pipe** option.
10. Check the option to **Preserve node rim elevations.**



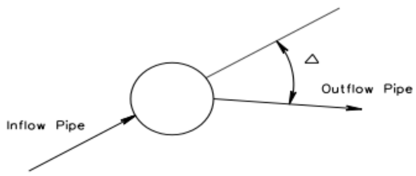
11. Click the **Analyze** button (please note the screenshot shown is in SI units, US units will differ).

12. Compare **Original Diameter** and **New Diameter** in the **Pipe Sizing** preview. Click on the **Nodes** tab to see the computed changes to the invert elevations due to pipe resizing.

13. Click **Apply** to implement the changes and click on the **Close** button to exit the **Pipe Sizing** tool and see the adjusted pipe diameters in the **Profile panel.**

2.11 Setting drops/losses

Now we need to adjust the drops across each manhole to take into account bend angles in SWMM, since the engine does not consider them. In this example, we will assign a drop of 0.1 (ft) | 0.03 (m) to have a loss coefficient of 0.15 for a straight-through pass (bends less than 15 degrees). And we will assign a drop of 0.49 (ft) | 0.15 (m) to have a loss coefficient of 1 for a 45 to 90 degree bend at a manhole. The bends at nodes J4 and J8 are greater than 90 degrees and so we will have 45 degree elbows installed immediately upstream (these elbows may not meet specs in your area and are not explicitly modeled for this preliminary design).



A bend angle in an outflow pipe in relation to the inflow pipe (VDOT Chap. 9, 2016).

For this process;

Use the **Set Drops/Losses** tool to set both drops and losses as shown in the following screenshots.

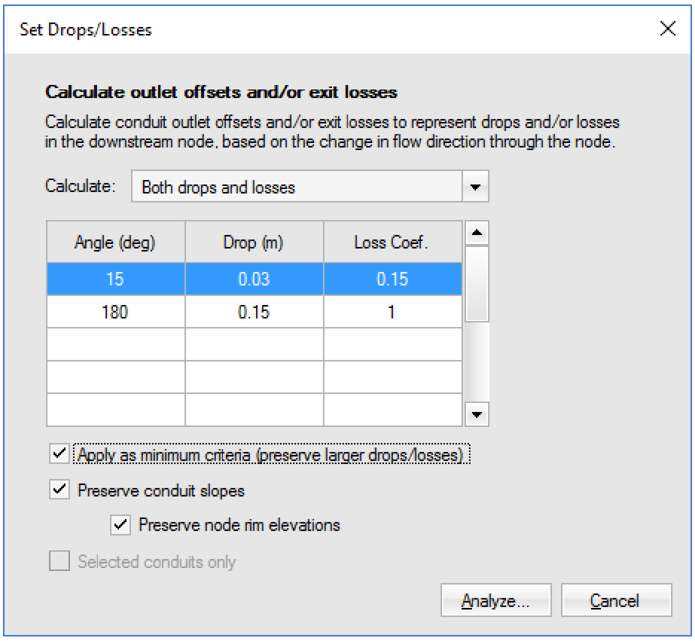
1. In the **Map panel** click on the **Tools**  button.

2. In the **Conduits** section, click on the **Set Drops/Losses** tool.

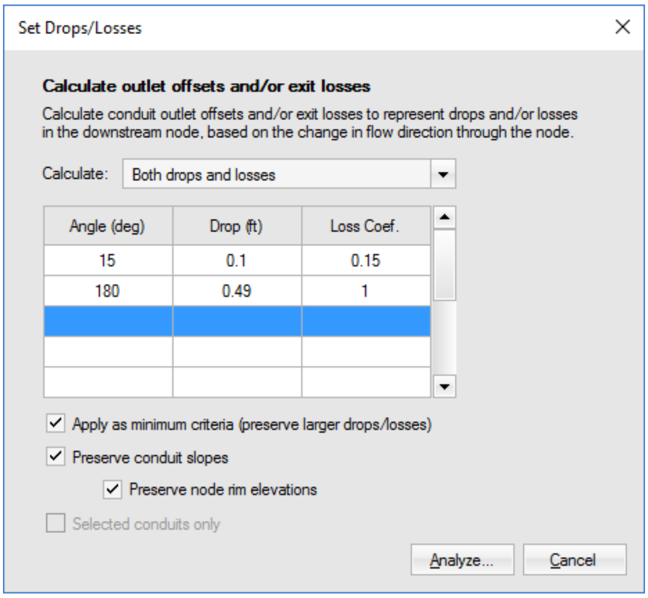
3. In the **Calculate** drop-down menu, select **Both drops and losses** (should be default).

4. Enter the **Angle**, **Drop** and **Loss Coef**. values as shown in the images, ensuring the units are correct.

SI units



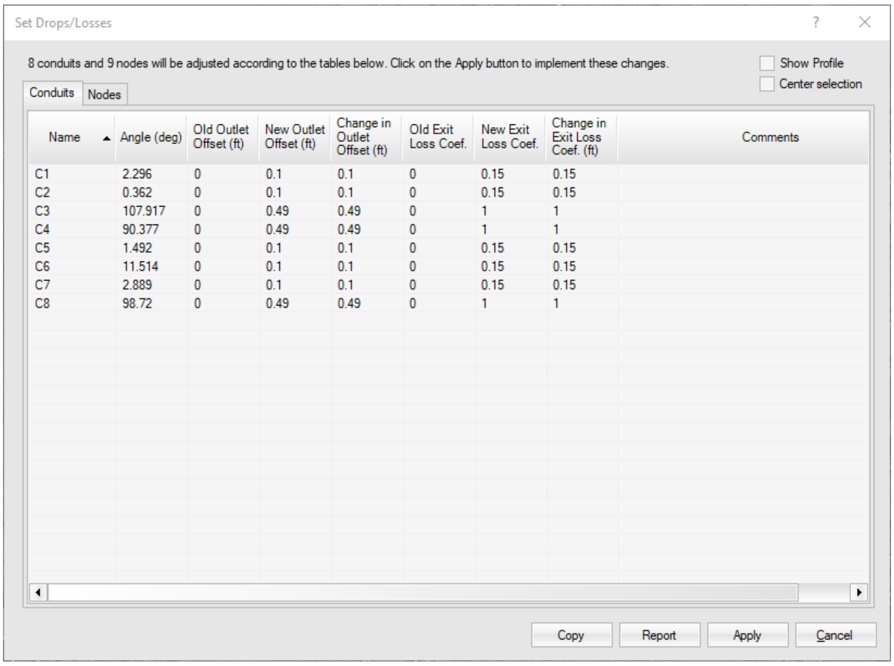
US Units



5. Check the **Apply as minimum criteria**, **Preserve conduit slopes**, and **Preserve node rim elevations** options.

6. Click **Analyze.**

7. View the **Set Drops/Losses** report to ensure that the **C3, C4** and **C5** exit loss coefficients are the same (please note the screenshot shown is in SI units, US units will differ).



Name	Angle (deg)	Old Outlet Offset (ft)	New Outlet Offset (ft)	Change in Outlet Offset (ft)	Old Exit Loss Coef.	New Exit Loss Coef.	Change in Exit Loss Coef. (ft)	Comments
C1	2.296	0	0.1	0.1	0	0.15	0.15	
C2	0.362	0	0.1	0.1	0	0.15	0.15	
C3	107.917	0	0.49	0.49	0	1	1	
C4	90.377	0	0.49	0.49	0	1	1	
C5	1.492	0	0.1	0.1	0	0.15	0.15	
C6	11.514	0	0.1	0.1	0	0.15	0.15	
C7	2.889	0	0.1	0.1	0	0.15	0.15	
C8	98.72	0	0.49	0.49	0	1	1	

8. Click **Apply** to apply the changes and then click the **Close** button.

9. Click on the **Run**  button to save the project and regenerate the results.

After the run completes, you can check the longitudinal profiles throughout the network (**Profile panel>Menu>Show Peak Values**) to ensure the pipes still have sufficient capacity to avoid surcharging with this design storm.

2.12 Viewing and interpreting the results

1. Clear the current plots in the **Graph panel.**

2. Click the **Graph** tab to open the **Graph panel**.

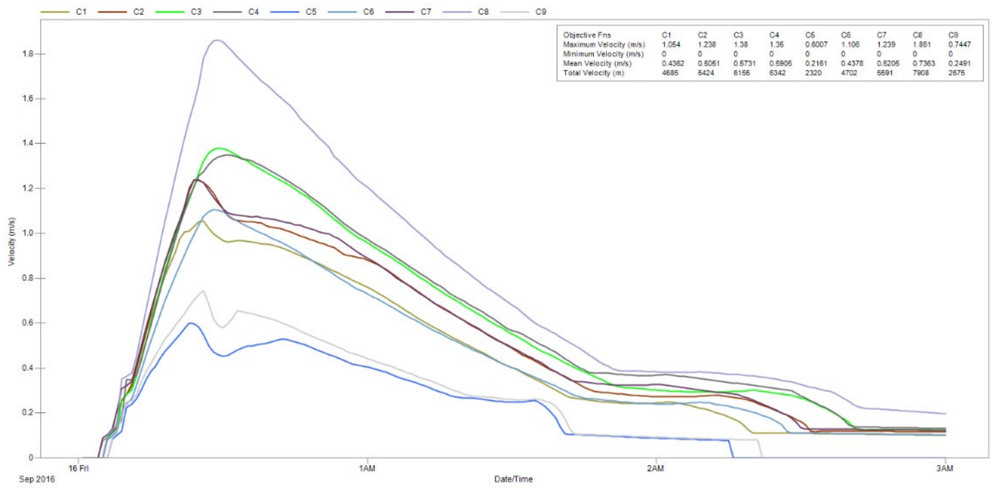
3. Uncheck the **AES_1h_Southwest_Quebec_36mm** if still plotted in the **Graph panel**.

4. Plot the velocity for all links.

5. In the **Time Series Manager**, under **SWMM5 output**, expand **Links** > **Velocity** to plot the stormwater velocities.

6. Select all the conduits (from 1 to 9) by right-clicking **Velocity** and choosing **Select All** from the pop-up menu.

The **Graph panel** can also be used to plot the runoff from all subcatchments and flows in all the conduits, as well as the depths in all nodes in the system (please note the screenshot shown is in SI units, US units will differ).



Note: Your plots of velocities should be similar to the above plots. The peak velocities should exceed the minimum self-scouring velocities specified by your local approval agency (commonly about 2 ft/s | 0.6 m/s). In this design, conduit C9 is a typical dead-end sewer with a small runoff area, so care needs to be taken to avoid sedimentation.

Finally, check the continuity errors reported in the Status panel to ensure that the numerical solution is behaving reasonably well. You can also experiment with radically different routing time steps (be sure to adjust the report time step accordingly) to examine the numerical effects.

2.13 Bonus: Alternative rainfall using IDF curves

The following section reviews the required steps for adding a design storm using IDF curves. For this example, we will be using IDF curve parameters available from the Windsor/Essex Regional Stormwater Management Standards Manual provided by the City of Windsor located in Ontario, Canada.

1. Duplicate the current scenario in the same folder, with the name "**Valleyfield IDF**", the abbreviation "**IDF**", and the description "**Bonus IDF rainfall example**".
2. Click on the **Plan** (scenario manager) button in the toolbar of the **Project panel**
3. In the **Scenario Manager**, click on the **Add** button and select **Duplicate Current Project**.
4. In the **Create Scenario(s)** dialog name the project **Valleyfield IDF**.
5. Specify the location where your exercise is located (e.g. **PCSWMM Exercises\K018\Initial).**
6. Enter in the **Abbreviation** to be **IDF**.
7. Edit the **Description (Title)** text to read **Pre-development peak flow estimation (Bonus IDF example).**
8. Click the **Create** button
9. Click on the **Graph** tab to open the **Graph panel**.

10. Click on the **Add** button in the **Graph panel** to open the **Design Storm Creator**.

11. Under **Type** click on **Chicago** and then select the **IDF...** button to open the **IDF Editor**.

12. Click on the **Add** button to add a new IDF curve and select **abc table** from the list of options.

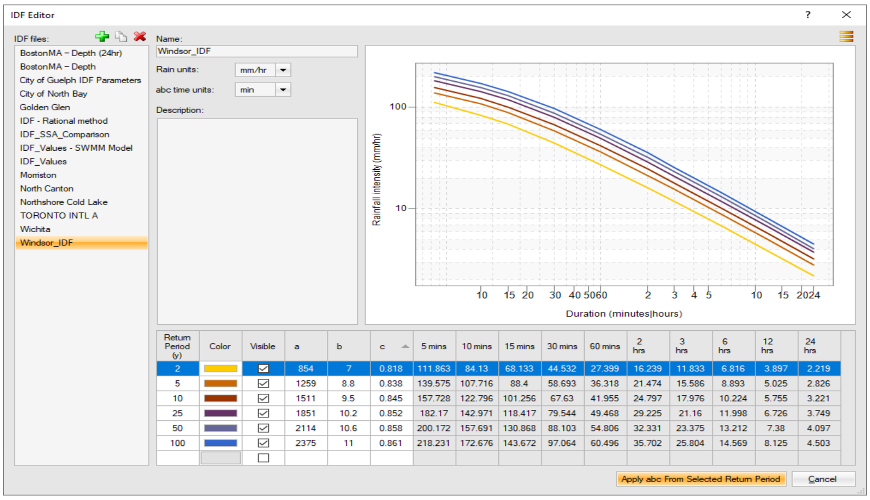
13. Change the name to **Windsor_IDF** and change the **Rain units** to **mm/hr.**

Enter the numbers presented in the tables below.

Note: to copy and paste the table directly into the IDF editor, copy the table to Excel, highlight the a, b, c values and paste them in the Editor (right mouse button click > paste).

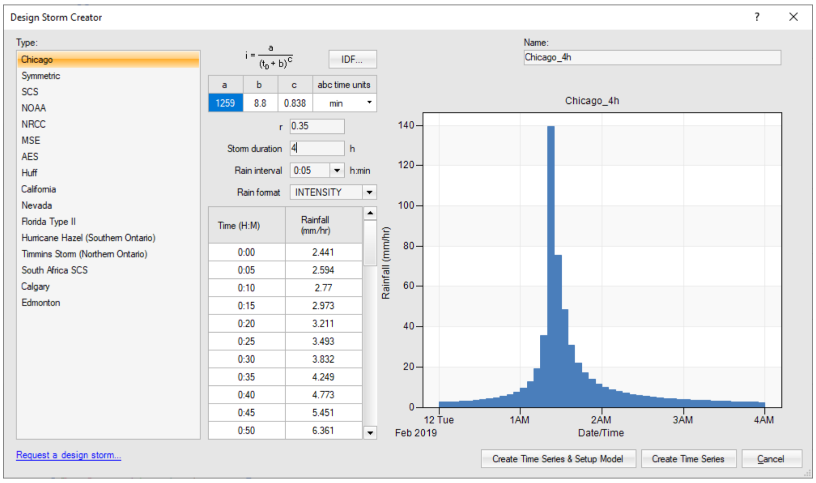
Return Period	a	b	c
2 - Year	854	7.0	0.818
5 - Year	1259	8.8	0.838
10 - Year	1511	9.5	0.845
25 - Year	1851	10.2	0.852
50 - Year	2114	10.6	0.858
100 - Year	2375	11.0	0.861

Toggle the **visible** boxes beside each Return Period to see the IDF curves.



14. Click on the **5 year** return period by selecting the row from the table and click on the **Apply abc From Selected Return Period**.

15. Change the **Storm duration** to **4 h**.



The **Design Storm Creator** tool will still be opened and the Chicago Type distribution will be selected, displaying the a, b and c values from the 10-year design storm.

16. Click on the **Create Time Series & Setup Model** button.

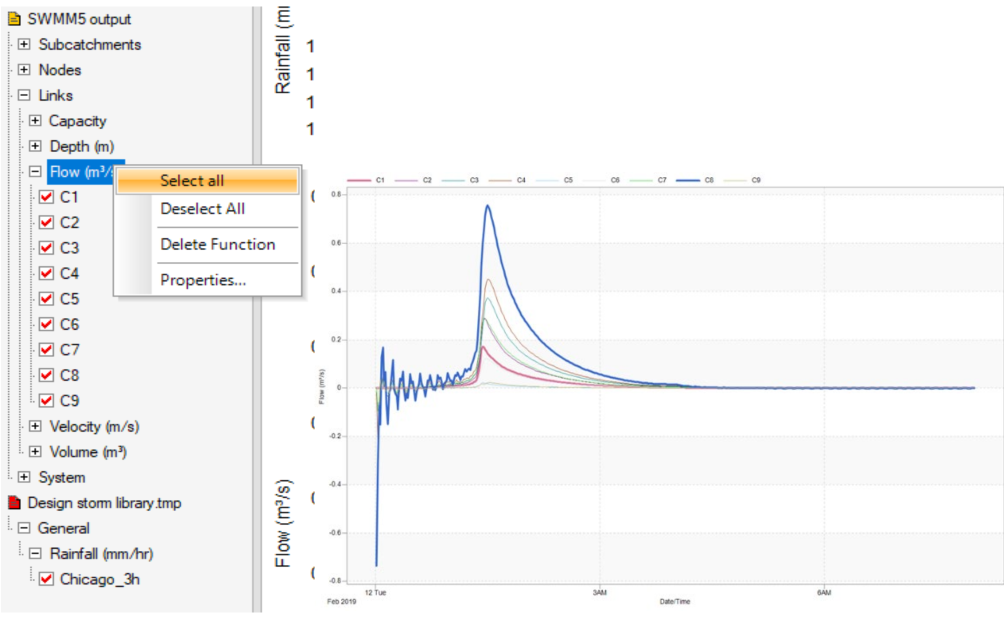
17. Click on the **Map** tab to open the **Map panel**.

For this model we will be using a fixed stage outfall. Fixed stage outfalls are used to model a constant boundary condition. When a fixed stage outfall is used, any upstream junctions with invert elevations equal to or less than the fixed stage will have water in them initially. If this is not accounted for, instabilities can result causing high routing continuity error values.

18. Select the outfall and, change the **Type** to **Fixed** and set the **Fixed Stage** to **152.5 ft** | **46.5 m**.

19. Click on the **Run** button to run a simulation and update the results.

In the **Graph panel**, expand the results and plot the new flows in all the conduits (**SWMM5 Output** > **Links** > **Flow** and right click > **Select All**). Take note of the backwater flow through C8.



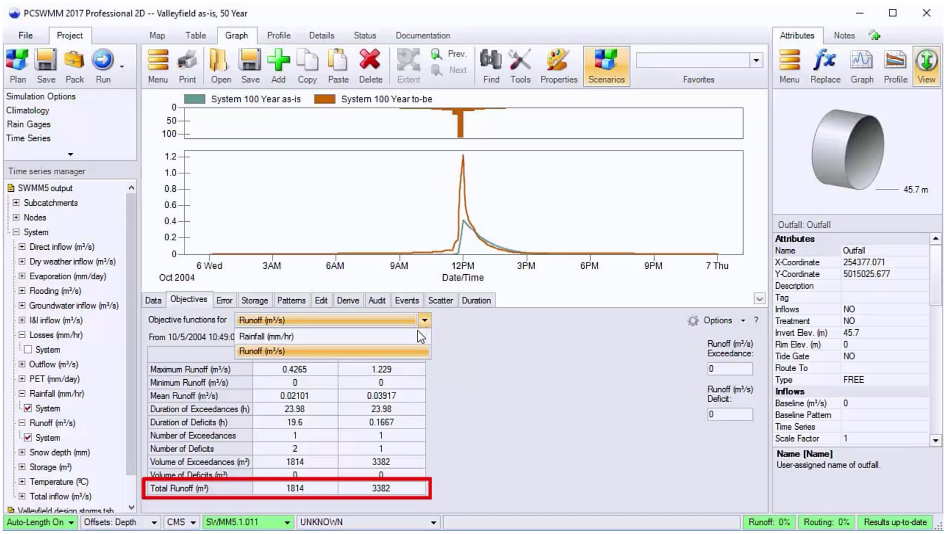
3. System evaluation using multiple return period design storms

This exercise illustrates how to quickly set up and evaluate model response to multiple design storms.

Stormwater infrastructure design should take into account watershed pre-development drainage patterns and watershed boundaries as a strategy to maintain natural site conditions.

In some cases unit relationships are available defining the unit peak flows for a boundary. If these are not available, it may be useful to compare the runoff response of the to-be model to an as-is model, as demonstrated here.

Written by request for the Toronto and Region Conservation Authority (TRCA), this chapter is based on requirements outlined in Section 3.2 of TRCA's Stormwater Management Criteria Manual and evaluates a new development design using 2, 5, 10, 25, 50 and 100-year design storms.



3.1 Opening and grouping as-is and to-be models

We will begin by linking the pre-development and post-development models as scenarios.

1. Open the **Valleyfield as-is.inp** file from the **PCSWMM Exercises\K021\Initial** folder.
2. In the **File** tab, click the **Open**  button.
3. Browse and open the project **PCSWMM Exercises\K021\Initial** and select **Valleyfield as-is.inp**.
4. A dialog will appear showing a default location to unpackage the model: click on the **Unpackage** button then the **OK** button.

5. Add **Valleyfield to-be** as a scenario.

6. Click on the **Plan** button in the Project panel to open the **Scenario Manager**.

7. In the **Scenario Manager**, click on the **Add** button and choose **Add Existing Project**.

8. Open the **Valleyfield to-be** scenario.

9. Browse to **K021\Initial** and select **Valleyfield to-be.inp** then the **Open** button.

10. Click the **Close** button to close the **Scenario Manager.**

3.2 Loading design events in the Graph panel

For this exercise we will be evaluating the pre- and post-development models using multiple design storms. Let's start by creating a 100-y storm.

1. Click on the **Graph** tab to open the **Graph panel**.

2. Select the **Add** button to open the **Design Storm Creator**.

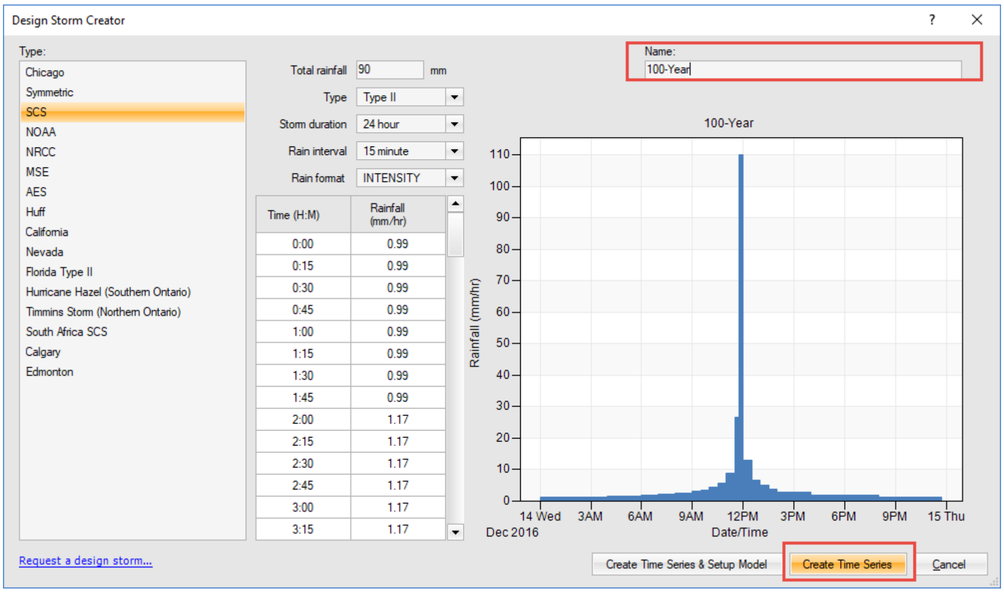
3. Select **SCS** from the list of design storms.

4. Change the total rainfall to be **3.5 in | 90 mm**.

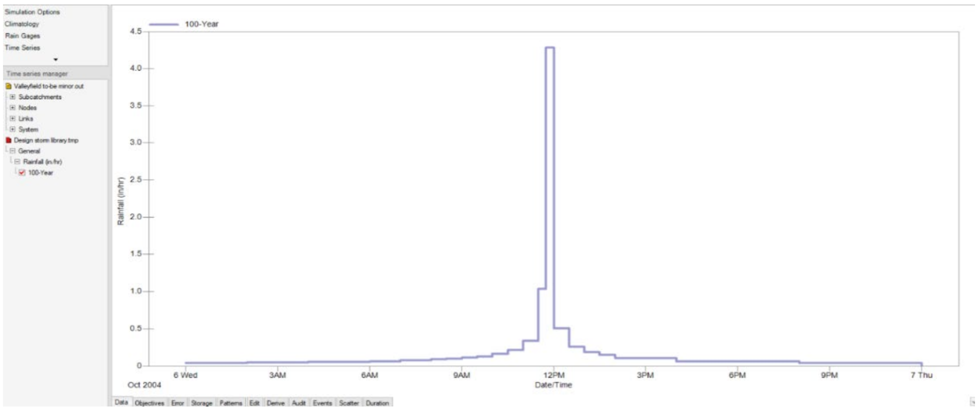
5. Choose a **Type II** storm and set the **Storm duration** to **24** hours.

6. Leave the **Rainfall interval** set to **15 min** and the **Rainfall format** set to **Intensity**.

7. Edit the **Name** to be **100-Year**.



8. Click on the **Create Time Series** button to create the 100-y time series (please note the screenshot shown is in SI units, US units will differ).



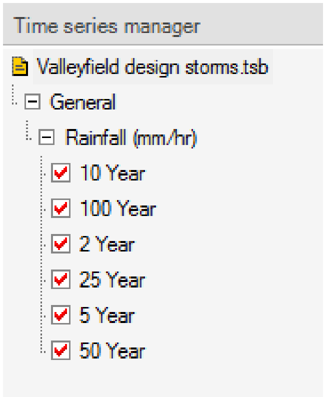
We will now repeat the steps 2-8 above to create the 50-y, 25-y, 10-y, 5-y and 2-y storms using the **SCS 24h Type II** design storm distribution. The table below summarizes the **Total rainfall** values and **Names** to be used for each storm.

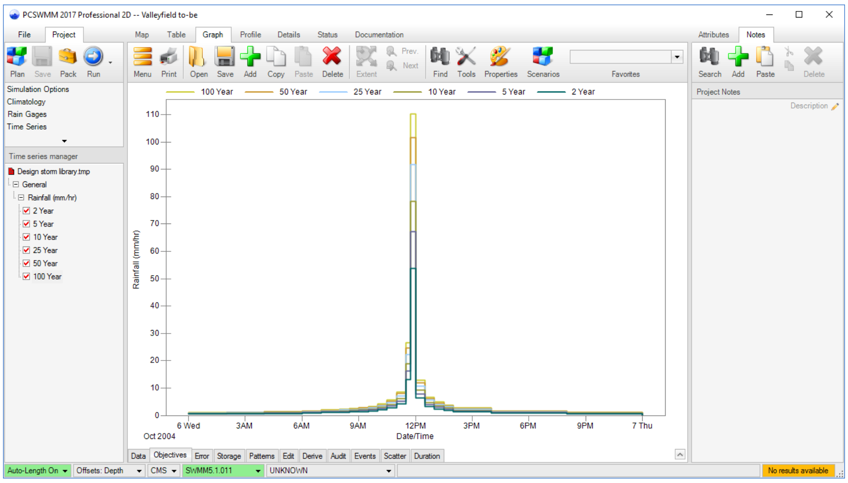
Name	Total rainfall - US (in)	Total rainfall - SI (mm)
50-Year	3.2	83
25-Year	3.0	75
10-Year	2.5	64
5-Year	2.2	55
2-Year	1.7	44

3.3 Saving design storms

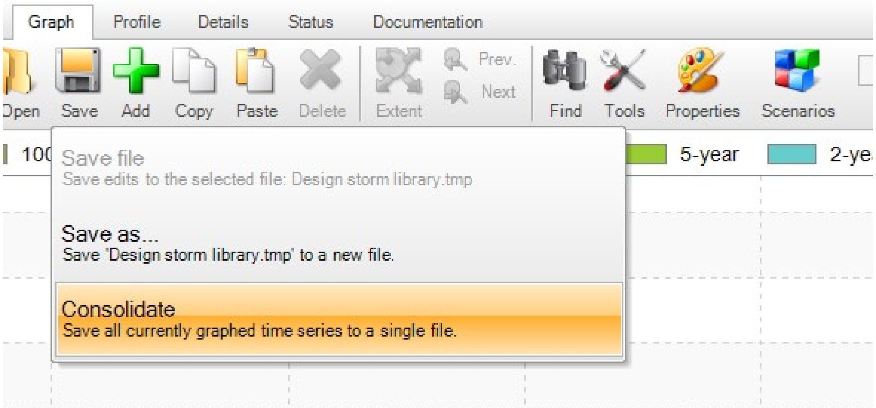
We will now save the time series as a single time series file by consolidating the graphed time series.

1. Click the check box next to each design storm time series to plot it in the **Graph panel** if it is not plotted already.





2. Click on the **Save** button in the **Graph panel** and select **Consolidate.**



3. Browse to the **K021\Initial** folder and name the file **Valleyfield design storms.tsb**. Click **Save**.

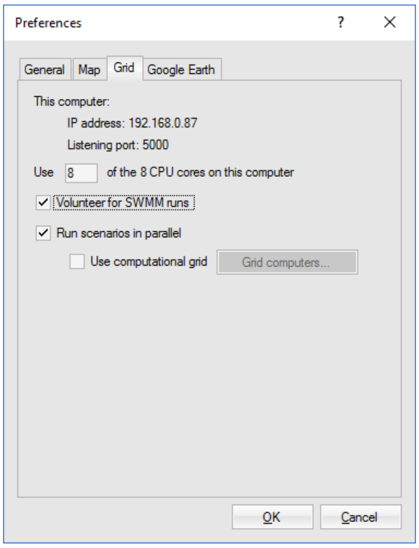
3.4 Generating return period storm scenarios for to-be model

In PCSWMM, scenarios can be created using graphed or saved design storms. This allows the user to quickly generate multiple scenarios and run them. In order to reduce computational time, it is useful to have multiple CPU cores in your computer. The first thing we will do is set up PCSWMM to leverage all of the available cores.

Set PCSWMM to use all of the CPU cores available on your computer.

1. Open the **Map panel**, click on the **Menu** button and select **Preferences...**

2. Click on the **Grid** tab and change the number of cores being used to the total number of cores available on your computer.



We will now generate 6 scenarios, one for each design storm.

3. Click on the **Run** button to run the current scenario.

4. Use the **Create Design Storm Scenarios** tool to create and run scenarios from the six currently graphed time series.

5. In the **Project panel** click on the **Plan** button.

6. Click on the **Add** button and select **Create Design Storm Scenarios** from the drop-down menu.

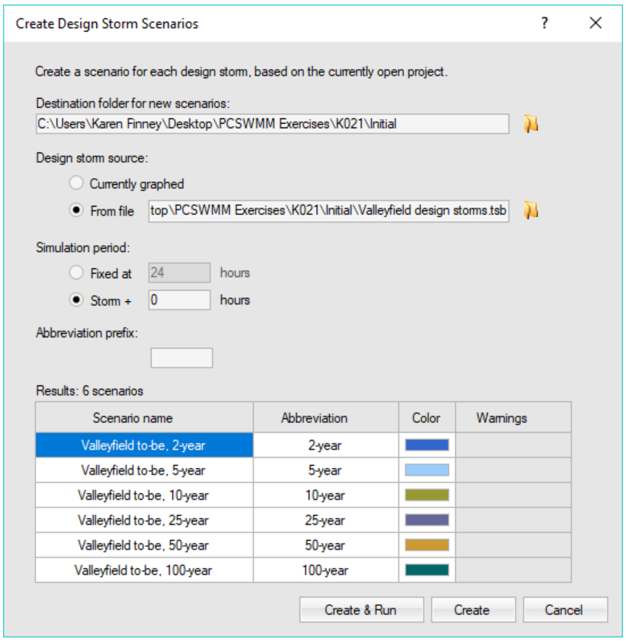
7. In the **Create Design Storm Scenarios** tool, use the **Browse** button and navigate to **PCSWMM Exercises\K021\Initial** and click **OK**.

8. Select the **Design storm source** to be **From file** and use the **Browse** button and navigate to **PCSWMM Exercises\K021\Initial**, select **Valleyfield design storms.tsb** and click **OK**.

9. Leave the **Simulation period** set to **Storm+ (0 hrs)**.

10. Click on the **Create & Run** button

This will create 6 new scenarios and run each one with their corresponding rainfall time series. If you have multiple cores, the scenarios will be run in parallel, reducing computational time.



3.5 Generating return period storm scenarios for as-is model

We will now create the design storm scenarios for the as-is model. We first need to switch to the as-is model scenario and then repeat the steps above to create the as-is design storm scenarios.

1. Click on the **Plan** button in the **Project panel** to open the **Scenario Manager**.
2. In the **Scenario Manager**, double click on the **Valleyfield as-is** scenario to open it.
3. If an option comes up to **Save time series**, click **Save time series.** In the **Graph panel**, click the **Save** button and choose **Save file** to save changes to the time series. Repeat steps 1 and 2.
4. If an option comes up to save the project click **Save project**.
5. Once the scenario has loaded, click on the **Run** button to run the model.
6. Click on the **Plan** button in the **Project panel** to open the **Scenario Manager** again.
7. Use the **Add** button to create and run 6 design storms scenarios for the as-is model using the **Valleyfield design storm.tsb** file. Refer to steps above in the previous section if you get stuck.

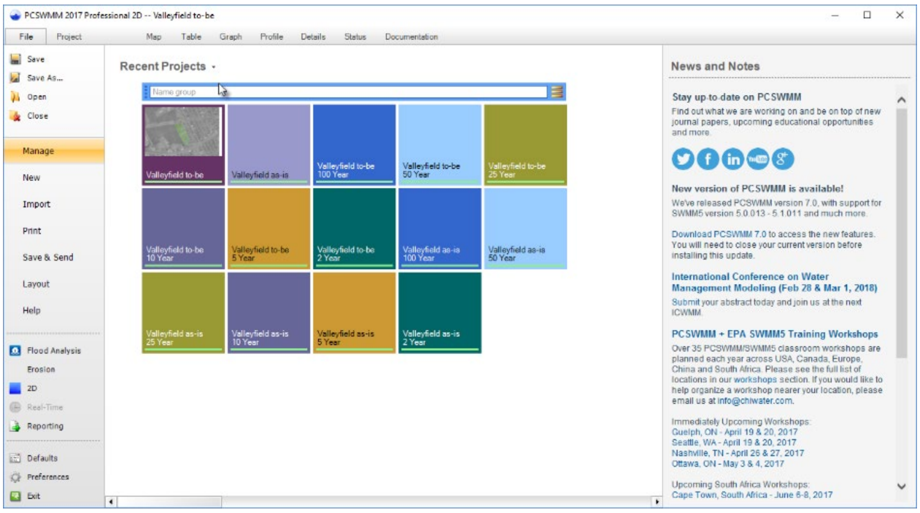
3.6 Recoloring scenarios

To better understand the results, we will recolor the scenarios to each have a unique color.

1. Click on **File** > **Manage**.

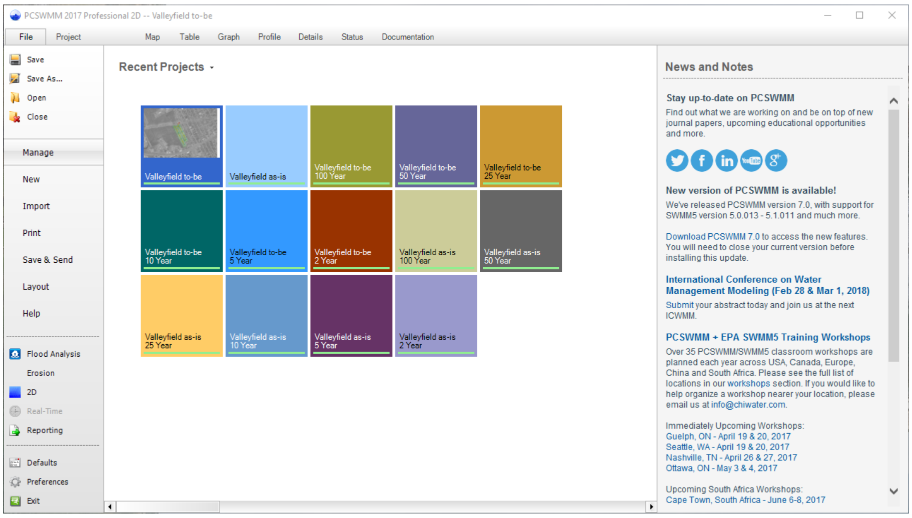
The scenarios will be grouped together in the **Manage** tab.

Hover on top of the group of scenario tiles until a blue box appears (as shown in the screenshot).



2. Click on the **Menu** button.

3. Choose **Color by > Sequential.** The scenarios will now be re-colored sequentially.



3.7 Evaluating results

Now that the design storm scenarios have all been created and run, their results can be compared with a number of tools in PCSWMM. As an example, we will do a hydrograph comparison.

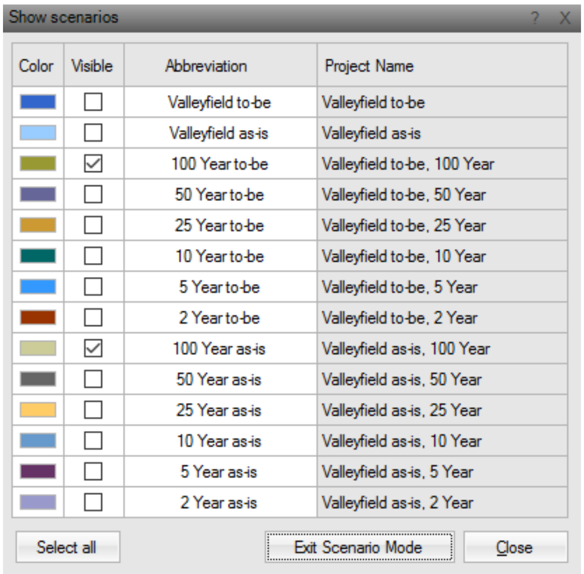
1. Switch scenarios by clicking on the **Plan** button in the **Project panel** and select one of the design storm scenarios.

2. Switch to the **Graph panel**.

3. In the **Time series manager**, graph **Nodes > Total inflow > Outfall**.

4. Click on the **Scenarios** button in the **Graph panel.**

5. Add a "to-be" or "as-is" suffix to the design storm depending on which parent scenario it came from. Refer to the screenshot.



Color	Visible	Abbreviation	Project Name
	☐	Valleyfield to-be	Valleyfield to-be
	☐	Valleyfield as-is	Valleyfield as-is
	☑	100 Year to-be	Valleyfield to-be, 100 Year
	☐	50 Year to-be	Valleyfield to-be, 50 Year
	☐	25 Year to-be	Valleyfield to-be, 25 Year
	☐	10 Year to-be	Valleyfield to-be, 10 Year
	☐	5 Year to-be	Valleyfield to-be, 5 Year
	☐	2 Year to-be	Valleyfield to-be, 2 Year
	☑	100 Year as-is	Valleyfield as-is, 100 Year
	☐	50 Year as-is	Valleyfield as-is, 50 Year
	☐	25 Year as-is	Valleyfield as-is, 25 Year
	☐	10 Year as-is	Valleyfield as-is, 10 Year
	☐	5 Year as-is	Valleyfield as-is, 5 Year
	☐	2 Year as-is	Valleyfield as-is, 2 Year

6. Un-check the current scenario and check both **100 year** events (to-be and as-is).

7. In the **Graph panel** click on the **Objectives** tab.

8. Compare the maximum and total inflows of the to-be model using the **Objective functions** table at the bottom of the screen.

9. Repeat steps 4-6 to compare each of the remaining design storms (**50 year**, **25 year**, etc.).

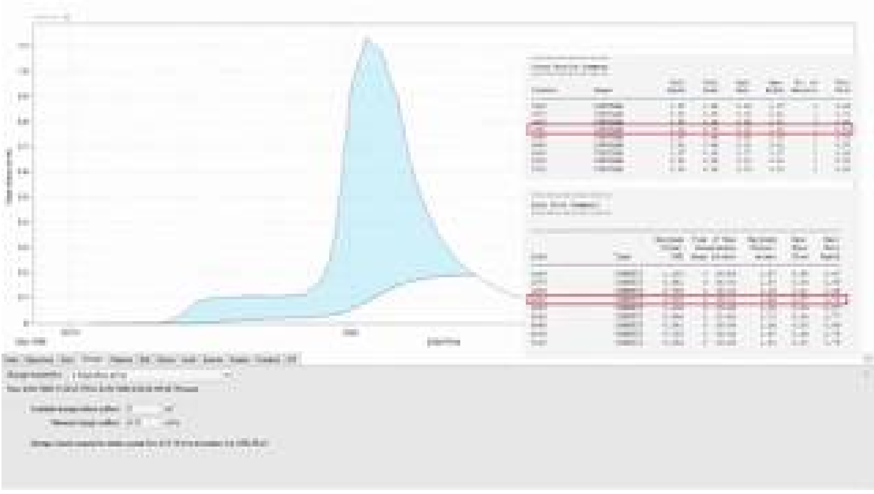
10. If any revisions to the model are required, make the changes in the original to-be scenario and regenerate the design storm scenarios – the existing scenarios will be overwritten.

3.8 References

Toronto and Region Conservation. 2012. Stormwater Management Criteria Draft. Version 1.0.

4. Simple Storage estimation

This exercise illustrates one method of reducing street flooding by sizing a detention pond to attenuate lateral inflow from a subcatchment.

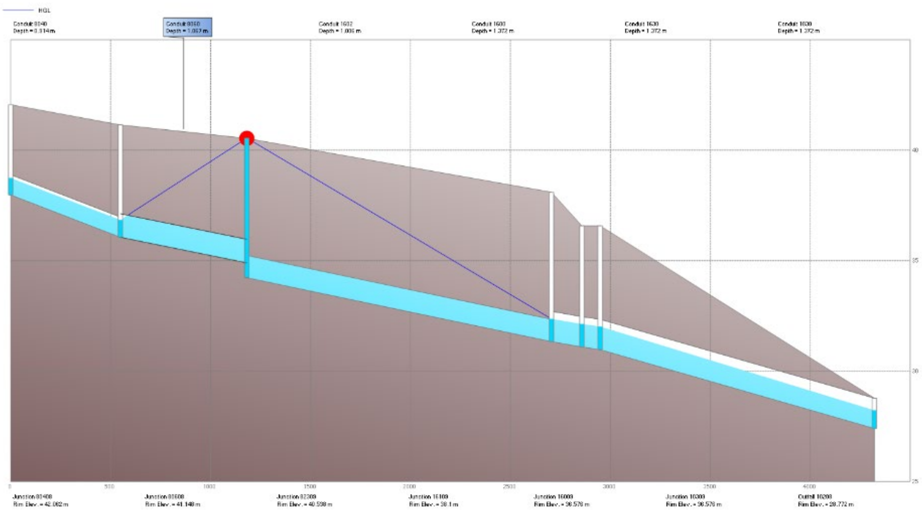


4.1 Open and run the project file

1. Open **Selected Conveyance.inp** from the **PCSWMM Exercises\K043\Initial** folder.
2. Launch PCSWMM.
3. Click **File** and then the **Open** button.
4. Browse to the folder **PCSWMM Exercises\K043\Initial**, select the **Selected Conveyance.inp** file and click the **Open** button.
5. Click the **Run** button in the toolbar of **Project panel**.

4.2 Find surcharging pipes

1. View the profile from the upper node **80408** to the outfall node **10208**.
2. In the **Map panel**, click junction **80408.**
3. Hold down the **Shift** key and select the outfall, **10208.**
4. Click the **Profile** tab to switch to the **Profile panel**.
5. In the **Profile panel** click on the **Menu** button and select **Show Peak Values** to view the peak HGL (hydraulic grade line) for the selected pathway.



In this model, conduit 1602 is undersized and causing pipe surcharging and surface flooding in the upstream branch of the drainage network. The purpose of this exercise is to try and reduce the flooding by inserting an off-line detention pond between subcatchment 1 and node 82309 to attenuate the flows coming into the system from subcatchment 1.

6. Return to the **Map panel** by clicking the **Map** tab.

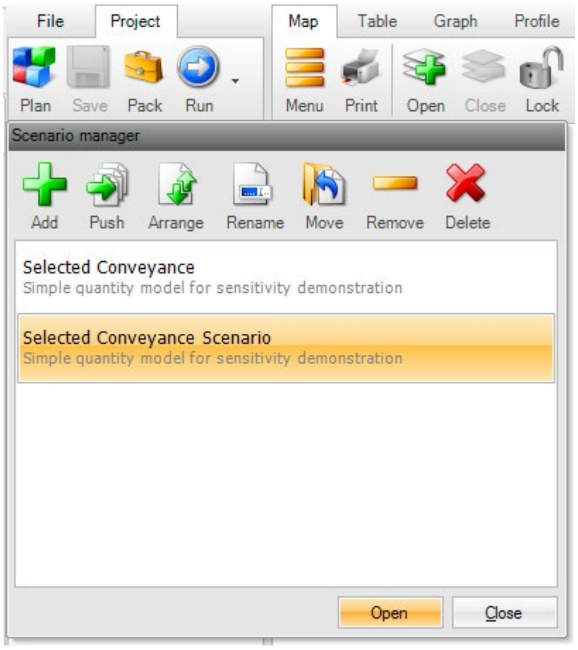
This approach involves determining the excess capacity of conduit 1602 without the contributing runoff from subcatchment 1. A storage pond will be then created for capturing the runoff from subcatchment 1, and an outlet will be designed to release a maximum peak flow to node 82309 that can be accommodated by conduit 1602. Alternative solutions may include digging up and replacing conduit 1602 with a larger diameter pipe, and/or placing a cross-connection to the other branch of the system upstream of conduit 1602.

4.3 Determine excess capacity of a conduit

To begin with, let's create a scenario model to work on:

Duplicate the current scenario, name it **Selected Conveyance Scenario**, and open it.

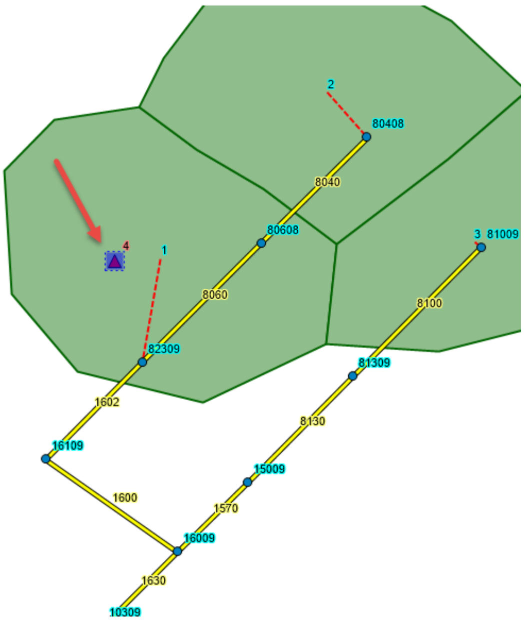
1. Click on the **Plan** button in the **Project panel**.
2. Click on the **Add** button and select to **Duplicate Current Project** .
3. Name the new scenario **Selected Conveyance Scenario.**
4. Click on the **Browse** button and navigate to **Desktop\PCSWMM Exercises\K043\Initial**.
5. Click on the **Create** button.
6. Click on the **Plan** button if the **Scenario manager** is not already open, select the newly created scenario and click on the **Open** button.
7. Click the option to **Save** the current project.



Now we will remove subcatchment 1 from the system by directing it to another inlet. In the **Map panel**:

8. Select the **Outfalls** layer item in the **Layers panel**.

9. Click the **Add** button in the toolbar in the **Map panel** and click in a blank area of the map northwest of node **82309**. The name of the outfall will be placed as **4**.



10. Click the **Select** button in the toolbar and select the Subcatchment **1**.

11. In the **Attributes panel**, change the **Outlet** attribute value to **4.**

12. Click the **Run** button in the toolbar of **Project panel** to update the results.

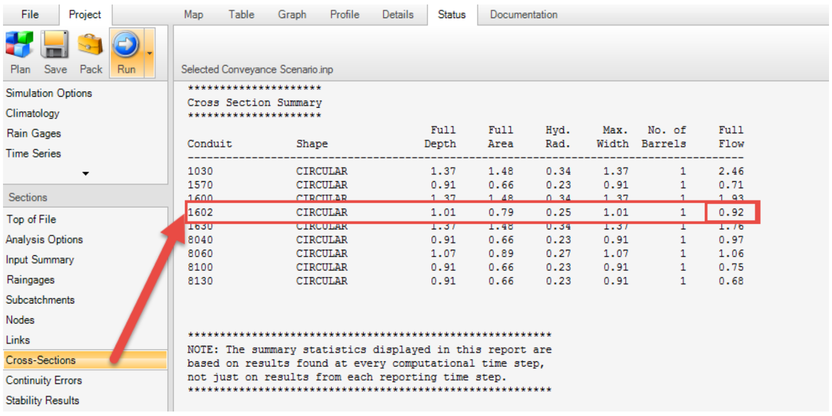
13. Select the pathway from the upper node 80408 to the outfall node 10208 and click the **Profile** tab to view the new peak HGL without the contributions from subcatchment **1**.

Conduit 1602 is almost at peak capacity; however, we should be able to attenuate the flow from subcatchment 1 enough with a detention pond to prevent surcharging.

Now we need to find the maximum hydraulic capacity of conduit 1602 and subtract the current peak flow to find the available capacity.

1. Click the **Status** tab to switch to **Status panel**.

In the **Sections** list of the **Status panel** select **Cross-sections.** The full flow for conduit 1602 should be **32.5** cfs | **0.92** m³/s.



2. Select **Link Flows** from the list of sections to jump to the **Link Flow Summary** section, you will see **Maximum Flow** for **Conduit 1602** is **35.15** cfs | **1.0** m³/s. This leaves **2.65** cfs | **0.08** m³/s as the maximum allowable contribution from subcatchment **1**.

Now we can design our pond and outlet structure. To do this we can apply one of the **Tools** in the **Graph panel** to compute runoff from subcatchment 1 (i.e. the flow at location 1).

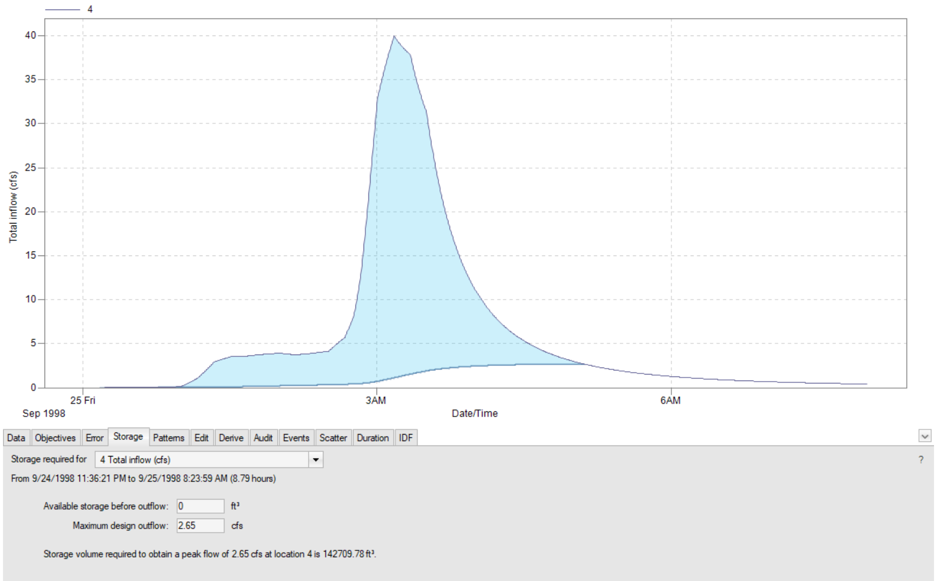
3. Switch to the **Graph panel.**

4. Click on the **Tools**  button and select **Storage Pond Calculator**.

5. In the **Time Series Manager**, expand **SWMM5 output > Nodes > Total inflow** and select the location **4** by clicking on the check box.

6. In the **Storage** tab at the bottom of the panel, enter a **Maximum design outflow** value of **2.65** cfs | **0.08** m³/s – The part of the Total inflow graph should be filled in blue.

For the sake of this exercise **Available storage before outfall** will be left at 0 (please note the screenshot shown is in US units, SI units will differ).

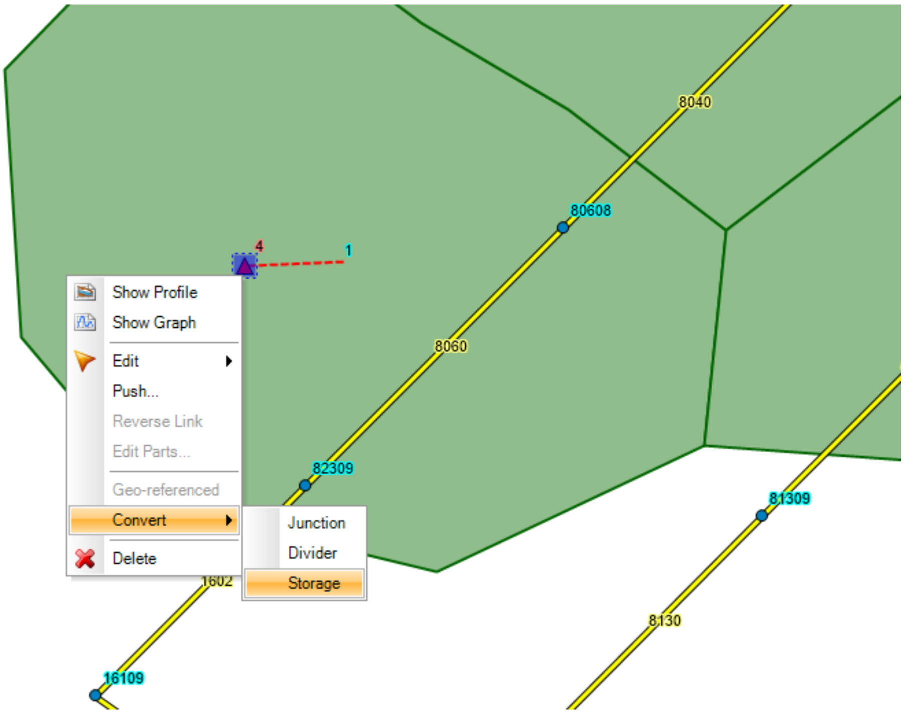


The storage function should report a required storage volume of about 142,709 ft^3 | 4,024 m^3. So we now know we need a pond that can store about 142,709 ft^3 | 4,024 m^3 and have a maximum outflow of 2.65 cfs | 0.08 m^3/s at its maximum capacity.

4.4 Insert a storage pond in the existing model

We will now insert a storage pond and link it to both subcatchment **1** and node **82309**.

1. Return to the **Map panel**.
2. Click the Outfall **4**, right click and select to **Convert > Storage**.



Now add an orifice to connect the storage facility to the drainage system:

1. Click the **Orifices** layer in the **Layers panel.**

2. Click the **Add** button in the **Map panel** toolbar.

3. Draw an orifice link between the new storage node and the existing node **82309**. This is done by clicking once the upstream node (our storage node **4**) and then clicking the downstream node (**82309**).

4. Click the **Select** button in the toolbar to escape from the **Add shape** mode.

Now we need to enter the attributes for the storage node and orifice link entities. The detention pond will be defined as a constant area pond (a tank) for simplicity.

5. Click the new storage node **4** to select it and edit its attributes.

6. Check to see if the **Depth** attribute is grayed out in the attributes panel. If it is, select the **Depth** attribute and click on the **Auto-expression** button that appears in the value field. In the **Auto-expression editor**, click on the **Calculate Rim Elev. Instead** button.

Update the attributes for storage node **4** as follows:

Invert El. = 113 (ft) | 34.5 (m)

Depth = 6.6 (ft) | 2 (m)

Storage Curve = FUNCTIONAL

Coefficient = 0

Constant = 21,623 ft^2| 2,012 m^2

For the orifice link, we will use a discharge coefficient of 1 (assuming a smooth round over entry for our circular orifice). We will place the orifice at the bottom of our tank, which will produce a head of approximately 7m. Using Q=AVK, where V=sqrt(2gh)=~6.3 m/s, we get an area of ~0.013 m^2 (using A=pi*r^2 this gives a diameter of about 0.13 metres).

7. Click the orifice link in the **Map** and enter the following attribute values in the **Attributes panel**:

Type = SIDE

Shape = CIRCULAR

Height = 0.42 (ft) | 0.13 (m)

Width = 0

Inlet Offset = 0

Discharge Coeff. = 1

8. Click the **Run** button in the toolbar to save your changes and execute a SWMM run.

After the SWMM run is completed, we are now ready to test this approach. In the **Map panel**,

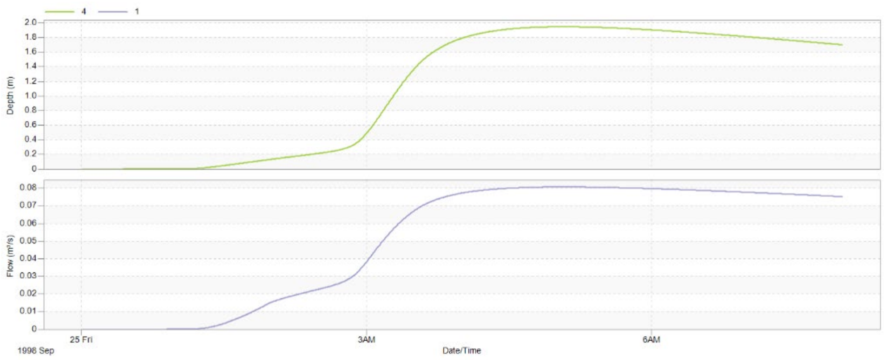
9. Select a pathway from storage node **4** to outfall **10208** and view the HGL (profile).

10. Click the **Profile** tab to switch to the **Profile panel**.

11. Click the **Play** button to view the peak HGL.

The peak HGL should not be surcharging. Note the peak flow through the orifice.

12. Click the **Graph** tab and expand **Nodes > Depth > 4** and **Links > Flow > 91001**.



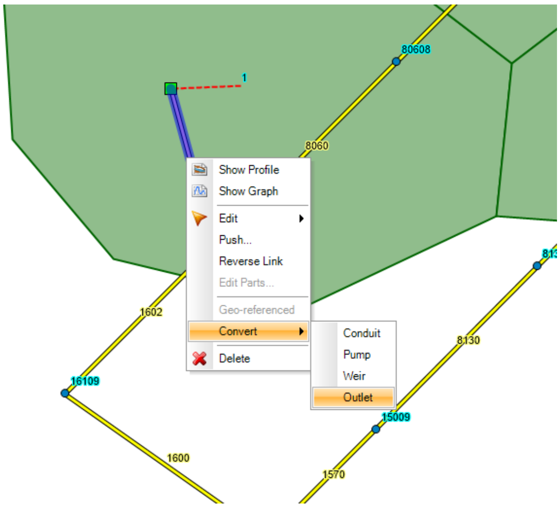
4.5 Substitute the orifice with an outlet

Replace the orifice connecting the storage tank 4 and node 82309. Outlets are used to control outflows from the storage tank by its rating curve as a function of the head difference across it.

1. Click on the **Map** tab to open the **Map panel.**

2. Select and right-click orifice **91001** in the **Map panel**.

3. In the pop-up menu, choose **Convert > Outlets**.



4. In the **Attributes panel**, change the **Rating Curve** to **TABULAR/DEPTH**.

5. Click in the **Curve Name** textbox and click the **ellipsis** button.

6. Create a new curve by clicking **Add** in the **Rating Curve Editor**.

7. Leave the name as **Curve1**.

Enter data as the outlet discharges no flow when zero head and maximum outflow of the storage tank (0.08 m3/s) at the maximum head. Enter the values as shown.

Head (m)	Outflow (m^3/s)
0	0
2	0.08
3	0.08

SI units

Head (ft)	Outflow (cfs)
0	0
6.6	35.3
7.6	35.3

US units

8. Cick the **Assign to Outlet91001** button.

Execute a SWMM5 **Run** and check the profile to ensure Conduit **1602** is not surcharging.

5. Modeling LIDs (Valleyfield, Quebec)

Low Impact Development (LID) is a term used to describe a sustainable storm water management approach that emphasizes conservation and water quality protection. LIDs are becoming more common in new developments and are also being implemented in older subdivisions in order to promote infiltration, reduce storm water runoff and improve receiving water quality. As such, LID devices are typically used to intercept, store and infiltrate some of the rainwater before it reaches the conveyance system. In this exercise we illustrate one way of assessing the potential reduction in storm water runoff by modeling a set of boulevard planters.

Multiple LIDs can be placed per subcatchment, and the LID function estimates overflows, infiltration flow and evaporation in Rain Barrels, Vegetative Swales, Porous Pavement, Bio Retention Cells and Infiltration Trenches. With some limitations, component process layers that can be used in each LID include:

Surface,

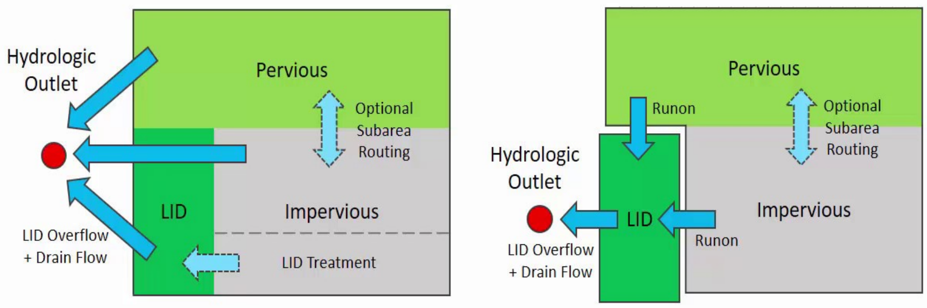
Pavement,

Soil,

Storage, and

Drain.

Output includes a LID summary in the Status report (.rpt) file and an external report file giving computed surface depth, soil moisture, storage depth, surface inflow, evaporation, surface infiltration, soil percolation, storage infiltration, surface outflow and the LID continuity error (which usually can be reduced by a smaller wet hydrology time step). The time series contained in the external report file can also be plotted in PCSWMM.



5.1 Opening the model

1. Open **Valleyfield to-be.inp** from the **PCSWMM Exercises\K031\Initial** folder**.**

   Launch PCSWMM.

2. Click on **File** and choose **Open** from the list of options.

3. Browse to the folder **PCSWMM Exercises\K031\Initial** and open **Valleyfield to-be.inp** file.

4. **Open** the **H64020_r14_Residential.dxf** and **photo-grande-ile.jpg** background layers from the same folder.

5. Click the **Open** button in the **Map panel**.

6. Click on the **Browse** button and browse to the folder **PCSWMM Exercises\K031\Initial**.

7. Select two files (holding the **Ctrl** key down) **H64020_r14_Residential.dxf** and **photo-grande-ile.jpg** and click on the **Open** button.

5.2 Creating a long term simulation

In order to assess the LID performance it is helpful to run a continuous time series. For this example, an external 5 year continuous rainfall time series will be used.

1. In the **Project panel**, click on **Rain Gages** to open the **Raingage Editor**.

2. Click **Add** to create a new rain gage object, and change the name to **1991-1996**.

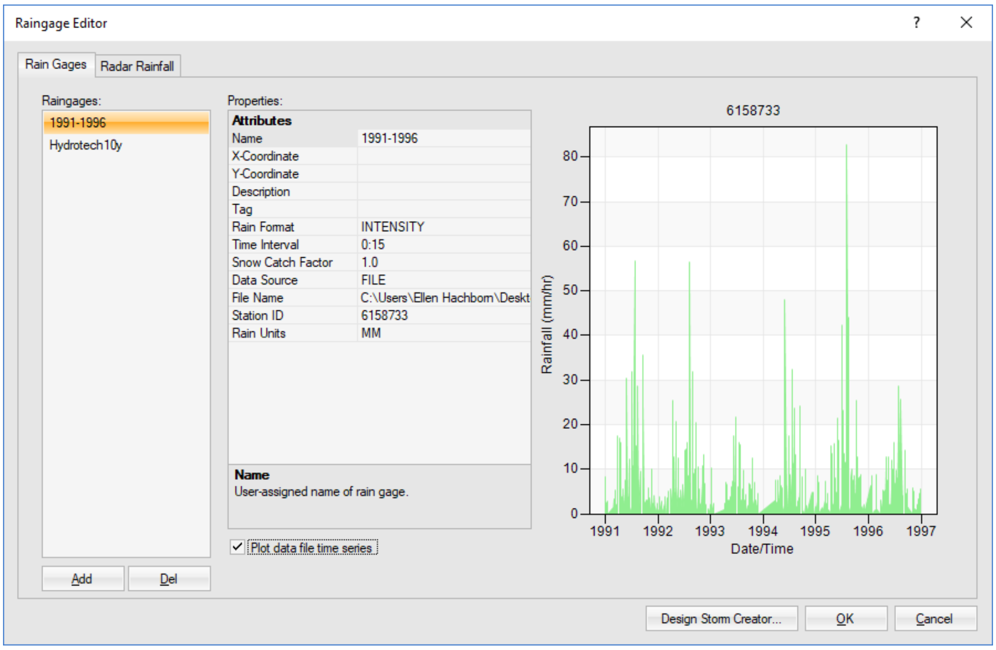
3. Set **Rain Format** to **Intensity**, and **Time interval** to **0:15.**

4. Change the **Data source** to be **File** and click on the **ellipsis** ... , browse to **PCSWMM Exercises\K031\Initial** and select **6158733_1991-1996_disaggregated_following_intensity.dat**.

5. In the **Station ID** box type in **6158733** and set the units to **mm** (even if working in US units)**.**

6. Check the **Plot data file time series** checkbox to preview the rainfall time series, as shown in the screenshot below.

7. Click the **OK** button to save your changes and close the **Raingage Editor**.



8. Assign the **1991-1996** rain gage to all subcatchments.

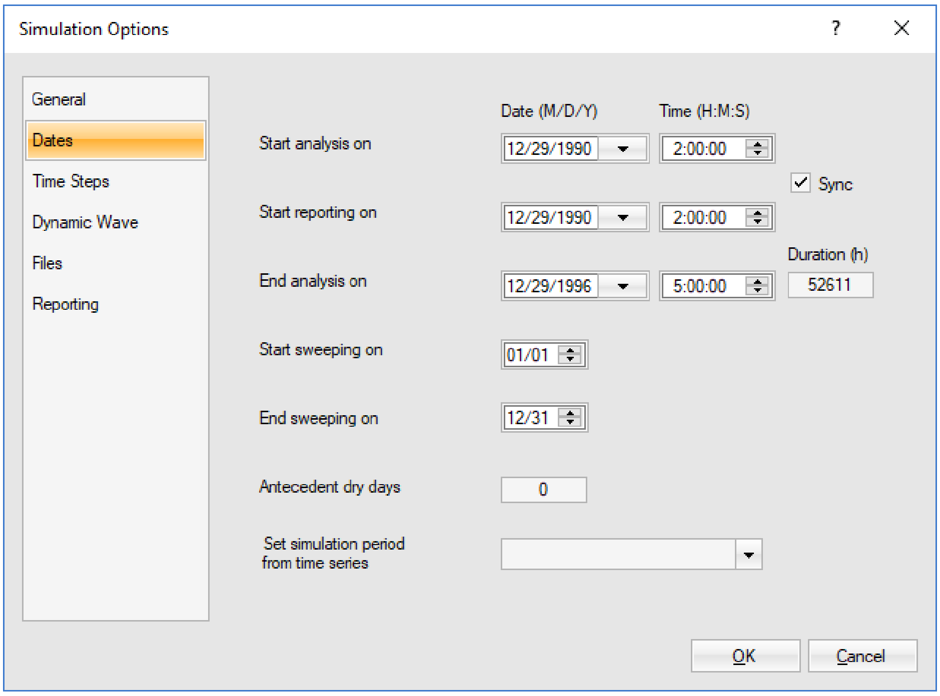
9. In the **Layers panel**, select **Subcatchments.**

10. Press the **Ctrl** + **A** keys to select all of the subcatchments.

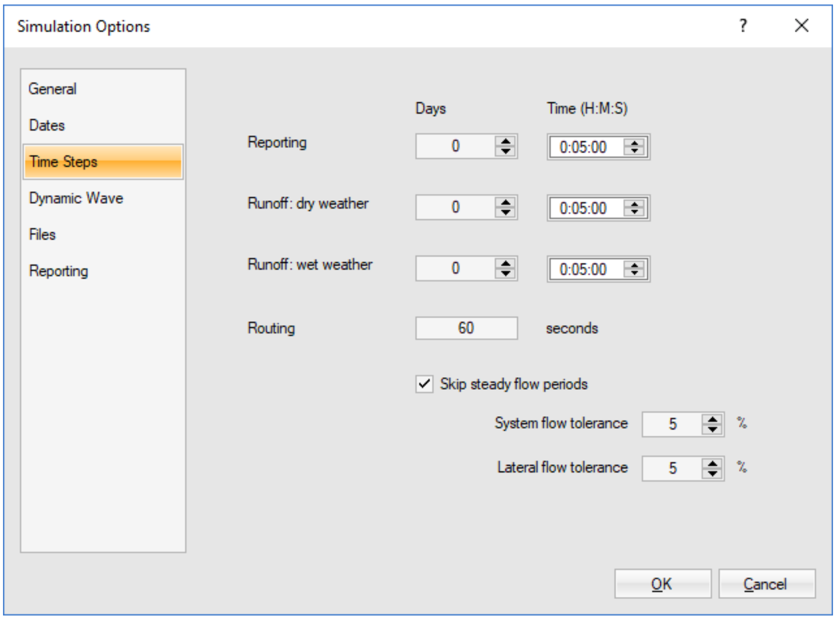
11. In the **Attributes panel** set **Rain gage** to **1991-1996**.

12. Click on **Simulation Options** in the **Project panel**.

13. Under the **Dates** tab, set the simulation options to be the same as shown in the following screen capture.



14. Open the **Time Steps** tab and change the **Routing** time step to **60** seconds if its not already.



15. Click on the **OK** button to save your changes and close the **Simulation Options** editor.

16. Click on the **Run** button in the **Project panel** to run the model.

5.3 Creating a LID Scenario

Now that we have a baseline model run for the 5 year period, we can create a new scenario to model the LID devices.

1. Create a duplicate scenario named **Valleyfield LIDs** and open the new project.

2. Click the **Plan** button in the **Project panel**, click on the **Add** button and select **Duplicate Current Project**.

3. **Name** the new scenario as **Valleyfield LIDs** and click **Create.**

4. Click the **Plan** button and open the new scenario by double-clicking on the new **Valleyfield LIDs** model.

5. Choose to **Save Project** if prompted.

6. Load the **LIDs.shp** background layer in the **Map panel.**

7. Click the **Open layer** button in the **Map panel**.

8. Click on the **Browse** button and browse to the folder **PCSWMM Exercises\K031\Initial** and open the **LIDs.SHP** layer.

9. In the **Layers panel**, drag the **LIDs.SHP** layer to the top of the list of layers.

10. Unlock the **LIDs** layer by clicking the **Lock/Unlock**  button.

Note: The LIDs layer is a polygon layer that was created in order to represent the boulevard planters (see image below), however the LID analysis can be performed without physically adding any new layers to the PCSWMM interface, allowing both small scale and large infrastructure projects to be easily modeled using the SWMM5 LID extension.

5.4 Defining LID attributes in LID toolbox

Each type of LID control is represented by a different combination of vertical layers. For example, the parameters defined when modelling porous pavement include surface, pavement and storage layers, but when modelling a vegetative swale only the surface layer is to be defined.

Parameter values for these layers can be obtained from the Reference tables on the PCSWMM support site, as well as through engineering drawings, and other hydrology literature values in textbooks and reports. The Reference tables include values for surface roughness, and soil infiltration parameters. In addition, the LID Control Editor reference article on the PCSWMM support site offers guidance for other LID parameters, such as clogging factors and drain coefficients.

The table in the image indicates which combination of layers applies to each type of LID (x means required, o means optional).

LID Type	Surface	Pavement	Soil	Storage	Drain	Drainage Mat
Bio-Retention Cell	x		x	x	o	
Rain Garden	x		x			
Green Roof	x		x			x
Infiltration Trench	x			x	o	
Permeable Pavement	x	x	o	x	o	
Rain Barrel				x	x	
Rooftop Disconnection	x				x	
Vegetative Swale	x					

In this example we will be modelling boulevard planters which would have similar properties as a bio-retention cell.

A bio-retention cell consists of three layers: **surface, soil** and **storage**, along with 8 modeled processes. Below is a diagram illustrating the layers and component processes to be defined in the LID editor.

Now we will define these three layer using PCSWMM.

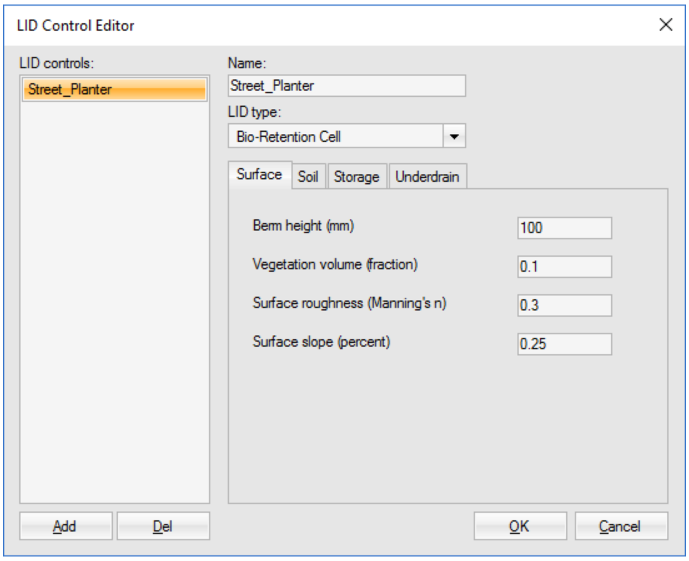
1. In the **Project panel** click on **LID Controls** to display the **LID Control Editor.**

2. In the editor click **Add** to create a new LID object, and change the **Name** of the LID to **Street_Planter.**

3. In the **LID type** drop down box select **Bio-Retention Cell**.

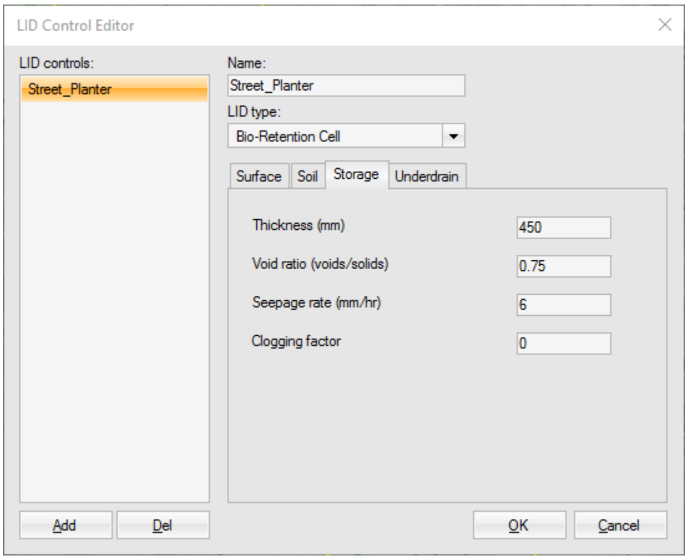
The image below shows an example of a bio-retention cell with approximately 4 in**|**100 mm of surface storage courtesy of a weir outlet. This design is similar to the boulevard planters that will be modeled.

Populate the values in the **Surface tab** as shown in the following screen capture. For this example each street planter will have a depth of **4 in|100 mm** consisting of a vegetation fraction of **0.1** a Manning's roughness of **0.3** and a slope of **0.25%** (please note the screen shot shown is in SI units, US units will differ).



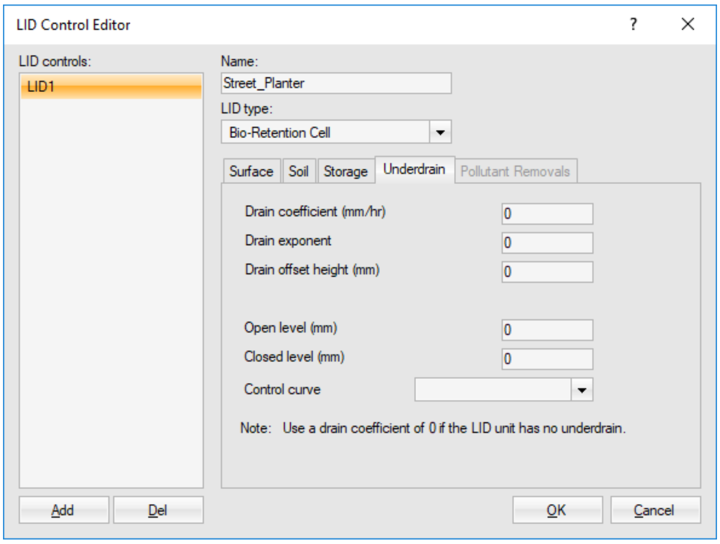
4. Click on the **Soil** tab in the **LID control editor** and populate the values as shown in the following screen capture. For this example each planter will have a soil surface thickness of **35 in|900 mm**, a porosity of **0.44**, a field capacity value of **0.11**, a wilting point value of **0.05,** a conductivity value of **1 in/hr|25 mm/hr**, a conductivity slope of **7.5** and a suction head of **3.5 in|88.9 mm** (please note the screen shot shown is in SI units, US units will differ).

5. Click on the **Storage** tab in the LID control editor and populate the values as shown in the following screen capture. In this example each rain garden will have a storage thickness of **18 in|450 mm**, with a void ratio of **0.75** and a conductivity of **0.24 in/hr|6 mm/hr** (please note the screen shot shown is in SI units, US units will differ).



6. Click on the **Underdrain** tab in the LID control editor and populate the values as shown in the following screen capture. In this model there will be no underdrain so **all** the attribute values will be set to zero.

7. Click **OK** to close the **LID Control Editor** (please note the screen shot shown is in US units, SI units will differ).



5.5 Assigning subcatchment LIDs

We will begin by specifying the LID details for each subcatchment:

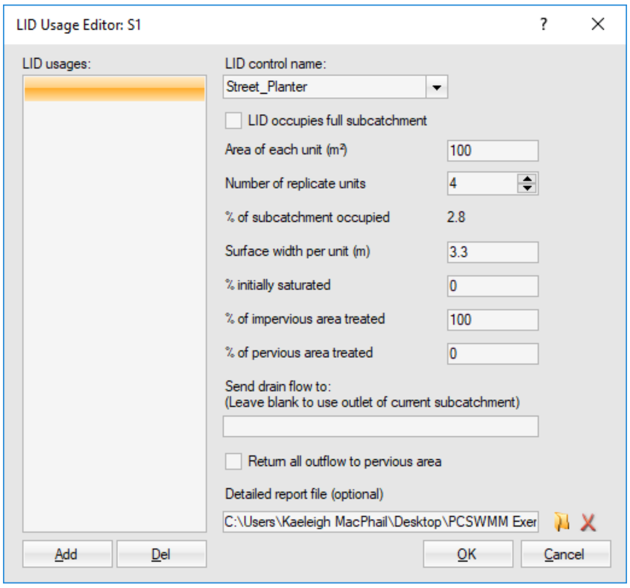
1. Select subcatchment **S1** in the **Map panel** and in the **Attributes panel** click on the **LID Controls** item (please note the screen shot shown is in US units, SI units will differ).

2. Click the **ellipsis**  button in the **LID Control** box to open the **LID Usage Editor.**

3. The **LID Control** box allows the user to specify the type and number of LID controls located in a subcatchment, as well as other subcatchment specific LID parameters.

4. In the **LID Usage Editor** click on the **Add** button.

5. Assign the properties of the LIDs located on Subcatchment **S1** as shown in the following screen capture. For **S1** there are 4 boulevard-planters to be built, each planter will occupy an area of **1076 ft² | 100 m²** and a top width of **10.8 ft | 3.3 m** and will be assumed to treat **100%** of the impervious area (please note the screen shot shown is in SI units, US units will differ).



6. To specify the creation of a **Detailed report file**, click on the **Browse** button, navigate to **PCSWMM exercises\K031\Initial\Valleyfield LIDs** and save the LID report as **S1-LIDs**.

7. Repeat step 3-4 for the remaining subcatchments, make sure each subcatchment has the correct number of planters as shown in the **LIDs** layer or using the table below.

Note: there are no LIDs on subcatchments **S5** and **S9** so the LID Controls attribute in the **Attributes panel** should be left as 0.

Subcatchment	Number of Planters
S1	4
S2	4
S3	4
S4	1
S6	4
S7	6
S8	1

Adjust subcatchment area and flow width to compensate for LIDs.

There are two approaches to placing LIDs:

Placing one or more LIDs in an existing subcatchment that will displace an equal amount of non-LID area from the subcatchment.

Creating a new subcatchment devoted entirely to a single LID and routing adjacent subcatchment runoff onto this LID subcatchment

When LIDs are added to a subcatchment, the subcatchment's Area property is the total area of the subcatchment (both non-LID and LID portions) while the percent imperviousness and width parameters apply only to the non-LID portion.

In this example we are adding LIDs to existing subcatchments so we will need to adjust the subcatchment percent imperviousness to compensate for the new LIDs.

1. Select **Subcatchments** from the list of tables in the **Tables panel** to display the subcatchment attributes.

2. Change the subcatchment **Imperv (%)** values to match the **New Imperv (%)** values in the table below:

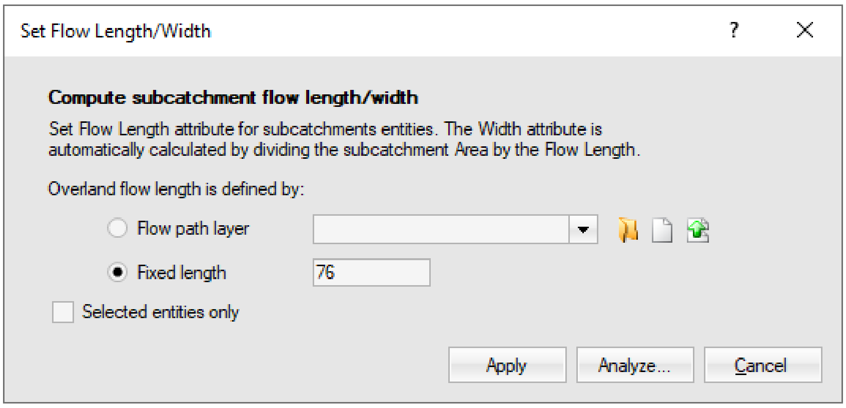
Name	Original Imperv (%)	New Imperv (%)
S1	30	27.1
S2	30	26.2
S3	30	26.2
S4	30	28.2
S5	30	30.0
S6	30	25.8
S7	30	25.9
S8	30	25.9
S9	30	30.0

3. Compute the **Width** attribute for the subcatchments, by dividing each subcatchment area by a approximate maximum length of overland flow. We'll assume the average maximum length of overland flow is the sum of the average lot depth (115 ft | 35 m) plus the curb length between planters (135 ft | 41 m), which equals 250 ft | 76 m.

4. Return to the **Map panel** by clicking on the **Map** tab.

5. Click on the **Tools** button and click on the **Set Flow Length/Width** tool (in the Subcatchments and Spatial Detail sections).

6. Select **Fixed length** in the dialog and enter the computed length found in the **Shape** section in the **Attributes panel**, **250 ft | 76 m**, in the textbox and ensure that **Selected entities only** is not checked.



7. Click **Apply** to execute it.

8. Click on the **Run** button.

5.6 Comparing LID results

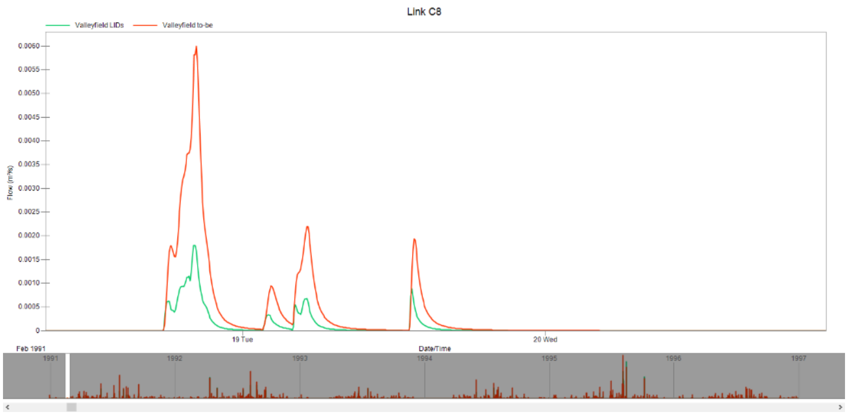
1. Open the **Graph panel.**

2. Click the **Scenarios** button (**next to Favorites**) and toggle on the **to-be** model, then select **Compare Scenarios**.

3. In the **Time series manager** expand it to show **Links > Flow > C8**.

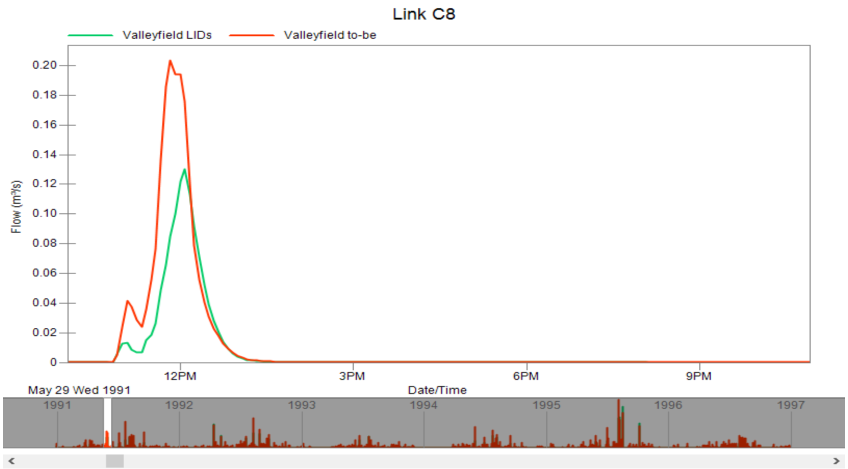
LID performance typically varies with rainfall event size. LIDs are typically more effective at reducing runoff for smaller events that are more frequent.

Zoom into a smaller event close to the beginning of the simulation (see screenshot below). Notice the reduction in total stormwater runoff leaving the subdivision in the LIDs scenario compared to the original to-be model.



4. Click the **Full Extent** button to zoom out to the entire simulation hydrograph time series.

5. Select a larger event close to the beginning of the simulation by holding down and dragging your mouse in the **Graph panel** (see screen shot below).

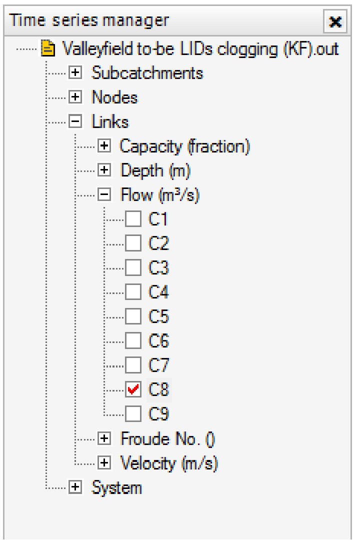


Notice that there is a smaller percent reduction in both total flow and peak flow. For these large events the planters become saturated and are no longer able to store and infiltrate runoff as effectively.

5.7 (Optional) Comparing LID performance with LID clogging factor

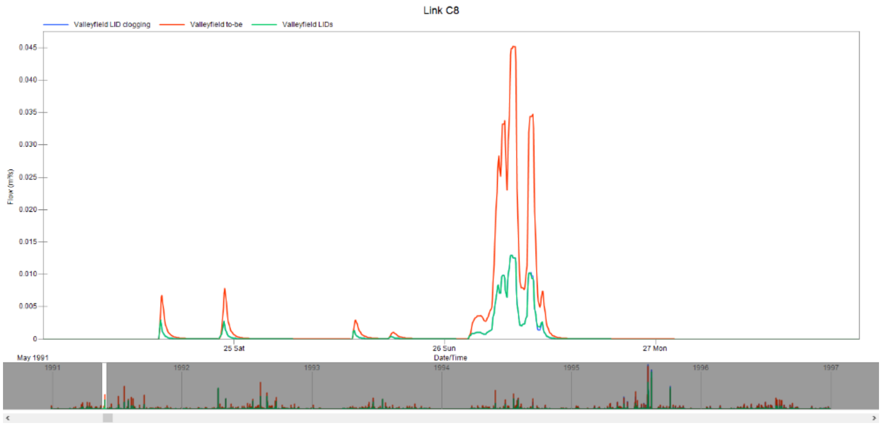
Over time LID performance may degrade due to clogging. Clogging is the reduction of infiltration capacity due mostly to fine particles. Clogging of LIDs in SWMM is modeled as a linear function of total infiltration. We will now create a new scenario to compare the performance of a LID with and without the addition of a clogging factor.

1. Duplicate the current scenario in the same folder, name it **Valleyfield LID clogging.**
2. With the **Valleyfield LIDs** model open, click on the **Plan** button and click the **Add** button and select **Duplicate current scenario**.
3. Name the scenario **Valleyfield LID clogging** and click **Create.**
4. Click on the **Plan** button and click on **Valleyfield LID clogging** and click **Open**.
5. In the new project, click on the **LID Controls** in the **Project panel** to open the **LID Controls Editor**.
6. Open the **Storage** tab for the **Street_Planter** LID control.
7. Change the value of the clogging factor to be **100**.
8. Click on the **OK** button to save your changes and exit the **LID Controls Editor**.
9. Click on the **Run** button in the **Project panel** to run the model.
10. Compare scenario results for the flow in **C8** again.
11. Open the **Graph panel.**
12. Click the **Scenarios** button (**next to Favorites**) and toggle on the **to-be** model.
13. Select the **Compare Scenarios** button.
14. In the **Time series manager** expand it to show **Links > Flow > C8** to compare the runoff hydrograph with and without clogging.

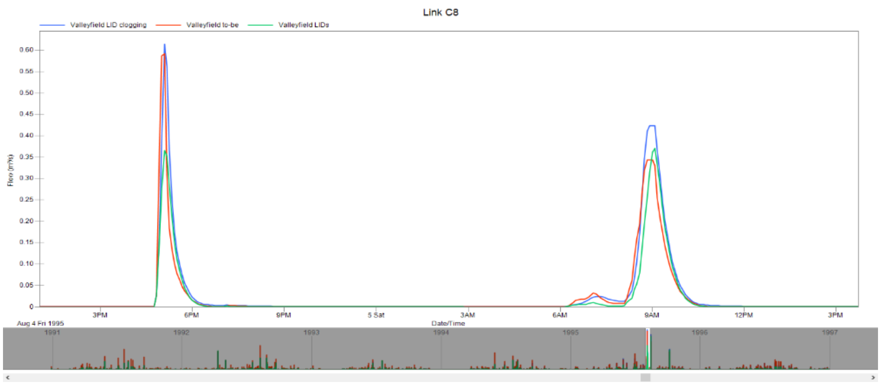


Zoom into a smaller event close to the beginning of the simulation (see screenshot below). Notice that the **LID clogging** factor has a minimal effect on the runoff response. This is because this event is at the beginning of the simulation period and there has been minimal clogging.

Note: To add the time line to the bottom of the plot in the Graph panel click on the Properties button and select the Zoom Window to be On.



Now select a period in the middle of the simulation and observe how over time the planters become clogged and are no longer able to reduce the runoff as much as when the LID was initially installed.



5.8 (Optional) Opening the LID report

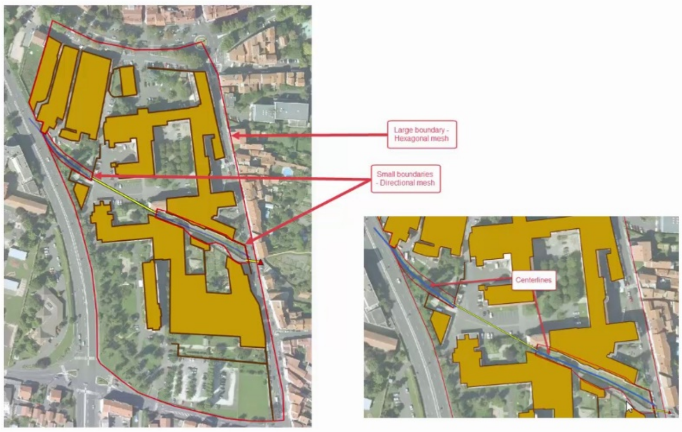
1. Click on the **Graph** tab to open the **Graph panel**.
2. Click on the **Open** button and browse to **PCSWMM exercises\K031\Initial\LIDs** and select **S1-LIDs**.
3. Expand **S1>Storage Level> Street_planter** and look at the changing storage depth over time.
4. Open the LID reports from the other subcatchments and look at the reported time series.

6. Combined 1D-2D urban flood analysis

European cities are commonly characterized as densely urbanized with small meandering streets and stone walls. Their drainage systems often consist of a mixture of open flow and underground conduits. One common stormwater problem in these cities is the frequent river flooding during heavy rains, due to urban development along the river flood plains in combination with walls acting as barriers to flood flows.

In this exercise we will be modeling an area in France where a hospital is located. Because flooding is a major concern, there are strict regulations and new construction is only authorized if it does not exacerbate the risk of flooding. In this example, the hospital is planning building improvements that require more precise information on the flow behavior (flow velocities and water levels) to adopt the best strategy for flood mitigation.

The objective of this study is to model the area with an assumed flow boundary condition at the upstream end of the river to compute potential flooding depths and flow velocities.



6.1 Create model and define 2D layers

When developing a 2D model in PCSWMM it is important to define the layers that will be used. The minimum input layers you need are a Bounding layer and a 2D nodes layer. A DEM is used to define the surface elevations of the modeled area (if DEM is not available PCSWMM assigns a uniform initial elevation of 0 m for all the 2D nodes).

In this exercise we will be using 4 existing layers: bounding, obstructions, river centerline, and DEM.

First, however, we need to create and setup the project:

1. Create a new project called **2D Model** in the **PCSWMM Exercises\K001\Initial** folder.
2. Open **PCSWMM** and create a new project by clicking **New** from the list of options on the left side of the **File** tab.

3. Select **SWMM5 Project**.

4. Name the model **2D Model**.

5. Click on the **Browse** button and navigate to **PCSWMM Exercises\K001\Initial**.

6. Click the **Create Project** button to create the new project.

This exercise is in SI units only so we need to set the units accordingly. If you are working outside of the US the units should already be set up to use SI units however if you are located in the US PCSWMM would have made the default US units.

7. Change the model flow units to **CMS.**

8. Click on the **Simulation Options** item from the **Project panel**.

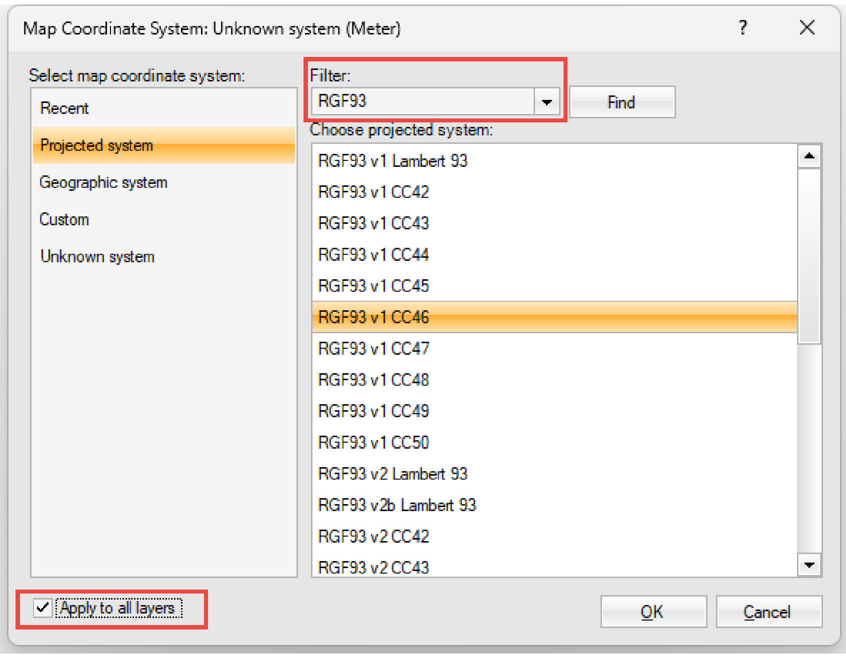
9. Under the **General** tab in the **Miscellaneous** section change the **Flow units** to be **CMS**. If you are completing this exercise outside of the US, the units should already be in CMS. Click **OK** to save changes.

10. Set the coordinate system for the map and all layers to **RGF93 v1 CC46**.

11. Click on the **Menu** button in the **Map panel** and select **Coordinate system**.

12. Select **Projected system** from the list of coordinate systems and type in **RGF93** in the **Filter** box and select **RGF93 v1 CC46** from the filtered list.

13. Toggle on the box to **Apply to all layers** and click **OK**.



14. Click on **Keep existing coordinate system** option if asked if you want to make this the default coordinate system for new projects.

15. Load the following background layers from the **PCSWMM Exercises\K001\Initial\GIS layers** folder: **high resolution photo.png, Bounding-2D.SHP, Obstructions-2D.SHP, River centreline-2D.SHP,** and **w001001.adf**.

16. Click on the **Open**  button in the **Map panel**.

17. Click on the **Open** button.

18. Browse to **PCSWMM Exercises\K001\Initial\GIS layers.**

19. Press the **Ctrl** key to select several different layers; **high resolution photo.png, Bounding-2D.SHP, Obstructions-2D.SHP, River centreline-2D.SHP**, and **w001001.adf**.

20. Click on the **Open** button to open the selected layers.

21. Click on the option to **Set coordinate syste**m to **RGF93 v1 CC46 (3946)**.

    Now we will enable the 2D modeling component and define the 2D layers.

22. Click on the **File** tab to open the PCSWMM backstage.

23. Click on the **2D** button located in the backstage menu (under **Professional Features**).

24. Check the **Enable 2D modeling** box to define the layers that will be used to set up the 2D model.

25. In the **Bounding layer** drop-down menu select the previously opened **Bounding - 2D** layer.

26. In the **Obstructions layer** drop-down menu select the previously opened **Obstructions - 2D** layer.

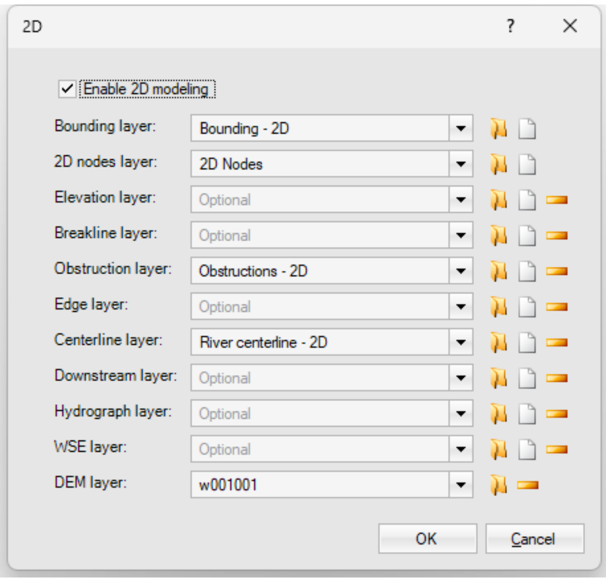
27. In the **Centerline layer** drop-down menu select the previously opened **River centerline - 2D** layer.

28. In the **DEM layer** drop-down menu select the DEM file **w001001**. **Do not close the 2D dialog.**

    We need to create a new shape file layer to store the 2D nodes.

29. Click on the **New** button beside the **2D nodes layer** drop-down menu.

30. Navigate to **PCSWMM Exercises\K001\Initial\GIS layers** and create the layer with the default name **2D Nodes**. Press **Save**.



Note: If you are using a PCSWMM version 7.5 or earlier, in the 2D editor, place a check next to **Include velocity post-processing**.

31. Click **OK** to save your changes to the 2D setup and close the **2D** editor.

A message will appear asking if you want to create the missing fields in the boundary layer. Click **Yes** to create these fields.

Rearrange the layers so the DEM layer (named "w001001") is positioned at the bottom (if it is not already) by dragging and dropping the layer in the **Layers panel**.

32. Click the **Save** button.

Note: There are up to 8 layers that can be used in a 2D model. For more information about the 2D layers in PCSWMM please visit our Overview.

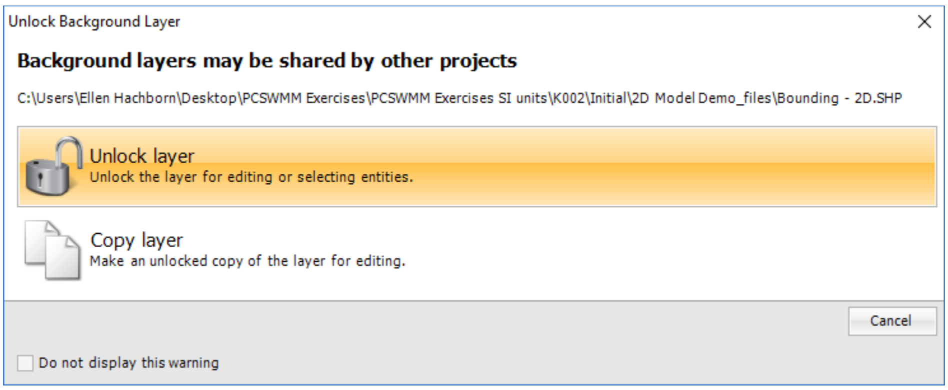
6.2 Generate Points

PCSWMM creates a 2D mesh using the 2D nodes layer to define the overland flow paths. A user can influence the point generation by specifying a mesh type and resolution in the **Boundary** layer polygon attributes. After the 2D nodes layer has been populated, points may be edited (moved, deleted, or added). For example, to add points, select the **2D nodes** layer, click on the **Add** button in the **Map panel** and add points of specific interest (i.e. bank stations, depressions, etc.).

In this model the Bounding – 2D layer consists of three polygons, one large one encompassing the hospital area and two smaller ones representing the river channels. We will define two different mesh types for these regions. A hexagonal mesh will be used to define the overland flooding area and a directional mesh will be generated to represent the river channels.

1. Click on the **Bounding – 2D** layer in the **Layer panel.** Unlock the layer by right-clicking on the layer and selecting **Unlock**. A message will be displayed asking if you want to unlock the layer or make a copy of the layer to edit. Select **Unlock layer**.

Note: When working with scenarios, one should be careful in editing background layers that is used by multiple scenarios as the changes will apply to all scenarios. Thus PCSWMM displays a message box confirming your intent when unlocking a layer, giving you the option to create a copy of the layer. The **2D Cells** layer is managed by PCSWMM for each project however other layers (bounding, 2D nodes, etc.) are background layers and can be used for more than one project.



2. Select the larger polygon in the **Bounding- 2D** layer.

3. In the **Attributes panel** specify the **Style** to be **Hexagonal**.

Since the map units are in SI units the 2D mesh has to also be defined in SI units. We will model the flood area at a 5 m resolution.

4. Next to **Resolution** type in **5** (m).

5. Next to **Roughness** type in **0.033** representing the roughness of overland flow.

6. Next to **Edge** select **Yes** from the drop-down menu. This has the same effect as the edge line layer.

Leave the rest of the attributes as shown.

Bounding - 2D	
Attributes	
Style	Hexagonal
Angle (deg)	0
Resolution (m)	5
Sampling Factor	1
Dist. Tolerance (r	0
Elev. Tolerance (	0
Roughness	0.033
Seepage Rate (m	0
Edge	YES
Other	
LENGTH	0
WIDTH	0
Shape	
Uid	1
Count	1
Vertices	38
Parts	1
Area (m²)	61171.16
Area (ha)	6.1171

We will now define a directional mesh for the smaller bounding polygons that encompass the open channels. To account for the greater variation in elevation along a cross-section, we will use a finer mesh resolution perpendicular to the mesh direction (i.e. a smaller mesh **Width**).

7. Select the two smaller boundary polygons by selecting one, holding down the **Ctrl** key and then selecting the other.

8. In the **Attributes panel** specify the **Style** to be **Directional**.

9. Next to **Resolution** enter **3** (m).

10. Next to **Roughness** type in **0.04** (the channel is heavily vegetated).

11. Next to **Edge** select **Yes** from the drop-down menu (this creates a clean edge between the two mesh types, suitable for delineating sudden topological changes such as overbank stations).

12. Click the **Save** button to save your project.

2 selected Bounding - 2D	
Attributes	
Style	Directional
Angle (deg)	0
Resolution (m)	3
Sampling Factor	1
Dist. Tolerance (r	0
Elev. Tolerance (	0
Roughness	0.04
Seepage Rate (m	0
Edge	YES
Other	
LENGTH	0
WIDTH	0
Shape	
Count	2
Total Points	35
Avg. Points	17.5
Total Parts	2
Avg. Parts	1
Total Area (m²)	1668.92613
Avg. Area (m²)	834.46307
Total Area (ha)	0.1669
Avg. Area (ha)	0.0834

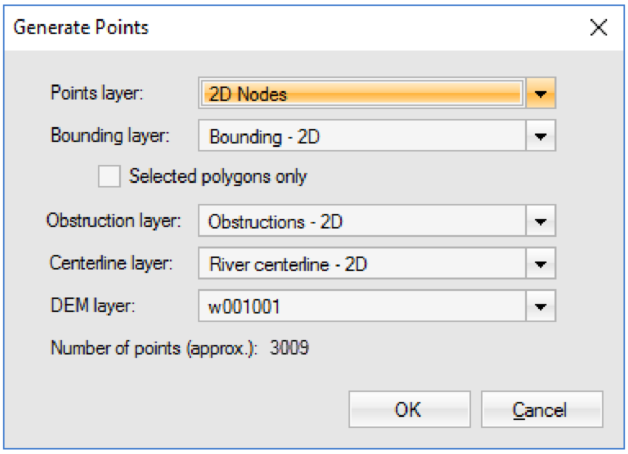
Now we can generate center points that will be used to create the mesh. These points are created on the **2D Nodes** layer.

13. Click on the **Tools**  button in the **Map panel**.

14. Select **2D modeling** from the list of tool categories.

15. Select the **Generate points** tool from the 2D modeling list.

16. Ensure that the **Points (2D Nodes)**, **Bounding**, **Obstruction** and **Center line** layers are correctly identified. Obstructions for this exercise include buildings as well as several walls in the area. The estimated number of points should appear at the bottom of the **Generate points** window.

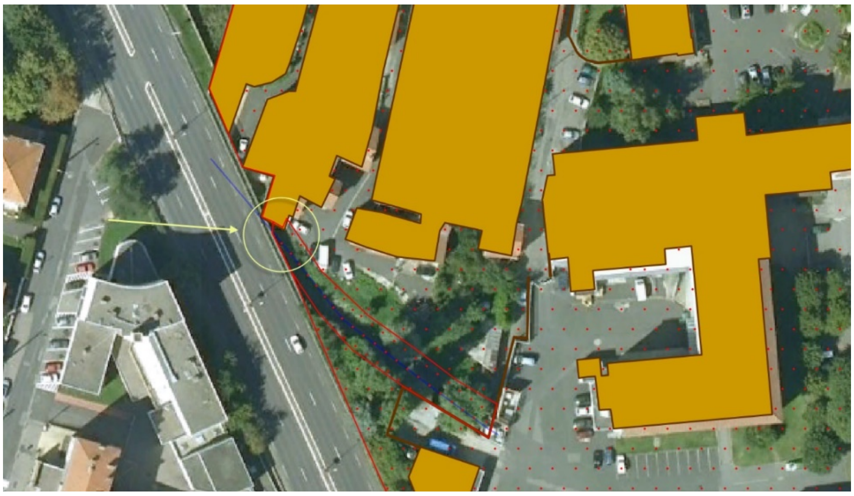


17. Click **OK** to generate the points. If you are unable to see the points after creating them toggle on the **2D Nodes** layer in the **Layers panel**.

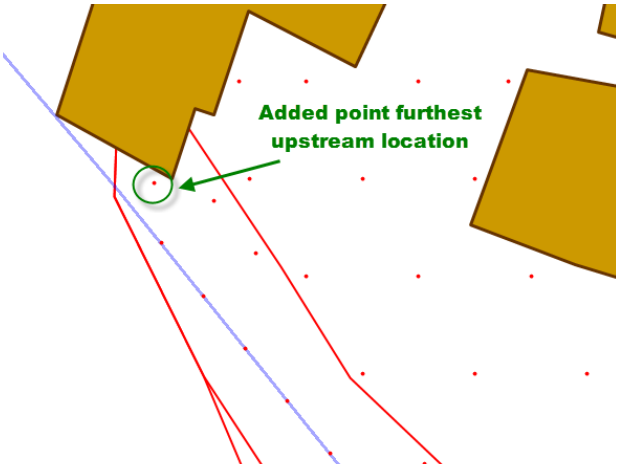
6.3 Create diverging/converging locations in stream

In this example, the two sections identified as being a directional river segment are connected with a culvert running under the parking lot. Also, inflow to the model will occur at the upstream end of the westerly directional river segment, and a downstream culvert will drain water from the east side of the easterly directional river segment. In order to connect the sectional meshes to upstream and downstream culverts, we need to create a 2D node point to force the mesh to converge.

Zoom into the upstream location of the westerly directional bounding polygon (NW boundary of the project site), as shown in the screenshot.

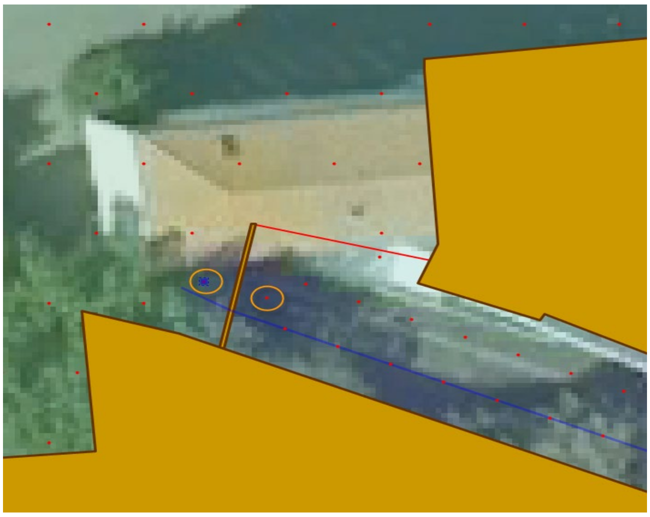


1. Select the **2D Nodes** layer in the **Layers panel**.
2. Unlock the layer (if locked) by right-clicking on the layer and selecting **Unlock**.
3. Click on the **Add** button in the **Map panel's** toolbar.
4. Put a point in the furthest upstream location in the polygon by clicking on the location in the map (see illustration below for guidance).
5. Click on the **Select**  button to get out of **Add** mode.

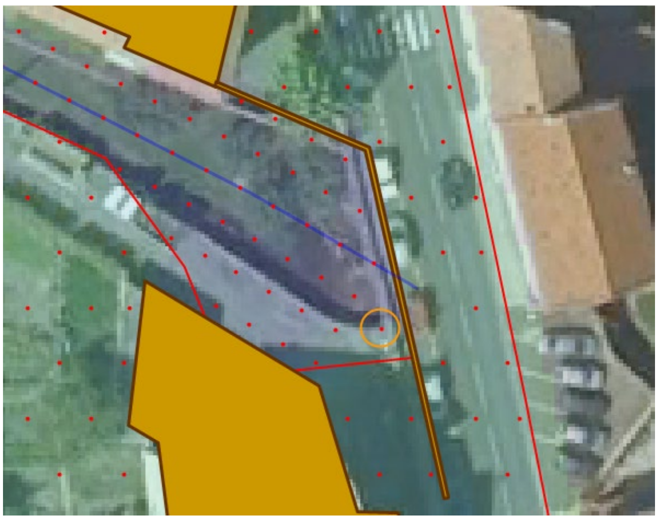


6. Zoom into the downstream location of the same directional bounding polygon.
7. Ensure there is a single point in the furthest downstream location; you may need to move or delete points. To move a point double click on the point and drag it to the new location.

Move to the upstream location of the second (easterly) directional bounding polygon and remove points so that there is a single point in the upstream location.



Finally move to the downstream location in the second directional polygon and add or remove points to make a single point.



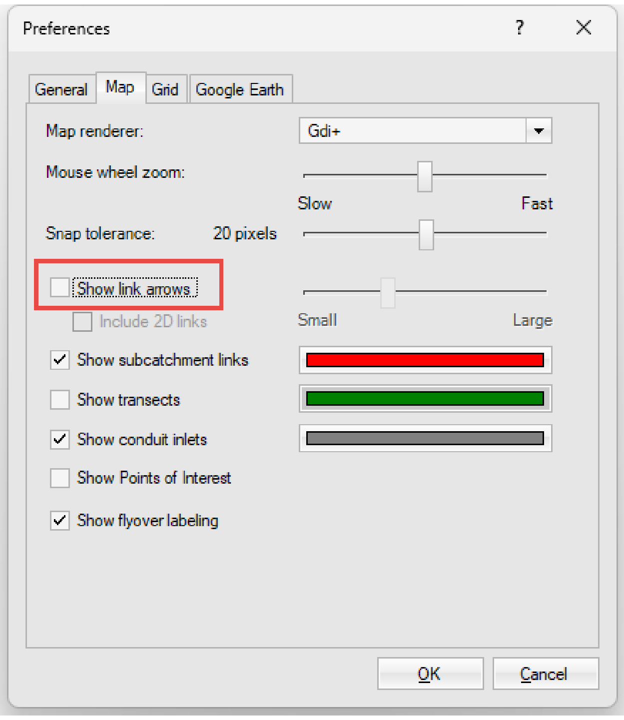
6.4 Create Mesh

Now that we have established the points for the 2D cells, we can create the 2D mesh. When generating the mesh the model will sample elevations from the DEM layer at the locations of the 2D nodes. The first thing we will do is turn off the option to **Show link arrows**, this will keep the mesh from not looking too cluttered.

1. Turn off the **Show link arrows** option in the **Preferences** dialog.

2. Click on the **Menu** button in the **Map panel** and select **Preferences**.

3. Under the **Map** tab toggle off **Show link arrows**.



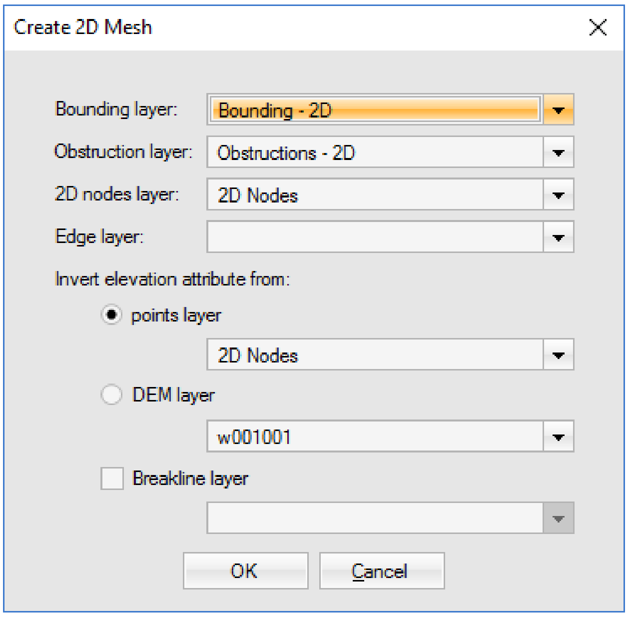
We will now create our 2D mesh.

4. Click on the **Tools** button, in the **Map panel**, and select **2D modeling.**

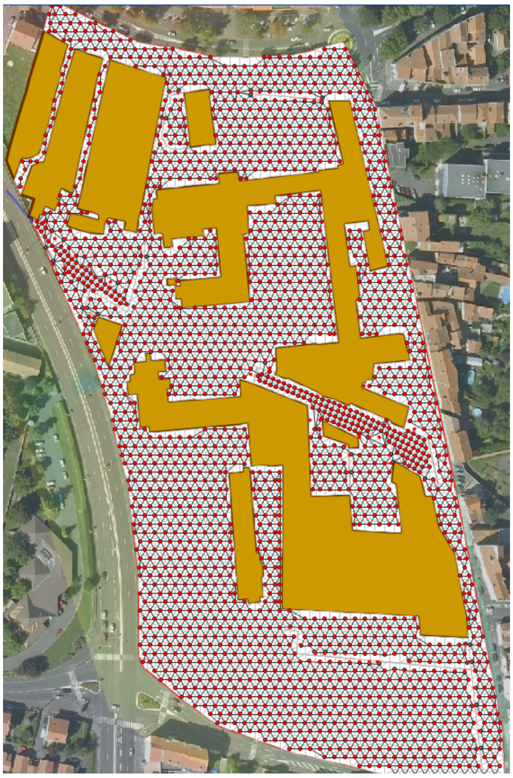
5. Select **Create Mesh** from the 2D modeling sub menu.

6. Ensure that the **Bounding, Obstruction,** and **2D Nodes** layers are defined in the **Create 2D mesh** window.

7. Click **OK** to generate the mesh.



PCSWMM will now automatically create the 2D mesh. This should only take a couple of seconds.



Once the mesh generation is completed a report will appear. You can close the report once you have read it. After mesh generation, the 2D mesh is rendered to display the cells as well as the links and nodes of the mesh.

8. Click on the **Close** button to close the **Operation Report**.

9. In the **Layers Panel** uncheck the **2D Nodes** layer to hide it (it is no longer needed).

10. Click on the **Save** button to save the project.

11. Link arrows can be turned back to see the direction of 2D links using the **Preferences** dialog.

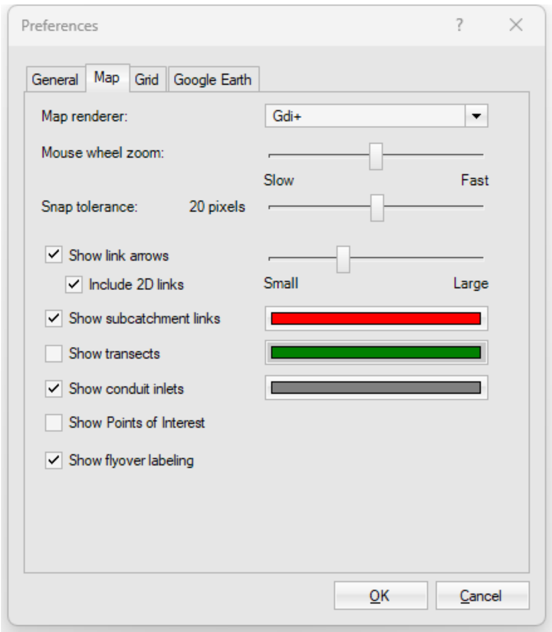
12. Turn on the **Show link arrows** option for 1D and 2D links in the **Preferences** dialog.

13. Click **Menu** > **Preferences**.

14. In the **Map** tab, check the option to **Show link arrows** if not checked already.

15. Toggle on the option to **Include 2D links**.

16. Click **OK**.



Zoom into the mesh to see the link arrows.

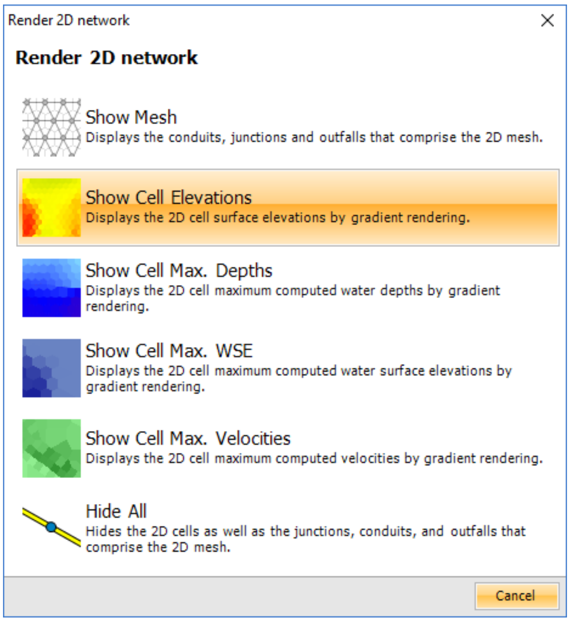
Repeat step 9, this time turning off the arrows to avoid crowding the screen.

We will now check the elevations assigned to the 2D cells through a map rendering. The **Render 2D network** tool allows you to quickly set the layer properties for a number of 2D layers to one of the default renderings.

17. Click on the **Tools**  button and select **2D modeling**.

18. Select **Render 2D network** from the 2D modeling sub menu.

19. Select **Show Cell Elevations.** Note how the color changes with the topography of the project area.



Let's return to a map rendering that displays the mesh.

20. Click on the **Tools** button and select **2D modeling**.
21. Select **Render 2D network** from the 2D modeling sub menu.
22. Select **Show Mesh**.
23. Now uncheck the **2D cells** layer in the **Layers panel** to hide it.

Zoom into the smaller bounding polygons and notice how the directional mesh is different from the hexagonal mesh shape in the larger polygon.

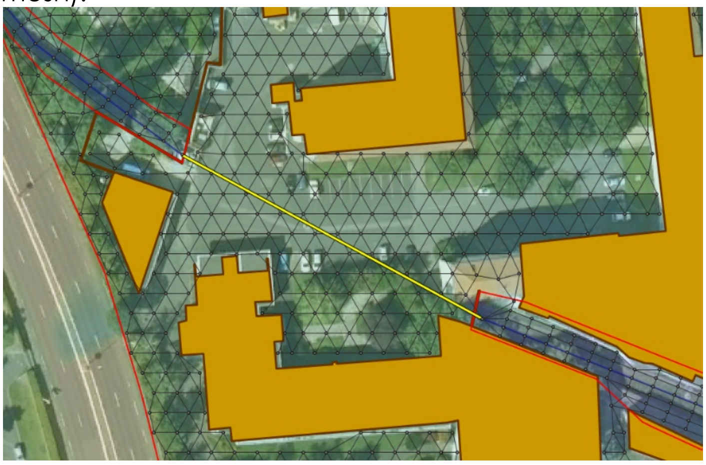
Note: that if you want to improve the layout of the mesh, you can use an iterative approach of adding/moving/deleting points in the **2D nodes** layer and regenerate the mesh with the **Create Mesh** tool. The Create Mesh tool will replace existing 2D entities in the model (2D cells, nodes and conduits) and reconnect the mesh to any existing 1D entities or 1D-2D links.

6.5 Add 1D culvert

We still need to add the culvert that links the two directional meshes under the hospital parking lot.

1. Select the **Conduits** layer from the **Layers panel**.
2. Click on the **Add** button.
3. Click on a mesh junction closest to the border of the upstream boundary and then on a junction close to the border of the downstream boundary (i.e. from northwest to southeast) to connect the upstream river segment to the downstream river segment (see the yellow conduit in the figure below).

4. Click the **No** button if asked if you would like to turn auto-length on. For 2D models, we do not want PCSWMM to determine conduits lengths from the map (the 2D mesh generator automatically sets appropriate conduit lengths for the 2D mesh).



Now we need to define the culvert attributes. For this model, the conduit just to link the two river segments is not one of the culvert shapes so the culvert shape will be defined using the custom culvert editor.

5. Click on the **Select** button to get out of **Add** mode.

6. Change the culvert length to **77** m and roughness to **0.033**.

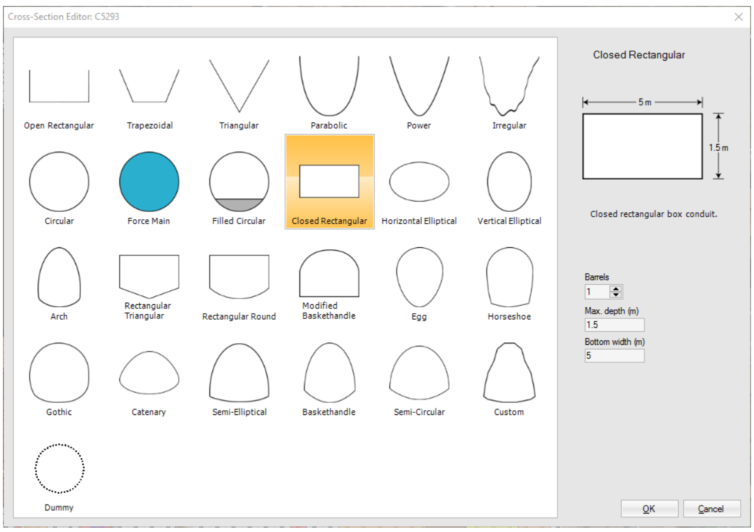
7. Click in the **Cross-section** attribute and click on the **Ellipsis** ... button.

8. Select **Closed Rectangular** from the cross-section editor.

9. Change the **Max. depth (m)** to **1.5** m.

10. Change the **Bottom width (m)** to **5** m.

11. Click the **OK** button.



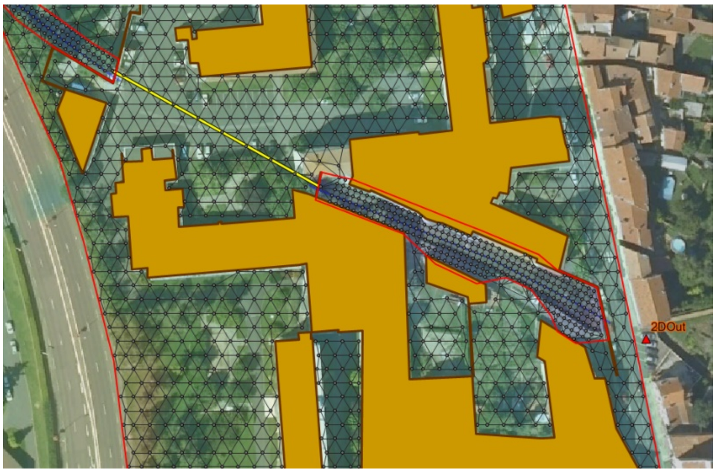
6.6 Create an outfall

An outfall is assigned at the downstream end of the river to define the model's downstream boundary condition. It will be placed at the exit of a culvert draining the downstream directional river segment.

1. Select the **Outfalls** layer from the **Layers panel**. The layer will appear grayed out as there are currently no Outfalls in the model.

2. Click on the **Add** button.

3. Click on the eastern edge of the project area downstream of the second directional river segment to add an **Outfall** (see screenshot for location).



4. Change the name of the **Outfall** to **2DOut**.

5. In the **Attributes panel** set the **Invert El. (m)** to **338.15** m.

6. In the **Attributes panel** set the **Rim Elev. (m)** to **348.15** m.

7. Change the **Type** to **FREE**.

Outfall: 2DOut

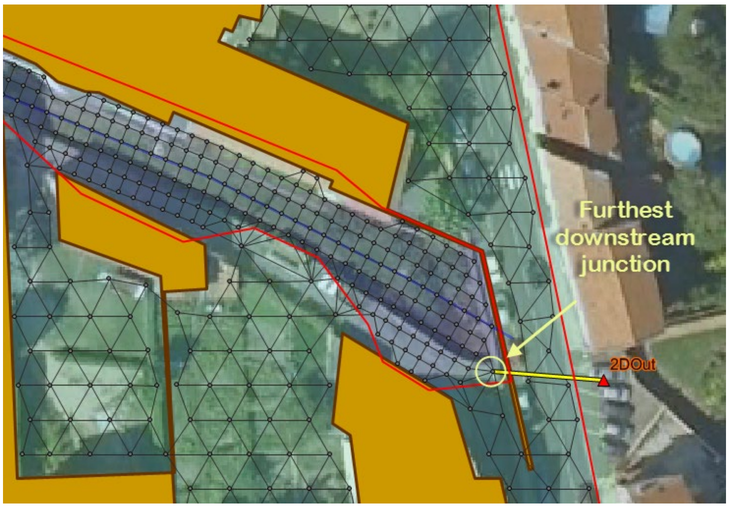
Attributes	
Name	2DOut
X-Coordinate	1708839.87835946
Y-Coordinate	5187615.55973739
Description	
Tag	
Inflows	NO
Treatment	NO
Invert El. (m)	338.15
Rim Elev. (m)	348.15
Tide Gate	NO
Route To	
Type	FREE

8. Select **Conduits** layer from the **Layers panel**.

9. Click on the **Add** button.

10. Click on the furthest downstream junction in the easterly directional river segment and then click on the **Outfall** to add a conduit between the two nodes.

11. Click the **No** button if asked if you would like to turn auto-length on.



12. In the **Attributes panel**, change the conduit's length to **12** m.

13. Set the **Roughness** to be **0.033**.

14. Change the conduit cross-section to a **Closed Rectangular** shape with a **Max Depth** of 1.5 m and **Bottom width** of 5.

15. Click on the **Cross-Section** attribute box and select the **Ellipsis** ... button.

16. Select the **Closed Rectangular** culvert shape.

17. Set the **Max depth** to **1.5** m.

18. Set the **Bottom width** to **5** m.

19. Click **OK** to close the **Cross-sectional Editor**.

20. Click on the **Select** button to exit the **Add** mode.

6.7 Assign an inflow time series

1. Create a new object in the **Time Series Editor** called **2D_Inflow**.

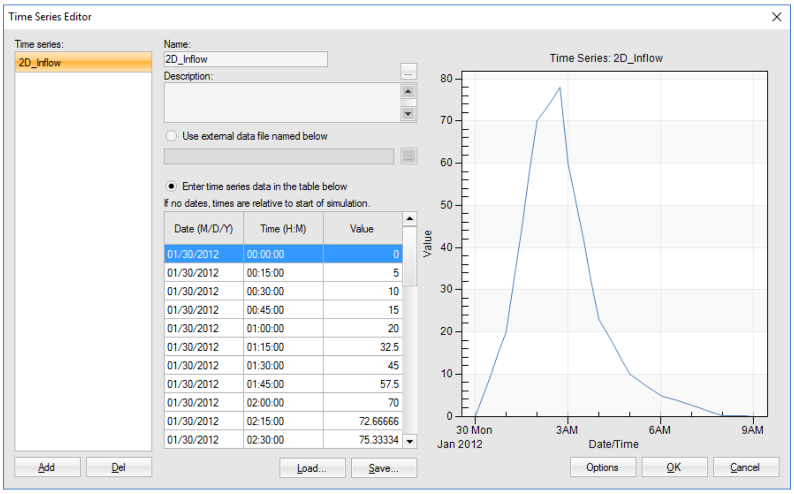
2. Click on the downward arrow in the **Project panel** and select the **Time Series** item. It will appear grayed out as there are currently no time series defined.

3. Click on the **Add** button.

   Name the new time series **2D_Inflow**.

4. Click on the **Load** button and browse to **PCSWMM Exercises\K001\Initial\Time series** and select **2D Inflow Time series.dat** and click **Open**. The design hydrograph will be displayed in the **Time Series Editor**.

5. Click **OK** to close the **Time Series Editor**.

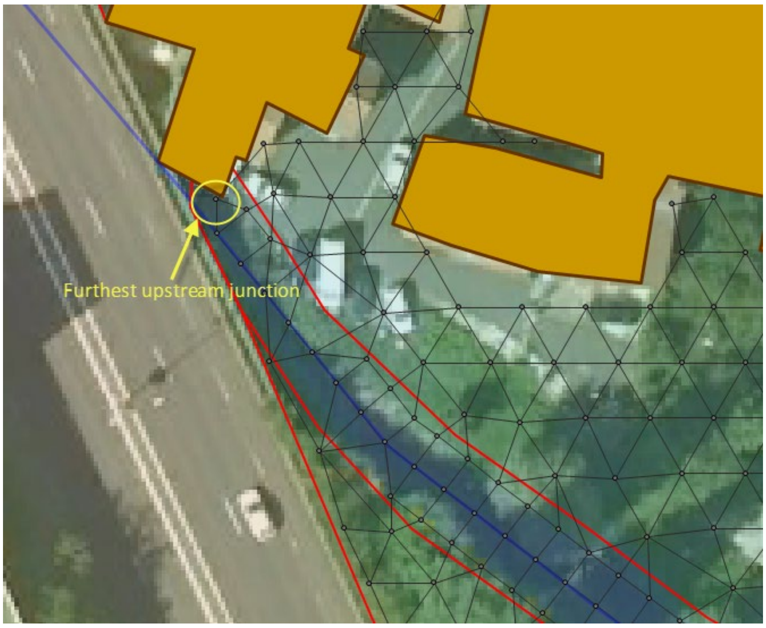


Note: there are multiple ways to assign an inflow boundary condition to a 2D model: 1. Assign a water surface elevation time series to one or more outfall(s) and connect the outfall(s) to the 2D mesh using a conduit. The downstream boundary layer and associated tool can be used to efficiently do this for a non-point boundary condition. 2. Apportion a flow time series to one or more 2D junctions directly in the junction's Time series attribute. In this case, the inflow can be apportioned to multiple nodes by setting the node's time series Factor attribute to 1/n, where n is the number of nodes the inflow is to be apportioned.

We now want to assign the inflow to the furthest upstream junction in the river boundary polygons. In this case, we can assign the entire inflow to a single junction in the 2D mesh.

6. Click on the **Junctions** layer from the **Layers panel**.

7. Select the furthest upstream junction in the upper river boundary polygon (as shown in the screenshot).

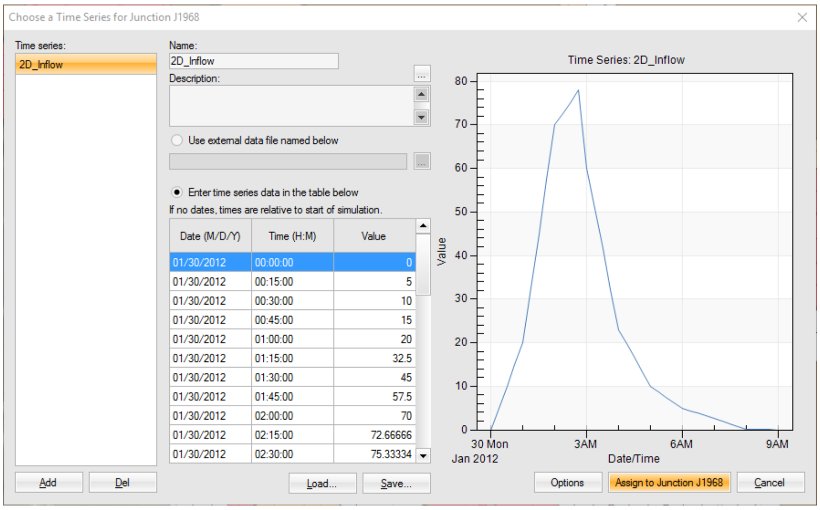


8. Assign the **2D_Inflow** time series to the selected junction as an inflow.

9. In the **Attributes panel**, click in the **Time Series** box and click on the **Ellipis**  button.

10. The **Time Series** editor will open, allowing you to specify the time series.

11. Select the desired inflow time series and click on the **Assign to Junction** button to assign it to the selected junction.



6.8 Run a simulation

We won't assign any initial conditions: the model will assume to start as dry bed. SWMM5 is able to transition a 2D mesh from a dry bed to a wet one without model stability issues. Let's set the simulation options and run the model.

1. Set the simulation period to match **2D_inflow** and change the simulation duration to 12 hours.

2. Click on **Simulation Options** from the **Project panel.**

3. Click on the **Dates** tab and select **2D_Inflow** under the **Set simulation period from time series** drop-down menu. This allows us to quickly set the simulation period to match the input time series.

4. Set the **Duration** to **12** (h).

5. Change the reporting time step to 30 seconds, and the routing time step to 0.5 seconds.

6. Still in the **Simulation Options**, click on the **Time Steps** tab and change the **Reporting** time step to be **00:00:30** (i.e. 30 seconds)

7. Change the **Routing** time step to be **0.5** s.

8. Set PCSWMM to ignore the inertial terms and use the maximum number of threads available.

9. Still in the **Simulation Options**, click on the **Dynamic Wave Tab.** Under **Inertial terms** select **Ignore.**

10. Under **Number of Threads** choose the maximum number available.

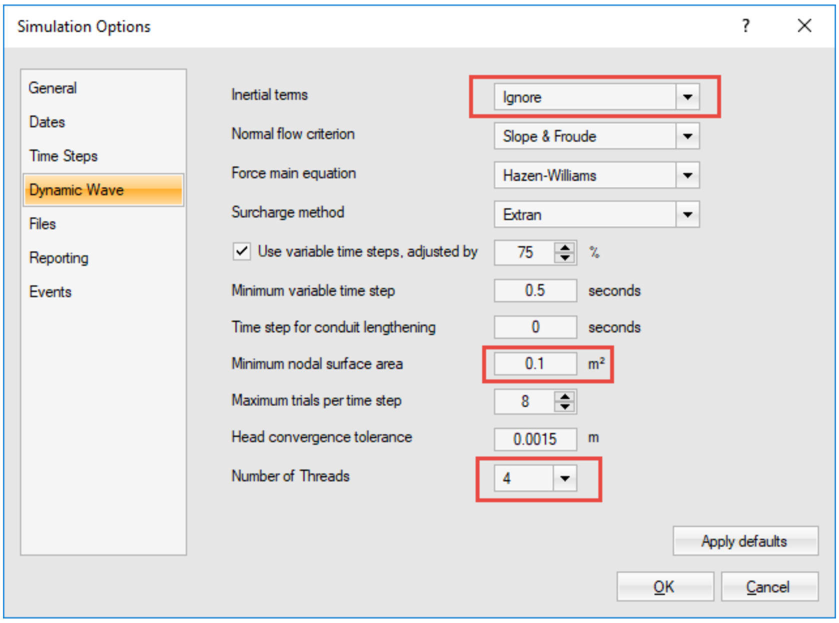
11. Click **OK** to close the **Simulation Options** editor.

12. Click on the **Run** button to run a simulation. It should take between 4 - 8 minutes to run, depending on the computational speed of your computer.

13. You should receive the message "**We recommend a minimum surface area of 0.1 m² or less for 2D projects**", select the option to "**Change the minimum surface area**".

14. When the **Simulation Options** window opens, change the **Minimum nodal surface area** to **0.1** (Square Meters).

15. Click **OK** to save your changes



16. Click on the **Run** button to run a simulation.

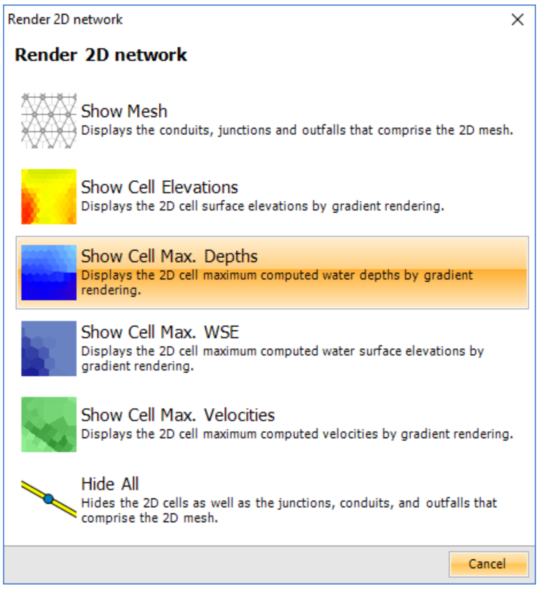
You may receive a message saying that we recommend not reporting the input summary for 2D projects, click on the **Continue anyway** button.

6.9 Render model to show maximum water surface elevation

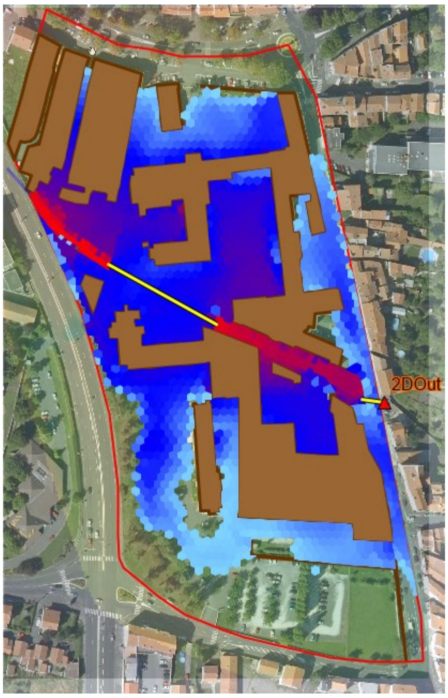
1. Click on the **Tools** button and select **2D modeling**.

2. Select **Render 2D network** from the 2D modeling sub menu.

3. Select **Show Cell Max. Depths.**



Your model should look something like this.



6.10 Add downstream boundary condition

You may have noticed when the model was rendered to the maximum water surface elevation that water was starting to buildup on the boundary of the 2D model. This is because the only way water can leave the model is through the outfall. To prevent this from happening we can make a downstream boundary condition allowing water to leave the boundary area through outfalls automatically generated using the **Create Boundary Outfalls** tool. This tool takes a user-drawn line and creates outfalls at the conduits crossing the line. In this example we have already created the boundary outfall line.

1. Add a 2D downstream layer called **DS boundary condition - 2D.SHP** from the **\Initial\GIS layers** folder.

2. Click on **File** and select **2D** from the options on the left side of the screen.

3. Click on the **Open** button beside the **Downstream layer** drop-down menu and navigate to **PCSWMM Exercises\K001\Initial\GIS layers** and select **DS boundary condition - 2D.SHP**.

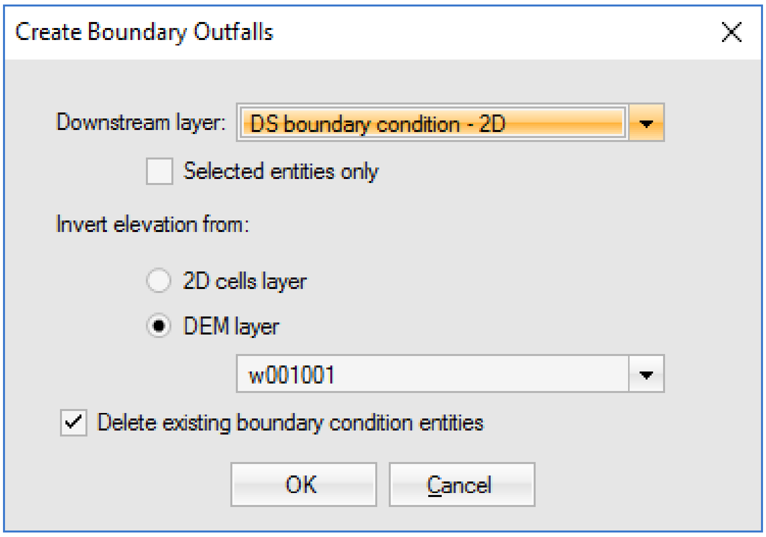
4. Click **Open** and then select **OK**.

You will notice that there are 6 green lines in the downstream boundary layer. These lines are indicating the locations where we want to add our downstream boundary condition.

5. Click on the **Tools** button and select **2D modeling** from the list of tool options.

6. Select the **Create Boundary Outfalls** from the list of tools.

7. Select **DS boundary condition - 2D** from the **Downstream layer** drop-down menu.



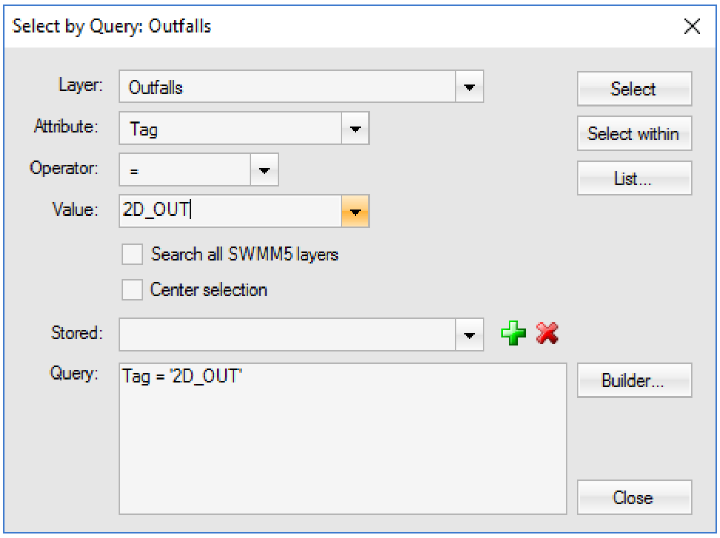
8. Click on the **OK** button.

A message will appear saying the number of outfalls and connected conduits were created. The 2D outfalls are set to be type = Normal meaning they are assigned the sampled ground elevation. To use Normal outfalls in a model it is important that the link connecting the model to the outfall has a positive slope. To ensure this in our model we will adjust the slopes of the conduits with invert slopes.

9. Select all outfalls with the tag **2D_Out**.

10. Click on the **Find** button and choose **Select by Query**.

11. Change the **Layer** to be **Outfalls**, the **Attribute** to be **Tag**, the **Operator** to be **=**, and the **Value** to be **2D_OUT**.



12. Click on the **Select** button then the **Close** button.

13. Click on the **Find** button and choose **Select connected** > **Immediate Upstream**.

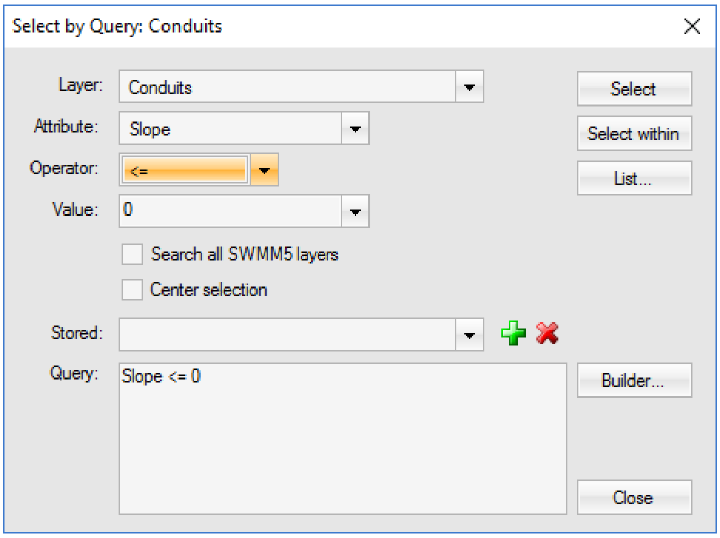
14. Click on the **Conduits layer**, the **Attribute panel** will indicate that you have multiple conduits selected.

Now we want to identify the links that have negative slopes. To do this:

15. Click on the **Find** button and choose **Select by Query**.

16. Set the **Layer** to **Conduits, Attribute** to **Slope**, **Operator** to **<=**, and **Value** to **0**.

17. Click **Select within** to limit the search to include only the selected conduits and click **Close**.

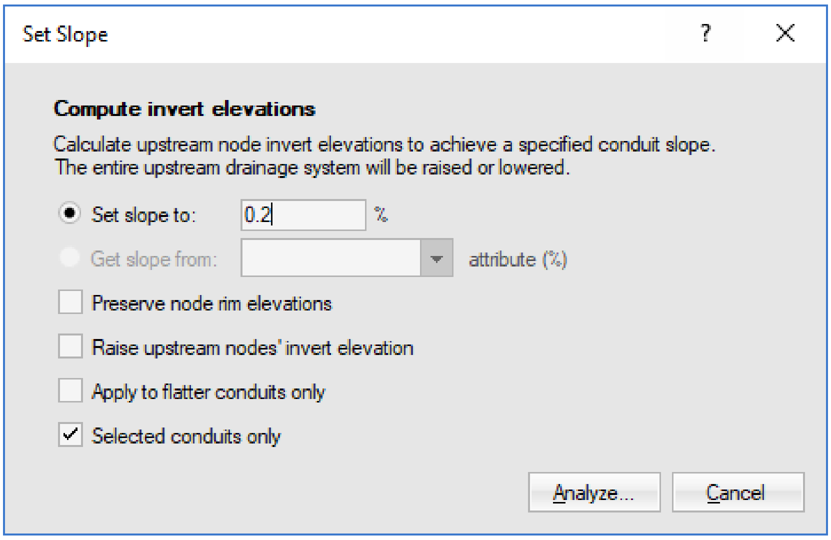


We now want to fix the inverse slopes using the set slope tool.

18. With the conduits still selected, click on the **Tools** button.

19. In the **Conduits** section, click on the **Set slope** tool located at the bottom of the list.

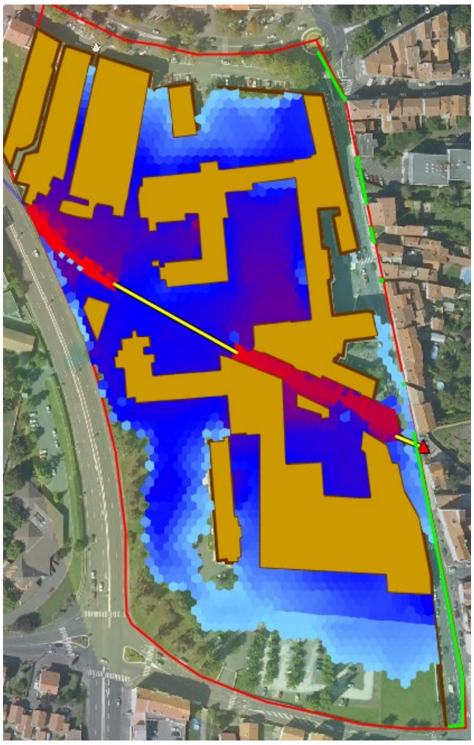
20. Set the slope to **0.2%**, uncheck **Preserve node rim elevations**, **Raise upstream nodes' invert elevation** and **Apply to flatter conduits only**, leave the **Selected conduits only** checked.



21. Click on the **Analyze...** button.

22. Click **Apply** and **Close**.

You can now re-run your model and render the 2D cells on Maximum depth and see how the flooding has changed as a result of the added downstream boundary layer.



The following chapter will review some of the post-processing and analysis tools available in PCSWMM for reviewing the results of a 2D simulation.

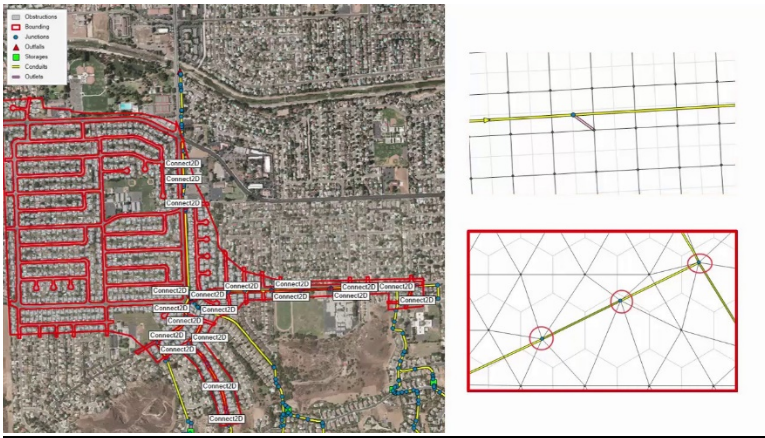
6.11 Troubleshooting

If you added a downstream layer to your model and after you ran and rendered the cells everything shows in red: The downstream layer uses free outfalls which does not permit the upstream conduit to have a slope less than or equal to zero. If you are having this issue you likely used the less than < symbol and not the <= symbol when selecting the upstream conduits.

If you have a high continuity error it is likely caused by the contraction and expansion nodes not being defined correctly.

7. Dual-drainage integrated 1D-2D flood modeling with obstructions

The following example illustrates a case study in an urban area in Southern California. This model represents a 2-square mile watershed area, and consists of 6 miles of storm drains, 6 detention basins, 2000 houses and structures and 20 miles of streets and roadways. In this exercise we will be starting with a 1D model representing the underground stormwater pipes, from there we will be adding a 2D overland mesh to represent the street and gutter flow.



7.1 Open existing 1D model

We will begin by opening the 1D model.

1. Unpackage **1D Simi Valley model.pcz** from the **PCSWMM Exercises/K003/Initial** folder.

2. Open **PCSWMM** and click on the **Open**  button.

3. Browse to **PCSWMM Exercises/K003/Initial** and select **1D Simi Valley model.pcz** and click **Open**.

4. Unpackage the model to **PCSWMM Exercises/K003/Initial/Model**

5. Click **OK**.

6. Load the DEM, **w001001.adf,** from the **PCSWMM Exercises/K003/Initial/GIS layers/erringer_1ft** folder.

7. Click on the **Open**  button and then the **Open**  button.

8. Navigate to **PCSWMM Exercises/K003/Initial/GIS layers/erringer_1ft** and select **w001001.adf**. and click **Open**.

9. In the **Layers panel**, click on the DEM layer (w001001).

10. Click on the **Render** button and look at the coordinate system. If you click on other layers, you will notice it is a different coordinate system.

When a DEM is used in a 2D model, the coordinate system must be the same as other layers. DEMs and other layers can be projected directly in PCSWMM.

11. Reproject the DEM from its current system to **NAD83 California zone 5ft US.** Rename it "DEM".

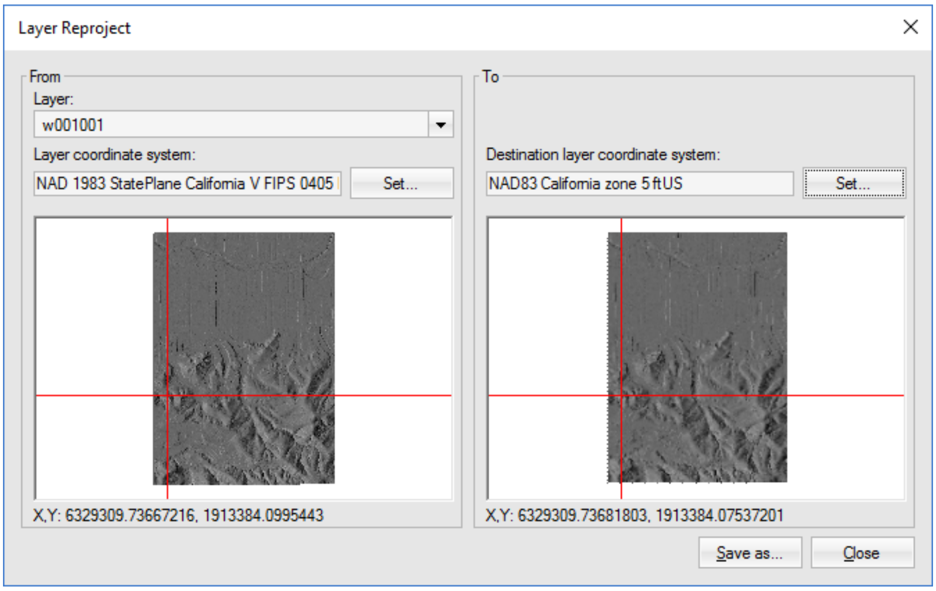
12. Click on the **Alter** button and choose **Reproject**. The current coordinate system will already be set in the **From** section.

13. In the **To** section, click on the **Set** button.

14. In the **Coordinate system editor**, click on the **Projected system** section.

15. Type in NAD83 California.

16. From the results choose **NAD83 California zone 5ft US** and click **OK**.



17. Click on the **Save as** button and name the new file "DEM."

18. Click **Close**. The new DEM will be loaded in the **Layers panel**.

19. In the Layers panel, right-click on the w001001 file and then **Close**.

20. Click the **Save** button to save your project.

7.2 Create 2D layers

When developing a 2D model in PCSWMM it is important to define the layers that will be used. In this exercise we will be using five 2D layers: bounding, 2D nodes, obstruction, downstream, and a DEM layer. We will start by creating and defining the 2D layers we will need for the model.

1. In the **File panel**, select the**2D** button near the bottom of the list.

2. Toggle on the **Enable 2D modeling** box.

3. Click on the **Open** button beside the **Bounding layer** drop-down menu and browse to **PCSWMM Exercises/K003/Initial/GIS layers**.

4. Select **Boundary.SHP** and click on the **Open** button.

5. Click on the **New** button beside the **2D nodes layer** drop-down menu.

6. Navigate to **PCSWMM Exercises/K003/Initial/GIS layers** and click on the **Save** button to save the layer with the default name (i.e. 2D Nodes).

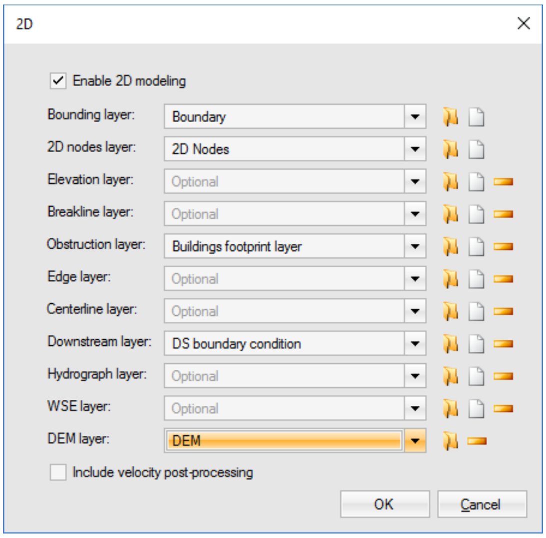
7. Click on the **Open** button beside the **Obstructions layer** drop-down menu and navigate to **PCSWMM Exercises/K003/Initial/GIS layers** and select **Buildings footprint layer.SHP** and click on the **Open** button.

8. Click on the **New** button beside the **Downstream layer** drop-down menu.

9. Navigate to **PCSWMM Exercises/K003/Initial/GIS layers** and click on the **Save** button to save the layer with the default name.

10. In the **DEM layer** drop-down menu, select the new **DEM** file that was just created.

11. Click **Yes** if asked if you want to set the map coordinate system to NAD83 California zone 5ft US.



12. Click on the **OK** button to save your 2D settings and close the 2D setup dialog.

13. Click the **Save** button to save your project.

7.3 Create second bounding polygon based on roads layer

In this example, a different Manning's roughness value will be assigned for the roads compared to the areas around the buildings. This can be done with two bounding polygons, the first being the overall study area and the second just the roadways. We have already defined the overall bounding polygon however we still need to create a bounding polygon representing the roads. To do this we will use an existing roads layer and use the **Split** tool to crop the roads layer to the overall boundary layer.

1. In the **Layers panel**, uncheck the **DEM** layer to improve the responsiveness of the **Map panel**.

2. Load the **Roads layer.shp** file from the **PCSWMM Exercises/K003/Initial/GIS layers** folder.

3. In the **Map panel** click on the **Open layer** button.

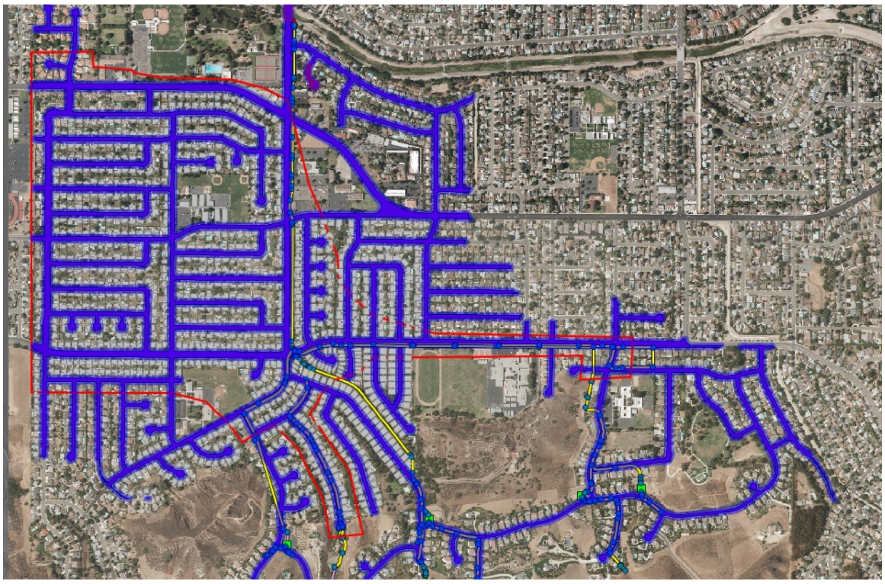
4. In the **Layer browser** click on the **Open** button.

5. Navigate to **PCSWMM Exercises/K003/Initial/GIS layers** and open the **Roads layer.SHP**.

6. Reproject "**Roads layer**" from its current system to **NAD83 California zone 5ft US** and save it as "**Roads**". **Close** the previous "Roads layer" file.

7. Right click on the **Roads layer** and click on the **Unlock option** to unlock the layer.

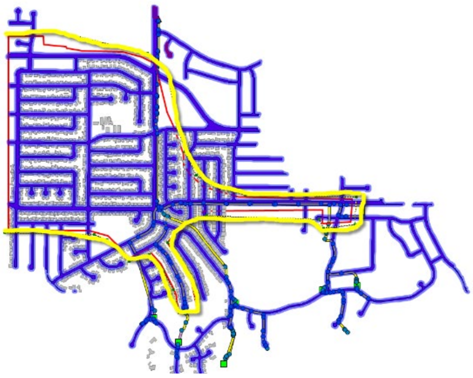
8. Click on the **Roads layer** polygon and press **Ctrl+A** to select all the roads.



9. Click on the **Edit** button in the **Map panel**. Select **Split** .

10. In split mode, choose **After active point** , and then click around the bounding polygon drawing a cut line just outside of the boundary. The following screenshot highlights the pathway to be followed with the split tool in yellow.

11. Click on the **Split** button in the sub-toolbar to split the selected roads layer polygon. It may take a few seconds to complete the split operation

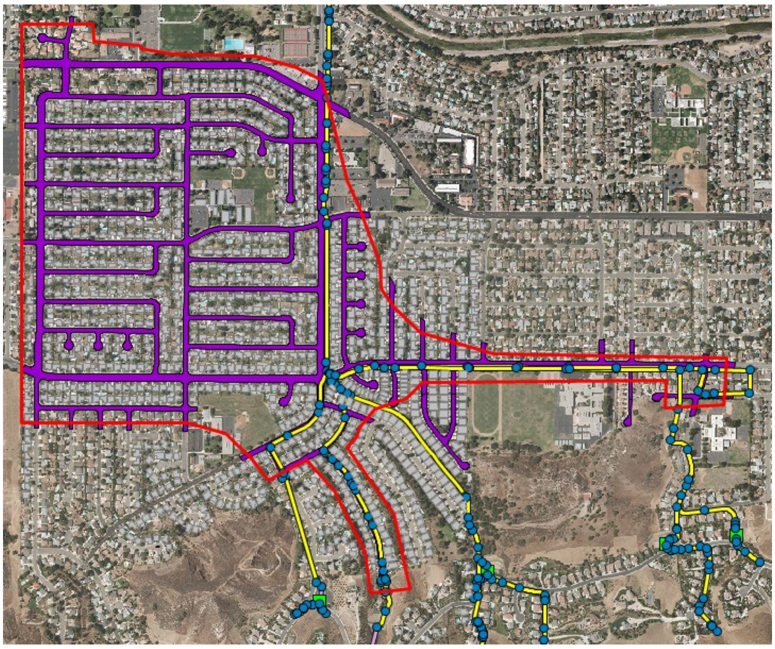


12. Delete portions of the Roads layer outside of the split area.

13. Click on the area of the roadway layer inside the overall boundary.

14. Click on the **Find** button and choose **Invert selection**.

15. Click on the **Delete** button to remove them from the roads layer.



Now we are going to add the edited roads polygon as a bounding polygon.

1. Copy and paste the **Roads layer** shape into the **Boundary layer**.

2. Click on the **Roads layer** in the **Layers** manager and click on the remaining roads polygon in the map to select it.

3. Click on the **Copy** button.

4. Select the **Boundary** layer from the list of layers**.**

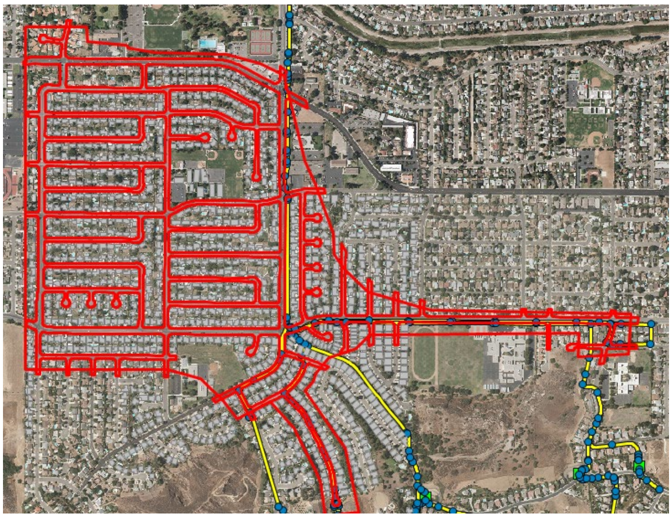
5. Click on the **Lock** button and select to **Unlock** the layer.

6. Click on the **Paste** button to paste the Roads layer into the **Boundary** layer.

7. Finally, let's close the **Roads** layer.

8. In the **Layer** manager click on the **Roads** layer and click on the **Close** button in the Map panel toolbar to close the layer.

9. Click **No** when asked if you want to save the unsaved work. This way you will still have the original roads layer intact.



7.4 Define bounding polygon attributes

We will now assign the resolution and Manning's roughness coefficients for the bounding polygons.

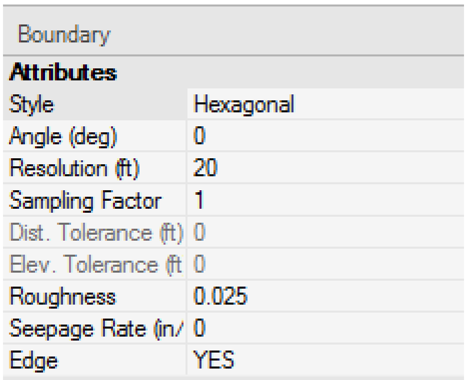
1. Click on the **Boundary** layer in the **Layers panel** and select the overall boundary polygon in the map by clicking on it.

2. Leave the **Style** set as **Hexagonal**, set the **Resolution** to **50** ft and change the **Roughness** to be **0.085**. Leave the **Edge** attribute set to **No**.

Boundary	
Attributes	
Style	Hexagonal
Angle (deg)	0
Resolution (ft)	50
Sampling Factor	1
Dist. Tolerance (ft)	0
Elev. Tolerance (ft	0
Roughness	0.085
Seepage Rate (in/	0.5
Edge	No

1. Select the boundary polygon marking the roads in the **Boundary** layer.

2. Leave the **Style** set as **Hexagonal**, set the **Resolution** to **20** ft and change the **Roughness** to be **0.025**. Change the **Edge** attribute set to **Yes**, this will cut the 2D cells cleanly at the curb.



7.5 Create points in 2D nodes layer

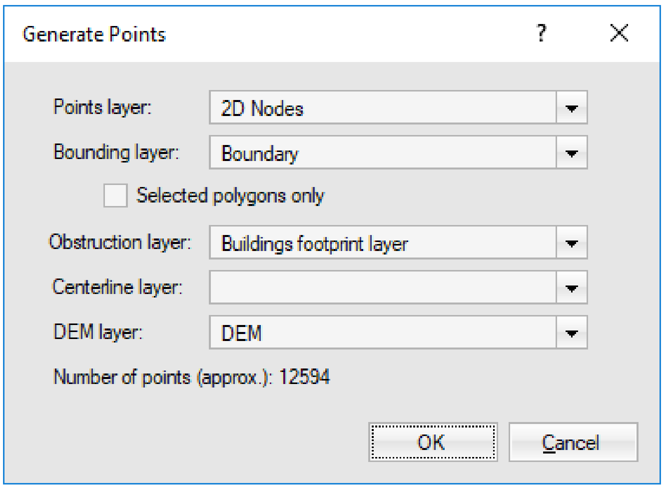
Depending on the number of obstructions and the complexity of your bounding polygons the 2D nodes may take some time to generate. For this particular exercise there are ~1000 buildings in the obstruction layer and the roadway bounding polygon adds complexity, so for the sake of time we will be providing the points for the 2D nodes layer.

1. In the **Map panel** click on the **Tools** button.

2. Click on the **2D modeling** section from the list of tool categories.

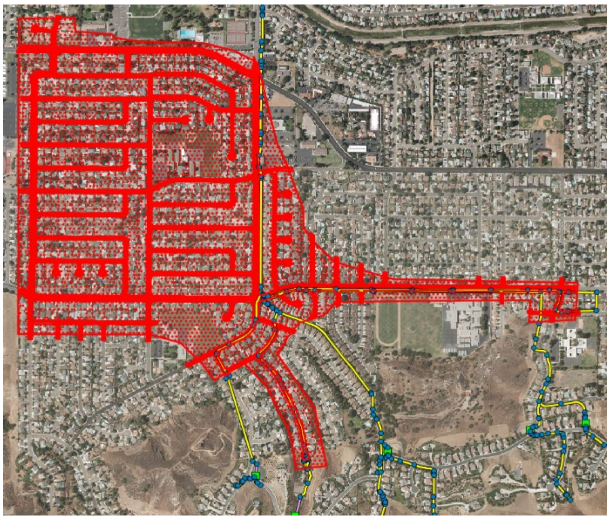
3. Select **Generate points** from the list of 2D tools.

4. Check that the layers have been correctly assigned for 2D node creation as shown in the screenshot. The number of points to be created will be displayed and should be approximately 12,000.



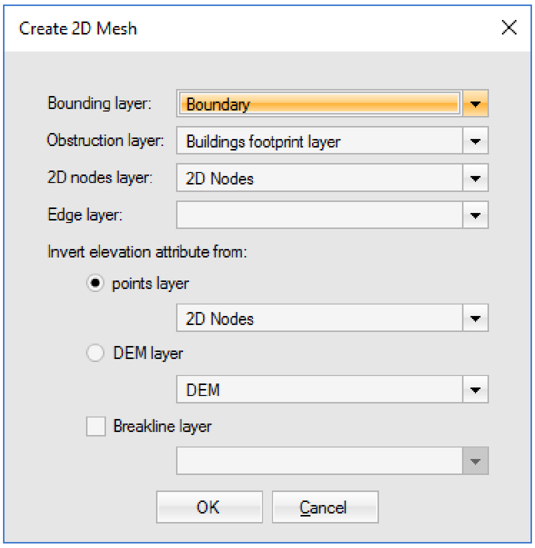
5. Click **OK**. It may take a few minutes for the 2D Nodes to be generated.

6. The new layer will automatically be added to the **Layers panel** and the **Map panel.** There should now be approximately 12,000 points in the 2D Nodes layer within the 2D model domain identified by the Bounding layers.



7.6 Generate 2D overland mesh

1. Click on the **Tools** button and select **2D modeling** from the list of tool categories.
2. Select **Create mesh** from the list of 2D tools.
3. Ensure the layers have been correctly assigned (see screenshot) and click on the **OK** button. It may take around 10 minutes to generate the mesh.



7.7 Identify 1D nodes to be connected to the 2D overland mesh

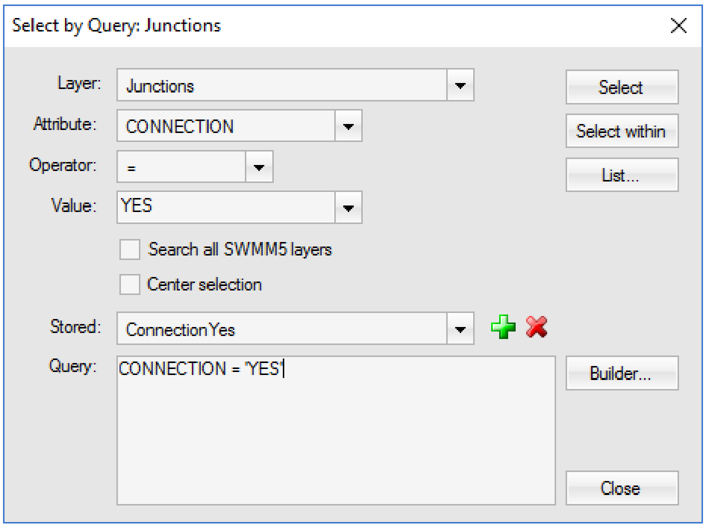
The next step is to connect the 1D portion of the model to the 2D overland mesh to connect the major and minor systems of our dual drainage model. To do this we need to first identify where the 1D connection points are and mark them by adding a **Connect2D** tag. Normally you would do this manually, however for this exercise we have identified the connection points using a user defined attribute.

1. Uncheck the **2D Nodes** layer in the **Layer panel**.

2. Select all junctions with the attribute **CONNECTION** set to **YES**.

3. In the **Map panel** click on the **Find** button and choose **Select by Query**.

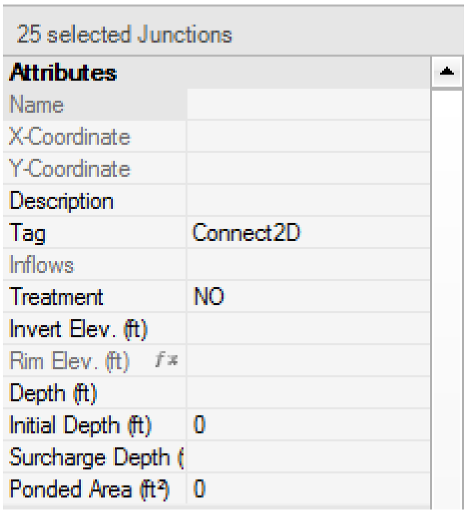
4. Set the **Layer** to be **Junctions**, the **Attribute** to be **CONNECTION**, the **Operator** to be **=** and the **Value** to be **YES**.



5. Click on the **Select** button, there should be 24 selected junctions.

6. Click on the **Close** button to close the **Select by Query** tool.

7. With the junctions still selected, in the **Attributes panel** type **Connect2D** in the **Tag** attribute.



Now let's connect these junctions to the 2D mesh:

1. Click on the **Tools** button, select **2D modeling** from the list of tool options and select **Connect 1D to 2D**.

2. Select **Connect to 1D nodes directly** and click **OK.**

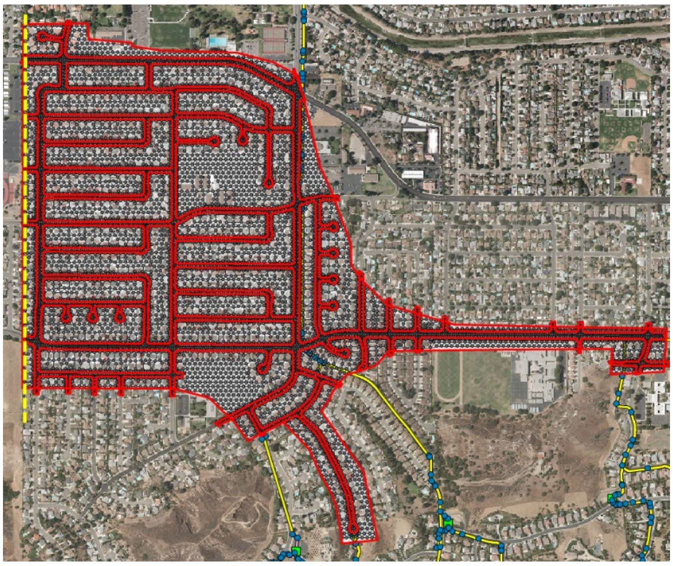
Note: For 2D dual drainage models, it is recommended to use bottom orifices to represent catch-basins. However, for this exercise, the catch-basin details were not available so direct connections were used.

7.8 Create downstream layer

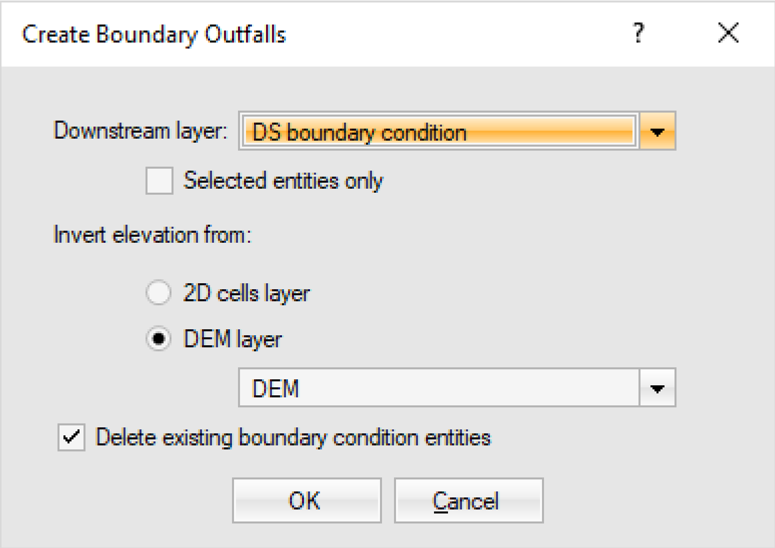
Currently the only way for water to leave the system is through the outfall at the north end of the 1D drainage system. In order to allow surface flow to exit the model across the 2D boundary, we need to add a downstream boundary line. In the steps below we will draw the boundary condition line in the **DS boundary condition** layer.

1. Click on the **DS boundary condition** layer in the **Layer** manager to select it.
2. Note: If the **DS...** layer is locked, unlock it.
3. Click on the **Add** button.

Draw a line across the west side of the 2D mesh, just inside the boundary area by first clicking on the desired starting position for the line, and then clicking on the desired ending position for the line. The line should run the full length of the west side of the mesh as shown by the yellow dashed line.

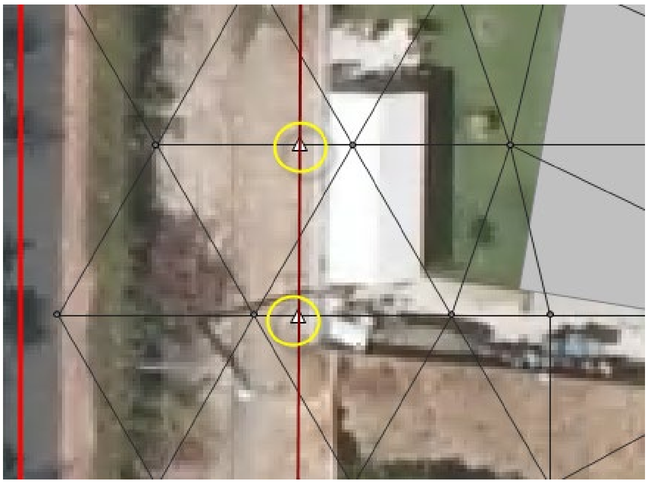


1. Click on the **Commit changes** button or press the **Enter** key to complete the creation of the line.
2. Click on the **Tools** button and select the **2D modeling** from the list of tool categories.
3. Select the **Create Boundary Outfalls** tool.
4. Ensure the **Downstream layer** has been assigned to **DS boundary condition**, the **DEM layer** is selected and click **OK**.



5. Click **Close** to close the **Create Boundary Outfalls** report.

6. Click on the **Zoom**  button and zoom into a section where the **DS boundary condition** was drawn. Make sure the 2D network is rendered to **Show Mesh**. Notice how there are now approximately 90 outfalls added to the mesh.



7. Zoom back out to the full extent of the SWMM model (click on the **Extent**  button and select **SWMM model** and then **View**).

7.9 Set simulation options

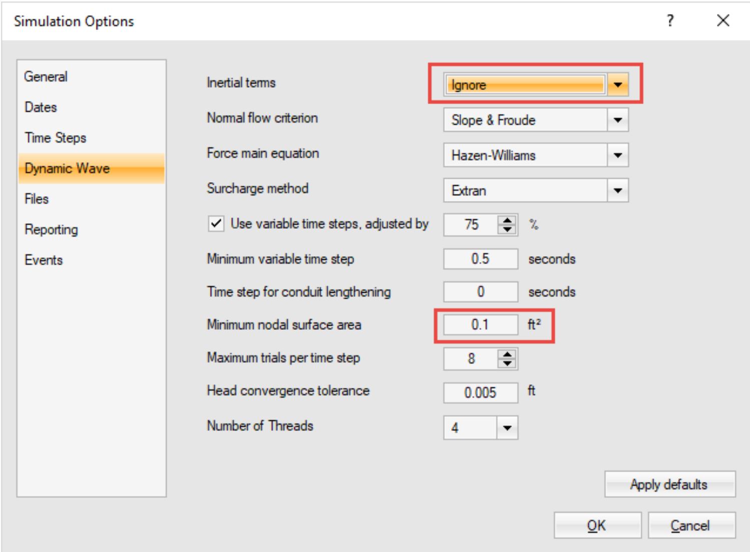
We will now set the simulation options to run the integrated 1D-2D model. When setting the simulation options we recommend you change the **Inertial terms** to **Ignore** (SWMM will use a Diffusive Wave routing method) and set the **Minimum surface area** to 0.1 ft. This will reduce the continuity error and provide a more reasonable estimate of the extent of flooding.

1. Click on the **Simulation Options** item from the **Project panel** and select the **Dynamic Wave** tab.

2. Set the **Inertial terms** option to **Ignore**.

3. In the **Minimum surface area** box, type in **0.1**.

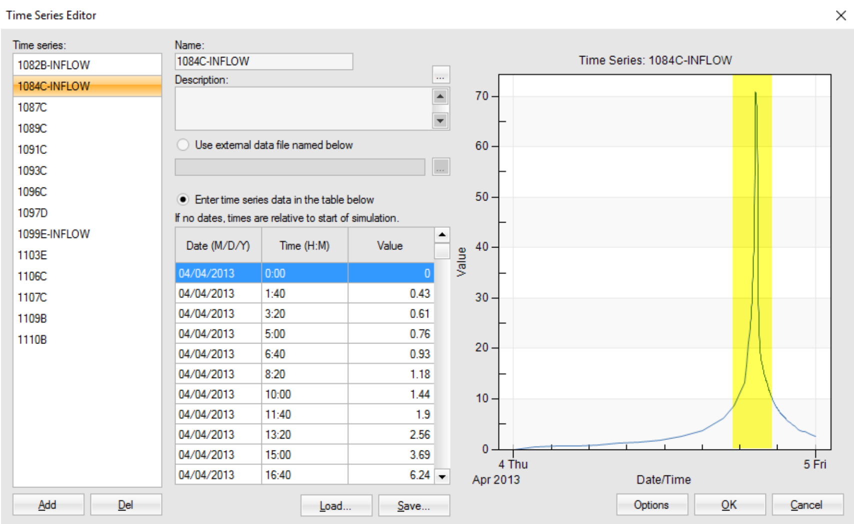
4. Click on the **OK** button to close the **Simulation Options** editor.



Since this model would take ~1 hour to run the full 24 hour simulation, we will change the simulation time to only include the peak of the event. Hydrology is not simulated in this model; instead, inflow hydrographs are added directly to the 1D hydraulic model nodes.

5. Click on the **Time Series** item in the **Project panel** to show the inflow hydrographs being used as input into the model.

Note where the peak of the event is for the inflow time series (from 18:00 to 20:00).

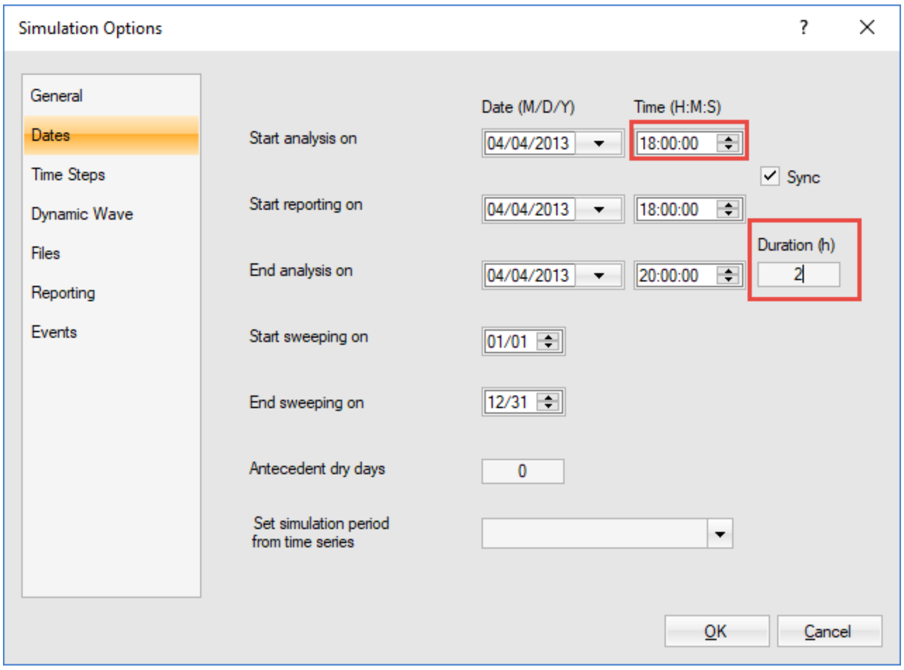


6. Click **OK** to close the **Time Series editor**.

7. Click on the **Simulation Options** item in the **Project panel**.

8. Click on the **Dates** tab to change the date of the simulation.

9. Change the **Start analysis on** time to be **18:00:00**, change the **Duration** to be **2** h and then click **OK.**



7.10 Run the model and visualize results

The model is now ready to run.

1. Click on the **Run** button to run the model.
2. When a warning message about the routing time step appears, click on the option to **Continue anyway**.

It may take between 10-15 minutes to run the model depending on the speed of your computer.

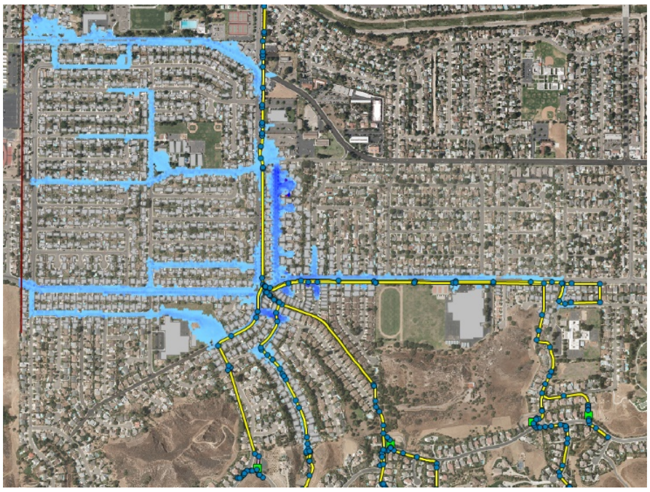
Once the run is complete, check that the routing continuity error is less than 5%. If it is between 3-5% it is likely a result of the model being run for such a small duration of the event.

We will now render our 2D cells to show the maximum extent of flooding.

Once the model is done running, uncheck the **Boundary** layer in the **Layers panel**.

Render the 2D mesh to show cell max. depths.

3. Click on the **Tools** button and select **2D modeling** from the list of tool categories.
4. Select **Render 2D network** and select **Show Cell Max Depths**.



You can also render the model based on Maximum Cell Max. Velocity and animate the 2D cells using the **Play** button in the **Map panel**.

7.11 Adding an elevations layer (Optional)

You may have noticed when the model was rendered to show the maximum depth, there is a section running from north to south showing odd rendering where the depths of water notably vary between adjacent cells. This is caused by the cell elevation being sampled from the center location of the 2D cell. In this case it is suggested that an elevation layer is used to sample the DEM at a higher resolution and from there assign the average elevation in the cell as the cell elevation.

1. Duplicate the project to create and then open a new scenario called **2D Simi model with elevations**.
2. Click on the **Plan** button and click on the **Add** button.
3. Select **Duplicate Current Project** and name the project **2D Simi model with elevations**.
4. Open the newly created scenario.
5. Click on **File** and select **2D** from the options on the left side of the screen.
6. Click on the **New** button beside the **Elevation layer** drop-down menu.
7. Navigate to **PCSWMM Exercises/K003/Initial/GIS layers** and click **Save** to save the file with the default file name **(i.e. Elevations.SHP)**.
8. Click **OK** to close the **2D** window.
9. Select the **Boundary** layer from the list of layers and unlock the layer.
10. Click on the overall boundary polygon.
11. Click on the **Tools** button.
12. In the **2D modeling** section, select **Generate Points**.

13. In the **Points layer** drop-down menu, specify **Elevations.**

14. Ensure that the **Selected polygons only** box is checked and the DEM layer has been assigned.

15. Click **OK**.

Now we want to regenerate the 2D overland mesh

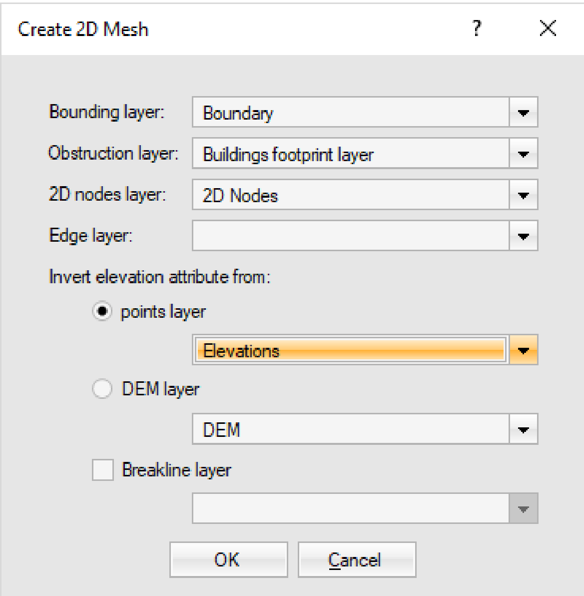
16. Click on **File** and select **2D** from the options on the left side of the screen.

17. Click on the **Tools** button.

18. In the **2D modeling** section, select **Create Mesh**.

19. Under **Invert elevation attribute from,** select **points layer**. Select **Elevations** from the drop-down menu.

20. Ensure the other 2D layers are properly assigned and click **OK**.



Because we are using direct connections we need to reconnect the model. Had we used orifice connections the orifices would have automatically been recreated.

21. Click on the **Tools** button.

22. In the **2D modeling** section, select **Connect 1D to 2D.**

23. Choose to **Connect to 1D nodes directly**.

24. Click **OK**.

25. Click on the **Run** button to run your model.

When a warning message about the routing time step appears, click on the option to **Continue anyway**.

It may take 10-15 minutes to run the model. Once the run is complete, note how the rendering in the one area has improved. As an additional step create a hydrograph layer in the backstage 2D options and compare the flow across this channel.

www.hydrosoft.co.kr

하이드로소프트

Telephone: 031.8017.8033

support@hydrosoft.co.kr

www.hydrosoft.co.kr

Follow Us