PCSWMM

Workbook (Advanced)

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. Design of an in-line stormwater pond to meet pre-development conditions

For new developments observed flow data for the "as-is" condition is usually absent, and an "as-is" model is developed to estimate pre-development runoff from the site. A detention pond is then sized to reduce the new or post-development ("to-be") peak runoff to the estimated pre-development value.

The limited time available for a workshop necessitate simplifying the steps required for this type of analysis. We'll start with the previously created "to-be" minor system model and remove the drainage structures to create the "as-is" model.

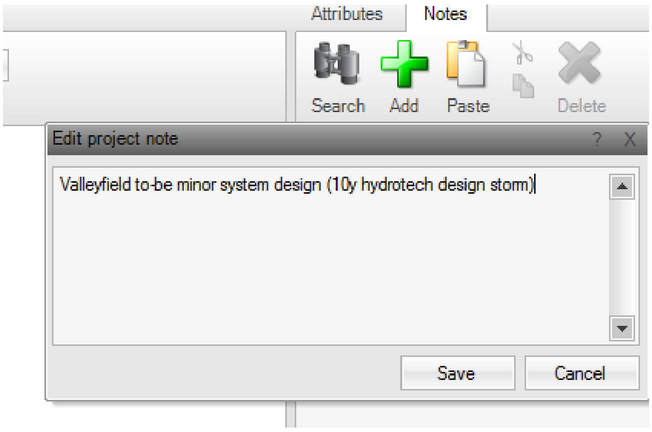
1.1 Creating an "as-is" scenario to estimate pre-development flow

First, let's run the current project to generate results and title the project to indicate its purpose more clearly. We are using a previously made solution file to ensure there are no errors from previous Valleyfield exercise files.

1. Open the **Valleyfield to-be minor system.pcz** from the **PCSWMM Exercises\K019\Initial** folder.
2. Click on the **File** tab and then the **Open** button**.**
3. Navigate to **PCSWMM Exercises\K019\Initial** and select **Valleyfield to-be minor system.pcz**.
4. Click the **Open** button.
5. A dialog will appear showing a default location to unpackage the model: click on the **Unpackage** button then the **OK** button.
6. Click on the **Run** button in the **Project panel** to generate computed results.
7. Add a project description to "**Valleyfield to-be minor system design (10y hydrotech design storm)**".
8. If no entity is currently selected, the **Notes panel** (located on the right side of the

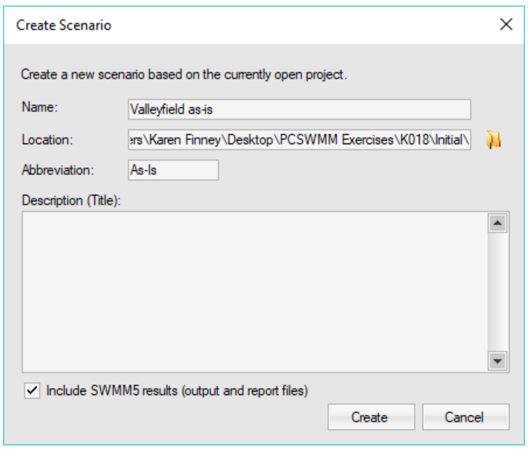
main PCSWMM window) should display the **Project Notes**. If an entity is selected, ensure the **Map panel** has the focus and press the **Esc.** key.

9. In the **Notes panel**, click on the **Edit** button in the **Description** entry.
10. Enter the following description: **Valleyfield to-be minor system design (10y hydrotech design storm)** and then click on the **Save** button (or press the **Enter** key) to save the note.

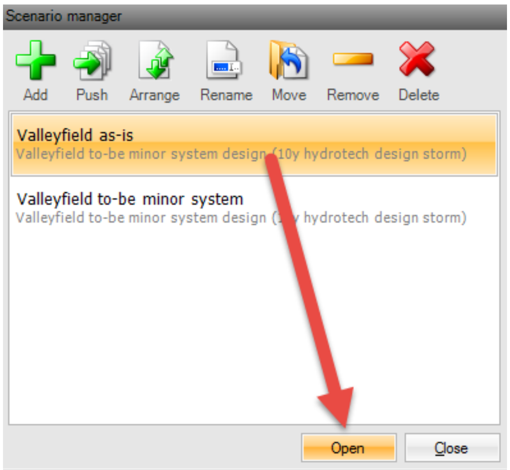


Now let's create a new scenario for the as-is model. A scenario is simply another SWMM5 project, however by adding it to the PCSWMM scenario manager it is easier to switch between the projects. In addition, a number of useful scenario comparison tools become available.

11. Duplicate the current scenario in the same folder, with the name "**Valleyfield as-is**", the abbreviation "**As-Is**", and the description "**Pre-development peak flow estimation (10y hydrotech design storm)**".
12. Click on the **Plan** (scenario manager) button in the toolbar of the **Project panel**
13. In the **Scenario Manager**, click on the **Add** button and select **Duplicate Current Project**.
14. In the **Create Scenario(s)** dialog name the project **Valleyfield as-is**.
15. Specify the location where your exercise is located (e.g. **PCSWMM Exercises\K019\Initial).**
16. Enter in the **Abbreviation** to be **As-is**.
17. Edit the **Description (Title)** text to read **Pre-development peak flow estimation (10y hydrotech design storm).**
18. Click the **Create** button



19. The current project as well as the newly created scenario should appear in the **Scenario Manager** (under the **Plan** button). Note that the currently loaded project is still the to-be project, as indicated by the name displayed at the top of the PCSWMM interface.
20. Open the newly created **As-is** scenario.
21. Click on the **Plan** button.
22. Select **Valleyfield as-is** and click **Open** to switch to the new scenario.



23. Click on the **Save Project** button in the **Save Project** message box to save the current scenario before switching.

Do not save time series (if prompted).

For the as-is model, we will remove the minor system drainage entities (conduits and junctions). We will then adjust the subcatchment routing and impervious area and run the model to produce an as-is runoff hydrograph at the outfall.

Deleting junctions will automatically delete the connected conduits (because conduits must be connected to junctions).

24. Select all the junctions in the **Map panel** and delete them.
25. Select the **Junctions** layer in the **Layer panel**.
26. Select all the junctions by pressing the **Ctrl + A** keys on the keyboard.
27. Click the **Delete** button or press the **Delete** key on your keyboard to remove the junctions.
28. Click **Yes** in the **Delete Shapes** confirmation message.

All conduits and junctions should be deleted, leaving only the subcatchments and an outfall.

There are two ways to model the hydrology for the pre-development condition:

aggregate the subcatchments into a single subcatchment, OR

leave the subcatchments as they are and route the runoff from each subcatchment onto the next subcatchment towards the outfall. This is termed run-on and can be useful when routing overland sheet flow onto areas of differing hydrologic properties – for example routing an impervious area (say a parking lot) onto a pervious area.

In our case, the hydrologic properties of the site are uniform and thus we will aggregate the subcatchments into a single subcatchment. Before aggregating the subcatchments, it is recommended that you ensure that the boundaries of the subcatchments coincide cleanly.

29. Use the **Join** tool to combine all the subcatchments into one.
30. Switch to the **Subcatchments** layer by clicking on the **Subcatchments** item in the **Layers panel**.
31. Press the **Ctrl** + **A** keys to select all subcatchments.
32. Click on the **Edit** button in the **Map panel** and select **Join** from the list of items in the **Edit browser**.
33. Click **Analyze** to preview the join operation.
34. Click on the **Apply** button to aggregate the subcatchments, and click **Close** to exit the **Join** tool.

If the boundaries of the subcatchments coincided, the resulting aggregated subcatchment should appear similar to the one illustrated below. The only remaining entities in the as-is model should be a single subcatchment and an outfall.

If you have stray vertices, you could edit the new subcatchment shape to clean them up, however please note that this is for aesthetics only – the area computed by PCSWMM will not be affected significantly and thus there should be no significant effect on the SWMM5 computed runoff.

Let's edit the attributes for the aggregated subcatchment. We will check the area, estimate the as-is overland flow path length and set the directly connected impervious area (DCIA) to 0.

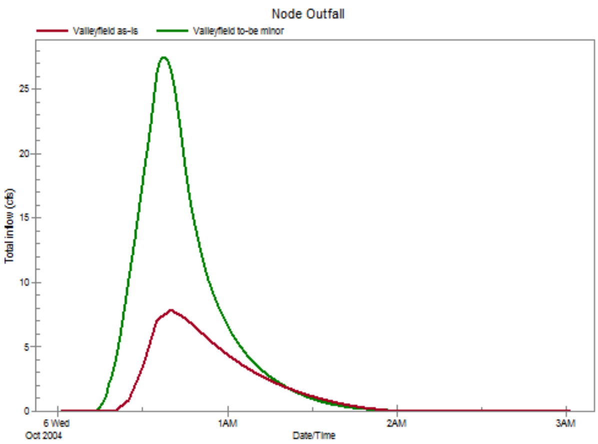
35. Click on the **Menu** button in the **Map panel**, select **Preferences** and in the **General** tab, check the **Calculate subcatchment width** box if it is not already checked.
36. Click **OK** to close the **Preferences**.
37. Select the subcatchment and check that the **Area** attribute is approximately 17 acres | 7 hectares (as shown in the **Attributes panel**).
38. In the **Attributes panel**, set the **Outlet** to **Outfall**.
39. Set the **Flow Length** to **500 ft** | **150 m**.
40. Set the **% Impervious** to **0**.

Now let's run the as-is model and compare the runoff to the to-be scenario.

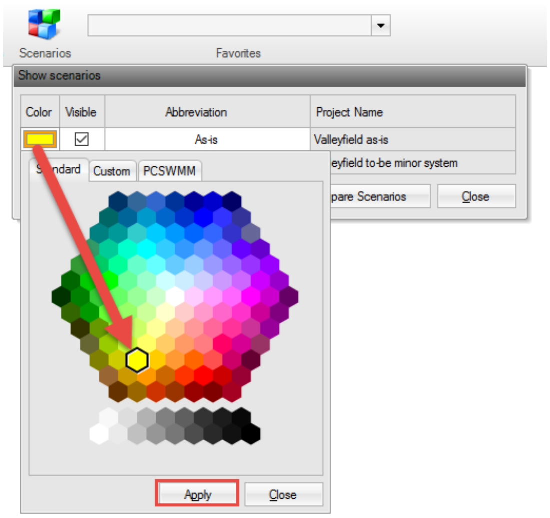
41. Click on the **Run**  button to run the model.
42. After the run, ensure the continuity error is reasonable (< 1 %) by checking the run summary status results located at the bottom right hand side of the screen.
43. In the **Graph panel**, plot the **Total inflow** for the **Outfall**.
44. Switch to the **Graph panel** by clicking on the **Graph** tab.
45. In the **Time Series Manager**, expand **SWMM5 output > Nodes > Total Inflow** and click on the checkbox for the **Outfall** location.
46. In the **Graph panel,** click on the **Scenarios**  button.
47. Check both scenarios are **Visible** and click on the **Compare Scenarios** button.

Note: If a message appears saying PCSWMM is unable to locate a scenario, click on the **Remove scenarios** option. If this happens we do apologize, this is a result of scenarios not being removed while updating exercise files.

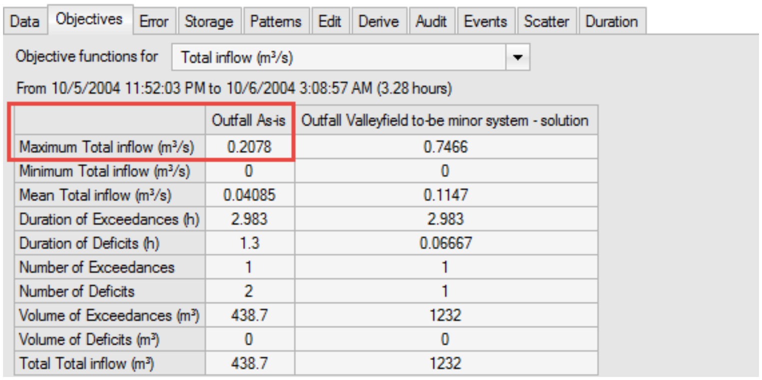
The **Graph panel's** scenario mode will turn on and the selected location will be plotted for both scenarios (as-is and to-be). For this rather frequent storm, the paved areas make a relatively large difference to the computed response (please note the screenshot is in US units, SI units will differ and your line colors may differ).



The **Show Scenario** button can be used to toggle the scenario mode on and off. The **Show Scenario** button will display the **Show Scenario** pop-up editor, which can be used to change the color for each scenario (click on a colored square to change it). The color for a scenario is saved and is consistent for all plots when in the scenario mode.



48. Make a note of the pre-development peak flow from the **Objectives** tab below the plot in the **Graph panel.** Please note the screenshot is in SI units.



	Outfall As-is	Outfall Valleyfield to-be minor system - solution
Maximum Total inflow (m³/s)	0.2078	0.7466
Minimum Total inflow (m³/s)	0	0
Mean Total inflow (m³/s)	0.04085	0.1147
Duration of Exceedances (h)	2.983	2.983
Duration of Deficits (h)	1.3	0.06667
Number of Exceedances	1	1
Number of Deficits	2	1
Volume of Exceedances (m³)	438.7	1232
Volume of Deficits (m³)	0	0
Total Total inflow (m³)	438.7	1232

In the **Graph panel** click on **Tools** **> Objective functions**.

The **Objectives** tab will appear in the bottom of the **Graph panel**. The **Objective functions** tool computes various statistics on one or more plotted time series on-the-fly in the **Graph panel**. Statistics include maximum, minimum, mean and total, as well as number, duration and volume of exceedances and deficits. Objective functions can be used to compare two or more time series, as well as quickly display statistics of events within the time series.

In the **Objectives** tab, take a note of the peak flow (**Maximum Total Inflow**) for the as-is scenario (probably between **7 and 8 CFS | 0.17 - 0.22 m³/s**).

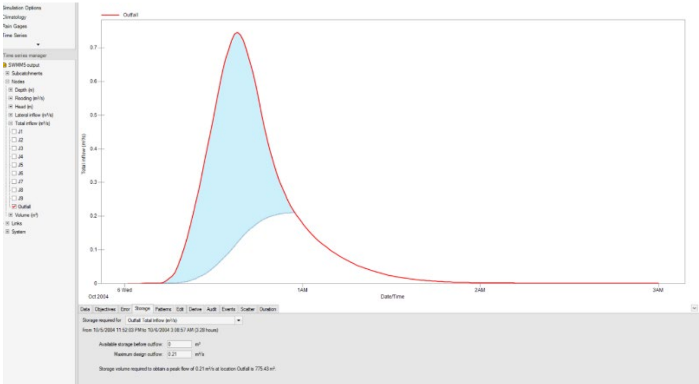
1.2 Sizing a detention pond to mitigate increases in peak flow

Now we are ready to add a detention pond to the drainage system design (to-be model). The volume required for a storage pond can be determined using the **Storage Pond Calculator** tool in the **Graph panel**.

1. Open the **Valleyfield to-be minor** scenario.
2. Click on the **Plan** button in the **Project panel,** select the **Valleyfield to-be minor** scenario and click on the **Open** button.
3. Choose to **Save** the project if prompted to.
4. In the **Graph panel**, exit **Scenario mode**.
5. Click on the **Graph** tab to open the **Graph panel** (if it is not open already).
6. Click on the **Scenarios** button in the **Graph panel** and click the **Exit Scenario Mode** button if you are still in scenario mode.
7. Plot the **Total inflow** at the **Outfall**.
8. In the **Time series manager**, expand the **SWMM5 output** to show **Nodes > Total Inflow > Outfall**.
9. Check the **Outfall** to plot the hydrograph at the outfall.
10. In the **Graph panel,** click on **Tools** and select **Storage Pond Calculator**.
11. In the **Storage** tab at the bottom of the panel, enter your recorded predevelopment peak flow (approx. ± **8 CFS | 0.21 m³/s**) in the **Maximum design outflow** box.

12. Ensure the **Available storage before outflow** is set to 0 m^3.

The approximate storage requirement to meet the predevelopment peak flow is calculated as the area under the hydrograph (shown in blue). This tool provides a rough estimate of the storage required – it makes some simple assumptions about the outflow from the storage facility (shown with the blue line: outflow 1 is assumed to increase linearly with volume in the storage unit), but can be useful for a first estimate. (Please note the screenshot is in SI units, US units will differ).



13. Make a note of the required storage volume. It should be approximately **27,000 ft³ | 800 m³** (likely less).

1.3 Creating a pond scenario

Now let's create a new scenario for developing the storage pond. As new scenarios created in the **Scenario Manager** are copies of the currently loaded scenario, it is important to ensure the current scenario is the **Valleyfield to-be minor system** project.

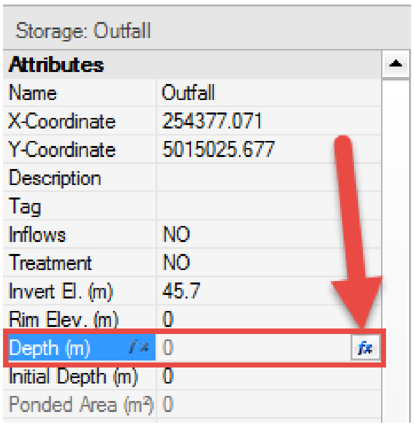
1. Create a duplicate scenario called **Valleyfield to-be pond**, with the abbreviation "**To-be Pond**" and description "**Valleyfield minor system detention pond design (10y hydrotech design storm)**".
2. Check that the model currently open is the **Valleyfield to-be minor** scenario (should be listed at the top of the interface).
3. Click on the **Plan** button in the **Project panel**.
4. In the **Scenario Manager**, click on the **Add** button and select **Duplicate Current Project** .
5. In the **Create Scenario** dialog, name the project **Valleyfield to-be pond**.
6. Set the **Abbreviation** to **To-be Pond**.
7. Edit the **Description** item text to read **Valleyfield minor system detention pond design (10y hydrotech design storm)**.
8. Click the **Create** button.
9. The new SWMM5 project should appear in the **Scenario Manager**.
10. Switch to the new scenario by selecting it in the **Scenario Manager** and clicking **Open**. If prompted to save the current project, click on the **Yes** button.

Now let's add a storage unit to the model at the current outfall location.

Convert the existing outfall to a storage node.

11. Open the **Map panel** and click on the **Outfall** entity.

12. Right-click on the **Outfall** and select **Convert > Storage** from the drop-down box.
13. If necessary, edit the storage layer to auto-calculate rim elevation instead of auto-calculating depth, as we want to enter the pond depth manually.
14. With the Storage entity selected, check the **Attribute panel** to see if the **Depth** attribute is disabled (gray), If not, skip the next 3 steps.
15. If disabled, the depth will be calculated based on the difference between the rim and invert elevations, which is not what we want. To change this, click on the **Depth** attribute to select it, and then click on the small *fx* button that appears in the **Depth** attribute to open the **Auto-expression editor.**



16. In the **Auto-expression editor**, click on the **Calculate Rim Elev. instead** button. This will allow you to edit the **Depth** attribute as opposed to being automatically calculated using the defined auto-expression.
17. Click **Close**.
18. The outfall entity has been converted to a storage unit, however we still need to set its attributes. Let's make the pond **6.56 ft** | **2 m** deep and have the sewer discharging near the top of the pond bank.
19. In the **Attributes panel** for the converted entity, change the following parameters:

Name = **Pond**

Invert Elev. = **146.51** (ft) | **44.5** (m)

Depth = **6.56** (ft) | **2** (m)

Normally we would now create a stage / area curve to define the pond, however in the interest of time we will use a simple functional relationship. (If you have time at the end of the exercise, try your hand at using a TABULAR shape curve and entering a depth vs. area curve).

Since we need a total storage volume of **27,000 ft³** | **800 m³**, we simply divide this value by the depth of the pond to arrive at a constant area for the pond (in fact now a vertical walled tank).

In the **Attributes panel** for the **Pond** storage entity, set:

Storage Curve = **FUNCTIONAL**

Coefficient = **0**

Exponent = **0**

Constant = **4116** (ft²) | **400** (m²)

Recall that we want the conduit discharging into the pond to have the same slope as the rest of the drainage pipe network and thus connect to the pond near the top. Since in the project offsets are specified in depths rather than elevations (see the **Offsets** setting in status bar in the lower left corner of the main PCSWMM window), we need to specify the depth above the pond invert at which the discharging conduit connects. We'll set this depth so that conduit **C8**'s outlet elevation is the same as it was before we added the pond. We can use the **Map panel** to do this, however it can also be completed in the **Profile panel**.

20. View the profile from node **J1** to the storage **Pond**.
21. In the **Map panel**, select a pathway through the system by clicking on the furthest upstream junction **J1** (at the bottom of the **Map panel**) and, holding the **Shift** key down, clicking on the storage **Pond**. All intermediate nodes and conduits should be selected (highlighted in blue).
22. Click on the **Profile** tab to switch to the **Profile panel** and see the drainage system profile along the selected pathway.
23. If a warning appears notifying you that there are no results, choose **Continue**.
24. You can see that since conduit **C8**'s outlet offset is 0, the conduit is currently steeply sloped and connected to the bottom of the pond. If the **Offsets** option was set to **Elevation** this would not have happened.
25. Set the **Outlet Offset** for **C8** to **3.06 (ft) | 1.1 (m)**.
26. In the **Profile panel**, select conduit **C8** by clicking on it in the profile plot. When selected the conduit lines turn red and its attributes are displayed in the **Attributes panel**.
27. In the **Attributes panel** for conduit **C8**, set the **Outlet Offset** = **3.06 (ft) | 1.1 (m)**. If the conduit slope became negative when you applied the offset it is likely because you did not change the invert elevation of the pond to 44.5 m as outlined earlier in step 5.
28. Press the **Enter** key or switch away from the **Outlet Offset** attribute item (e.g. click on another attribute or in the profile plot) to commit the change and see the new profile.

Note: You can also graphically edit the outlet offset for conduit **C8** directly in the **Profile panel**. Before doing this it is a good idea to **save** your project first. To do this, entering Edit mode (click on the Edit button in the toolbar), click on conduit C8 in the profile plot to select it and then drag the right-most handle (red dot) up and down to set the Outlet offset attribute. This method does not give as much accuracy, but can be faster, especially when used for sizing conduits (in which case the diameter of the pipe snaps to standard pipe sizes).

Now we need to create an outfall below the pond and an outlet entity to connect the pond to the outfall and define the head / outflow relationship of the pond's outlet structure.

Note: you can model the physical structure of the outflow devices using a combination of one or more orifices and weirs, as an alternative to defining a head vs. outflow relationship. One advantage of this latter approach is that it can more accurately model reverse flow conditions and/or backwater effects from the water surface elevation downstream of the pond. (If you have time at the end of the exercise, try your hand at replacing the outlet entity with an orifice and weir combination. You can also add a second, higher and larger weir to model a spillway i.e. have one orifice and 2 weirs draining the pond in parallel).

In the **Map panel**, switch to the **Outfalls** layer and click on the **Add**  button.

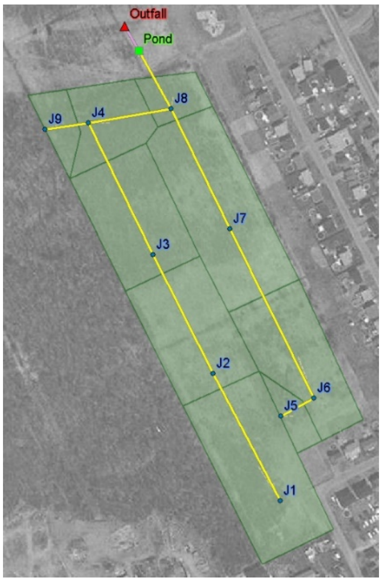
Add an outfall somewhat north of the pond and set its attributes as follows:

Name = **Outfall**

Invert El. = **144.36** (ft) | **44** (m)

Type = **FREE**

29. Switch to the **Outlets** layer and click on the **Add** button. The **Outlets** layer will appear grayed out in the **Layers panel** as there are not **Outlets** in the model as of yet.
30. Draw an outlet joining the **Pond** by clicking on the pond then on the new **Outfall**, and set its attributes as follows:



Name = **PondOutlet**

Rating Curve = **TABULAR/DEPTH**

We'll use a tabular rating curve to illustrate the method of creating and assigning curves.

31. Create an outlet rating curve **PondOutlet** and assign it to the outlet using the Rating Curve info in the screenshot below.
32. With the Outlet selected, click on the **Curve Name** attribute in the **Attributes panel** and an ellipsis ... button will appear. Click on this button to launch the **Rating Curve** editor.
33. In the **Rating Curve** editor, click on the **Add** button to create a new curve.
34. Name the new curve **PondOutlet** and enter the following data in the **Head** vs. **Outflow** table (click the image below to see the data).
35. Click on the **Assign to Outlet PondOutlet** button to save your changes.

Rating curve info:

US units		**SI units**	
Head (ft)	**Outflow (CFS)**	**Head (m)**	**Outflow (m³/s)**
0	0	0	0
6.56	8	2	0.23

The **Rating Curve** editor opens in a mode that allows you to set the rating curve for the selected entity. You should notice the **Rating Curve** editor's title bar indicates the selected entity. PCSWMM presents the **Rating Curve** editor, rather than a simple drop-down list of curve names, to provide more information during the selection of the curve and also to provide the option of adding or editing curves during the selection process. This mode only applies if a curve editor is launched from the **Attributes panel** for a selected entity. Launching a curve editor from the **Project panel** opens it in a neutral mode (i.e. doesn't affect the assigning of curves).

Please note that normally a pond outlet rating curve will be more complex than the one just specified.

Now let's run the model to evaluate the performance of the pond.

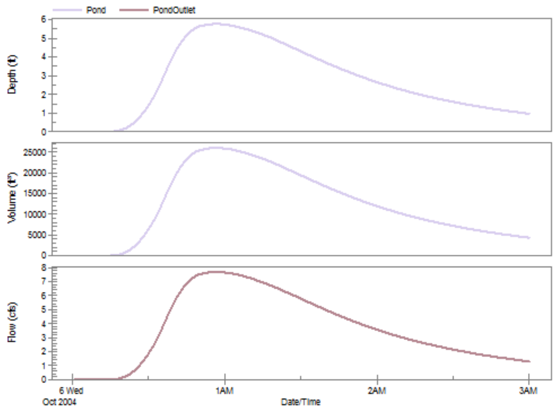
36. In the **Map panel**, click the **Select** button to exit edit mode.
37. Click on the **Run** button to execute a SWMM5 run.
38. Plot the **Flow** for the **PondOutlet** and **Volume** and **Depth** for the **Pond**.
39. Switch to the **Graph panel**.
40. In the **Time series manager** expand the following SWMM5 output, and plot:

Links > **Flow > PondOutlet**

Nodes > **Volume > Pond**

Nodes > **Depth** > **Pond**

Please note the screenshot is in US units, SI units will differ.

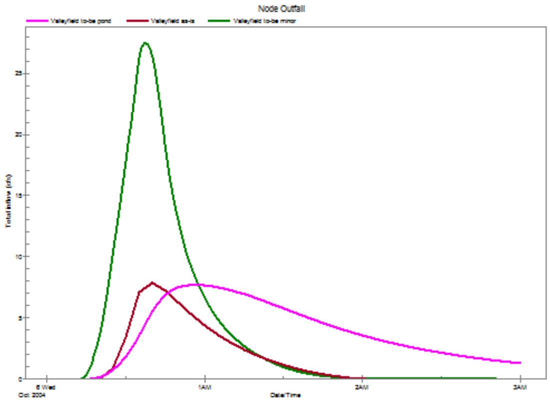


The outflow from the pond will hopefully meet the pre-development target and the pond's water surface elevation should not reach its bank elevation.

We can use the scenario mode to compare the outfall hydrograph for all three scenarios.

41. In the **Graph panel**, click on the **Menu** button and select **Clear Graph**.
42. Plot the **Total Inflow** for the **Outfall** node (**Nodes > Total Inflow > Outfall**).
43. View and compare the hydrographs for all three scenarios.
44. Click on the **Scenarios** button in the **Graph panel.**
45. In the **Show Scenarios** editor, place a check in all three scenario check-boxes and adjust the colors as necessary.
46. Click on the **Compare Scenarios** button to enter scenario mode.
47. Click **Close** to exit the **Show Scenarios** editor (please note the screenshot is in US units, SI units will differ)

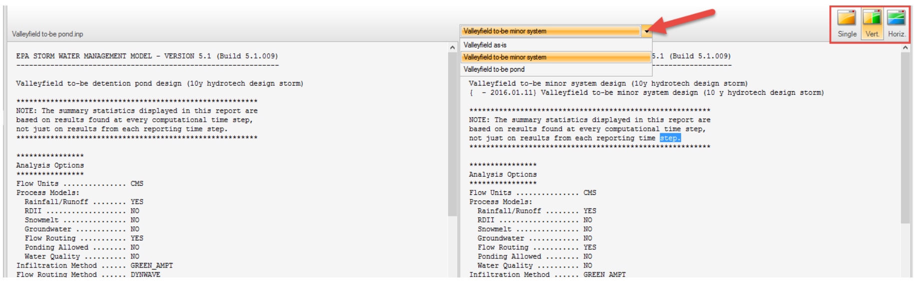
The comparison plot should look similar to the one here (line colors may vary). The objective functions below the plot can be used to view statistics on the three time series, including their peak (maximum) values.



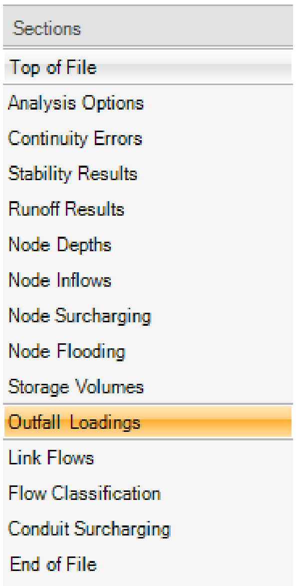
It is important to note that the objective functions are computed on the plotted time series and are thus dependent on the **Reporting** time step set in the **Simulation Options** editor. They may differ from the statistics reported in the **Status panel**, which are computed using the routing time step being used by the SWMM5 engine. The statistics reported in the **Status panel** (i.e. the SWMM5 report file) should take precedence in all cases. The plotted time series can more accurately reflect the computed values by reducing the **Reporting** time step, however this creates more data points to be loaded and plotted.

Let's compare the outfall loading summary table in SWMM's status report.

48. Switch to the **Status panel** by clicking on the **Status** tab
49. Click on the **Split window vertically**  button in the upper right corner of the **Status panel** (alternatively, if your screen resolution is small, you may prefer to choose the **Split window horizontally**  button).
50. Click on the **Scenario drop-down** menu button and select the **Valleyfield to-be minor** scenario.



51. In the **Sections** list on the left (under **Project panel)** select **Outfall Loadings**.



52. The **Status panel** should display the peak outfall flows for the two scenarios (in the **Outfall Loading Summary** tables).

Let's look at the **Project Summary/Comparison** tool, which generates tables of both model inputs and results for multiple scenarios.

53. Switch to the **Map panel** and open the **Summary/Comparison** tool (Tools menu).
54. Click on the **Map** tab and click on the **Tools** button in the **Map panel**.
55. In the **Auditing** section, click on the **Summary/Comparison** tool.
56. In the **Summary/Comparison** tool, click the **Show Scenarios** button and select all three scenarios.
57. In the **Summary Tables** list, select **Results statistics.** Scroll down to see the full results table.
58. In the results table, compare the scenario values for:

Max. subcatchment total runoff

Max. subcatchment peak runoff

Max. subcatchment runoff coefficient

Num. conduits surcharged

Max. storage percent full

Max. outfall peak flow

Total outfall volume

Valleyfield as-is scenario is missing some values from this table as it did not contain any hydraulic routing components. For more insight to how the **Valleyfield as-is** model performed, examine the **Runoff quantity continuity** table:

59. Select the **Runoff quantity continuity** table in the **Summary Tables** list.
60. Compare the mass balance values in this table and ensure they make sense.
61. Close down the **Project Summary/Comparison** tool when finished.

62. Finally, let's take a look at the animation of the profile through the pond.
63. Animate the results in the **Profile panel** for **J1** to the **Outfall**.
64. Select a pathway (using the **Shift** key) from junction **J1** to outfall **Outfall**.
65. Switch to the **Profile panel** to see the peak values for head displayed in the plot (hint... Menu | Show Peak Values).
66. Click on the **Play** button in the bottom left corner of the **Profile panel** to playback the computed HGL in the profile plot.

The slider bar at the bottom of the profile has start and stop pointers that can confine the playback to a specific time period of interest. Playback speed can be adjusted in the **General** section of the **Profile Properties** display (click on **Properties** button in the **Profile panel**). Playback can also be manually controlled by dragging the slider.

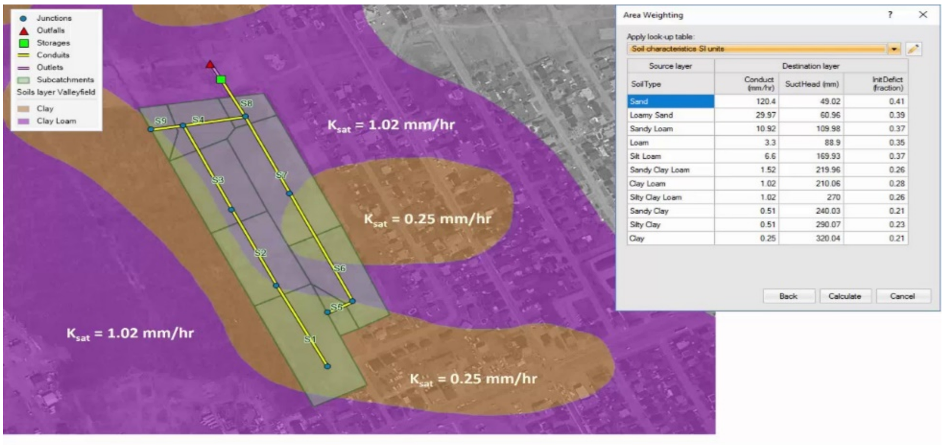
2. Estimating subcatchment attributes based on land-use and soils layers

This example illustrates the steps a planner or engineer may consider if there was a change in a subdivision design to incorporate a small park. In this example, 8 of the subcatchment attributes will be calculated based on a land-use and soils map.

Normally the user would re-discretize subcatchments however, for simplicity sake, this example uses the same subcatchment discretization created in earlier Valleyfield exercises.

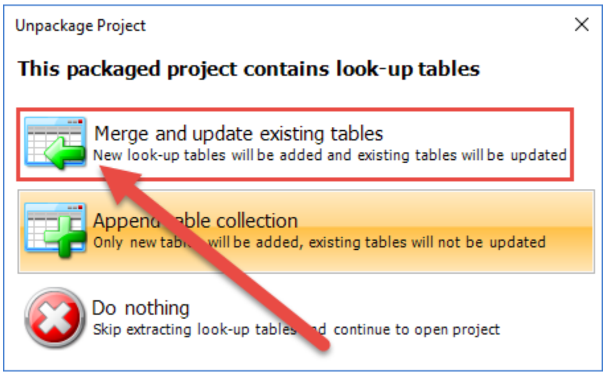
Subcatchment attributes can be calculated based on a land-use layer. User-defined attributes matching those in the subcatchment layer can be added to a land-use layer in order to obtain an area-weighted average of the subcatchment parameters. In this section the user-defined attributes IMPERV, NIMPERV, NPERV, PSPERV and DSIMPERV have already been added to the land-use layer for you.

Another approach for estimating SWMM attributes is through the use of look-up tables. In the second part of this exercise we will estimate subcatchment infiltration using a soils layer in combination with a look-up table.



2.1 Rendering a land-use map

1. Unpackage the **Valleyfield to-be pond - solution.pcz** file from the **PCSWMM Exercises\K022\Initial** folder. Ensure you **Merge and update existing look-up tables.**
2. Click **File** and select the **Open** button.
3. Browse to the folder **PCSWMM Exercises\K022\Initial.** Open **Valleyfield to-be pond - solution.pcz.**
4. Click the **Open** button.
5. A dialog will appear showing a default location to unpackage the model: click on the **Unpackage** button then the **OK** button.
6. If PCSWMM notifies you of look-up table information, choose to **Merge and update existing tables.**

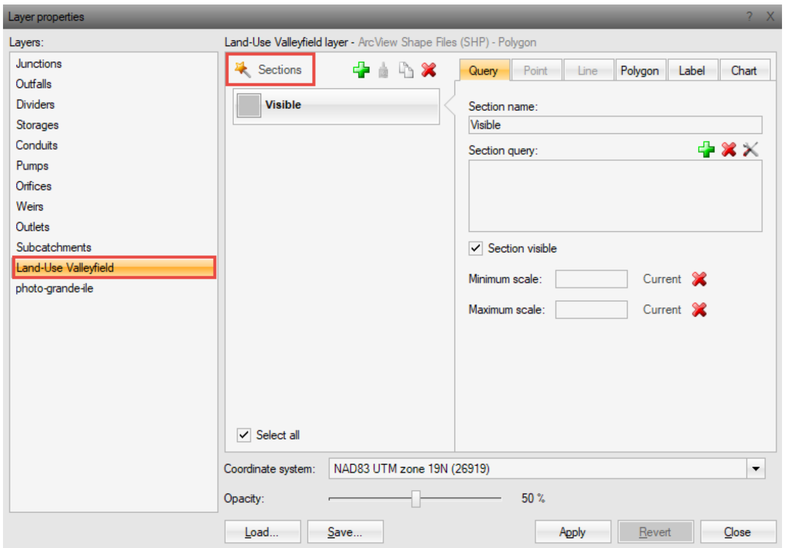


7. Click on the **Run** button.
8. **Open** the background layers **photo-grande-ile.jpg** and **Land-Use Valleyfield.shp** files from the **PCSWMM Exercises\K022\Initial** folder.
9. Click the **Open** button in **Map panel**.
10. Click the **Browse** button and navigate to **PCSWMM Exercises\K022\Initial.**
11. Select the raster image **photo-grande-ile.jpg**.
12. Hold down the **Ctrl** key and select the **Land-Use Valleyfield.shp** file.
13. Click on the **Open** button.
14. In the **Layers panel** arrange the layers (by dragging-and-dropping) so that **Land-Use Valleyfield.shp** overlies **photo-grande-ile**.
15. Click on the **File** tab and select **Save As...**
16. Change the name of the project to **Valleyfield Area-weighting** and click **Save.**

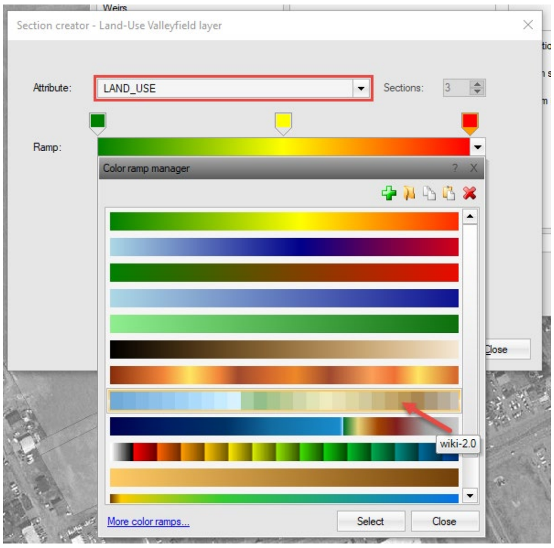
It is a good habit to render a layer based on the attribute of interest. In this case the assigned land-use is of greatest interest.

Render the land use layer with sections based on the **LAND_USE** attribute and the **wiki-2.0** color ramp. Make the layer 50% transparent and add a land use label.

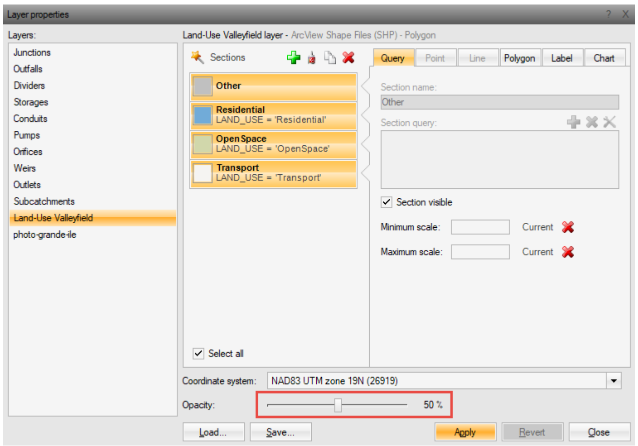
17. Click on the **Render** button in the **Map panel**.
18. Select the **Land-Use Valleyfield** layer from the list of layers.
19. Click on the **Sections** drop-down box to open the **Section creator**.



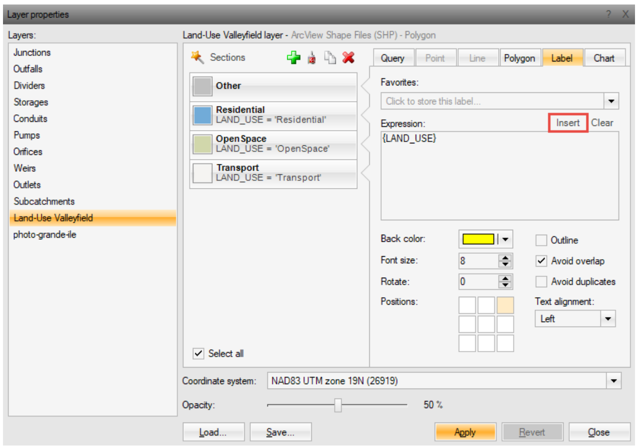
20. Click on the **Attribute** drop-down list and select **LAND_USE** as the attribute to render on.
21. Click on the color ramp drop-down box and click **wiki-2.0** from the list of available ramps (you can see the ramp name by holding your mouse over a ramp) and click **Select**.



22. Click on the **Create sections** button.
23. Change the **Opacity** slider bar to around 50%.



24. Click on the **Label** button and click on **Insert**.



25. Place a check beside the **LAND_USE** attribute.
26. Click **Insert, Apply** and then **Close**.

The **Map panel** should look something like the illustration below (colors may vary).

2.2 Performing area-weighting from Land-use layer attributes

1. Click the **Table** tab to open the **Table panel** and select **Land-Use Valleyfield** from the **Layers panel**.

Examine the values in the table. You should notice that there are 8 attributes, 5 of which we will use to calculate the subcatchment attributes (please note the screenshot shown is in SI units, US units will differ).

Land-Use Valleyfield

LAND_USE	IMPERV	NIMPERV	NPERV	DSPERV	DSIMPERV	GIS_LENGTH (m)	GIS_AREA (m²)	GIS_PARTS
Residential	50	0.015	0.3	3.81	1.9	671.9	16768.333	1
Residential	50	0.015	0.3	3.81	1.9	289.475	3352.367	1
Residential	50	0.015	0.3	3.81	1.9	610.479	9313.99	1
Residential	50	0.015	0.3	3.81	1.9	709.884	14761.976	1
OpenSpace	0	0.015	0.3	5.08	1.9	252.926	3434.51	1
OpenSpace	0	0.015	0.3	5.08	1.9	933.797	34003.701	1
Transport	90	0.015	0.3	3.81	1.9	3364.203	25786.388	2
Residential	50	0.015	0.3	3.81	1.9	1110.109	31529.524	1
OpenSpace	0	0.015	0.3	5.08	1.9	2595.261	295281.429	1

The 5 attributes that will be used for the area-weighting can be found in following table (click view to open the image).

Attribute Name	Attribute Definition
IMPERV	Imperv (%) - Percent of directly connected impervious area
NIMPERV	Mannings N for impervious area
NPERV	Mannings N for pervious area
DSPERV	Depth of depression storage on pervious area (mm)
DSIMPERV	Depth of depression storage on impervious area (mm)

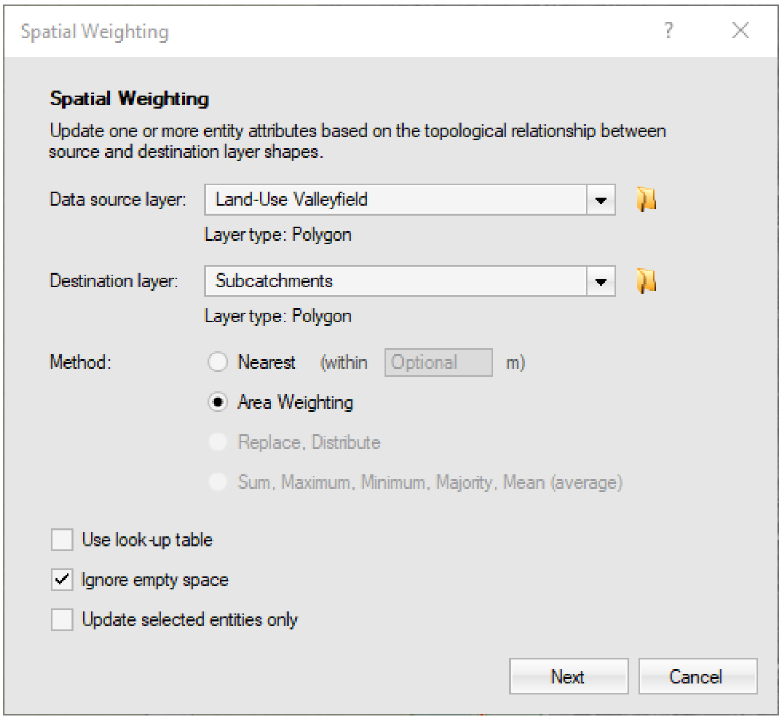
Some of these values were estimated using the following Reference Tables listed on the PCSWMM support site:

Manning's N - Overland flow for N Imperv, N Perv

Depression storage for Dstore Imperv, Dstore Perv

Note: You should notice how the attributes are named. These names are the subcatchment field names recognized by PCSWMM. By having these attributes named with the subcatchment field names in the land-use map, the area-weighing tool will automatically recognize the attribute and match it with the same attribute in the subcatchments layer. Otherwise the user can manually select the matching layer attributes.

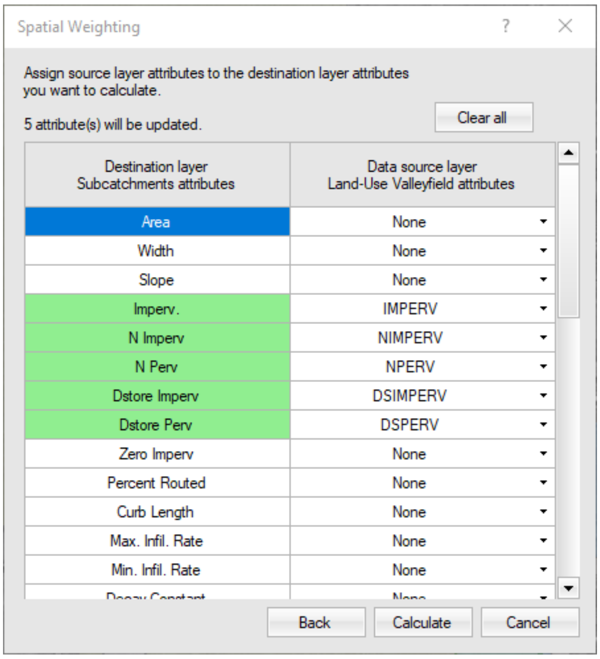
2. Click on the **Tools** button in the toolbar of the **Map panel** to open the **Tools Browser**.
3. Click on the **Spatial Weighting** tool (in the Subcatchments, Nodes and Conduits sections).
4. In the **Spatial Weighting** tool, set the **Data source layer** to **Land-Use Valleyfield** and the **Destination layer** to **Subcatchments.** Change the **Method** to **Area Weighting.**
5. Click on the **Next** button.



The area weighting tool option in the Spatial Weighting tool calculates the value of one or more attributes for entities in a polygon layer (e.g. the SWMM5 Subcatchments layer, or any other layer) by area-weighting matching attributes from the intersecting entities on a source polygon layer.

Note: The Spatial Weighting dialog displays a table with two columns. The first column displays the destination layer attributes (Subcatchment attributes) while the second contains drop-down lists of the attributes in the data source layer (Land-Use Valleyfield attributes). Because the land-use and subcatchment attributes have identical field names, the area-weighing tool can identify the 5 attributes to be used for the area-weighting calculation.

Ensure the destination layer (**Subcatchment**) attributes are properly matched with the data source layer (**Land-Use Valleyfield**) as shown below:



6. Click on the **Calculate** button to perform the area-weighting operation. A report should appear saying that 9 entities on the **Subcatchment** layer have been updated.

In the report, you should notice that for each of the subcatchments both the old and new values are displayed for each of the subcatchment attributes updated. You will also notice, in the **Spatial weighing** report, the number of **Land-Use** components (polygons) located in the boundaries of each of the subcatchments. Also listed is the percentage of area for each component located in each subcatchment. The sum of these fractions should equal 1.0, meaning that 100% of the subcatchment area has been accounted for in the **Land-Use Valleyfield** layer.

7. Click the **Close** button to close the report.
8. Select the **Land-use Valleyfield** layer and click the **Close** button to remove it from the list of layers.

2.3 Opening up a soils background layer

Soils layers commonly consist of the spatial distribution of soil types for a specific area. Soil layers can be valuable in setting up a SWMM model, as they allow for the infiltration properties for the subcatchments be estimated in setting up a SWMM model.

1. **Open** and unlock the **Soils layer Valleyfield.shp** background layer in the **PCSWMM Exercises\K022\Initial** folder.
2. Click the **Open layer** button in the **Map panel**.
3. Click on the **Browse** button and open **PCSWMM Exercises\K022\Initial\Soils layer Valleyfield.shp.**
4. Click on the **Soils layer Valleyfield** layer in the **Layers panel** and click the **Lock/Unlock** button and then click **Unlock layer.**

The soils layer should look like the screen capture below (you may have to move the soils layer up in the layers list or un-check the Land-Use Valleyfield.shp layer).

Examine the attributes for the soils layer in the **Table panel**.

5. Click on the **Table** tab to open the **Table panel**.

If the attributes for the **Soils layer Valleyfield** are not already being displayed, click the **Soils layer Valleyfield** layer from the **Layers panel** (should be located close to the bottom of the layers list).

Examine the given attributes and notice the attributes provided including the **UNIT, ID, SOILTYPE** and **GIS** coordinates

The infiltration type selected for the Valleyfield model was Green-Ampt. The Green-Ampt equations require three input attributes: **Suction Head** (Ψ), **Conductivity** (K) and **Initial Deficit** (WP).

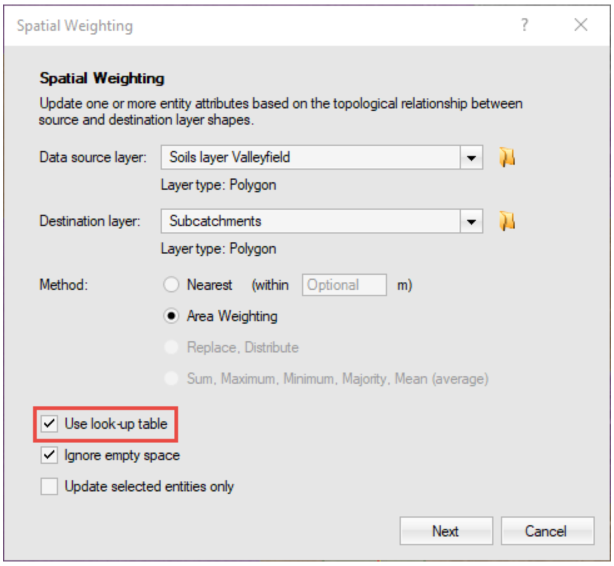
2.4 Estimating subcatchment infiltration based on the soils layer

The subcatchment Green-Ampt infiltration attributes can be estimated using the soils layer in conjunction with a lookup table. This is also done using the **Spatial Weighting** tool.

The infiltration attributes required by the subcatchment layer are the **SUCTHEAD** (Suction head)**, CONDUCT** (Conductivity) and **INITDEFICT** (initial deficit)**.** Infiltration parameter values related to soils can be found on the PCSWMM support site in the Soil characteristics and NRCS hydrologic soil group definitions reference tables.

The **Spatial Weighting** tool will match up the attributes between the soils layer and the subcatchment layer and can be used to estimate the subcatchment infiltration attributes.

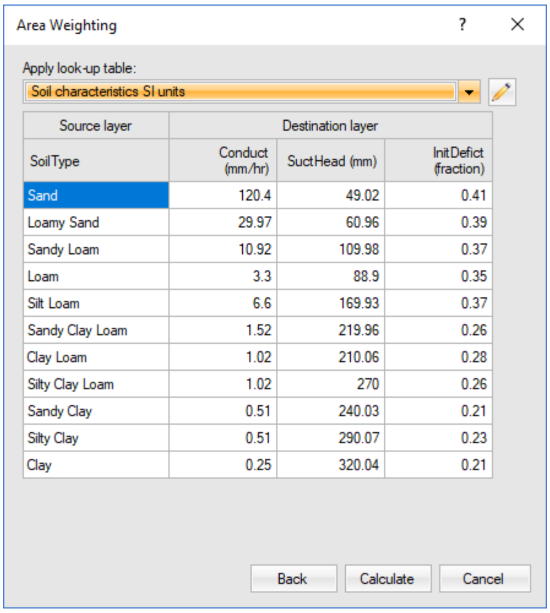
1. Open the **Map panel** and click on the **Tools**  button in the toolbar of the **Map panel** to open the **Tools Browser**.
2. Click on the **Spatial Weighting** tool, (in the Subcatchments, Nodes and Conduits sections).
3. In the **Spatial Weighting** tool, set the **Data source layer** to **Soils layer Valleyfield**, the **Destination layer** to **Subcatchments,** and change the **Method** to **Area Weighting.**
4. Place a check next to the **Use look-up table** option.



5. Click on the **Next** button.

At the top of the **Spatial Weighting** dialog there is a look-up table drop down box. Select the **Soil characteristics SI units** or **Soil characteristics US units** depending on what units you are using. The **Spatial Weighting** tool will display a table of default infiltration values for different soil types.

Note: In this example we are using a pre-defined lookup table however you can create a new one for your custom layers. For instructions on how to create a look-up table for the Spatial Weighting tool see the following article from our support site: https://support.chiwater.com/77973/area-weighting-with-a-look-up-table



Note: The look-up table uses attributes that match the destination layer (i.e. SOILTYPE, CONDUCT, SUCTHEAD and INITDEFICT). If the look-up table attribute names were not found in the destination layer an error message would have appeared in the **Spatial Weighting** dialog.

6. Click **Calculate** to perform the area weighting calculation with the selected look-up table.

The **Spatial Weighting** report shows the number of **Soils layer Valleyfield** components (polygons) intersecting each subcatchment. Also listed is the percentage of area for each component located in each subcatchment. The sum of these fractions should equal 1.0, meaning that 100% of the subcatchment area has been accounted for in the **Soils layer Valleyfield.**

7. Click the **Close** button to close the report.
8. In the **Map panel**, select a subcatchment and inspect the Green-Ampt Infiltration attributes to ensure they are reasonable.
9. Click on the **Run** button the model and check that the continuity results are reasonable.

2.5 Troubleshooting

A common problem users have with this exercise is not being able to see or select the correct look-up table from the Spatial Weighting tool. This error is the result of not selecting the option to **Merge and update look-up tables** indicated in step 1. This step is important as the look-up tables needed for this exercise are stored in the packaged model.

You can correct this by doing the following:

Close the currently opened PCSWMM model.

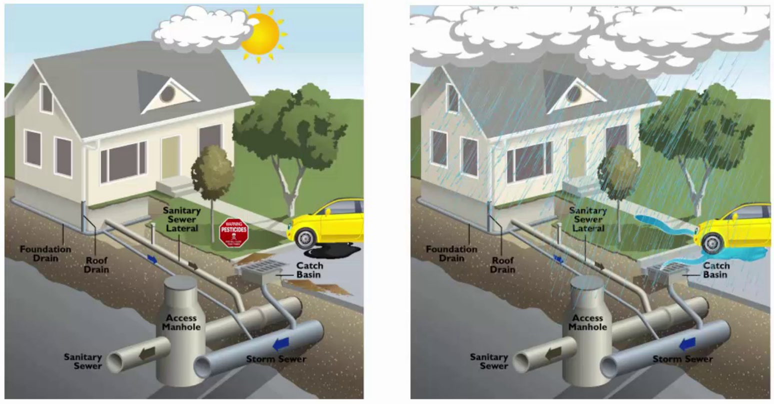
Re-open the packaged project located in PCSWMM Exercises\K022\Initial and unpackage it in a different location. This will prevent you from loosing your work on the first model. Ensure you select the option to **Merge and update look-up tables**.

Close the newly un-packaged model and re-open the original model.

You should now be able to see the look-up tables from the drop-down box.

3. Simulating water quality (Valleyfield, Quebec)

This exercise illustrates how to model the water quality aspect of an urban residential stormwater drainage system. In this example, we define the pollutants of interest, assign land uses to each subcatchment, and simulate pollutant removal from a stormwater pond.



Build Up

Wash off

3.1 Setting up a model

First, let's create a new scenario and title the current project to indicate its purpose more clearly.

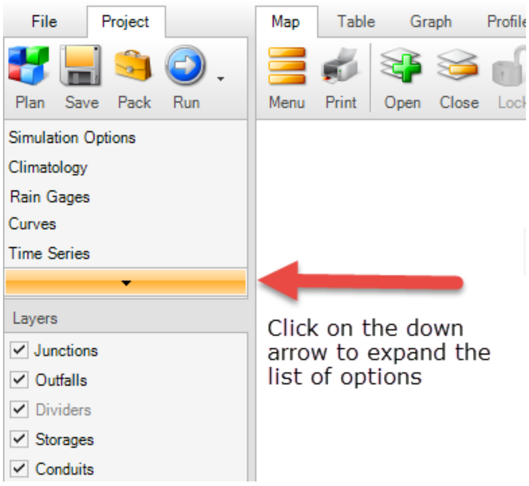
1. Open **Valleyfield to-be pond-solution.inp** from the **PCSWMM Exercises\K023\Initial** folder.
2. Launch PCSWMM and click **File > Open**.
3. Browse to **PCSWMM Exercises\K023\Initial** and select **Valleyfield to-be pond-solution.inp.**
4. Make a duplicate scenario of the current project named **Valleyfield to-be water quality** and open it.
5. Click on the **Plan** button in the toolbar of the **Project panel**.
6. In the **Scenario Manager,** click on the **Add** button and select **Duplicate Current Project.**
7. Name the project **Valleyfield to-be water quality**.
8. In the **Description (Title) box** edit the text to read **Valleyfield water quality analysis.**
9. Click the **Create** button to close the **Create Scenario** window.
10. Click on the newly created **Valleyfield to-be water quality** scenario from the **Scenario manager** and click **Open.**
11. If the background image did not automatically open, load **photo-grande-ile.jpg** from the **PCSWMM Exercises\K023\Initial** folder.
12. Click the **Open layer** button in **Map panel**.

13. Click on the **Open** button and navigate to **PCSWMM Exercises\K023\Initial**, select the raster image **photo-grande-ile.jpg** and click **Open**.

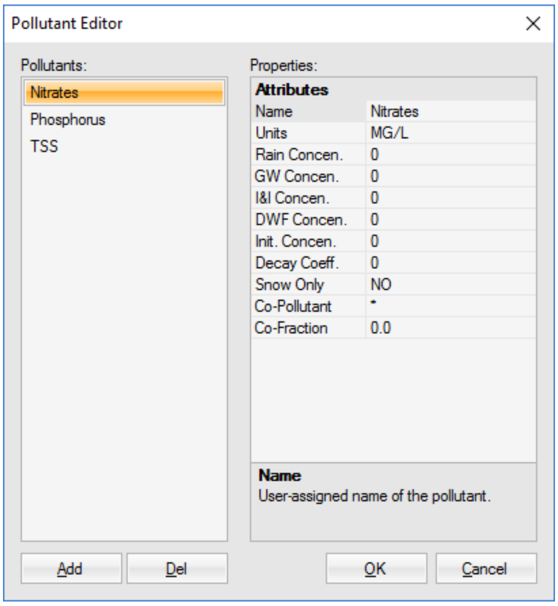
3.2 Adding pollutants to the Pollutant Editor

The pollutants to be modeled in this example are TSS, phosphorus, and nitrates.

1. In the **Project panel**, click on the downward arrow and select **Pollutants** (see screenshot provided). It will be grayed out as there are currently no pollutants defined.



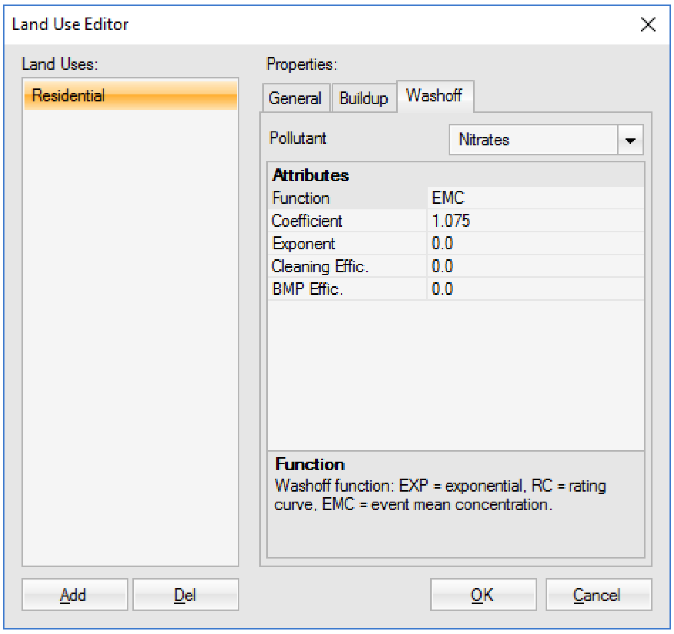
2. In the **Pollutant Editor**, click the **Add** button.
3. In the **Name** attribute type **TSS** and set the units to be **MG/L**.
4. Repeat steps 2-3 to add **Phosphorus** and **Nitrates** pollutants all in units of **MG/L**.



5. When complete click on the **OK** button to close the **Pollutant Editor**.
6. Save your changes to the model by clicking on the **Save** button in the **Project panel**.

3.3 Adding land uses to the land-use editor

1. In the **Project panel** click on **Land Uses** to open the **Land Use Editor**. It will appear grayed out as there are currently no land uses assigned to the model.
2. Click the **Add** button and type **Residential** in the **Land Use Name** attribute.
3. Switch to the **Washoff Tab** and select **Nitrates** in the pollutant drop-down menu.
4. In the **Attributes** table ensure that the **Function** is set to **EMC,** and set the **Coefficient** to **1.075**, as shown in the screenshot provided.



5. While still in the **Washoff** tab, select **Phosphorus** from the drop-down menu and set the coefficient to **0.28**. Set the **TSS** coefficient to **72**.
6. Click **Add** two times to define two new land uses (select **Yes** if prompted to save the changes).
7. Name the new land uses: **Transportation** and **OpenSpace.**
8. Enter the Washoff coefficients for these land uses using Washoff information from the table shown in the image. (Click to view the table).

Pollutant	Washoff Coefficients		
	Residential	Transportation	OpenSpace
Nitrates	1.075	1.16	0
Phosphorus	0.28	0.3	0.3
TSS	72	67	100

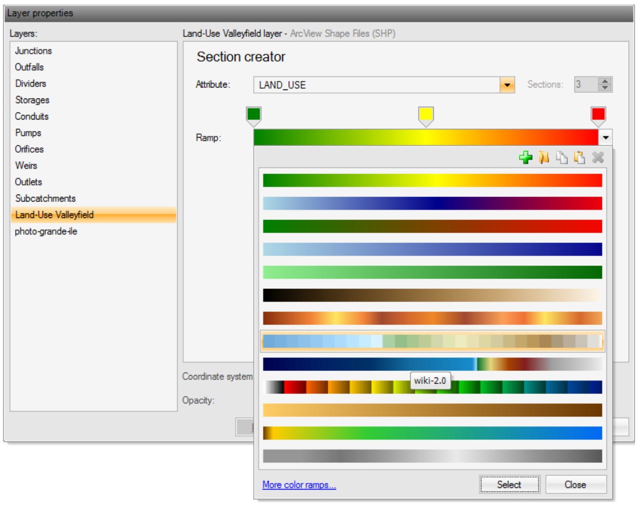
Disclaimer: The values presented were delivered by a third party and are solely provided for the purpose of this example and are not necessarily representative of conditions outside of this study area.

9. Click **OK** to close the **Land Use Editor**.

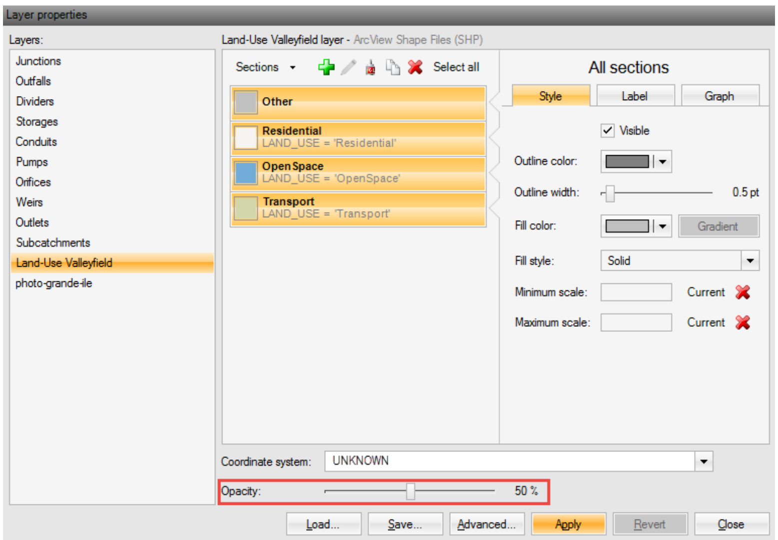
3.4 Assigning subcatchment land-uses for pollutant modelling

This section illustrates one of the many uses for the **Spatial Weighting tool**. In this case, a land-use layer will be used to determine the percentages of each land use in each subcatchment. To do this, we will edit the background land-use layer to add the attributes in preparation for the area-weighting operation.

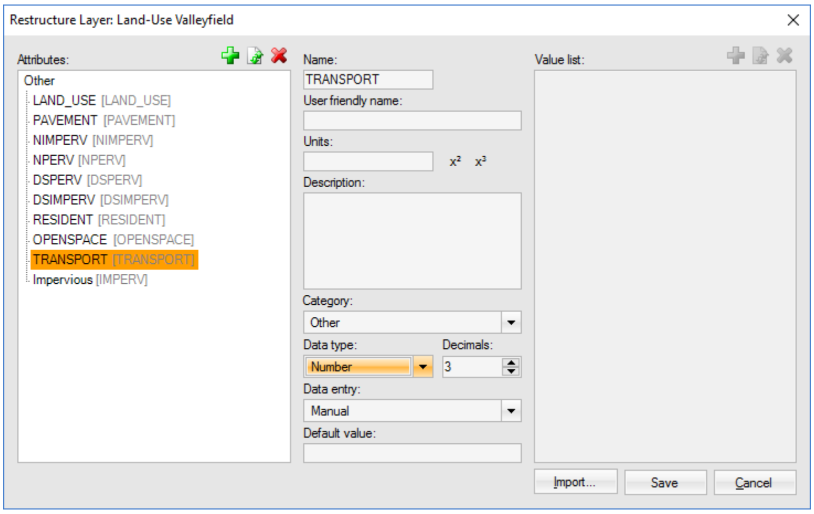
1. Open the **Land-Use Valleyfield.shp** background layer from the **PCSWMM Exercises\K023\Initial** folder.
2. Click the **Open**  button in the **Map panel**.
3. Click on the **Browse**  button and open **PCSWMM Exercises\K023\Initial** and select **Land-Use Valleyfield.shp**.
4. Click on the **Render**  button and ensure the **Land-Use Valleyfield** layer is selected.
5. Click on the **Sections** button and select **LAND_USE** as the attribute to render on.
6. Choose **wiki – 2.0** as the color ramp to render to and click on the **Select** button.



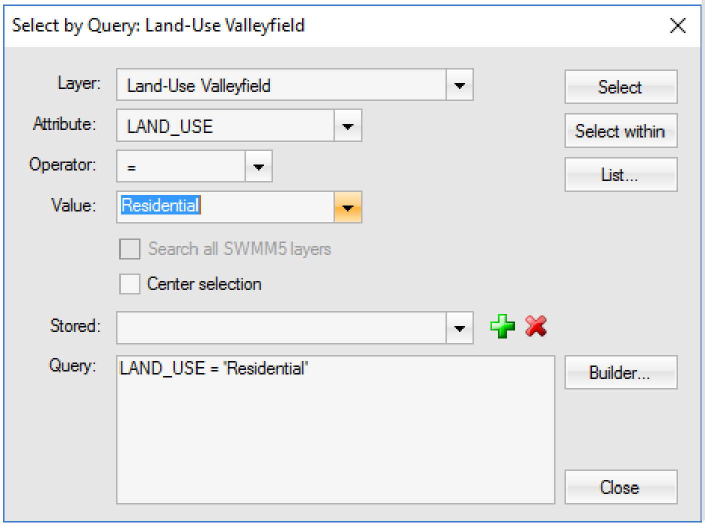
7. Put a check beside the **Random sampling** option and click on the **Create sections** button.
8. Slide the **Opacity** slider bar to around 50% and click **Apply** and **Close**.

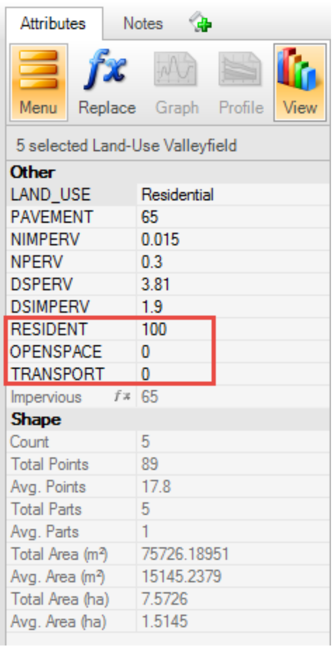


9. Select the **Land-Use Valleyfield** layer from the **Layers panel**.
10. Unlock the layer by right-clicking the layer and selecting **Unlock** .
11. Add three Number-type attributes to the land-use layer: **RESIDENT**, **OPENSPACE** and **TRANSPORT**.



12. Click the **Alter** button in the **Map panel** and select **Restructure**.
13. Click on the **Add** button and select **Attributes** from the drop-down menu
14. Under **Name** type **RESIDENT** and under **Data Type** select **Number.**
15. Add two more attributes by repeating steps 10 and 11 for **OPENSPACE** and **TRANSPORT** and define the two new attributes with the **Type** defined as **Number.**
16. Click **Save** to add the user-defined attributes to the **Land-use** layer and close the dialog.
17. Select all the land-use polygons named **Residential**, and set the **RESIDENT** attribute value to 100.





18. Click on the **Find** button in the **Map panel** and choose to **Select by Query**. Under **Layer** select **Land-Use Valleyfield**, under **Attribute** select **LAND_USE,** for **Operator** select =**,** and for **Value** select **Residential** and click **Select.**
19. Keeping the **Select by Query** dialog open, go to the **Attributes panel** and change the attribute **RESIDENT** to 100 (to represent the areas that are 100% residential area). Leave the other attributes **OPENSPACE** and **TRANSPORT** at 0.
20. Select all the land-use polygons named **OpenSpace**, and set the **OPENSPACE** attribute value to 100.

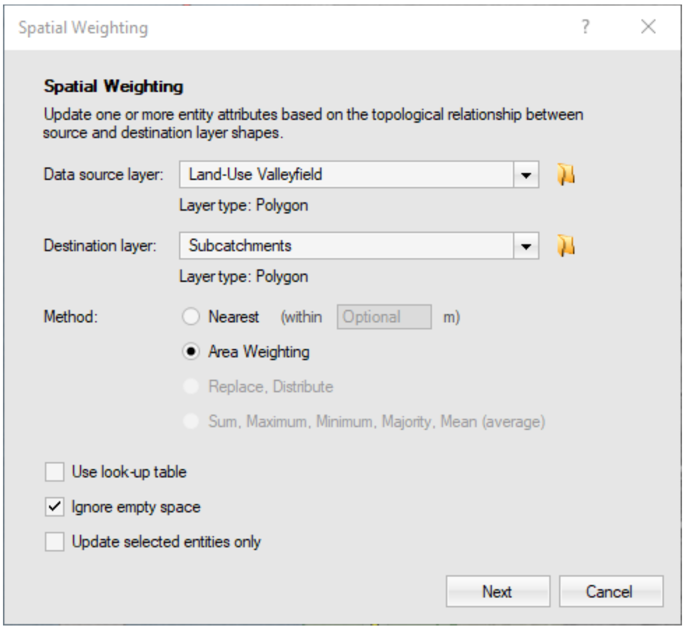
< style="display:none;" p>

Now in the **Select by Query** window, change the **Value** to **OpenSpace** and click **Select.**

21. Without closing the **Select by Query** dialog, in the attribute **Attributes panel** set **OPENSPACE** to **100**. Leave the other attributes at 0
22. Select all the land-use polygons named **Transport**, and set the **TRANSPORT** attribute value to 100.

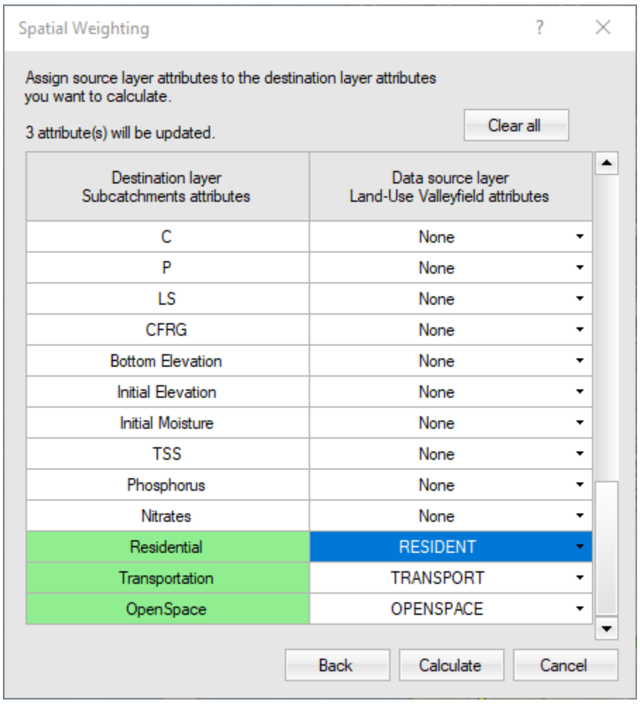
< style="display:none;" p>

23. Back in the **Select by Query** window change the **Value** to **Transport** and click **Select.**
24. You can now click **Close** to close the **Select by Query** window and, in the **Attributes panel**, set the attribute **TRANSPORT** to **100**. Leave the other attributes at 0.
25. Now we will apply the percentage of each land use to the **Subcatchment** attributes we created using area weighting.
26. Click on the **Tools** button and in the **Subcatchments** section select the **Spatial Weighting** tool.
27. In the **Spatial Weighting tool**, set the **Data source layer** to **Land-Use Valleyfield,** the **Destination layer** to **Subcatchments** and the **Method** to **Area Weighting**. Click **Next**.

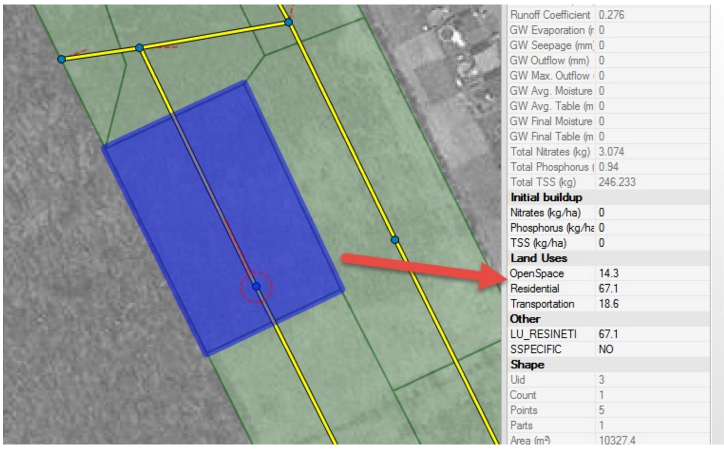


28. Click the **Clear all** button to clear the assumed matched parameters.

29. Match up the attributes **Transportation** with **TRANSPORT**, **Residential** with **RESIDENT**, and **Open Space** with **OPENSPACE** in the **Spatial Weighting** dialog.



30. Click **Calculate** to apply the area-weighted subcatchment attributes.
31. Click the **Close** button to close the summary.
32. Select **Subcatchments** in the **Layers panel**, select various subcatchments, and ensure that there are now values for the **Land Uses** attributes in the **Attributes panel**.



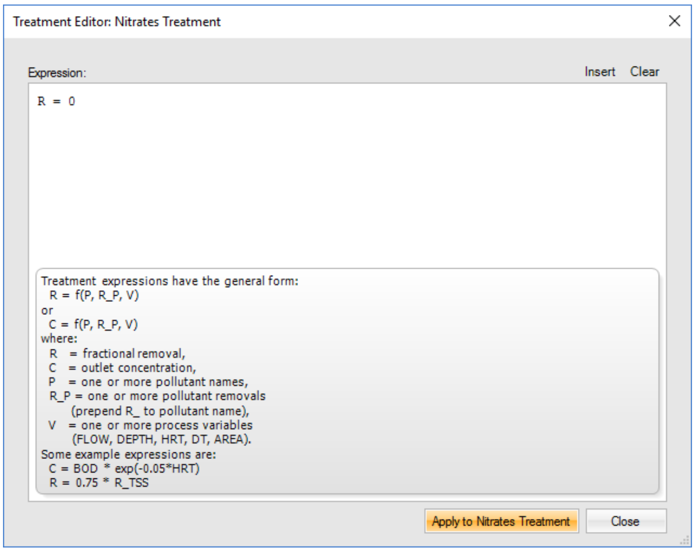
3.5 Simulating pollutant removal from pond

Pollutant removal in a storage or junction node can be modeled in PCSWMM using treatment functions entered by the user in the **Treatment Editor**. For more information please see our <u>Treatment</u> article available from our support site.

Let's assign the pond's removal efficiency for each pollutant.

1. Select the **Pond** entity in the **Map panel**.

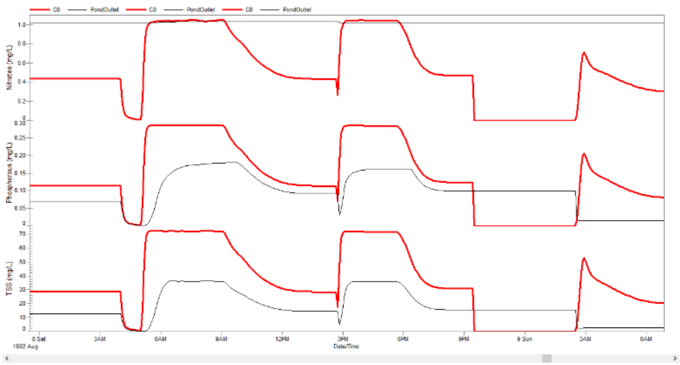
2. In the **Attributes panel,** under the **Treatment** attribute heading, click on the **ellipsis** ... button to the right of the **Nitrates Treatment** attribute to display the pond **Treatment Editor**.
3. At the bottom of the **Treatment Editor** window there is a brief explanation of how to input the pond treatment expressions.
4. In order to express the removal efficiency as an expression in the **Treatment Editor,** type in the expression **R = 0**. This means that 0% of the nitrates in the stormwater will be removed in the pond.
5. Click **Apply to Nitrates Treatment**.



6. Click on the **ellipsis** ... button next to **Phosphorus Treatment**, enter **R=0.2** to indicate that 20% of Phosphorus will be removed in the pond.
7. Click **Apply to Phosphorus Treatment**.
8. Click on the **ellipsis** ... button for **TSS Treatment**, enter **R=0.5** to indicate that 50% of TSS will be removed in the pond.
9. Once complete, click the **Apply to TSS Treatment**.
10. Run the simulation by clicking on the **Run** button in the **Project panel**.
11. Note that the project is automatically saved when a simulation is run.

3.6 Interpreting results

1. Check water quality **Continuity Errors** (runoff and routing) in the **Status Report**.
2. Switch to the **Graph panel**.
3. In the **Time Series Manager**, select **Links > Nitrates > C8** and **PondOutlet**.
4. Repeat step 3 for **Phosphorus** and **TSS** and compare the concentration differences. Zoom in on a couple of events and check to see that the concentration differences make sense.



The plot above is zoomed in to show how the concentration of pollutants decreased in the pond.

3.7 References

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

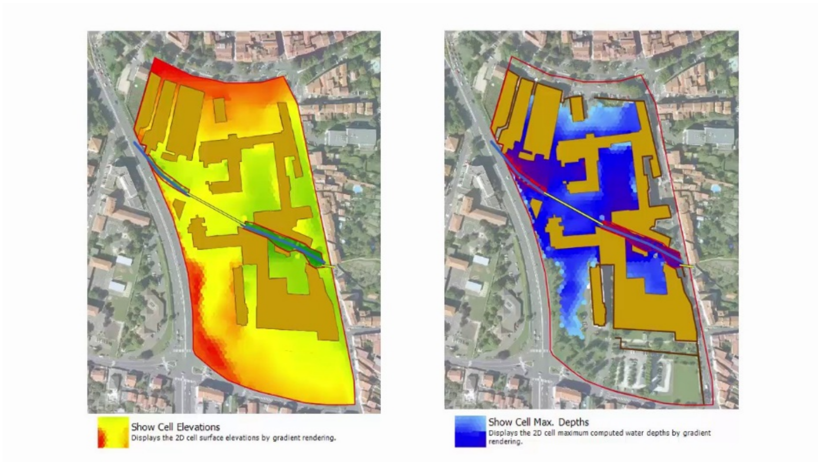
Ministry of the Environment, Ontario. 2003. Stormwater Management Planning and Design Manual.

Toronto and Region Conservation. 2012. Characterization of particle size distributions of runoff from high impervious urban catchments in the Greater Toronto Area

4. Post-processing 1D-2D urban flood analysis

This exercise reviews the post-run analysis options available in PCSWMM for 2D modeling, including plan view animation, risk map generation, video production and velocity analysis.

You will need to have Google Earth downloaded on your computer to complete this exercise. Google Earth can be downloaded at the following link: http://www.google.com/earth/.



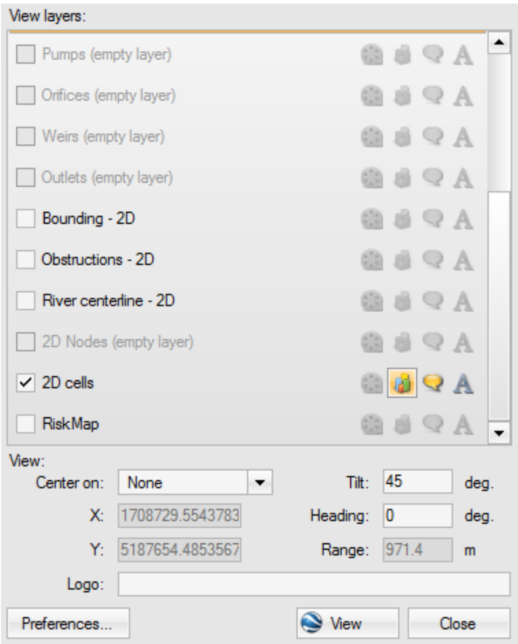
4.1 Post processing

In the previous chapter we built an integrated 1D-2D model and ran it. Now we are ready to analyze the results, starting with thematically rendering the 2D cells to indicate maximum flood extent and maximum flood depths. There are several default rendering options that can be easily applied to the 2D entities. Let's begin by rendering the 2D cells using the **Max depth** attribute of the **2D cells** layer.

1. Unpackage **2D Model.pcz** from the **PCSWMM Exercises\K002\Initial** folder (or continue with your completed 2D model from the previous chapter if you prefer).
2. Click **File > Open** .
3. Browse to the **PCSWMM Exercises\K002\Initial** folder and select **2D Model.pcz.**
4. Unpackage the model to **PCSWMM Exercises\K002\Initial\.**
5. Click **OK**.
6. Render the 2D cells to show cell maximum depths.
7. Click on the **Tools** button and select **2D modeling** from the list of tool categories.
8. Select **Render 2D network** from the **2D modeling** sub menu.
9. Select **Show Cell Max. Depths** to render the cells based on the maximum depth reached during the simulation.

Now we will project model results onto Google Earth, by rendering on the maximum water depth computed for the 2D cells. We will use the Extrude function to provide a 3D visualization of the 2D cells (this is optional). To do this we will first need to set the coordinate system.

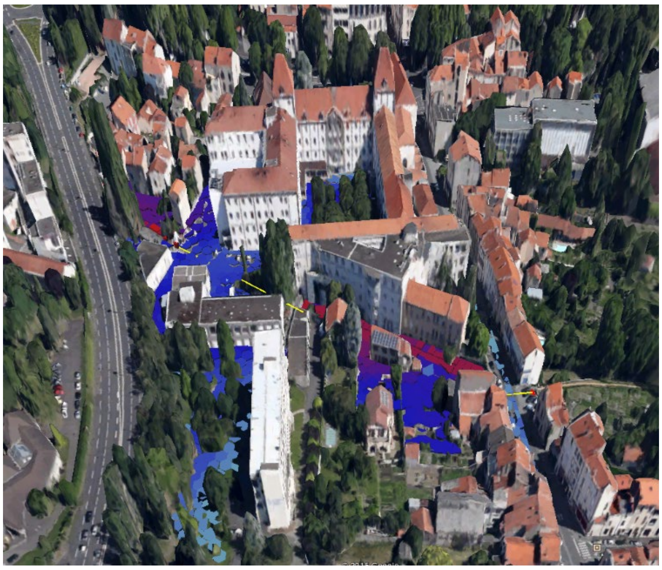
10. Set the coordinate system to **RGF93 CC46.**
11. Click on the **Menu** button in the **Map panel** and select **Coordinate system**.
12. Select **Projected system**.
13. Delete the word **All** from the filter box and type in **RGF93** into the filter and press the **Find** button.
14. Select **RGF93 CC46** from the list of options and click **OK**. A message may appear asking if you would like to make this the default coordinate system. Click on the option to **Keep existing coordinate system**.
15. Another message may appear asking if you wish to apply the coordinate system to the other layers. Click on the **Yes** button to set the other layers coordinate system.
16. Click on the down arrow beside the **Google Earth** button in the **Map panel**. If you are using a low resolution, you will not see an arrow; simply click on the button.
17. Under **View layers**, scroll down and check the **2D cells** layer.
18. Ensure this is the only layer checked (uncheck all other layers).
19. Click the **Export Extrude** button beside the **2D cells** layer.



20. Click on the **View** button to see the results in Google Earth.

In Google Earth you can select the exported model results in the Places tree list (on the left side of the Google Earth interface) and reduce the opacity with the Adjust Opacity button. You can also pan and rotate the Google Earth view as required.

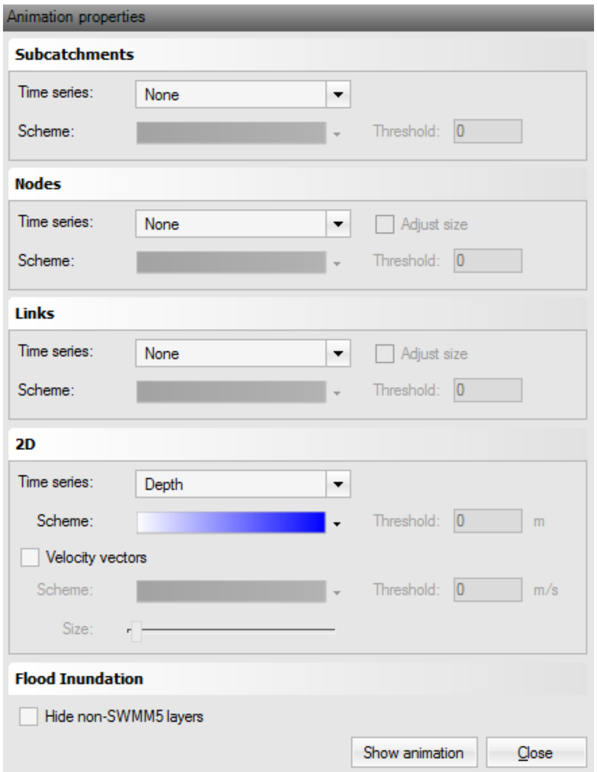
Optionally, you can return to PCSWMM and regenerate the layer in Google Earth without the extrude option. Simply click on the drop-down beside the **Google Earth** button in the **Map panel**, uncheck the **Export Extrude** button, and click on the **View** button. The existing model layer in Google Earth will be updated.



4.2 Animate 2D simulation

An advantage to modeling in 2D is the ability to animate the computed flood inundation area. We will start by dynamically rendering the map to show instantaneous computed water depths during the simulation. Please note that when you are in 2D playback mode you cannot pan, zoom or edit within the **Map panel**.

1. In the **Map panel**, click on the drop-down menu beside the **Play**  button.



2. Ensure that only the **Depth** box is checked and the **Scheme** is rendered from white to dark blue.

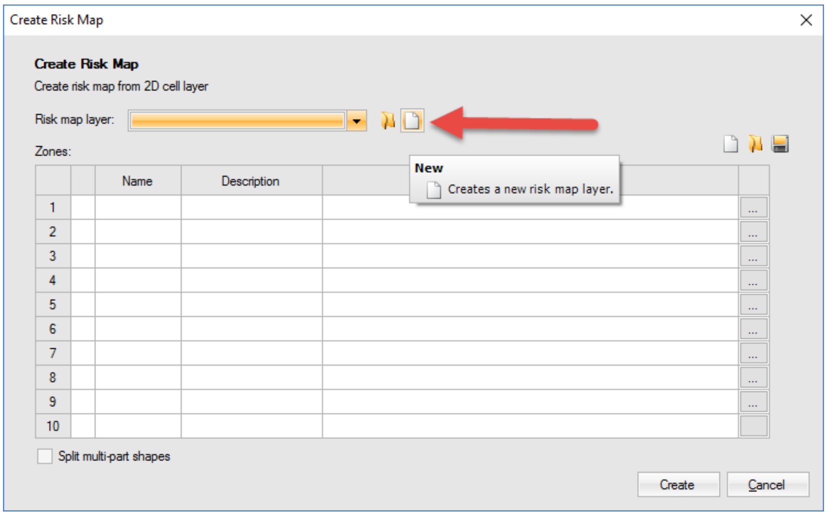
3. Click **Show animation**.
4. Click the **Play** button on the bottom of the **Map panel** to start the animation.
5. Click on the **Pause** button to stop the animation
6. You can also use the slider to quickly move back and forth through the animation.
7. Drag the slider bar forward and backwards through time.
8. Stop the slide bar at around 01/30/2012 2:30 AM.

Exit the animation view by clicking on the **Animate/Play** button. The 2D cell attributes including Depth, Volume and Time (time that the computed water depth was reached) will be updated to correspond with the time at which the animation was stopped.

4.3 Create a flood risk map

One of the 2D capabilities in PCSWMM is the ability to create contour maps and flood risk maps. Risk maps use SQL query statements and can be based on any of the attributes of the 2D cells layer (including user-defined attributes that the user creates and populates). The generated map layers can be accessed by any GIS or CAD program, as they can be saved in any of the supported GIS/CAD file formats. They can also be exported as georeferenced raster images, or viewed in Google Earth / Google Maps.

1. Click on the **Tools** button and select **2D modeling**.
2. Choose **Create Risk Map** from the 2D modeling sub menu.
3. In the **Create Risk map** window, click on the **New** button beside the **Risk map layer** drop-down menu.

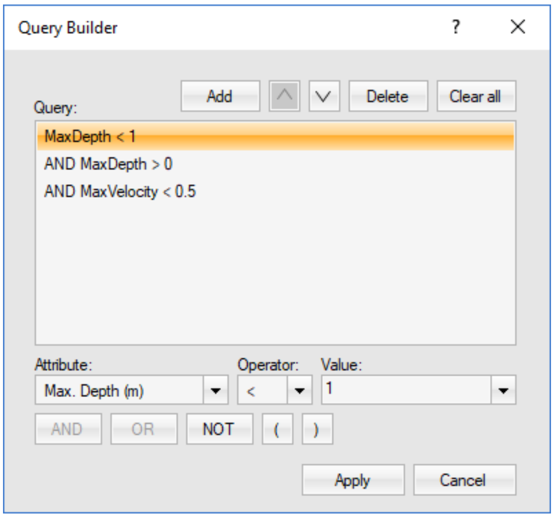


4. Navigate to **PCSWMM Exercises\K002\Initial** and save the newly created layer as a shape file using the default name **RiskMap** and click **Save**.
5. In the **Zones** table, click on the white box beside the 1 (i.e. first row, second column). This allows the user to define the colour that will be displayed for the defined zone 1. In this case we will start with yellow and progress to red.
6. Select a bright **yellow** colour and click **OK**.
7. Click in the **Name** box for **zone 1** and type in **Low**.
8. Click in the **Description** box for **zone 1** and type in **Low risk**.
9. Click in the **SQL** box for **zone 1** and click on the **ellipsis** ... button.

We will now build a query that specifies the criteria for zone 1. For this exercise we will define zone 1 to include 2D cells with a maximum depth between 0 m and 1 m and a maximum velocity less than 0.5 m/s. Note that you can simply type in the SQL query directly in the SQL box, however in this exercise we will use the **Query Builder**.

Set up a query for 2D cells with a maximum depth greater than 0 m, less than 1 m and a maximum velocity less than 0.5 m/s.

10. Click on the **Add** button to add a statement.
11. Select **Max. Depth** under the **Attribute** drop-down menu, select greater than **>** under the **Operator** drop-down menu and leave the **Value** set to **0**.
12. Click on the **Add** button to add a second statement.
13. Change the **Operator** to less than **<** and change the value to be **1**.
14. Click on the **Add** button to add to the statement.
15. Change the **Attribute** to **Max. Velocity**, the **Operator** to be less than **<** and the **Value** to be 0.5.
16. Click **Apply** to apply the query to risk zone 1.



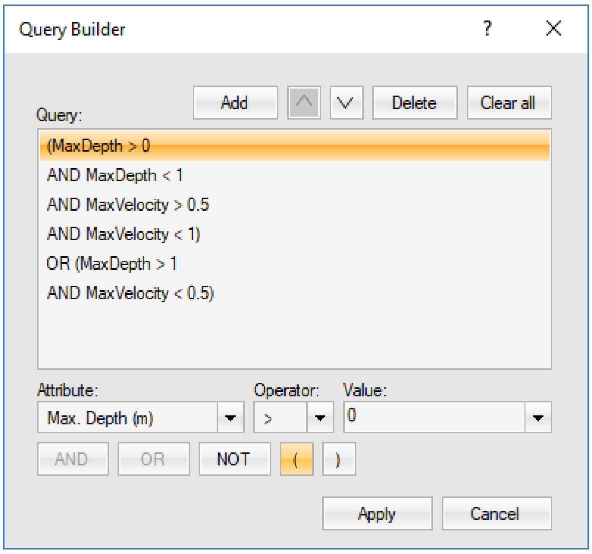
Now we will build a query for zone 2. We will define this zone to include 2 sets of criteria: cells with a maximum depth between 0 m and 1 m and a maximum velocity between 0.5 m/s and 1 m/s, and cells with a maximum depth greater than 1 m but with a maximum velocity less than 0.5 m/s.

17. In the **Zones** table click on the white box beside the 2 (i.e. second row, second column).
18. Select an **orange** color and click **OK**.
19. Click in the **Name** box for **zone 2** and type in **Medium**.
20. Click in the **Description** box for **zone 2** and type in **Medium risk**.

Now we will define our criteria for zone 2.

21. Click on the **ellipsis** ... button for **zone 2** to open the Query Builder**.**
22. Set up a query for cells with either 1) a maximum depth greater than 0 m, less than 1 m and a with maximum velocity greater than 0.5 m/s and less than 1 m/s or 2) a maximum depth greater than 1 m and a with maximum velocity less than 0.5 m/s.
23. The SQL query should read as follows...
24. **(MaxDepth > 0 AND MaxDepth < 1 AND MaxVelocit > 0.5 AND MaxVelocit < 1) OR (MaxDepth > 1 AND MaxVelocit < 0.5)**

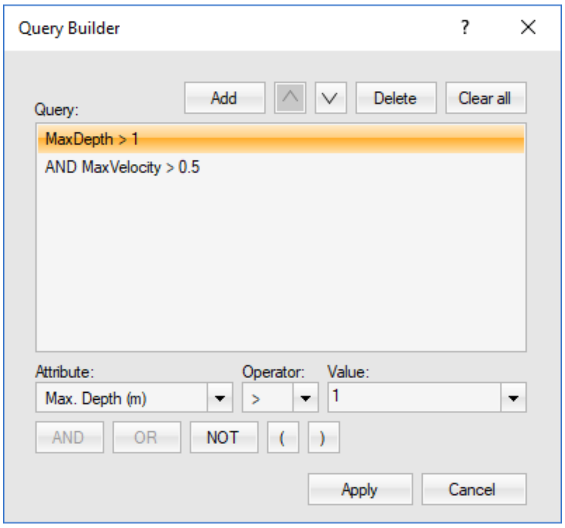
25. Click on the **Add** button to add a statement.
26. Click on the **Left bracket** (button
27. Select **Max. Depth** under the **Attribute** drop-down menu, select greater than **>** under the **Operator** drop-down menu and leave the **Value** set to **0**.
28. Click on the **Add** button to add a second statement.
29. Change the **Operator** to less than **<** and change the value to be **1**.
30. Click on the **Add** button to add to a third statement.
31. Change the **Attribute** to **Max. Velocity**, the **Operator** to be greater than **>** and the **Value** to be 0.5.
32. Click on the **Add** button to add another statement.
33. Change the **Operator** to be less than **<** and the **Value** to be **1**.
34. Click on the **Right bracket**) button to complete the first criterion.
35. Click on the **Add** button to add another statement.
36. Click on the **OR** button.
37. Click on the **Left bracket** (button.
38. Change the **Attribute** to be **Max. Depth**.
39. Change the **Operator** to be greater than **>** and the value to be **1**.
40. Click on the **Add** button to add another statement.
41. Change the **Attribute** to be **Max. Velocity**.
42. Change the **Operator** to be less than **<** and the value to be **0.5**.
43. Click on the **Right bracket**) button to complete the second criterion.
44. Click **Apply** to apply the query to risk zone 2.



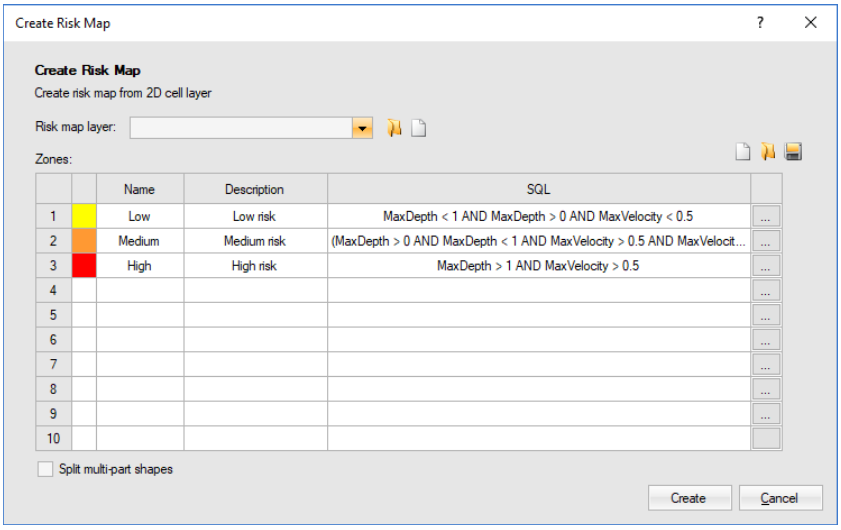
Now we will build the query for the final (third) zone. This zone represents the highest risk and includes areas where the maximum depth is greater than 1 m and the maximum velocity is greater than 0.5 m/s.

45. In the **Zones** table click on the white box beside the 3 (i.e. third row, second column).
46. Select a red color and click **OK**.
47. Click in the **Name** box for **zone 3** and type in **High**.
48. Click in the **Description** box for **zone 3** and type in **High risk**.
49. Click on the **ellipsis** ... button for z**one 3** to open the Query Builder.

50. Set up a query for cells with a maximum depth greater than 1 m and a maximum velocity greater than 0.5 m/s. The SQL query should read as follows: **MaxDepth > 1 AND MaxVelocit > 0.5**
51. Click on the **Add** button to add a statement.
52. Select **Max. Depth** under the **Attribute** drop-down menu, select greater than **>** under the **Operator** drop-down menu and change the **Value** set to **1**.
53. Click on the **Add** button to add a second statement.
54. Change the **Attribute** to **Max. Velocity**, the **Operator** to be **>** and the **Value** to be **0.5**.

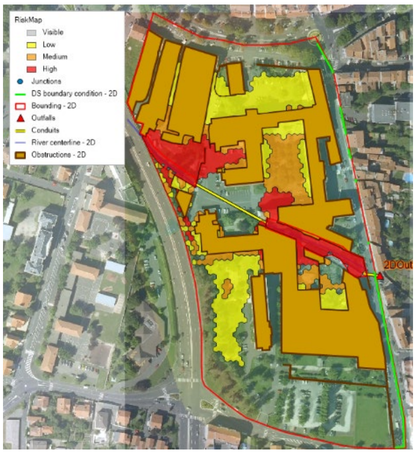


55. Click **Apply** to apply the query to risk zone 3.



56. Click on the **Create** button to generate the risk map layer and exit the **Create Risk Map** tool.

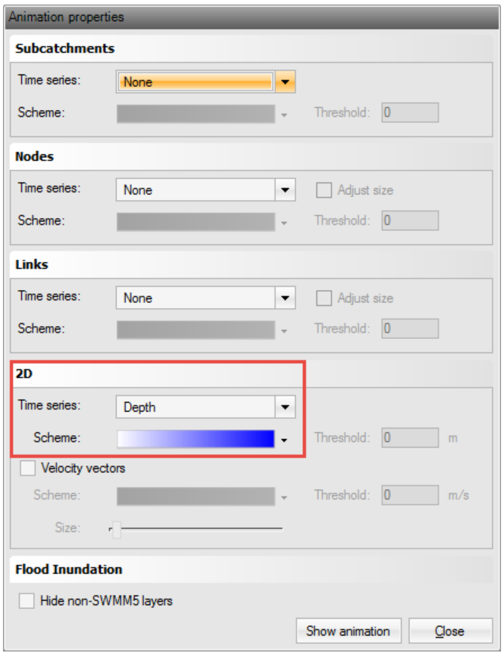
In the Layers panel, uncheck the **2D cells** layer (to hide it) and in the **Map panel**, select the **Menu** button and then check **Show Legend**. Observe the extent of each of the risk zones in the new risk map layer, and note that this may also be viewed in Google Earth as described above.



4.4 Create a video

Now we will record a video showing the water depth. Videos can be created from either plan view animations in the **Map panel**, or profile animations in the **Profile panel**. Video creation is part of the PCSWMM Professional version and can be used for normal 1D model animations as well as the 2D animations we will be recording here. To record a video, simply click on the **Record** button when in animate/playback mode in either panel.

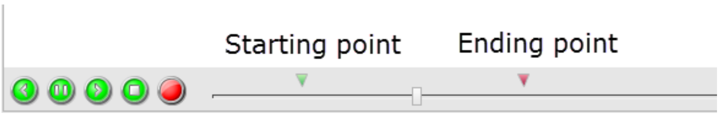
1. Turn off the **RiskMap** layer from the **Layers panel** (i.e. uncheck it).
2. Render the 2D network to hide all 2D entities.
3. Click on the **Tools** button and select **2D modeling.**
4. Under **2D modeling** select **Render 2D network.**
5. Click on **Hide All** to hide the 2D junctions, conduits and outfalls.
6. Zoom in on the **Map** to show the extent of the area you wish to include in the video.
7. Click on the drop-down menu beside the **Animate/Play** button.
8. Ensure that only the **Depth** box is checked and the **Scheme** is rendered from white to dark blue.



9. Click the **Show animation** button.

Now let's choose the start and ending times for the video. Note that this step is optional and you can record a video of any length, including the full simulation period.

10. Use the playback scroll bar to find a suitable starting point for the video.
11. Move the **Start marker** (green arrow) to the selected starting point (see screenshot).



Again, use the playback scroll bar to find a suitable ending point for the video.

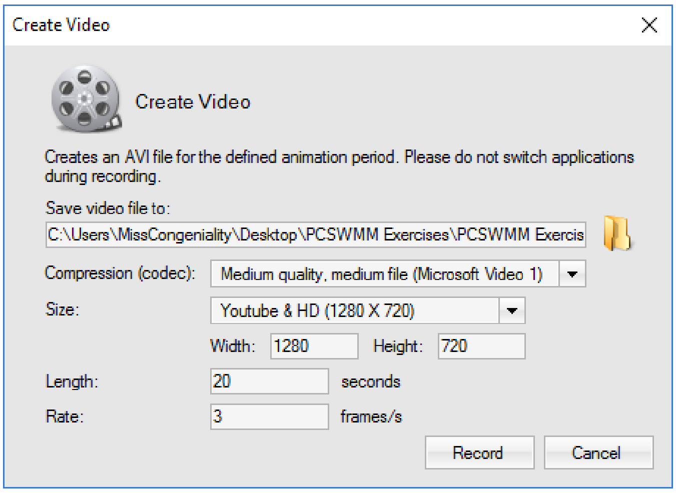
12. Move the **End** marker (red arrow) to the chosen stopping point.
13. Click on the **Stop** button to move the scroll bar to the **Start** marker position. Note the Stop button will be grayed out if the scroll bar is already at the starting point.

Now we are ready to record a video showing the animated water depth for the selected time period.

During the recording session, the **Map panel** will playback the simulation and record the screen. The **Create Video** tool allows you to specify the video quality, the video speed (i.e. duration) and the size of the video. PCSWMM provides a list of standard video sizes such as Full HD (1080p), HD (720p), XGA, etc. and also provides support for any custom video resolution. However, the video resolution cannot be greater than the screen resolution. Thus we recommend a high resolution monitor for recording Full HD or greater resolution videos.

Let's setup and record the video:

14. Click the **Record** button on the bottom of the **Map panel**.
15. In the **Create Video** dialog click on the folder button to browse.



16. Browse to **PCSWMM Exercises\K002\Initial,** name the video **2D video** and click the **Save** button to return to the **Create Video** tool.
17. Set the **Compression** (i.e. video quality) to **Medium quality, medium file**.
18. Set the **Size** to **Youtube & HD (1280 X 720)** if you can. If your screen resolution doesn't allow this, use the default custom size (i.e. the current **Map panel** resolution) or choose a smaller size from the list.

19. Set the **Length** to 20 seconds to speed up the playback. Note that the Rate (in frames per second) adjusts accordingly.
20. Click on the **Record** button. The recorder will automatically stop recording once the simulation duration is complete.

Once complete a message will appear asking if you wish to view the video (e.g., using Windows Media Player) or show the file location. Choose to **View video**.

Once you have finished watching the video, return to PCSWMM and save your model.

4.5 Animate with velocity vectors

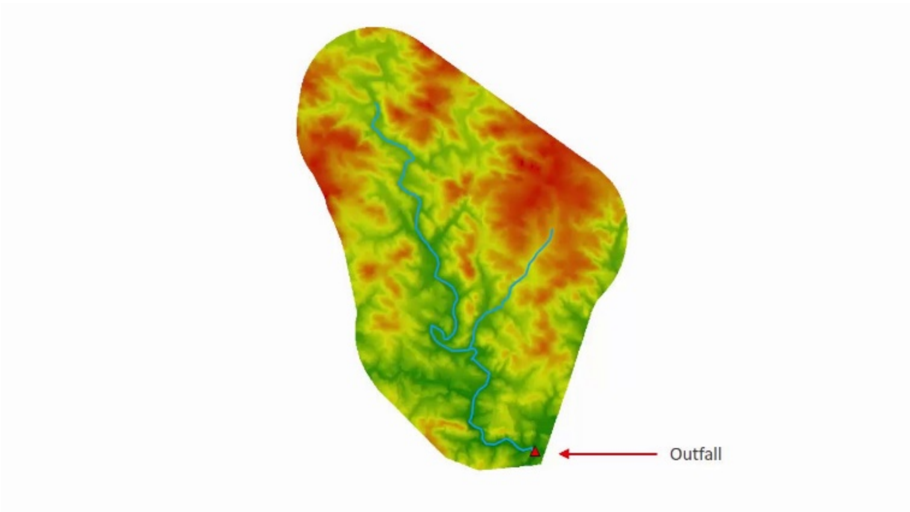
1. Click on the drop-down menu beside the **Animate/Play** button.
2. Check the **Velocity vectors** check box.
3. Click the **Update animation** button.

Animate your results and observe the velocity vectors.

Note that to zoom into a specific location for a detailed view of the velocity vectors, you will need to temporarily exit the animation mode. Also note that you can control the size of the velocity vectors in the Animation/Play pop-up editor.

5. Automatic watershed delineation with a DEM

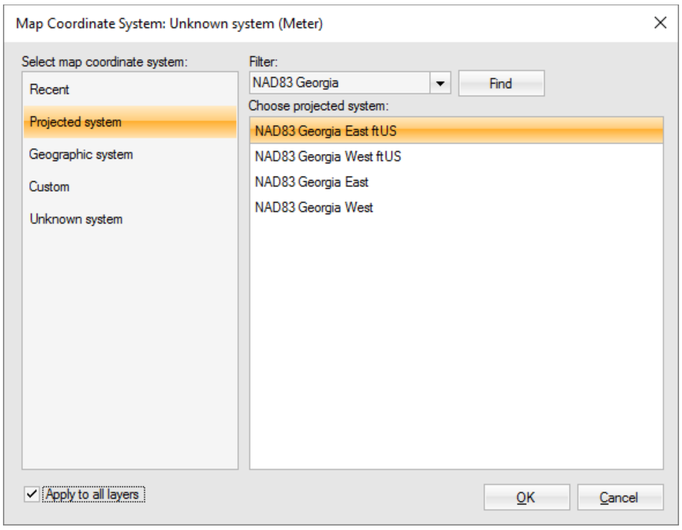
This exercise will demonstrate the creation of a DEM-based open channel model in an urban setting. For simplicity, only watercourses will be represented in the model, other hydraulic structures such as bridges, culverts, and pipes will not be included. The watershed delineation tool will be used to generate the SWMM5 layers for this model. Note that this model is for instructional purposes; it is fictional and therefore only suitable for use in this exercise.



5.1 Create the project and load the DEM and burn-in streams

1. Create a new project called **YellowRiver** in the **PCSWMM Exercises\K1190\Initial** folder.
2. Launch PCSWMM.
3. Click the **File** tab then click the **New** button.
4. Choose to create a new **SWMM5 Project.**
5. Name the project **YellowRiver.**
6. Browse to the folder **PCSWMM Exercises\K1190\Initial** as the location to save the project.
7. Click the **Create Project** button.
8. Set the **Map coordinate system** to **NAD83 Georgia East ft US** and **Apply to all layers**.
9. In the **Map panel**, click on the **Menu** button and select **Coordinate system**.
10. Click on **Projected system**.

11. Under **Filter** type in "**NAD83 Georgia**" and press the **Find** button.

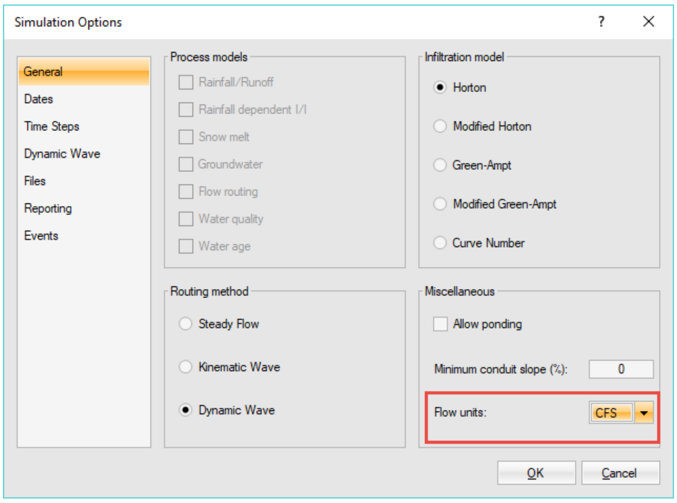


12. Choose **NAD83 Georgia East ft US.**
13. Check **Apply to all layers.**
14. Click **OK**.

Since this exercise is located in the U.S., we will set up the project to use U.S. units.

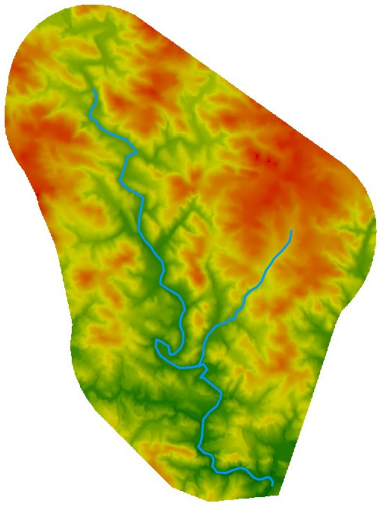
Set the flow units to **CFS**. If you are working in the United States you can skip this step as your flow units should automatically be set to CFS.

15. Click on **Simulation Options** from the **Project panel**.
16. In the **Simulation Options** dialog, under the **General** tab, change the **Flow units** to **CFS**.



17. Click on the **OK** button.
18. Select the **Switch** button when asked.
19. A burn-in streams layer and DEM will be loaded for use with the watershed delineation tool and flood inundation analysis.
20. Open the **Burn-in Streams.shp** and **ClippedDEM.FLT** layers from the **PCSWMM Exercises\K1190\Initial** folder.
21. Click on the **Open layer**  button and press the **Open**  button in the top right corner.

22. Navigate to **PCSWMM Exercises\K1190\Initial.**
23. Hold down the **Ctrl** key and select **Burn-in Streams.shp** and **ClippedDEM.FLT** and click **Open**.

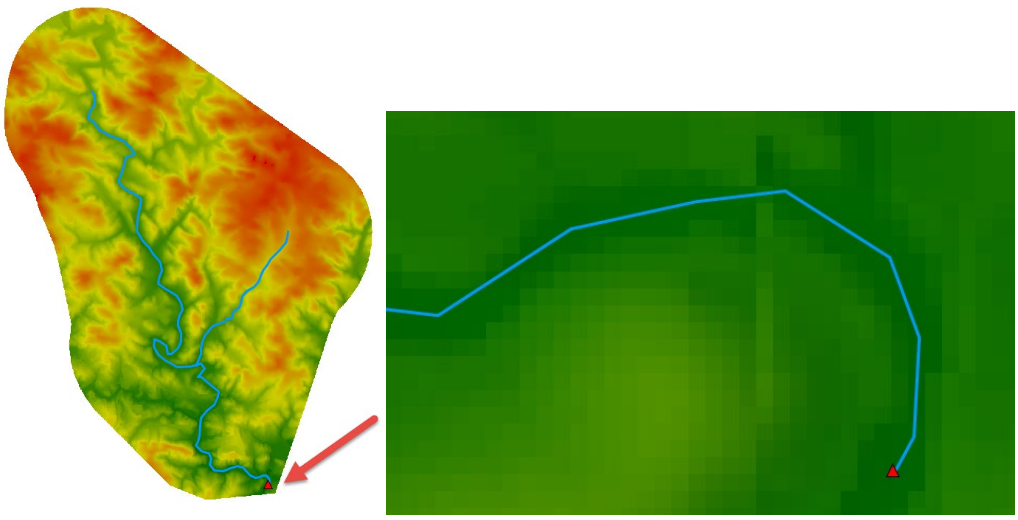


5.2 Add an outfall

First, an outfall must be placed to indicate to the tool the most downstream point for the hydrology/hydraulics model.

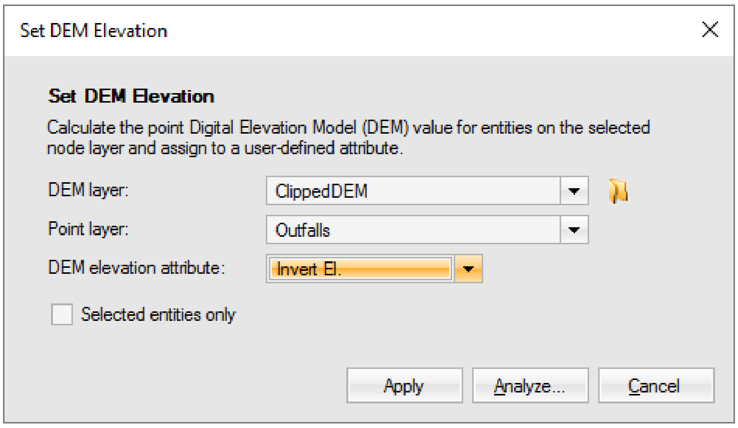
In the **Layers panel**, click on the **Outfalls layer.** It will be grayed out to indicate the layer is currently empty.

1. Click on the **Add** button and add an outfall at the southern end of the "Burn-in Streams" line layer. See the screenshot for reference. Note that it is important the outfall is located in the darker green channel portion of the DEM as shown (i.e., the elevation should be in the range of 216-217 ft, as indicated by the Z value in the bottom status bar).



2. Click on the **Select** button to exit add mode.

3. Now the invert elevation of the outfall can be updated from the DEM.
4. Click on the **Tools** button.
5. In the **Nodes** section, choose E**levation from DEM**.
6. Select the DEM layer to be **ClippedDEM**.
7. Choose the Point layer to be **Outfalls**.
8. Choose the DEM elevation attribute to be **Invert Elev**.



9. Click **Apply** then **OK** when a report appears. The outfall invert elevation will be updated to match the DEM (approximately 216-217 ft, depending on where the outfall is located).

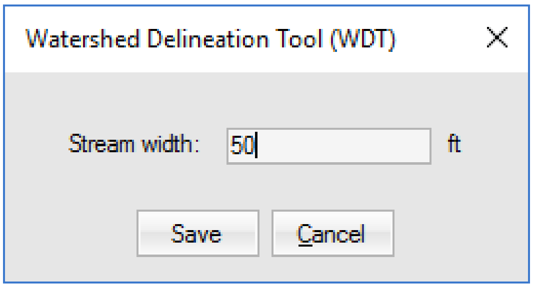
5.3 Automatically generate hydrology and hydraulic model parameters

Now the junctions, conduits, and subcatchments will be created for the model using the **Watershed Delineation Tool** (WDT) in PCSWMM.

1. Click on the **Tools** button.
2. In the **Subcatchments** section, choose **Watershed Delineation**.
3. In the **WDT** dialog, ensure the **DEM layer** has been set as **ClippedDEM.**

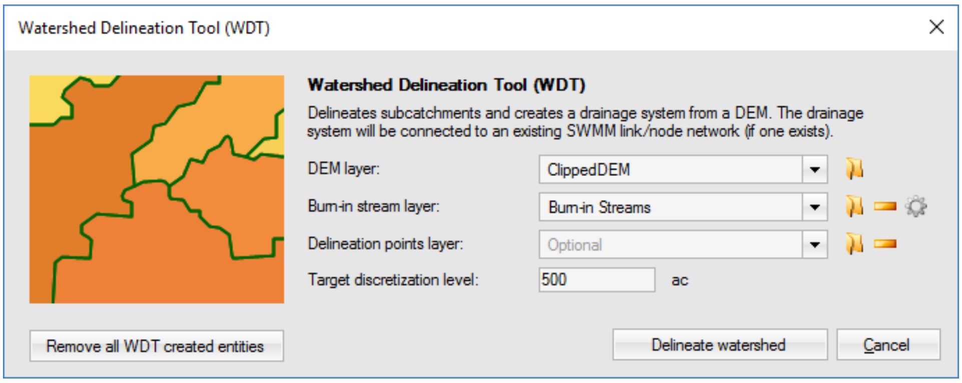
A burn-in stream layer specifies the line layer to be used for delineating flow paths. The burn-in streams option carves a trench in the original DEM along the specified line layer and removes any obstructions in the flow such as bridges or dams.

4. Choose the **Burn-in stream layer** to be the **Burn-in Streams** layer from the drop-down menu.
5. Click on the **Advanced** button and set the stream width to **50** feet.



6. Click **Save**.

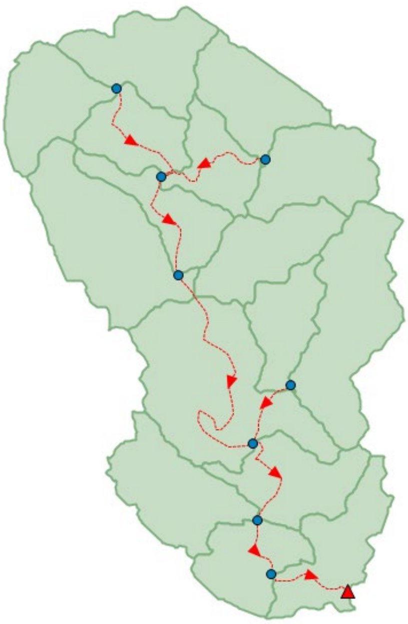
7. Change the **Target discretization level** to **500** acres. This represents the desired average area for the created subcatchments. This coarse resolution was chosen for illustration purposes, normally a higher resolution would be used.



8. Click the **Delineate watershed** button to create the SWMM5 layers for the model.
9. It may take a minute for PCSWMM to create the entities. When the operation is complete, a summary report will appear. Click **Close** to close the report.

Note: If the WDT tool doesn't generate any entities (0 subcatchments, 0 conduits, etc.), check that the coordinate system for all the layers and map is set to NAD83 Georgia ftUS under **Menu** > Coordinate systems. Additionally, check that the outfall is located in the dark green portion of the DEM. Re-try using the WDT tool without the Burn-in Streams layer selected.

10. The model will appear in the **Map panel**. In the **Layers panel**, uncheck the **WDT Flow Paths**, **ClippedDEM** and **Burn-in Streams** layer to view the model. It may appear slightly different than the one shown in the image.



11. Click on a subcatchment entity and in the **Attributes panel,** view the different attribute values. Using the WDT, the subcatchment area, slope, width, flow length, and outlet attribute values have all been automatically assigned.

Elevations for junctions and conduit inlet/outlets have also been assigned based on the DEM. Conduit lengths have been calculated from the coordinate system and GIS properties.

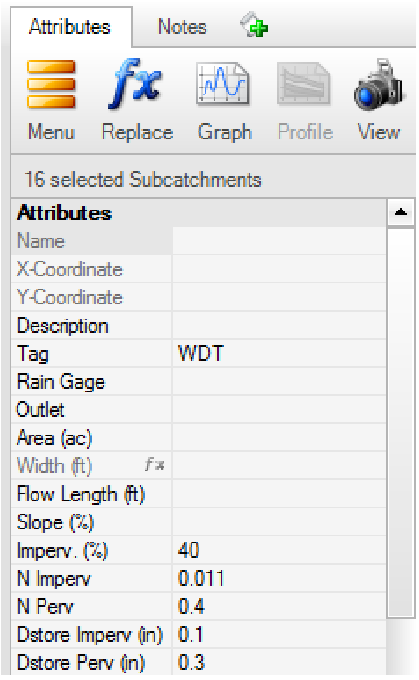
Note: The WDT generates several different layers that will be visible in the **Layers panel**. Turn on and off these layers to view the different components that are used to generate the watershed hydrology and hydraulic parameters. For more information on these layers and how the tool creates them, please see the **Watershed Delineation** article on the PCSWMM support site.

Other attributes such as roughness, depression storage, percent impervious and initial depths must still be specified by the user as the WDT does not automatically generate these parameter values (SWMM5 default values are used).

In this example, values from the PCSWMM support site Reference Tables will be used to parameterize the subcatchment values. For simplicity and to save time in this exercise, the same values will be applied for all the subcatchments. For the Dstore Imperv and Dstore Perv values, the Depression storage article was used to determine values for the depression storage based on an assumed land use from satellite imagery (Bing Map satellite tiles). The satellite imagery was also used to determine the ground cover for assigning the N Imperv and N Perv values with the aid of the Manning's N - Overland flow article. Percent impervious was also estimated based on the satellite imagery.

In a real model, a land-use and soils layer would be used to determine the exact locations of soils and land use to accurately assign these values. The values can then be weighted using the soil and land use polygons with the **Area weighting** tool.

12. Click on the **Subcatchments** layer in the **Layers panel**.
13. Press the **Ctrl+A** keys on the keyboard to select all of the entities.
14. In the **Attributes panel**, enter in the following values for the attributes.



Conduit roughness values will also be assigned based on the estimated roughness of the Yellow River channel. Roughness value for natural channels can be obtained from the Manning's N - Open channels article on the PCSWMM support site.

15. Click on the **Conduits** layer in the **Layers panel**.
16. Press the **Ctrl+A** keys on the keyboard to select all of the entities.
17. In the **Attributes panel**, enter in a value of 0.05 for **Roughness**.

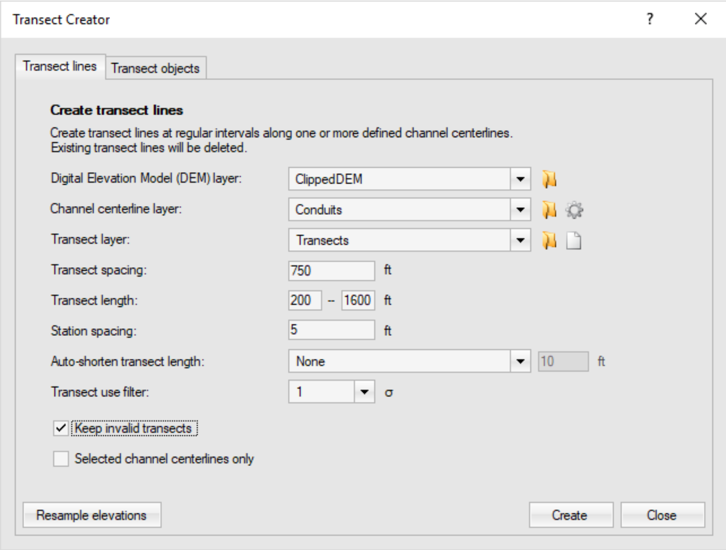
5.4 Create Conduit Cross-Sections

Now that the initial hydraulic model has been generated, the Transect Creator tool can be used to quickly create and assign conduit transects. Conduits are assigned a default cross-section that is circular with a set diameter. The Transect Creator offers a fast and accurate way to assign channel cross-sections to conduits based on the DEM.

1. Click on the **Tools** button, select **Conduits**, then select **Transect Creator**.
2. Click the **Transect lines** tab.
3. Ensure **ClippedDEM** has been identified as the **DEM layer** in the first drop down box.
4. Set the **Channel centerline layer** to be **Conduits**.
5. Click on the **Advanced** button beside the **Channel centerline layer** drop-down box.

The advanced options allows a user to apply smoothing for the centerline. Since the lines are initially drawn perpendicular to the flow path line, straightening this line can result in improved transects, especially in the case of meandering rivers. The advanced options also allow a user to specify the number of flow lines that will be used to draw the full transect. These lines are used to better represent the river shape and flood plains, by following the river centerline and drawing the transect to the closest vertex on each flow line. PCSWMM also allows the flow lines to be straightened, for better representation of the floodplains. For this exercise we will use the default advanced options.

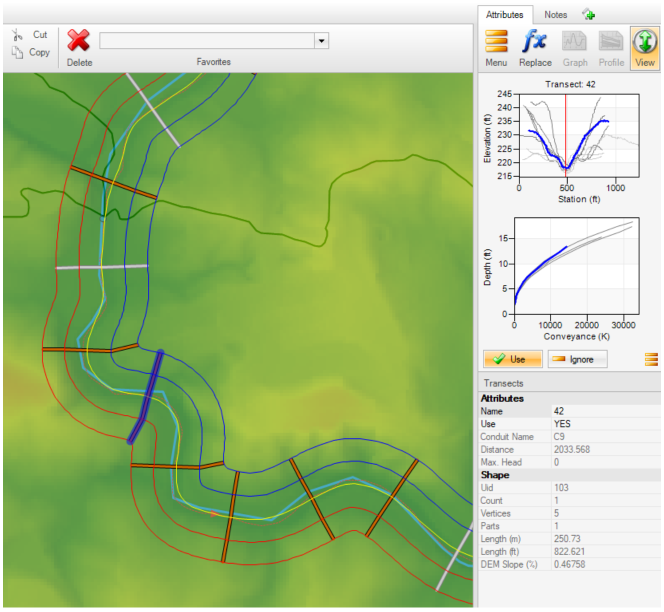
6. Click on the **OK** button to close the advanced options.
7. Beside the **Transect layer** drop-down menu click on the **New** button.
8. Navigate to **PCSWMM Exercises\K1190\Initial** and type **Transects** as the **File name.**
9. Click on the **Save** button.
10. Beside **Transect spacing** enter a value of **750 ft**.
11. Beside the **Transect length** enter a range of **200** - **1600** ft**.**
12. Beside the **Station spacing** box enter a value of **5** ft.
13. For **Auto-shorten transect length** select the **None** option.
14. Set the **Transect use filter** to **1**.
15. Toggle on the checkbox to **Keep invalid transects**.



16. Click **Create**. If a warning that "-- **invalid floodplain transect lines were found**" appears, click **Yes** to delete them.
17. An Error Report will appear. Click **Close** followed by **Close** to close the **Transect editor**.
18. Transects are based on the DEM and grouped according to the nearest conduit. These will be automatically assigned and added to the **Map panel**.

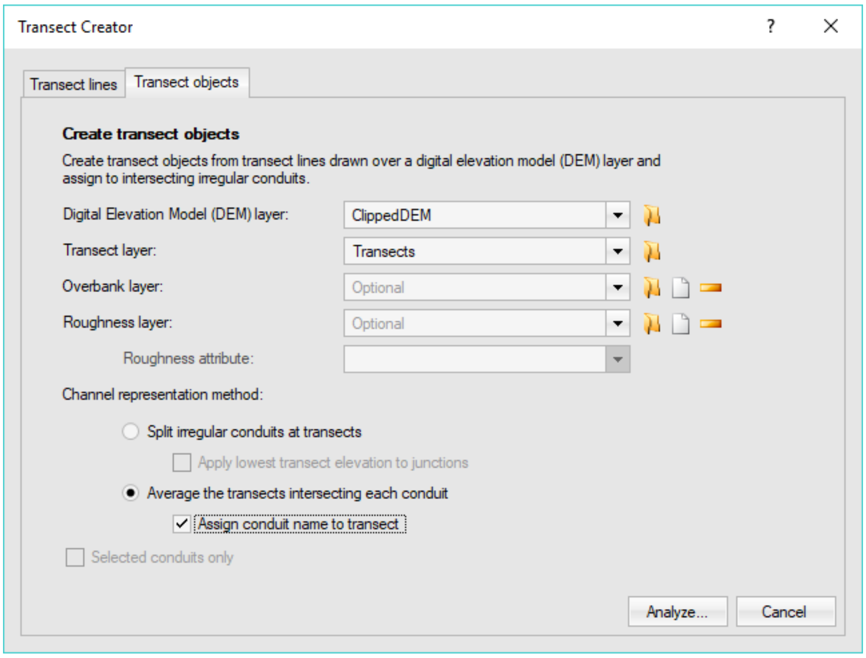
Note: Transects can also be manually drawn using the **Ruler** tool. The transect data can then be copied from the **Attributes panel Cross-section** and pasted to the **Transect Editor** as station elevation data to create transect objects for irregular-type conduits.

19. Click on a transect to view the elevation profile as shown in the **Attributes panel**. Transects are drawn from left to right facing downstream. If the transect isn't shown, click on the **View** button and choose **Cross-Section**. Transects shown in the screenshot may not exactly match those in your model.

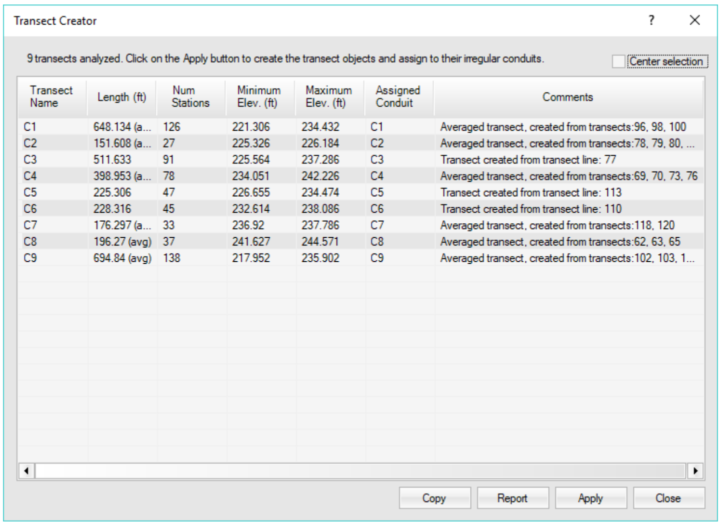


Now Transect Objects can be created. This adds the station and elevation data from each transect line to the input file and allows transects to be assigned as cross-sections for irregular type conduits.

20. Click on the **Tools** button, select **Conduits**, then select **Transect Creator**.
21. Click the **Transect objects** tab.
22. Ensure the **ClippedDEM** has been identified as the DEM layer in the first drop-down box.
23. Beside the **Transect layer** drop-down select the **Transects** layer that was created above.
24. Leave the **Overbank layer** and **Roughness layer** blank.
25. For the **Channel representation method**, select to **Average the transects intersecting each conduit**.
26. Check the option to **Assign conduit name to transect**.



27. Click **Analyze**.
28. If the Transect Creator dialog indicates that "**One or more intersecting conduits are not irregular type conduits"** (i.e. the default conduit type is circular shape), select **Change conduit type**.
29. A Transect Creator report will appear with the details of the analysis. Click **Apply** then **Close**. The transects will now appear in the **Transects Editor** which can be accessed from the **Project panel**.



Transect Name	Length (ft)	Num Stations	Minimum Elev. (ft)	Maximum Elev. (ft)	Assigned Conduit	Comments
C1	648.134 (a...	126	221.306	234.432	C1	Averaged transect, created from transects:96, 98, 100
C2	151.608 (a...	27	225.326	226.184	C2	Averaged transect, created from transects:78, 79, 80, ...
C3	511.633	91	225.564	237.286	C3	Transect created from transect line: 77
C4	398.953 (a...	78	234.051	242.226	C4	Averaged transect, created from transects:69, 70, 73, 76
C5	225.306	47	226.655	234.474	C5	Transect created from transect line: 113
C6	228.316	45	232.614	238.086	C6	Transect created from transect line: 110
C7	176.297 (a...	33	236.92	237.786	C7	Averaged transect, created from transects:118, 120
C8	196.27 (avg)	37	241.627	244.571	C8	Averaged transect, created from transects:62, 63, 65
C9	694.84 (avg)	138	217.952	235.902	C9	Averaged transect, created from transects:102, 103, 1...

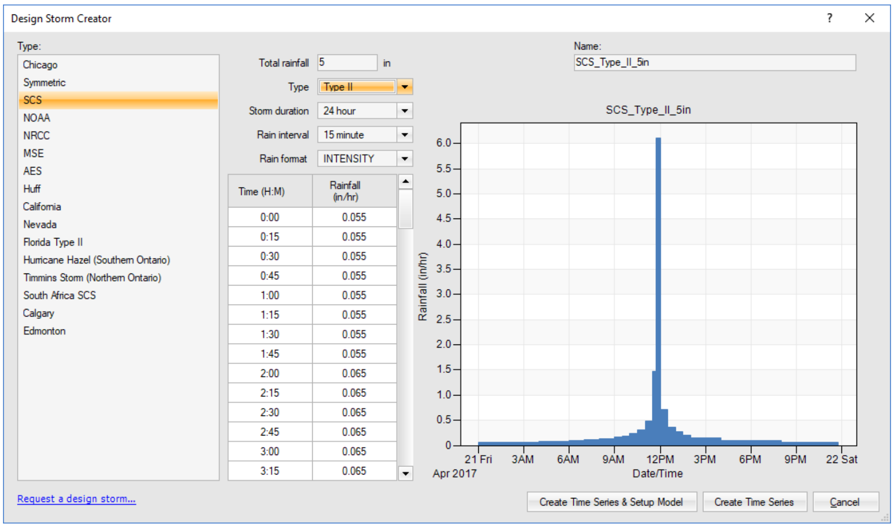
30. In the **Map panel** select the **conduits** layer and click **Ctrl** + **A** to select all the conduits in the layer. Check all the conduits have been assigned **IRREGULAR** cross-section.

Note: To assign different roughness coefficients to portions of the transects, you can define overbank stations in the **Transect editor** (accessed through the **Project panel**). For this exercise, a single roughness value will be used with no defined bank stations.

5.5 Assign a rain gage to subcatchments

To run the model, subcatchments must have a rain gage assigned to them.

1. Click on the **Graph** tab to open the **Graph panel**.
2. In the **Graph panel**, click on the **Add** button to open the **Design Storm Creator**.
3. In the **Design Storm Creator** dialog, choose an **SCS** storm.
4. Set the **Total rainfall** to **5 inches**.
5. Change the type to **Type II.**
6. Set the **Storm duration** to **24 hour**.
7. Set the **Rain interval** to **15 minutes**.
8. Choose the **Rain Format** to be **Intensity**.



9. Click the button to **Create Time Series & Setup Model**. This option assigns the rain gage to all the subcatchments and sets the Simulation Options, Dates tab to match the design storm selected.
10. Click the **Map** tab to return to the **Map panel**.
11. In the **Project panel**, click on the **Simulation Options**.
12. In the **Dates** tab, change the **Duration** to **36 hours**. Since large subcatchments were generated, the simulation duration must allow enough time for the water to fully runoff the subcatchments and to be routed through the hydraulic network.
13. Click **OK**.
14. Click on the **Run** button to run the model. There should be flooding indicated in the status bar.

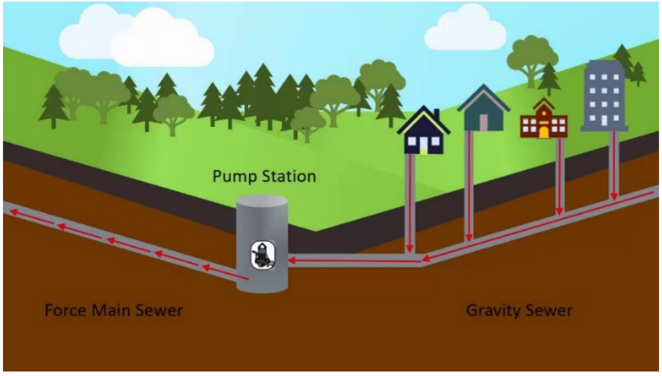
5.6 Next steps

If this was a real model, the flooding should be corrected as it indicates a loss of water in the system which can lead to high continuity errors. By clicking on the **Flooding** notification in the **Status bar**, the **Status panel** will open and display the status report for the model. The **Node flooding** section will indicate which nodes are flooding and for how long and by how much. To model the flooding, the 1D flood inundation analysis, or integrated 1D-2D flood modeling can be used. Please see those exercises for further instructions.

In addition, in a real scenario, the user would need to go through each transect line in the model and ensure the transect has been correctly sampled. To edit a transect, select the transect, double click to enter edit mode and drag the left and right vertices so the full transect is captured.

6. Design of a sanitary sewer system with a force main (Valleyfield, Quebec)

This exercise illustrates the design of a sanitary sewer system using PCSWMM. In this example, we develop a preliminary design for a sanitary sewer drainage system. The system comprises manholes and pipes and a pumping station with a force main - capable of handling the dry weather flow with inflow and infiltration - for a new development of approximately 80 single-family detached residences in Valleyfield, Quebec.



6.1 Set up a new SWMM project

Let's begin by opening a project with the background layers for the Valleyfield sanitary sewer design:

1. Unpackage **PCSWMM Exercises\K027\Initial\Valleyfield_sanitary.pcz.**
2. Launch PCSWMM and click the **Open** button in the **File** menu.
3. Browse to the **PCSWMM Exercises\K027\Initial\Valleyfield_sanitary.pcz** and click **Open**.
4. Unpackage the model to **PCSWMM Exercises\K027\Initial**
5. Click the **Open** button.

The CAD drawing appears, delineating the proposed lots and roadways in yellow.

Note: Model units are set to GPM | LPS as the flows associated with the model are small in magnitude (i.e., compared to the default CFS | CMS flow units).

For this exercise we need to design the sanitary sewer network that will convey wastewater from all residential buildings on the proposed lots. The area is relatively flat and must be graded to ensure that minimum slopes are achieved. We will be entering the invert elevation at the downstream location and from there the manhole invert elevations will be calculated by setting the slope using the set slope tool in PCSWMM.

6.2 Locate the sanitary sewer manholes (junctions)

To begin with, we will locate an outfall (to simulate a pump wet well) 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. We will begin by adding an outfall.

1. In the **Layers panel**, click the **Outfalls** item, the layer will appear grayed out as there are currently no Outfalls in the project.
2. Click the **Add** button in the toolbar and click just outside of the blue CAD boundary on the west side of the subdivision (see screenshot provided).

3. Click the **Select** button in the toolbar to exit the **Add** mode.
4. With the outfall node selected, go into the **Attributes Panel** and change the **Invert Elevation** to **147.6 ft | 45 m**.

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

5. Open the **Junctions outline.SHP** layer from the **PCSWMM Exercises\K027\Initial** folder.
6. Click the **Open layer** button in the **Map Panel.**
7. Click the **Open** button in the Layer Browser window.
8. Navigate to the folder **PCSWMM Exercises\K027\Initial**, select **Junctions**

outline.SHP and click **Open**.

Keep in mind the junctions outline layer is just a background points layer and does not represent SWMM5 junctions. We will now copy and paste the junctions outline points and paste them to the SWMM5 junctions layer and draw conduits connecting the junctions.

9. Unlock the **Junctions outline** layer and select all the nodes.
10. Select the **Junctions outline** layer from the **Layers panel**.
11. Right click on the layer and select **Unlock** . A message will appear asking if you want to unlock or copy the layer, click on **Unlock layer**.
12. Press **Ctrl** + **A** to select all of the points in the **Junctions outline** layer.
13. Press **Ctrl + C** to copy the junctions.
14. Select the **Junctions** layer from the **Layers panel,** the layer will appear grayed out as there are currently no Junctions in the project**.**
15. Press **Ctrl** + **V** to paste the points into the **Junctions** layer.
16. Right click on the **Junctions outline.SHP** layer in the **Layers panel** and click the **Close** .
17. We will now add conduits to connect the junctions.
18. Click on the **Conduits** layer in the **Layers panel**. It will appear grayed out as there are currently no conduits in the model.
19. Click the **Add** button in the toolbar.
20. Click on **J1** and then click on **J2**. A pop up may appear asking if you wish to turn Auto length on. Click **Yes** to turn it on.
21. Repeat the above step linking **J2** to **J3**, **J3** to **J4**, **J4** to **J5**, **J5** to **OF1** and so on, as shown in the supplied screenshot.

Note: 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 match the screenshot provided. You can also double click on the entity you are wanting to change and drag it to the desired position.

22. Click the **Select** button in the toolbar to escape from the **Add shape** mode.
23. Turn on link arrows in the **Map panel**.
24. Click on the **Menu** button in the **Map panel** and select **Preferences...**
25. Under the **Map** tab, check that the **Show link arrow** box is checked, if not, put a check in the box.
26. Click **OK** to close.
27. Check that the conduit flow directions match the screenshot provided. If you need to change the direction of a conduit click on the conduit, right click it and select **Reverse link**.

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 **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.

For this exercise junction depths will be set to 6 ft | 2 m and inverts will be calculated assuming a constant slope.

28. Set all the junction depths to be **6 (ft) | 2 (m).**
29. Select the **Junctions** layer in the **Layers panel** and press **Ctrl** + **A** to select all the junctions.
30. Enter **6 (ft) | 2 (m)** for the **Depth** in the **Attributes panel**. If the **Depth** attribute is grayed out, then click on the **Expression** button that appears in the value field. In the Auto-expression editor, click on the **Calculate Rim Elev. instead** button. Now you can enter the junction depth values.
31. Click the **Save** button.

6.3 Set conduit attributes

1. Select all the conduits.
2. Click the **Conduits** layer in the **Layer Panel**.
3. Select all of the conduits by pressing the **Ctrl** and **A** buttons on the keyboard.
4. Change the attributes for the selected conduits in the **Attributes Panel** as follows:

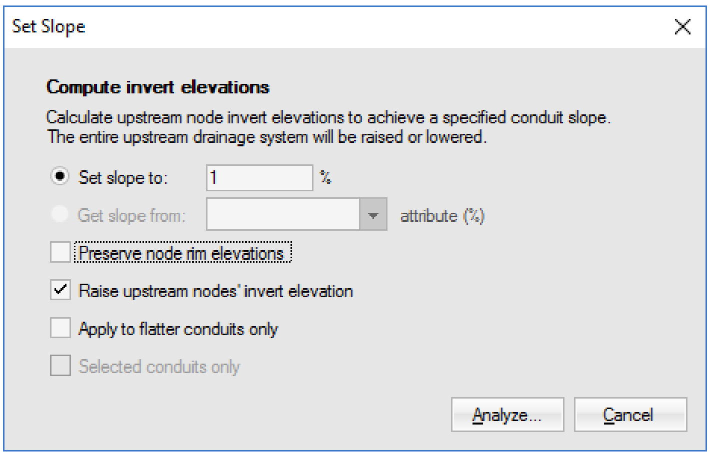
Roughness = 0.013 (new concrete pipe material)

Cross-Section = CIRCULAR

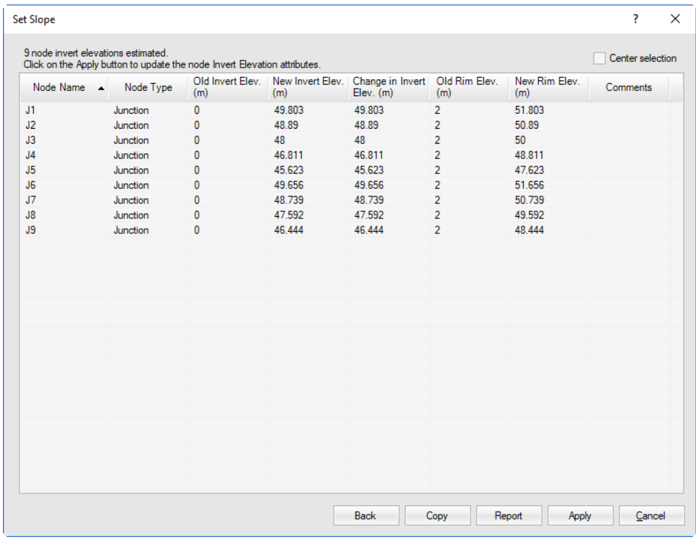
Geom1 = 3.5 (ft) | 1 (m)

Now we need to assign invert elevations for all manholes in the system. As we need to meet the pipe slope design criteria of at least 1% and the outfall invert elevation is 146.7 (ft) | 45 (m), PCSWMM can calculate the required invert elevations of all the other manholes in the drainage network.

5. Click on the **Tools** button.
6. Click on the **Set Slope** tool listed at the bottom (in the Nodes, Conduits or Vertical detail sections).
7. In the **Set Slope** tool, set the **Set slope to 1** (%)
8. Make sure that the **Preserve node rim elevations** box is turned **off**.
9. Make sure that the **Raise upstream nodes' invert elevation** box is turned **on**.
10. Make sure that the **Apply to flatter conduits only** box is turned **off**.
11. **Selected conduits only** should be grayed out or left unchecked.
12. Click on the **Analyze…** button.



13. A table of calculated changes appears for review; click **Apply** to implement them, and then on the **Close** button (please note that the screenshot provided is showing SI units, US units will differ).



Set Slope

9 node invert elevations estimated.
Click on the Apply button to update the node Invert Elevation attributes.

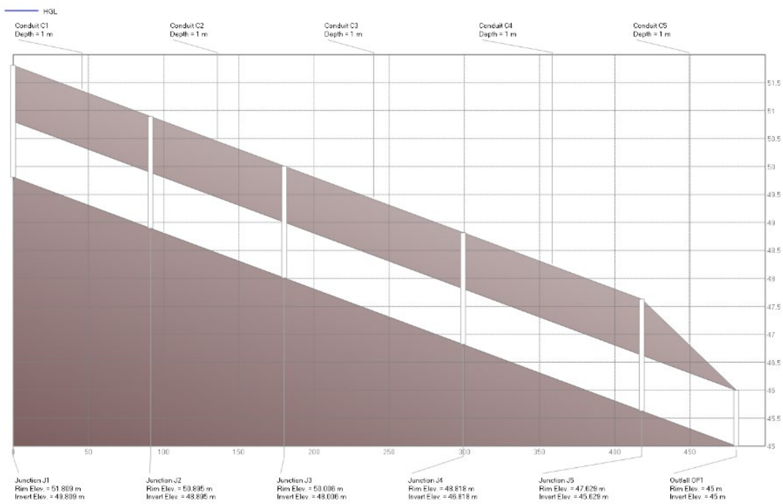
Center selection

Node Name	Node Type	Old Invert Elev. (m)	New Invert Elev. (m)	Change in Invert Elev. (m)	Old Rim Elev. (m)	New Rim Elev. (m)	Comments
J1	Junction	0	49.803	49.803	2	51.803	
J2	Junction	0	48.89	48.89	2	50.89	
J3	Junction	0	48	48	2	50	
J4	Junction	0	46.811	46.811	2	48.811	
J5	Junction	0	45.623	45.623	2	47.623	
J6	Junction	0	49.656	49.656	2	51.656	
J7	Junction	0	48.739	48.739	2	50.739	
J8	Junction	0	47.592	47.592	2	49.592	
J9	Junction	0	46.444	46.444	2	48.444	

Back Copy Report Apply Cancel

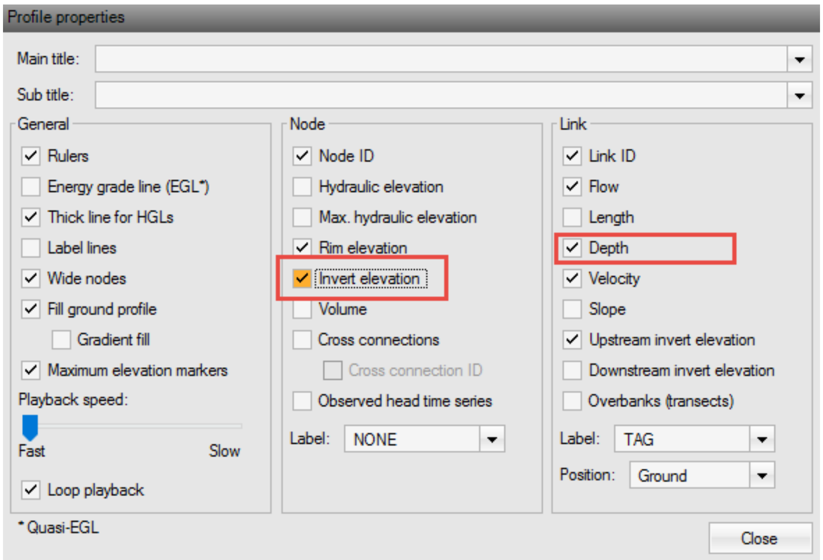
Note: If all 9 nodes are not listed under the Set Slope dialog, it may be because the flow direction is incorrect. You can display flow arrows to check flow direction.

14. Select a pathway from **J1** to **OF1**.
15. In the **Map Panel**, click on junction **J1**
16. Hold the **Shift** key down and click on the outfall **OF1**. A path of connecting entities should be selected.
17. Click the **Profile** tab to view the profile.



To display the invert elevations and pipe diameters as shown in the screenshot:

18. Click the **Properties** button in the **Profile Panel**.
19. In the **Profile Properties** editor, under the **Node** section, ensure **Invert elevation** is checked.
20. In the **Link** section, ensure **Depth** is checked.
21. Click the **Close** button to exit the **Profile Properties**.



6.4 Assign inflows to junctions

Sanitary sewer systems receive inflows based on the wastewater generation from contributing areas and any rain derived infiltration/inflow (RDII). This is in contrast to stormwater collection systems, where inflows are generated from the runoff response due to rainfall/snowmelt. For sanitary sewers, inflows are entered at junctions considering contributing area (sewershed) for each junction. Note the difference between the words subwatershed (contributing area generating runoff) and sewershed (contributing area generating wastewater and RDII).

Inflows can be assigned as direct flow (i.e., a constant value or a variable time series), dry weather flow (average flow with time patterns), or RDII using triangular hydrographs. RDII is discussed in a hydrology article and explored in more detail in the hands-on exercise titled "Derive triangular unit hydrographs".

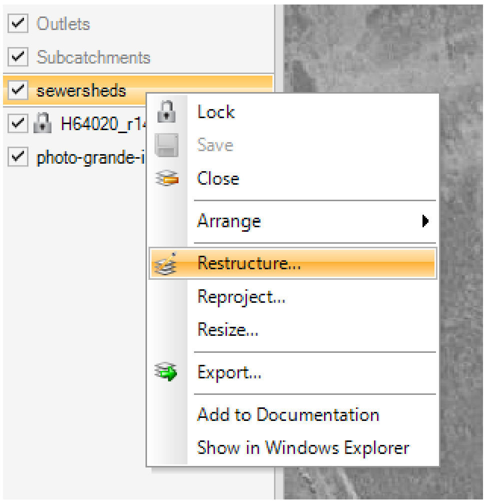
For the first step, sewersheds should be delineated for each junction. You can create an empty shapefile and then add polygons to represent sewersheds

Note: To create a new GIS layer please see the following how-to article: https://support.chiwater.com/77980/creating-a-new-layer.

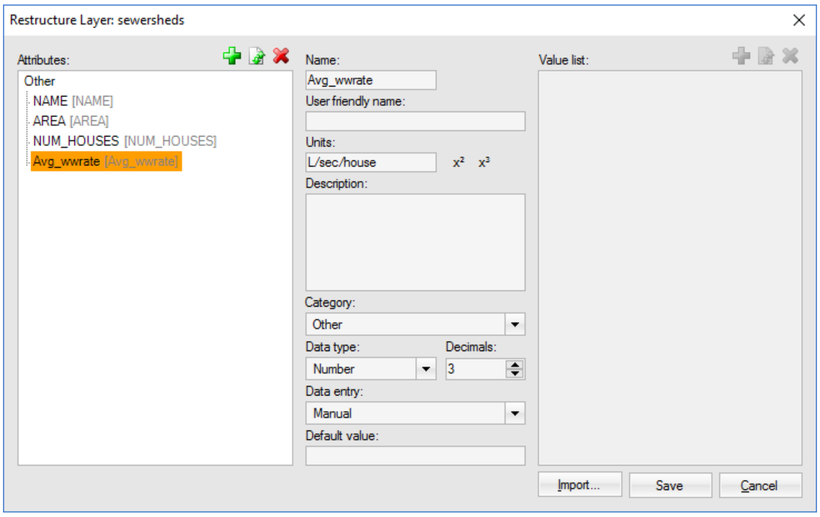
Once the sewersheds are delineated, wastewater generation can be determined in several ways. The most common method involves the sewershed population multiplied by the average daily wastewater generation rate. This can be based on the number of buildings in each sewershed multiplied by the average people per building or area multiplied by population density. Different land uses may also be considered. Another approach is to estimate wastewater generation based on water consumption (fraction of metered water data)

For this example we are going to import an external GIS layer containing sewersheds for each junction. Wastewater generation rates will be calculated within this shapefile and assigned to the associated junctions (note the sewersheds are named to match junctions for easy data importation). We will create an additional two attributes in the shapefile to calculate average wastewater generation rate and inflow/infiltration. Alternatively you can import inflow values from an Excel/text file.

1. Open the **Sewersheds.SHP** layer from the **PCSWMM Exercises\ K027\Initial** folder.
2. Open the **Map panel** and click the **Open layer** button.
3. Click on the **Open** button and navigate to the folder **PCSWMM Exercises\ K027\Initial**, select **Sewersheds.SHP** and click **Open**.
4. Click on **Table panel** and click on the **sewersheds** layer in the **Layers panel**. You should see attributes called **AREA** and **NUM_HOUSES** indicating the area and number of houses in each sewershed.
5. Return to the **Map panel** and click on the **sewersheds** layer in the **Layers panel**, right click and select **Unlock**. A message will appear asking if you want to **Unlock or Copy layer**, click on the **Unlock layer** button.
6. Right click the **sewersheds** layer in the **Layers panel** and select **Restructure**.



7. In the dialog box that appears, click the **Add** button and select **Attributes** to create a new attribute.
8. Type name as **Avg_wwrate.**
9. Enter the units to be **gal/min/house** (US) or **L/sec/house** (SI).
10. Change the **Data type:** to **Number**.



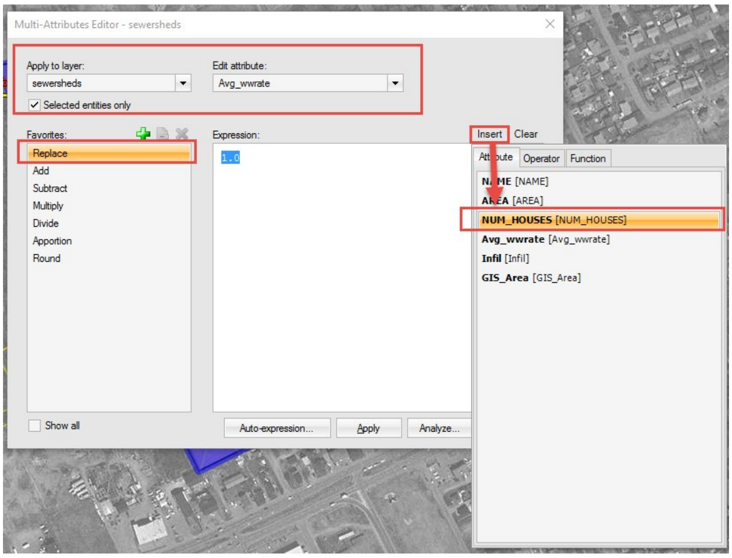
11. Click on the **Add** button again and create another attribute called **Infil.**

12. Change the **Data type:** to **Number**.
13. Click the **Save** button.

Now to populate the two attributes created, we will use the **Replace tool**. The average wastewater generation rate is calculated assuming each house has on average 3.5 persons and a unit wastewater generation rate is 95 gal/person/day | 350 L/person/day (average daily flow can vary from 225 – 425 L/person/day, source: Gravity sanitary Sewer – Design and Construction, Second edition, 2007, ASCE Manuals and Report on Engineering Practice No. 60, WEF Manual of Practice No. FD-5).

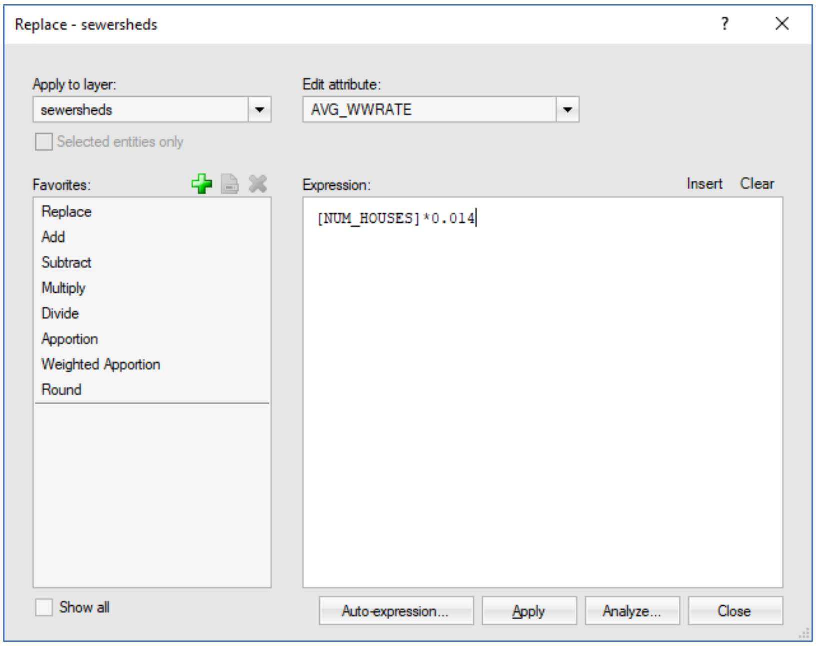
This value can be used as the average value in dry weather flow. The sewer infiltration rate is assumed to be constant at 0.2 L/s/house and can be entered as a direct flow. Inflow and infiltration can be dependent on rainfall with a multiplier (L/s/ha/cm of rain) or it is possible to use RDII algorithms. It is recommended to use local data based on wet weather and dry weather flow monitoring.

14. Click on **sewersheds** in **Layers panel** and press the **Ctrl** and **A** keys to select all the sewersheds.
15. Click **Replace**  button in the **Attributes panel**.
16. Ensure the layer selected is **sewersheds**, under **Edit attribute** select **Avg_wwrate**.
17. Select **Replace** from the list of **Favorites**.
18. Click on the **Insert** button and select **NUM_HOUSES**, as shown in the screenshot provided.

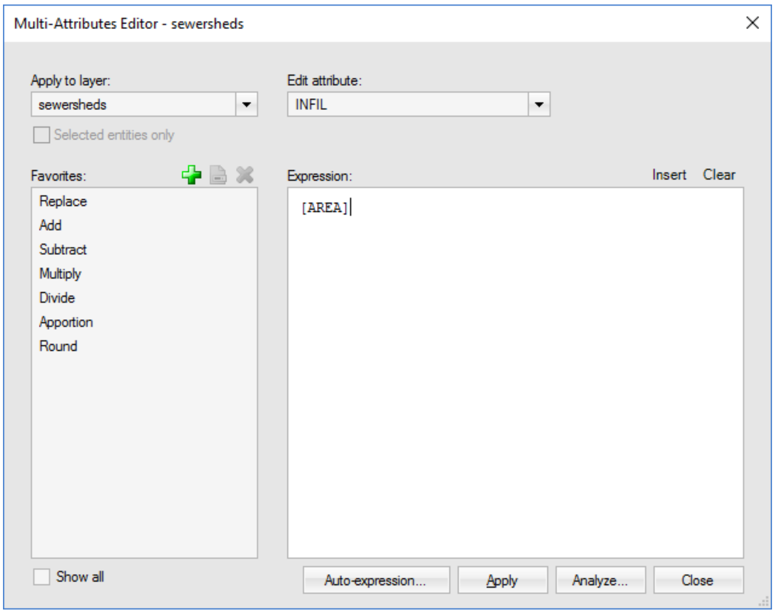


The average wastewater generation rate (Avg_wwrate) for each sewershed will be replaced with the no. of houses within the sewershed. Now we will multiply that value by 0.23 gal/min/house | 0.014 L/s/house (95 gal/person/day x 3.5 persons/house | 350 L/person/day x 3.5 persons/house) using the same Replace function.

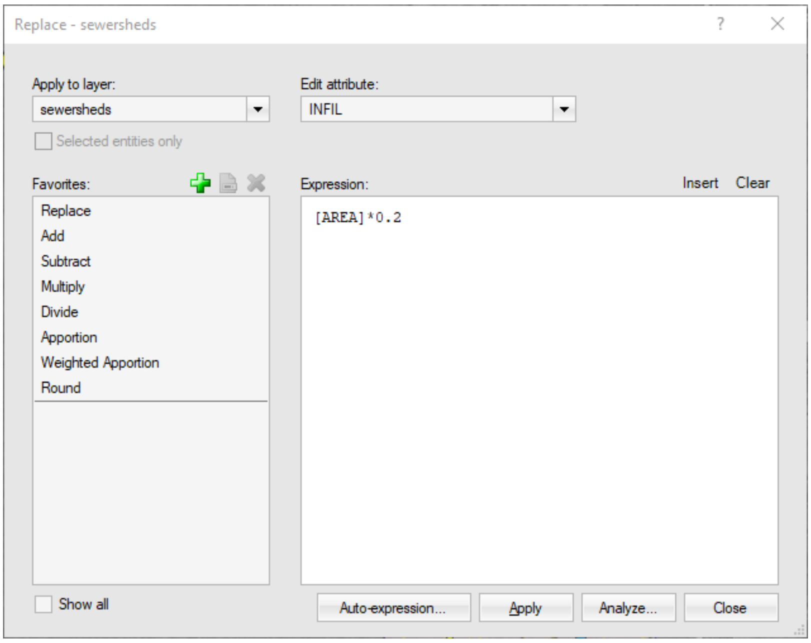
19. Next to [NUM_HOUSES] enter in an asterisk * for multiplication.
20. Type the **Value** to be **0.23** gal/min/house | **0.014** L/sec/house (please note screenshot is in SI units, US will differ).



21. Click **Apply**.
22. Similarly we will assign the **Infil** attribute using the **Replace tool**.
23. If you have already closed the **Replace tool**, click on the **Replace** *fx* button again.
24. Ensure the layer selected is **sewersheds**, under **Edit attribute** select **INFIL**.
25. Select **Replace** from the list of **Favorites**.
26. Click on the **Insert** button and select **AREA**.



27. Next to [AREA] enter in an asterisk * for multiplication.
28. Type in the **value** to be **1.28** gal/min/ac | **0.2** L/sec/ha, as shown in the screenshot provided (please note screenshot is in SI units, US will differ).



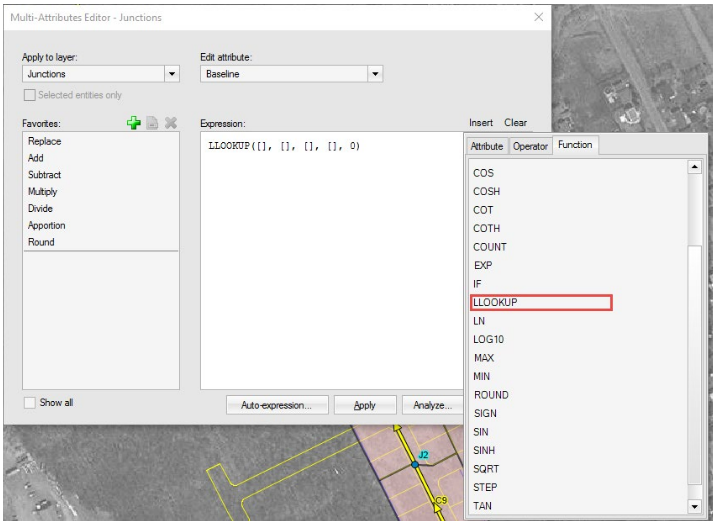
29. Click **Apply**.
30. Click **Close** to close the **Replace tool**.

6.5 Assign inflows at junctions

We will now import inflow values from the sewersheds GIS layer using the same **Replace tool**. In this case we are going to assign the infiltration rates to the associated junction by matching the **Name** attribute in the sewershed and Junctions layers.

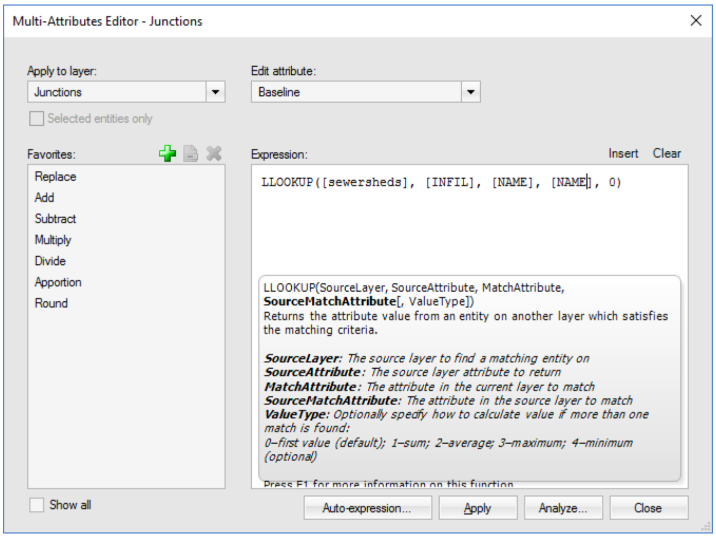
1. Select all the junctions.
2. Click on **Junctions** in the **Layers panel**.
3. Press **Ctrl** + **A** to select all the junctions.
4. In the **Attributes panel** click **Replace**  button.
5. Ensure the layer selected is **Junctions**, under **Edit attribute** select **Baseline**.
6. Select the **Insert** button located above the **Expression** box and click on the **Function** tab.
7. Select **LLOOKUP** from the list of functions.

The LLOOKUP function will import attribute values from another layer provided it can match the ID of one layer with another. In this exercise we will set the **Junction's Baseline** value from the **sewersheds** layer **INFIL** attribute. Entities will be matched based on their names.



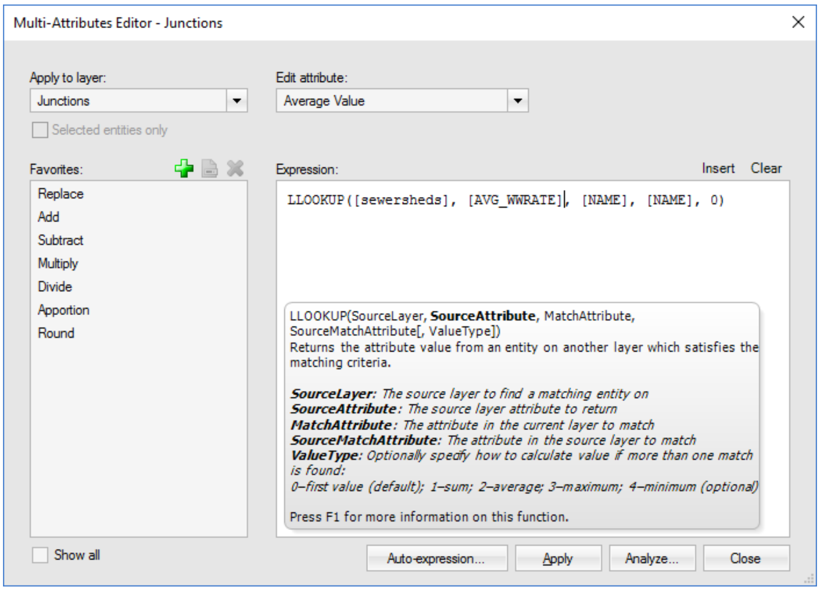
8. Put your cursor in the first set of square brackets, this is where the **SourceLayer** is defined, type in **sewersheds**.
9. Put your cursor in the second set of square brackets, this is where the **SourceAttribute** is defined, type in **INFIL**.
10. Put your cursor in the third set of square brackets, this is where the **MatchAttribute** is defined, type in **Name**.
11. Put your cursor in the last set of square brackets, this is where the **SourceMatchAttribute** is defined, type in **NAME**.

Note: the entities are matched by the **Name** attribute (the sewersheds were named to match junction names).



12. Click **Apply**.
13. Click **Replace**  button if the **Replace tool** was previously closed.
14. Repeat the process from above. This time, edit the **Junction's Average Value** attribute to use the **LLOOKUP** function to match the **sewersheds AVG_WWRATE value**. Names will be used to match entities.
15. Ensure the layer selected is **Junctions**, under **Edit attribute** select **Average Value**.
16. Ensure the **Selected entities only** box is not checked.
17. Select the **Insert** button located above the **Expression** box and click on the **Function** tab.

18. Select **LLOOKUP** from the list of functions.
19. Put your cursor in the first set of square brackets, this is where the **SourceLayer** is defined, type in **sewersheds**.
20. Put your cursor in the second set of square brackets, this is where the **SourceAttribute** is defined, type in **AVG_WWRATE**.
21. Put your cursor in the third set of square brackets, this is where the **MatchAttribute** is defined, type in **Name**.
22. Put your cursor in the last set of square brackets, this is where the **SourceMatchAttribute** is defined, type in **NAME**.

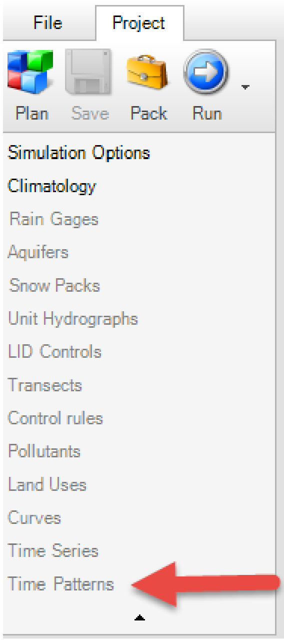


23. Click on **Analyze**, **Apply** and **Close**.

6.6 Assign time patterns

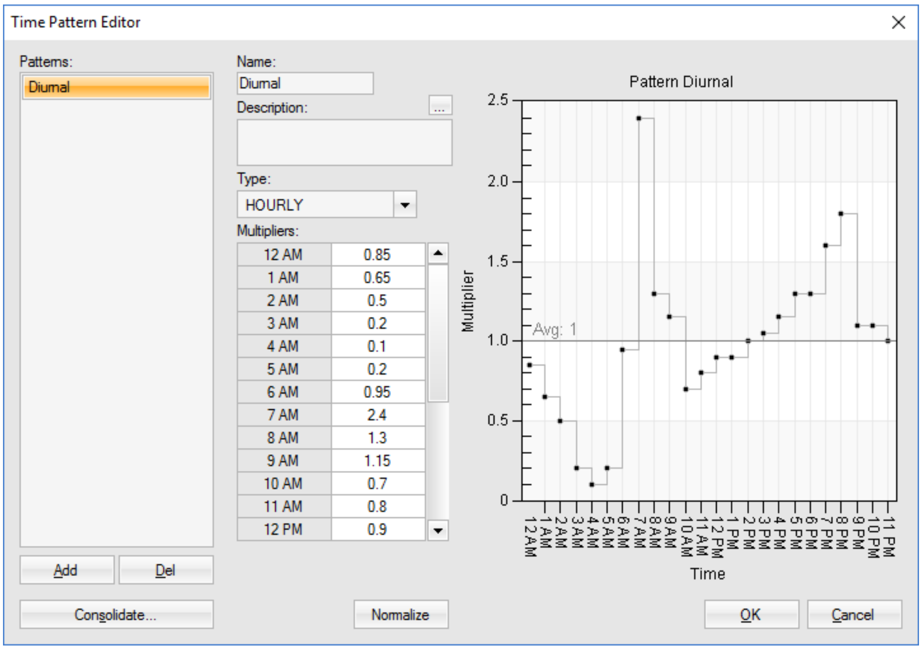
Assigning inflows at junctions can be done in many ways. If a GIS layer containing sewersheds has not been prepared, values can be entered manually or imported from an Excel worksheet. Also data can be extracted from a GIS layer containing water meter readings, or based on land use/population density etc.

1. Click on the downward arrow in the **Project panel** and click on **Time Patterns**. It will appear grayed out as there are no time patterns defined.

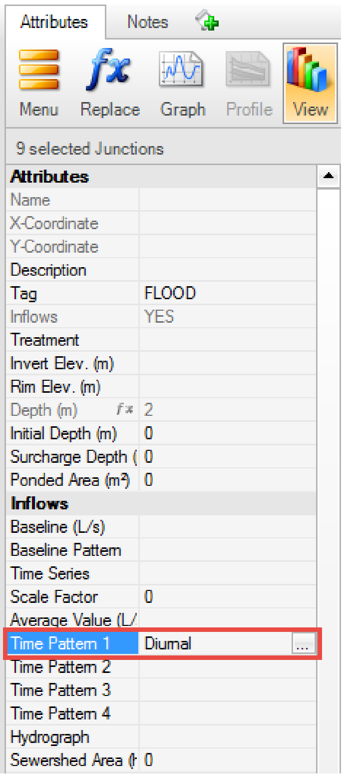


2. Click the **Add** button and under **Name** type **Diurnal** and set the **Type** to **HOURLY**.
3. Enter the numbers from the table provided in the **Time Pattern Editor.**

Time	Multiplier	Time	Multiplier
12:00 AM	0.85	12:00 PM	0.9
1:00 AM	0.65	1:00 PM	0.9
2:00 AM	0.5	2:00 PM	1
3:00 AM	0.2	3:00 PM	1.05
4:00 AM	0.1	4:00 PM	1.15
5:00 AM	0.2	5:00 PM	1.3
6:00 AM	0.95	6:00 PM	1.3
7:00 AM	2.4	7:00 PM	1.6
8:00 AM	1.3	8:00 PM	1.8
9:00 AM	1.15	9:00 PM	1.1
10:00 AM	0.7	10:00 PM	1.1
11:00 AM	0.8	11:00 PM	1



4. Click **OK**.
5. Assign the **Diurnal** pattern to all the junctions.
6. Click on **Junctions** in the **Layers panel** and press **Ctrl** + **A**.
7. In the **Attributes panel**, click on the **Time Pattern 1 ellipsis** ... .
8. Select **Diurnal** and click on the **Assign to the 9 selected junctions** button.



Note: The average wastewater generation rate and time patterns depend on the location as well as the land use of the contributing area. Therefore, you can have different average values and time patterns in the same model. If the data is available, SWMM5 allows using multiple time patterns to consider variability depending on the day or month of the year.

6.7 Run the model

Change the simulation duration to 150 hours.

1. Click on **Simulation Options** in the **Project panel**.
2. Click on the **Dates** tab and change the **Duration** to be **150** hrs.
3. Note: If you examine the **General** tab, you will see only **Flow routing** is checked as in this exercise only hydraulics are simulated (i.e., there are no subcatchments involved, so hydrology is not simulated).
4. 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 in the bottom right corner of the PCSWMM window with the message, 'Run was successful' along with Continuity Error information.
5. Check that the continuity errors are reasonable (say less than 5%, depending on the design accuracy required).
6. Click on the **Status** tab to open the **Status panel**.

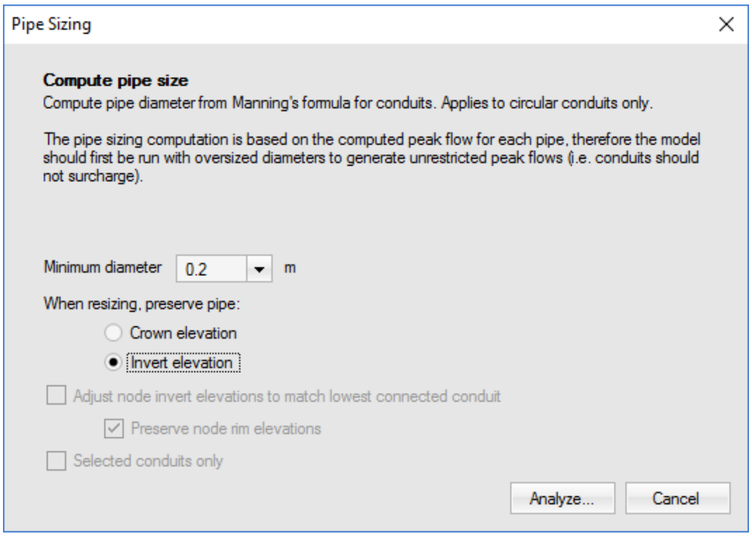
7. Click on the **Continuity Errors** section.
8. Check the routing continuity.

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 as this 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 adjusted 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.

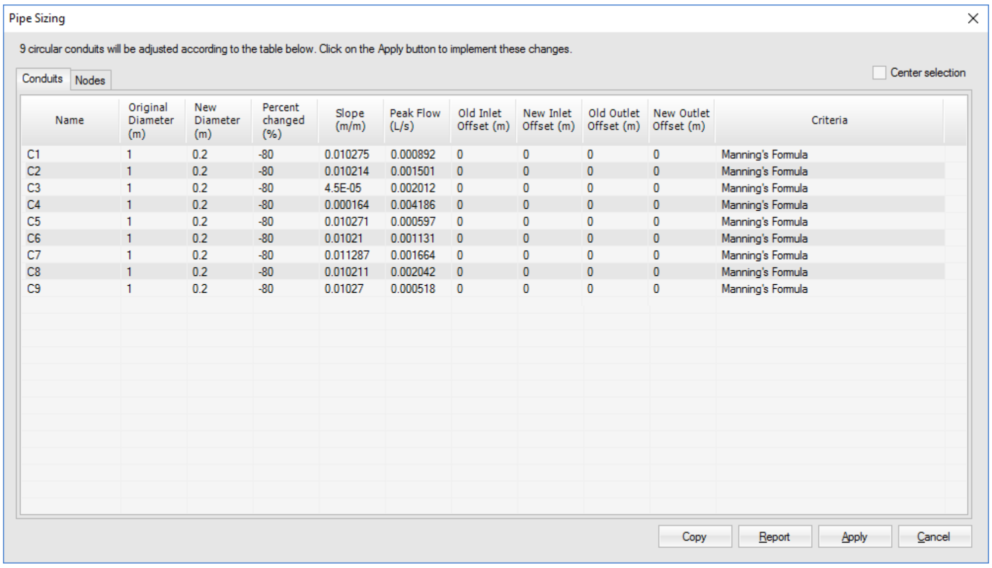
6.8 Size the pipe diameters

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).

1. In the **Map panel**, click on the **Tools** button and in the **Conduits** sections, click on the **Pipe Sizing** tool.
2. Set the **Minimum diameter** to **0.667** ft | **0.2** m, select the option to preserve pipe **Invert elevation**. Ensure **Selected conduits only** is not checked.
3. Click the **Analyze…** button (note that the screenshot is in SI units, US units will differ).



4. Compare **Original Diameter** and **New Diameter** in the **Pipe Sizing** preview table, and click on the **Nodes** tab to see the computed changes to the invert elevations due to pipe resizing. You may see most if not all of the pipes are resized to the minimum pipe diameter.



Name	Original Diameter (m)	New Diameter (m)	Percent changed (%)	Slope (m/m)	Peak Flow (L/s)	Old Inlet Offset (m)	New Inlet Offset (m)	Old Outlet Offset (m)	New Outlet Offset (m)	Criteria
C1	1	0.2	-80	0.010275	0.000892	0	0	0	0	Manning's Formula
C2	1	0.2	-80	0.010214	0.001501	0	0	0	0	Manning's Formula
C3	1	0.2	-80	4.5E-05	0.002012	0	0	0	0	Manning's Formula
C4	1	0.2	-80	0.000164	0.004186	0	0	0	0	Manning's Formula
C5	1	0.2	-80	0.010271	0.000597	0	0	0	0	Manning's Formula
C6	1	0.2	-80	0.01021	0.001131	0	0	0	0	Manning's Formula
C7	1	0.2	-80	0.011287	0.001664	0	0	0	0	Manning's Formula
C8	1	0.2	-80	0.010211	0.002042	0	0	0	0	Manning's Formula
C9	1	0.2	-80	0.01027	0.000518	0	0	0	0	Manning's Formula

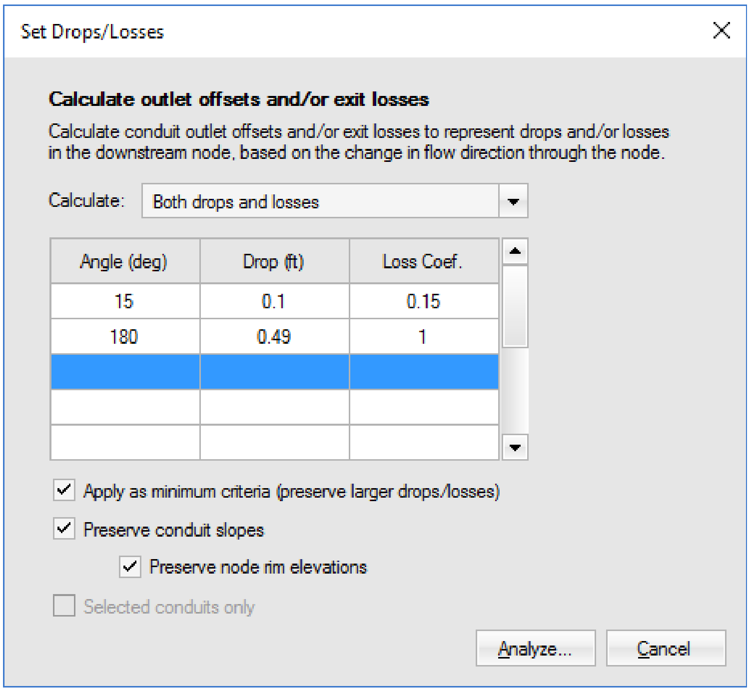
5. Click **Apply** to implement the changes and click on the **Close** button to exit the **Pipe Sizing** tool.

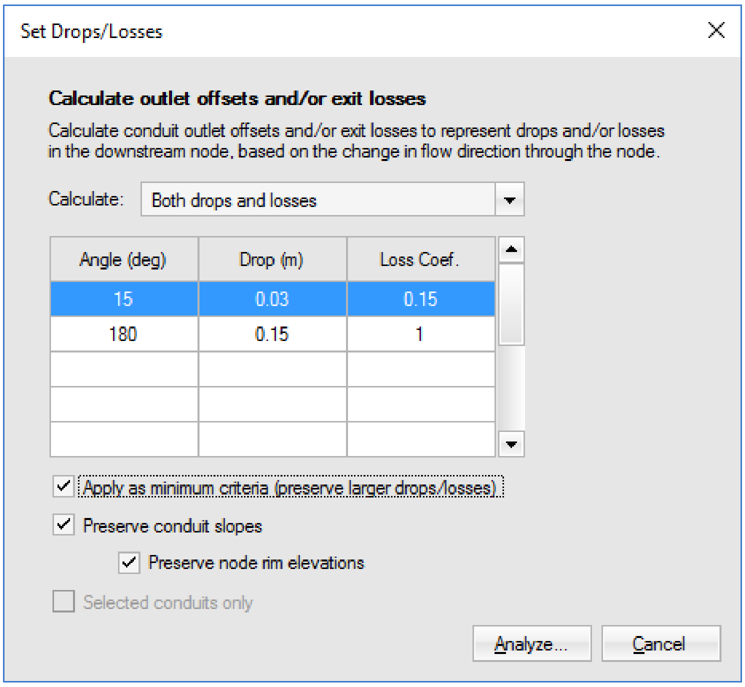
6.9 Set Drops/Losses

Now we need to adjust the drops across each manhole. In this example we will assign a drop of **0.1 ft | 0.03 m** for a straight-through pass and **0.49 ft | 0.15 m** for a 45 degree to 90 degree bend at a manhole. For this process;

1. In the **Map Panel** click on the **Tools** button and click on the **Set Drops/Losses** tool (in the Conduits or Vertical detail sections).
2. In the drop-down menu **Calculate:** select **Both drops and losses** (should be default).
3. Enter the **Angle**, **Drop** and **Loss Coef.** values as shown in figures provided (please note that two screenshots showing the values for both US and SI units are provided).

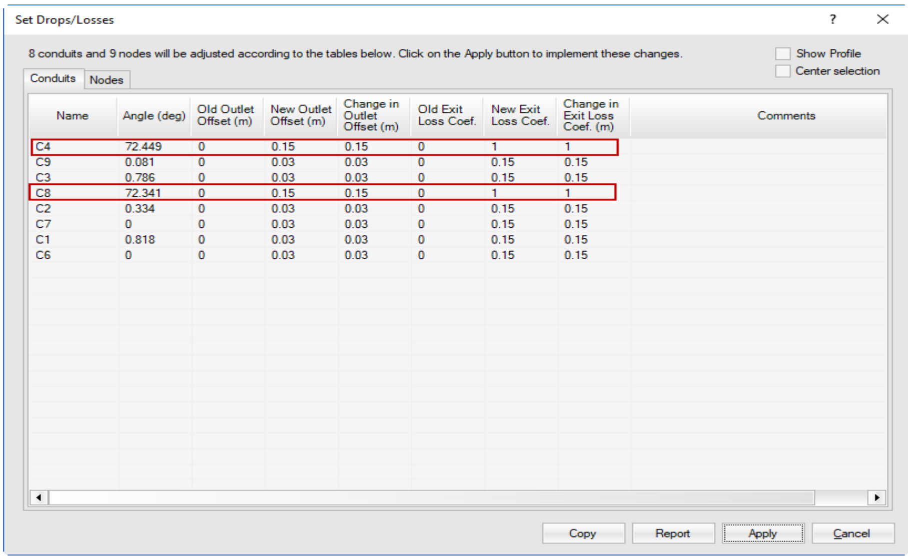
Note: It is recommended to enter drops/losses for other additional angle ranges however for the sake of the exercise the two angles are sufficient to illustrate the process.





4. Make sure that the **Apply as minimum criteria** and **Preserve conduit slopes** boxes are all checked.
5. Click **Analyze**.

View **Status report** to ensure that **two of the outlet offsets** and **exit loss coefficients** are the same (If you see 2 different conduits with the same values you may have simply created your conduits in a different order, and thus they have different names. Check the original set up of the conduits to ensure they are in the same location as shown above - in this example, the conduits with the largest exit angles are C4 and C8).



Set Drops/Losses

8 conduits and 9 nodes will be adjusted according to the tables below. Click on the Apply button to implement these changes.

Show Profile
Center selection

Conduits | Nodes

Name	Angle (deg)	Old Outlet Offset (m)	New Outlet Offset (m)	Change in Outlet Offset (m)	Old Exit Loss Coef.	New Exit Loss Coef.	Change in Exit Loss Coef. (m)	Comments
C4	72.449	0	0.15	0.15	0	1	1	
C9	0.081	0	0.03	0.03	0	0.15	0.15	
C3	0.786	0	0.03	0.03	0	0.15	0.15	
C8	72.341	0	0.15	0.15	0	1	1	
C2	0.334	0	0.03	0.03	0	0.15	0.15	
C7	0	0	0.03	0.03	0	0.15	0.15	
C1	0.818	0	0.03	0.03	0	0.15	0.15	
C6	0	0	0.03	0.03	0	0.15	0.15	

Copy | Report | Apply | Cancel

6. Click **Apply** to apply the changes and then click the **Close** button.
7. Set an **Entry Loss Coefficient** of 0.1 for all the conduits.
8. Click on the **Conduits** layer in the **Layers panel** and click **Ctrl** + **A** to select all of the conduits.
9. In the **Attributes panel** enter an **Entry loss coefficient** of **0.1**.
10. Click on the **Run** **Simulation** button on the toolbar to save the project and regenerate the results.

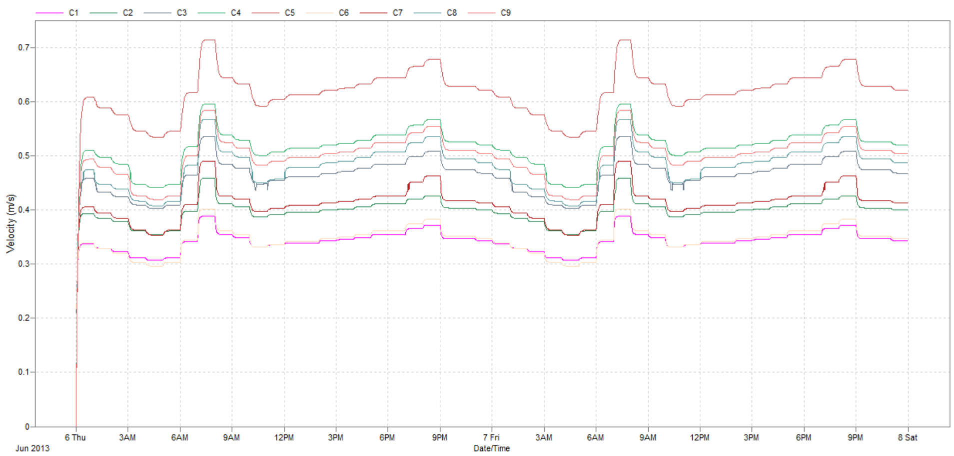
6.10 View and interpret the results

We will now ensure that the minimum velocity criteria are being met,

Plot the velocity through all the conduits.

1. Click the **Graph** tab to open the **Graph Panel**.
2. In the **Time Series Manager**, expand **Links > Velocity.**
3. Select all the conduits (from 1 to 9) by right-clicking **Velocity** and choosing **Select All** from the pop-up menu.

Note: The **Graph Panel** can also be used to plot the flows in all the conduits as well as the depths in all nodes in the system.



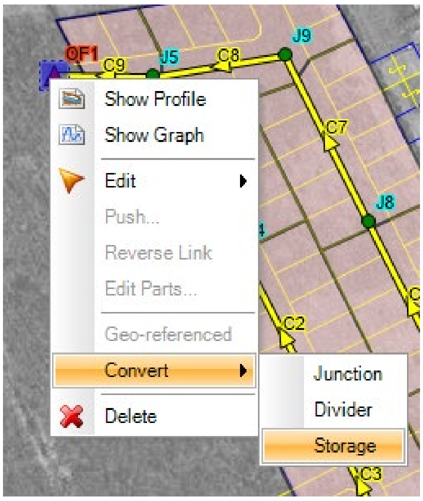
Note: Your plot of velocities should be similar to the above plots. The peak velocities should exceed the minimum self-scouring velocities specified by your local approval agency (commonly about 2 ft/s | 0.6 m/s). In this design, not all upstream pipes achieve their respective self-cleansing velocities due to small drainage areas, so maintenance may be required to avoid sedimentation.

4. Check that the continuity errors are less than 5%.
5. Click on the **Status** tab to open the **Status panel**.
6. Click on the **Continuity Errors** section.
7. Check the routing continuity.

6.11 Setup pump and the force main

Now for this simple exercise, we will assume wastewater is collected at a sump and pumped to a wastewater treatment plant. This will include adding a pump to the model and a force main that will convey wastewater to the treatment plant. For this purpose we will use Hazen-Williams equation (options available are Hazen-Williams, Darcy-Weisbach or simply Manning's equation).

1. Click on the **Map** tab to open the **Map panel**.
2. Right click on the outfall **OF1** and select **Convert > Storage** from the drop-down menu.



3. In the **Attributes panel**, change the **Name** to **SU1**, the **Invert Elevation** to **131.2** ft | **40** m, the **Depth** to **23** ft | **7** m, the **Storage Curve** to **FUNCTIONAL**, the **Coefficient** to **0** and change the **Constant** to **40 ft²** | **4 m²** as shown in the table provided.

Name	SU1
Invert Elevation	131.2 (ft) | 40 (m)
Depth	23 (ft) | 7 (m)
Storage Curve	FUNCTIONAL
Coefficient	0
Exponent	0
Constant	40 (ft^2) | 4 (m^2)

4. Select the conduit discharging into the pond/outfall (**C5** in this case) and change the **Outlet Offset** to **16.4** ft | **5** m

We will now add an outfall representing the wastewater treatment plant.

Use your mouse scroll wheel to zoom out (scroll down) until you can see the river.

5. Select **Outfalls** in the **Layers panel**, It will appear grayed out as there are currently no outfalls in the model.
6. Click the **Add** button and click beside the river to add an outfall as shown in the screenshot provided.

7. In the **Attributes panel**, change the **Name** to **Outfall** and set the **Invert El.** to **144** ft | **44** m.
8. Click on the **Select** button to get out of Edit mode.

We will now add the force main by drawing a conduit

9. Click on the **Junctions** layer in the **Layers panel** and click on the **Add** button.
10. Click beside the storage unit **SU1** to draw a junction and, while holding the **Shift** key, click midway between **SU1** and the **Outfall** to add another junction and a conduit link, as shown in the screenshot provided.

Note: The new junction, J10, represents the header of a pump while J11 is an intermediate location in the force main. The conduit C10 joining J10 and J11 represents half of the force main.

We will now add a second conduit linking J11 to the outfall. Conduits C10 and C11 represent the force main.

1. Click on the **Conduits** layer in the **Layers panel** and click on the **Add** button if you are not already in add/edit mode.
2. Click once on **J11** and then click on the **Outfall** to add a conduit.
3. Click on the **Select** button to get out of Edit mode.
4. Select conduits **C10** and **C11** by first selecting **C10,** and while holding down the **Ctrl** key, selecting **C11**.
5. In the **Attributes panel** change their lengths to **2450** ft **|** **750** m.
6. In the **Attributes panel**, click in the **Cross-section** attribute box and click the **ellipsis** button to open the **Cross-section Editor**.
7. Change the cross-section to **Force-Main** and change the Max. Depth to **0.5** ft | **0.15** m.
8. In the **Roughness** box enter a value of **120**, this roughness value represents the Hazen-Williams C-factor to **120**. Click **OK**.

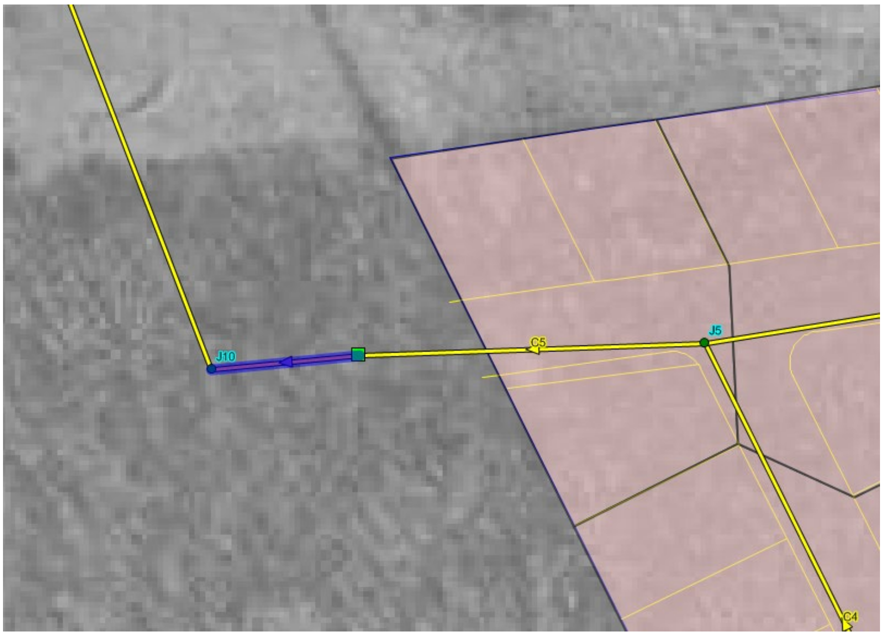
Note: By default PCSWMM uses the Hazen-Williams equation for force mains. You can change this by selecting **Simulation Options** under the **Project panel** and, under the **Dynamic Wave** tab, selecting **Darcy-Weisbach** under the **Force main equation** section.

9. Select junctions **J10** and **J11** (while pressing **Ctrl** key) and change the **Surcharge depth** to **160** ft | **50** m. Assigning a large surcharge depth allows pressurized flow at the nodes without flooding.

10. Click on **J10** and assign it an **Invert Elevation** of **138** ft **|** **42** m.
11. Click on **J11** and assign it an **Invert Elevation** of **164** ft | **50** m.

We will now add a pump between storage node SU1 and Junction J10.

12. Click on the **Zoom** button and draw a box around **J10** and **SU1**.
13. Select the **Pumps layer** from the **Layers panel** and click **Add** button. The pumps layer will appear grayed out as there are currently no pumps in the model.
14. Click first on **SU1** and then **J10** to draw the pump.



15. In the **Attributes panel**, specify the **Startup** depth as **13** ft **|** **4** m.
16. Specify the **Shutoff depth** as **1.6** ft | **0.5** m.
17. In the **Pump Curve** attribute click on the **ellipsis** button.
18. Click on the **Add** button in **Pump Curve Editor**.
19. Change the **Name** to **Pump1** and select **Pump type** as **TYPE3**.

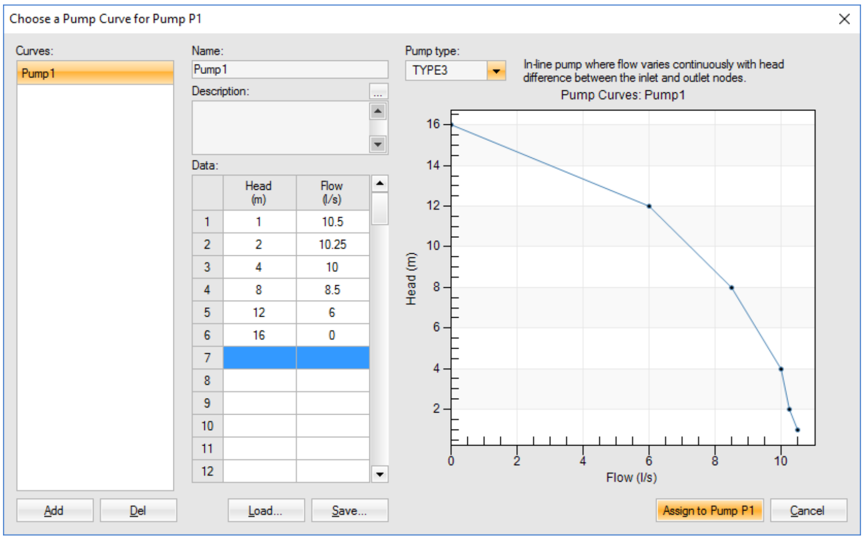
Enter the following numbers from the screenshots provided in the image, please note both US and SI units have been provided:

Head (ft)	Flow (gal/min)
3	166.4
6	162.5
12	158.5
24	134.7
36	95.1
48	0

US Units

Head (m)	Flow (L/s)
1	10.5
2	10.25
4	10
8	8.5
12	6
16	0

SI Units



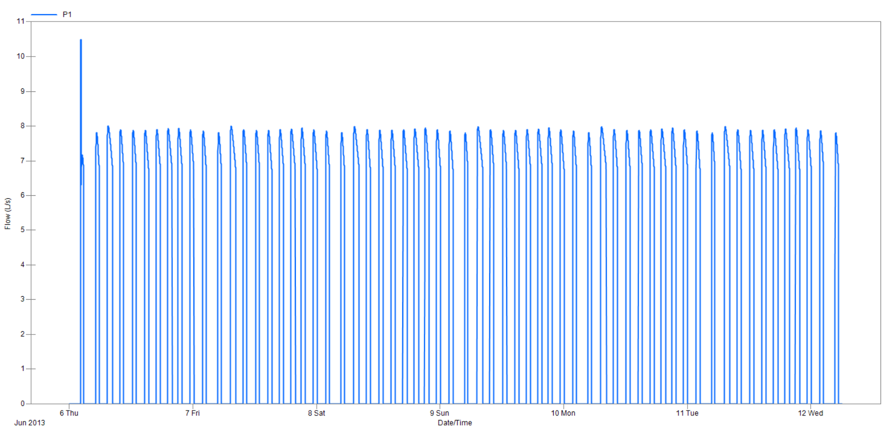
	Head (m)	Flow (l/s)
1	1	10.5
2	2	10.25
3	4	10
4	8	8.5
5	12	6
6	16	0
7		
8		
9		
10		
11		
12		

20. Click **Assign to Pump P1** button.

Note: Pump operation can also be modified using Control Rules.

6.12 Run the model

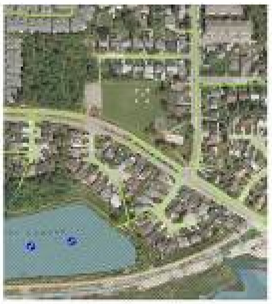
1. Click the **Run** button in the toolbar to save the project and run a simulation. If the run is successful, a pop-up window will show the message, 'Run was successful', with Continuity Error information. A model containing pumps and force mains may experience instabilities and high continuity errors. If this happens, you may have to try smaller routing time steps to get acceptable results.
2. Check that the continuity errors are reasonable (say less than 5%, depending on the design accuracy required).
3. Click on the **Status** tab to open the **Status panel**.
4. Click on the **Continuity Errors** section.
5. Check the routing continuity.
6. Plot flow in the pump to see how it was operated.
7. Switch to the **Graph panel.**
8. Expand the **Time Series Manager** > **Links > Flow > P1**), please note that the screenshot provided is in SI units, US units will differ.



7. Evaluation of in-line stormwater pond TSS removal

Water quality has become an important design criteria for many stormwater practices. Stormwater pond effectiveness is often based on the total suspended solids (TSS) removal, for example Section 4.2 of the Toronto and Region Conservation Authority (TRCA) Stormwater Management Criteria (2012) states that all watercourses and water bodies within TRCA's jurisdiction require an enhanced level of water quality protection, equivalent to 80% TSS removal.

TRCA guidelines, based on Ontario Ministry of the Environment Stormwater Management Planning and Design Manual (2003), provide water quality storage requirements to achieve long-term suspended solids removal. Providing an alternative modeling approach, this exercise illustrates how to determine the loading rates at the inlet and outlet of a stormwater pond and thereby determine the percentage removal for TSS considering particle settling in the pond.



7.1 Opening and running the to-be model

1. Unpackage **Valleyfield pond TSS removal.pcz** from the **PCSWMM Exercises\K024\Initial** folder.
2. Open **PCSWMM Exercises\K024\Initial** and select **Valleyfield pond TSS removal.pcz.**
3. Click the **Open** button.
4. A dialog will appear showing a default location to unpackage the model: click on the **Unpackage** button then the **OK** button.
5. Click on the **Run** button in the **Project panel** to run the model.

7.2 Evaluating the performance of the pond

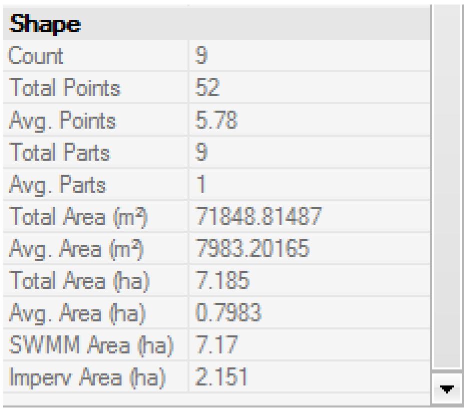
TRCA guidelines, based on Ontario Ministry of the Environment Stormwater Management Planning and Design Manual (2003), provides water quality storage requirements to achieve long-term suspended solids removal.

To determine water quality storage requirement for long-term SS removal at the pond, it is required to estimate total area and the imperviousness of the catchment.

1. Select all the subcatchments.
2. Select the **Subcatchments** layer in the **Layers panel.**
3. Press **Ctrl+A** to select all the subcatchments in the map.

Note: In this exercise all the subcatchments drain to the pond. In a larger model only a few subcatchments may be draining to any particular pond. In that case the catchment of the pond can be selected by selecting the pond storage unit in the map and using Find tool. Click Select Upstream and then select Subcatchments in the Layers panel.

4. The **Attributes panel** should indicate all the subcatchments are selected (9 in this exercise).
5. Scroll down in the **Attributes panel** to **Shape** to obtain statistics for all the subcatchments.



Shape	
Count	9
Total Points	52
Avg. Points	5.78
Total Parts	9
Avg. Parts	1
Total Area (m²)	71848.81487
Avg. Area (m²)	7983.20165
Total Area (ha)	7.185
Avg. Area (ha)	0.7983
SWMM Area (ha)	7.17
Imperv Area (ha)	2.151

6. On a scrap piece of paper, calculate imperviousness using the **SWMM Area** and **Imperv. Area** values using the equation below:
7. Determine the required storage requirement as per the table below. This table is for context only and is only provided in SI units.
8. **Shown in SI units**

Note: The table below shows the water quality storage requirements for Ontario, Canada, as specified by the Ministry of the Environment. Please note this figure was added to add context to the exercise and is not required to complete the exercise. Source: Stormwater Management Planning and Design Manual, March 2003, Ministry of the Environment, Ontario, Canada.

Table 3.2 Water Quality Storage Requirements based on Receiving Waters[1, 2]

Protection Level	SWMP Type	Storage Volume (m^3/ha) for Impervious Level 35%	55%	70%	85%
Enhanced 80% long-term S.S. removal	Infiltration	25	30	35	40
	Wetlands	80	105	120	140
	Hybrid Wet Pond/Wetland	110	150	175	195
	Wet Pond	140	190	225	250
Normal 70% long-term S.S. removal	Infiltration	20	20	25	30
	Wetlands	60	70	80	90
	Hybrid Wet Pond/Wetland	75	90	105	120
	Wet Pond	90	110	130	150
Basic 60% long-term S.S. removal	Infiltration	20	20	20	20
	Wetlands	60	60	60	60
	Hybrid Wet Pond/Wetland	60	70	75	80
	Wet Pond	60	75	85	95
	Dry Pond (Continuous Flow)	90	150	200	240

[1]*Table 3.2 does not include every available SWMP type. Any SWMP type that can be demonstrated to the approval agencies to meet the required long-term suspended solids removal for the selected protection levels under the conditions of the site is acceptable for water quality objectives. The sizing for these SWMP types is to be determined based on performance results that have been peer-reviewed. The designer and those who review the design should be fully aware of the assumptions and sampling methodologies used in formulating performance predictions and their implications for the design.*

[2]*Hybrid Wet Pond/Wetland systems have 50-60% of their permanent pool volume in deeper portions of the facility (e.g., forebay, wet pond).*

Note: As an alternative to the above procedure, PCSWMM provides a modeling approach to evaluate the performance of a stormwater management facility using water quality simulation. Long-term SS removal can be evaluated considering treatment occurring at the facility. This exercise presents SS removal modeling in PCSWMM/SWMM5 in a stormwater management pond simulating particle settling as described in the previous chapter. SS removal represents the reduction in TSS loading in the outflow of the pond in comparison to TSS loading in the inflow.

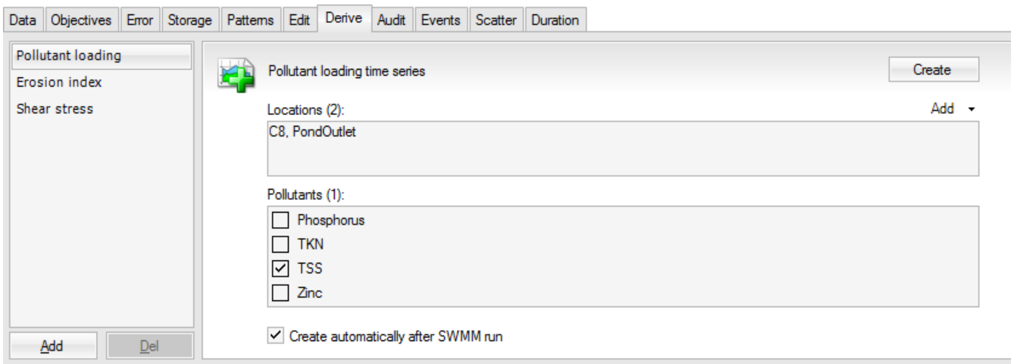
7.3 Deriving TSS load estimates

It is possible to estimate system wide TSS percent removal by considering inflow TSS load and mass reacted:

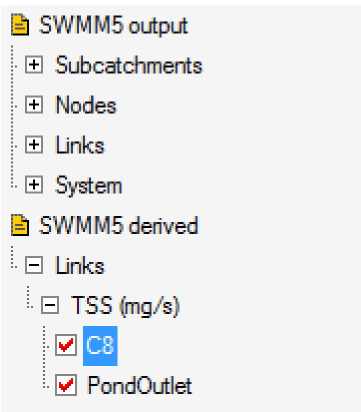
$$\%\ SS\ removal\ = \frac{Mass\ reacted}{Total\ wet\ weather\ inflow\ mass} x100$$

SWMM5 outputs water quality time series as concentrations (mg/L). To obtain pollutant loadings, we need to multiply the concentration by the flow. This is done after the SWMM run by deriving the TSS loading from SWMM5 output with the **Graph panel's Derive** tab.

1. Open the **Status panel** and check that the water quality runoff and routing continuity errors are acceptable.
2. To generate the TSS loading time series upstream and downstream of the pond:
3. Switch to the **Map panel** and select the links immediately upstream and downstream of the pond.
4. Click on conduit C8.
5. Click on the **Find** button.
6. Choose **Select Downstream**. Conduit **C8** and **PondOutlet** outlet link will be selected.
7. Switch to the **Graph panel** and click on **Tools** **> Derive Time Series**.
8. Select **Pollutant loading** from the list of functions.
9. Click on the **Add** button next to the **Locations** text box and choose **Selected Map Entities** from the menu. The two selected entities should appear in the list of locations.
10. Check the box next to **TSS** item in the list of pollutants in the screenshot.



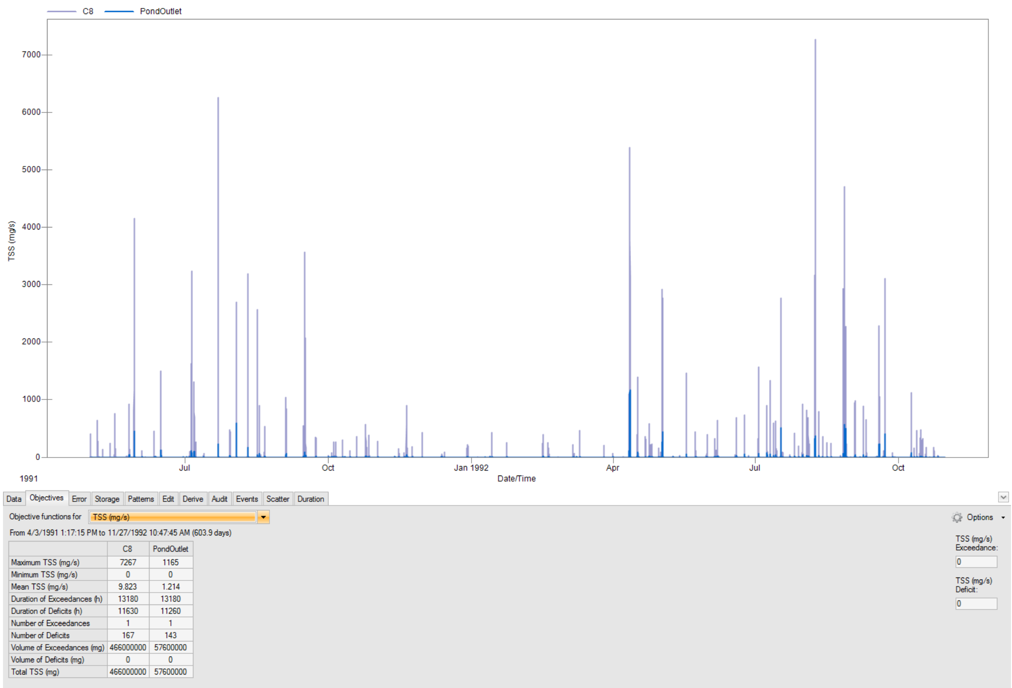
11. Click on the **Create** button to create the derived pollutant loading time series for the selected pollutants and locations.
12. Plot the newly created TSS loading time series by selecting them from the created .tsd file in the **Time Series Manager**.



7.4 Long-term SS removal based on TSS loads

In the **Graph panel** click on **Tools**  **> Objective Functions**.

Select **TSS (mg/s)** from the **Objectives functions for** drop-down menu.



Zoom into an event to see the performance of the pond at an individual event level.

It is possible to determine SS percent removal by considering the total TSS loads at inlets (i.e. conduit C8) and outlets of the pond (i.e. conduit PondOutlet) from the **Objectives** tab):

$$\% \, SS \; removal \; at \; the \; pond = \frac{(SS \; load \; at \; inlets - SS \; load \; at \; outlets)}{SS \; load \; at \; inlets} x100$$

Since the long-term removal is of interest, the SS load should be extracted from the whole simulation period (not on an event basis). If there is more than one inlet or outlet, they should be considered to calculate % SS removal. As it is, the current pond design may not meet the required % SS removal (approximately 60% SS removal).

1. Return to the **Map panel** and click on the pond to select it.
2. In the **Attributes panel** for the pond, revise the pond storage curve by changing the **constant** in the storage function from **400** m^2 **|** **4300** ft^2 to **800** m^2 **|** **8600** ft^2 (making the pond area larger, thereby reducing water depths).
3. **Run**  the model again.
4. Switch to the **Graph panel** and recalculate % SS removal. Notice the % removal increased from ~60% to ~71%.

Changing the storage curve as well as outlet settings, permanent pool volume etc. also affects the pond performance. Using different particle size ranges considering particle size distribution (see Chapter 21) will also affect the results.

8. Water quantity with LIDs

The objective of this exercise is to demonstrate how different LID practices are represented in SWMM5/PCSWMM and how their performance can be evaluated. In this model there are 9 subcatchments, each representing a single LID or homogeneous land surface. The treatment train aims to route runoff from impervious surfaces (streets, roofs and parking lots) to pervious and lot level LID subcatchments.

8.1 Setting up the model

For this model the subcatchments have already been drawn and the subcatchment parameters have been estimated, however the subcatchment connectivity (where subcatchment runoff is routed to) must first be defined.

1. Unpackage the **OldsCollege-LIDs(1).pcz** file from the **PCSWMM Exercises\K026\Initial** folder.
2. Launch **PCSWMM** and select **Open** from the **File** tab menu**.**
3. Browse to **PCSWMM Exercises\K026\Initial** and open **OldsCollege-LIDs(1).pcz.**
4. Click **Unpackage** followed by **OK**. The model will be unpackaged in the correct location (\PCSWMM Exercises\K026\Initial).

In this exercise we will be using the LID editor to define 4 different LIDs:

a) Rainwater Harvesting (Rain barrel),

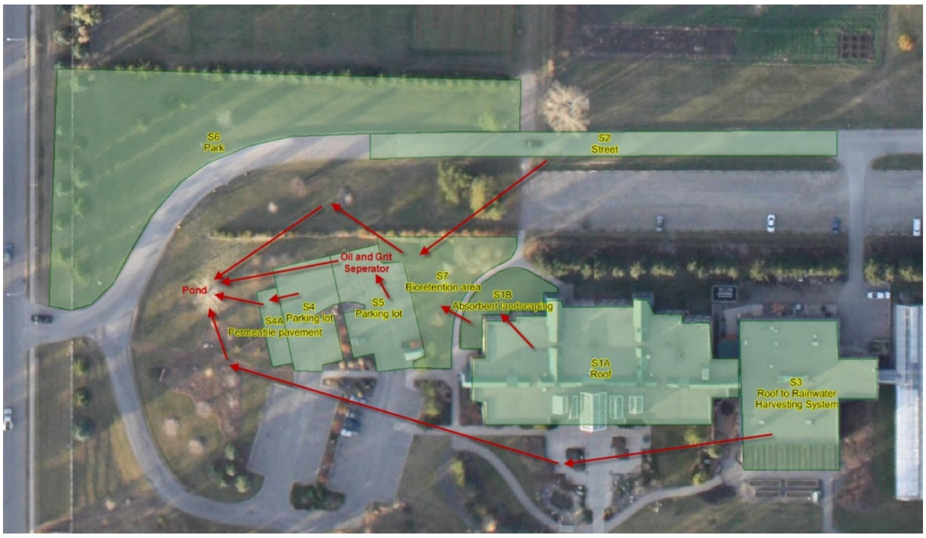
b) Bio-retention Cell,

c) Absorbent Landscaping, and

d) Permeable parking lot (Porous Pavement).

For this model the connectivity is set up as follows:

Subcatchment S3 (Roof) is collected by a rainwater harvesting system and from there routed to a stormwater pond via conduit.



Subcatchment S1A (Roof) is routed first to subcatchment S1B (Absorbent landscaping) from there routed to subcatchment S7 (Bio-retention area) and finally to the stormwater pond.

Subcatchment S2 (Street) is also routed to subcatchment S7 (Bio-retention area) and from there routed to the stormwater pond.

Subcatchment S5 (Parking lot) is routed first to an oil and grit separator and from there to the stormwater pond.

Subcatchment S4 (Parking lot) is first routed to subcatchment S4A (Permeable pavement) and from there to the stormwater pond.

For this exercise we have already created a model and setup the connectivity. You will start by setting up the LIDs.

Parameter values for the LID 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.

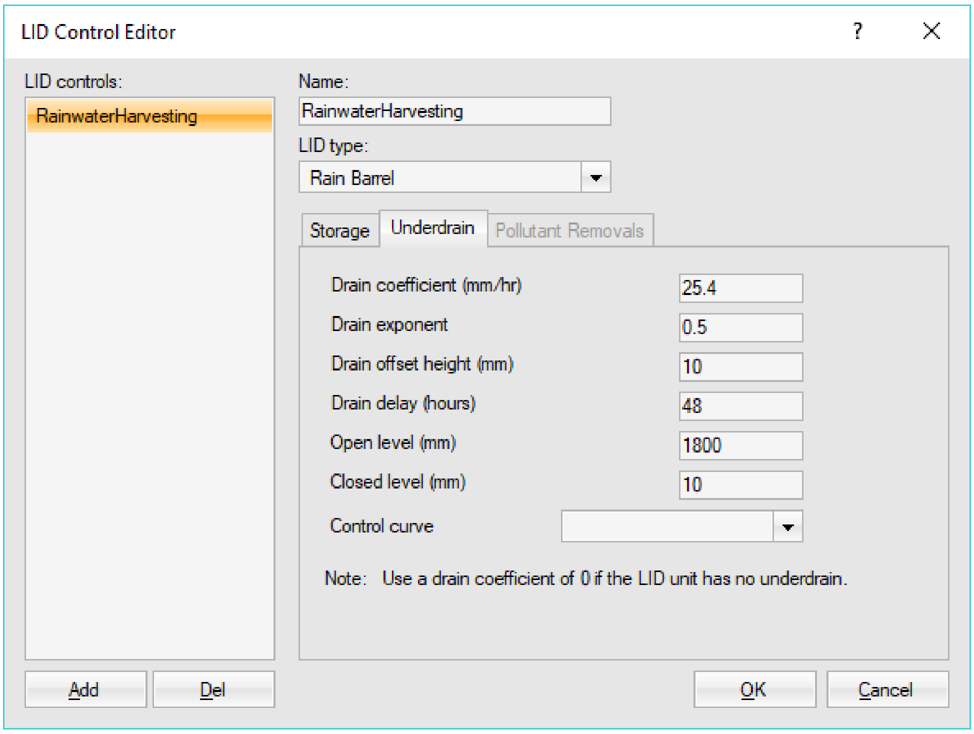
We begin by defining the rainwater harvesting LID.

5. Click the downward arrow in the **Project panel** and select **LID Controls** to open the **LID Control Editor**. It will appear grayed out as there are currently no LID objects defined in the model.
6. Click the **Add** button to create a LID control.
7. Re-name the newly created LID to **RainwaterHarvesting**.
8. Set the **LID type** to **Rain Barrel**.
9. Enter the properties for the **Storage** and **Underdrain** layers as shown.

Properties of the Rainwater Harvesting LID

Storage	
Barrel height (in|mm)	78 in | 2000 mm
Underdrain	
Drain coefficient (in/hr|mm/hr)	1 in/hr | 25.4 mm/hr
Drain exponent	0.5 (pg. 104 SWMM Reference Manual Volume III)
Drain offset height (in|mm)	0.39 in | 10 mm
Drain delay (hours)	48
Open level (in|mm)	70.87 in | 1800 mm
Closed level (in|mm)	0.39 in | 10 mm

10. Once entered keep the **LID Control Editor** open (please note the screenshot shown is in SI units, US units will differ).



Now we will set up a bioretention area LID.

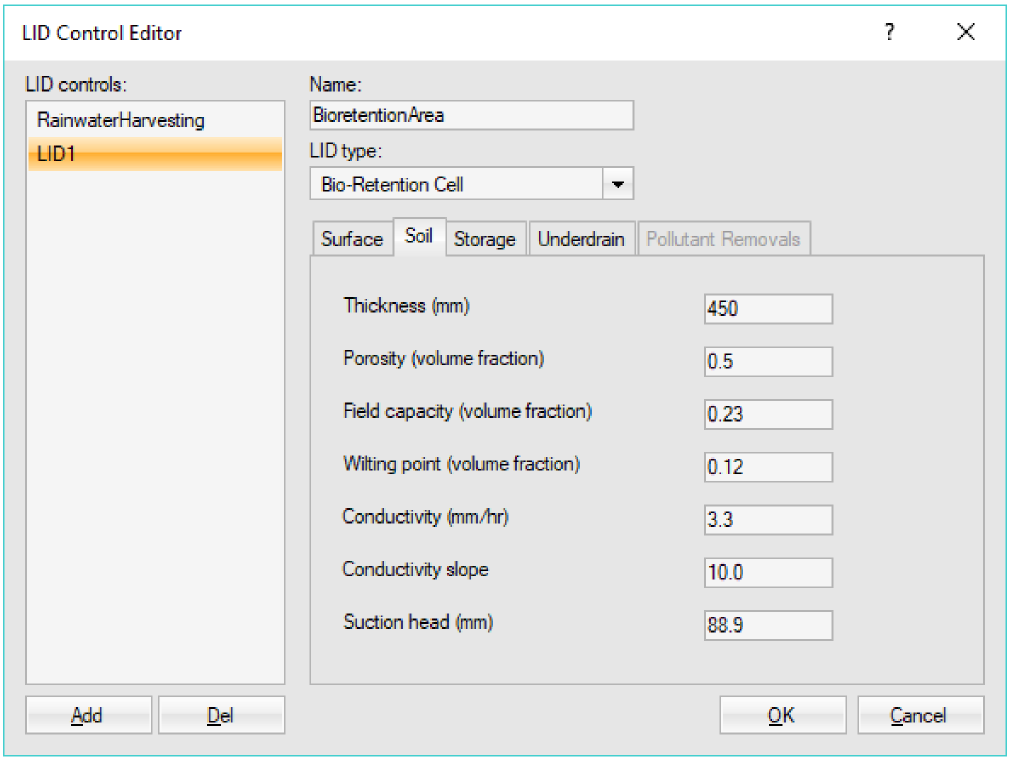
11. Add another LID named **BioretentionArea**.
12. Click the **Add** button again in the **LID Control Editor** to add a second LID.
13. Re-name the newly created LID to **BioretentionArea.**
14. Select the **LID type** to be **Bio-Retention Cell**.
15. Enter the properties for the **Surface, Soil, Storage** and **Underdrain** layers as shown in the image.

Parameters for the Bioretention Area LID

Surface	
Berm height (in|mm)	2 in | 50 mm (assumed)
Vegetation volume (fraction)	0.1 (assumed)
Surface roughness (Manning's n)	0
Surface of slope (percent)	0
Soil	
Thickness (in|mm)	18 in | 450 mm (page 136 of SWMM5 User's guide)
Porosity (volume fraction)	0.5 (assumed)
Field capacity (volume fraction)	0.23 (FC for loam – page 816 SWMM User's guide)
Wilting point (volume fraction	0.12 (WP for loam– page 816 SWMM User's guide)
Conductivity (in/hr | mm/hr)	0.13 in/hr | 3.3 mm/hr (K for loam – page 816 SWMM User's guide)
Conductivity slope	10.0 (page 136 SWMM User's guide)
Suction head (in|mm)	3.5 in | 88.9 mm (SH for loam – page 816 SWMM User's guide)
Storage	
Thickness (in | mm)	8 in | 200 mm (assumed ~0.7 ft | 0.2 m depth)
Void ratio (voids/solids)	0.75 (page 137 SWMM User's Guide)
Seepage rate (in/hr | mm/hr)	0.4 in/hr | 10 mm/hr
Clogging factor	0
Underdrain (not being modeled)	
Drain coefficient (in/hr | mm/hr)	0 in | mm
Drain exponent	0.5
Drain offset height (in | mm)	0 in | mm
Open level (mm | in)	0
Closed level (mm | in)	0

Provided as an example only. Your values or units may differ.

16. Once entered keep the **LID Control Editor** open (please note the screenshot shown is in SI units, US units will differ).



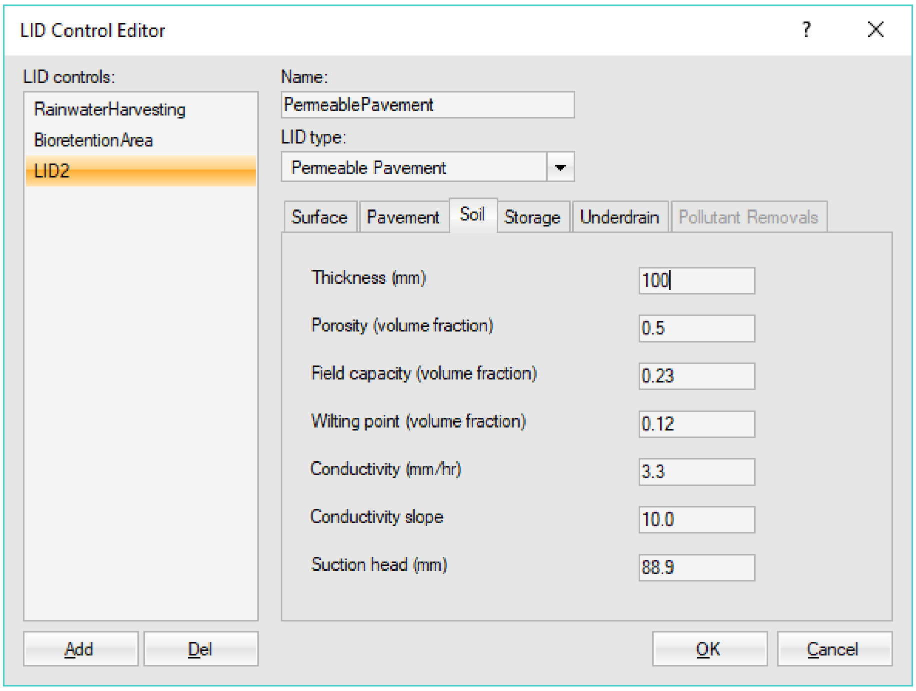
We will now set up the parameters for the Permeable pavement LID.

17. Add another LID called **PermeablePavement**.
18. Click the **Add** button again in the **LID Control** editor to add a third LID**.**
19. Re-name the newly created LID to be **PermeablePavement**.
20. Select the **LID type** to be **Permeable Pavement**.
21. Enter the properties for the **Surface, Pavement, Soil, Storage** and **Underdrain** layers as shown.

Properties for the Permeable Pavement LID

Surface	
Berm Height (in | mm)	0.8 in | 20 mm (assumed)
Vegetation volume (fraction)	0
Surface roughness (Manning's n)	0.02
Surface slope (percent)	1.0 (assumed)
Pavement	
Thickness (in| mm)	5.9 in | 150 mm (page 135 of SWMM5 User's Guide)
Void ratio (voids/solids)	0.21 (page 135 of SWMM5 User's Guide)
Impervious surface (fraction)	0 (assume continuous pavement system – page 135 of SWMM5 User's Guide)
Permeability (in/hr | mm/hr)	78.7 in/hr | 200 mm/hr (assumed from page 134 of SWMM5 User's Guide)
Clogging factor	83
Regeneration interval (days)	0 (new to 5.1.013, not used in this exercise)
Regeneration fraction	0 (new to 5.1.013, not used in this exercise)
Soil	
Thickness (in | mm)	4 in | 100 mm
Porosity (volume fraction)	0.5 (assumed)
Field capacity (volume fraction)	0.23 (FC for loam – page 816 SWMM User's Guide)
Wilting point (volume fraction)	0.12 (WP for loam – page 816 SWMM User's Guide)
Conductivity (in/hr | mm/hr)	0.13 in/hr | 3.3 mm/hr (K for loam– page 816 SWMM User's Guide)
Conductivity slope	10.0 (page 136 SWMM User's Guide)
Suction head (in|mm)	3.5 in | 88.9 mm (page 137 SWMM User's Guide)
Storage	
Thickness (in | mm)	11.8 in | 300 mm (page 137 SWMM User's Guide)
Void ratio (voids/solids)	0.75 (page 137 SWMM User's Guide)
Seepage rate (in/hr | mm/hr)	0.4 in/hr | 10 mm/hr (assumed)
Clogging factor	0 (not modeling clogging)
Underdrain	
Drain coefficient (in/hr | mm/hr)	0.008 in/hr | 0.2 mm/hr (page 138; n=0.5 y, Hd = 30 mm, q = 3.3, K for loam; page 816 SWMM5 User's Guide)
Drain exponent	0.5 (assumed)
Drain offset height (in|mm)	1.2 in | 30 mm
Open level (mm)	0 mm
Closed level (mm)	0 mm

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

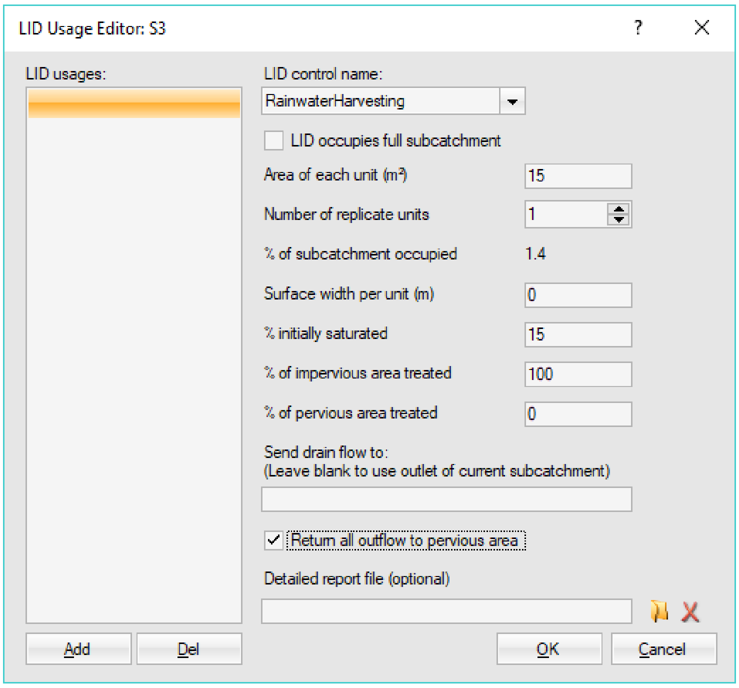


23. Click the **Save** button in the **Project panel** to save your model**.**

8.2 Assigning the LIDs to subcatchments

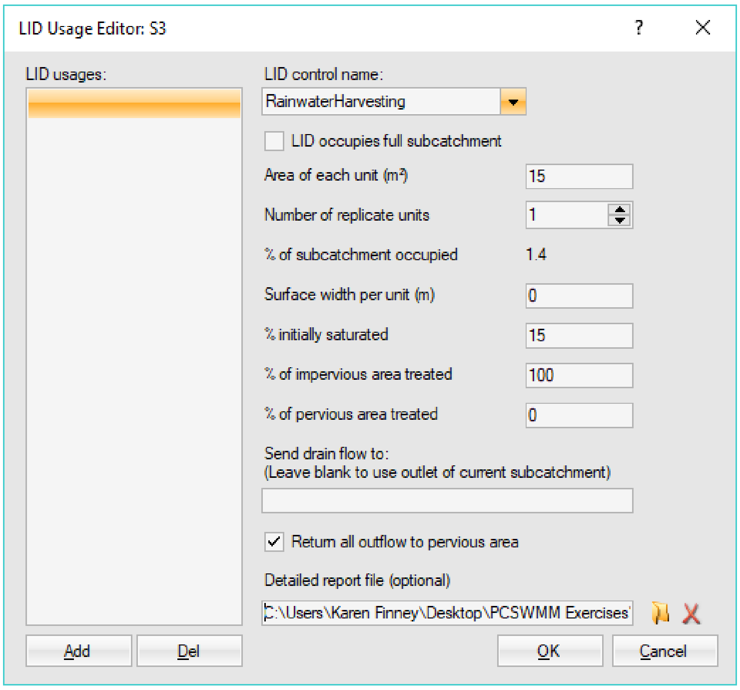
We now assign the LID subcatchments from the list of newly created LIDs. We'll begin with subcatchment **S3**. Subcatchment **S3** represents the roof of an entire building that was retrofitted with a rainwater harvesting unit, which will be represented as a rain barrel LID. Please note that it is assumed that this building has a green roof that makes up 20% of the roof area and thus the subcatchment has an imperviousness of 80%.

1. Select **S3** from the **Subcatchments** layer in the **Map panel**.
2. Click in the **LID Controls** in the **Attributes panel** and click on the **ellipsis** button.
3. Click on the **Add** button.
4. Select **RainwaterHarvesting** from the **LID control name** drop-down list.
5. Assign the **Area of each unit** as **161** ft^2 | **15** m^2 (assumes a 2 m high rainwater collection tank with 15 m^2 surface area).
6. The **Surface width per unit (ft | m)** is not applicable for rain barrels, so we will leave the default value of 0.
7. Change the **% initially saturated** to **15** and the **% of impervious area treated** to be **100**.
8. Leave the **% of pervious** area treated as 0.
9. Check **Return all outflow to pervious area**.



By sending outflow to a pervious area we simulate the situation where collected rainwater is used for irrigation within the subcatchment. If the harvested water is reused for other purposes you can use a storage unit with a pump. Now we'll specify a location to save the detailed LID report for S3.

10. In the **LID Usage Editor c**lick on the **Open**  button adjacent to the **Detailed report file**.

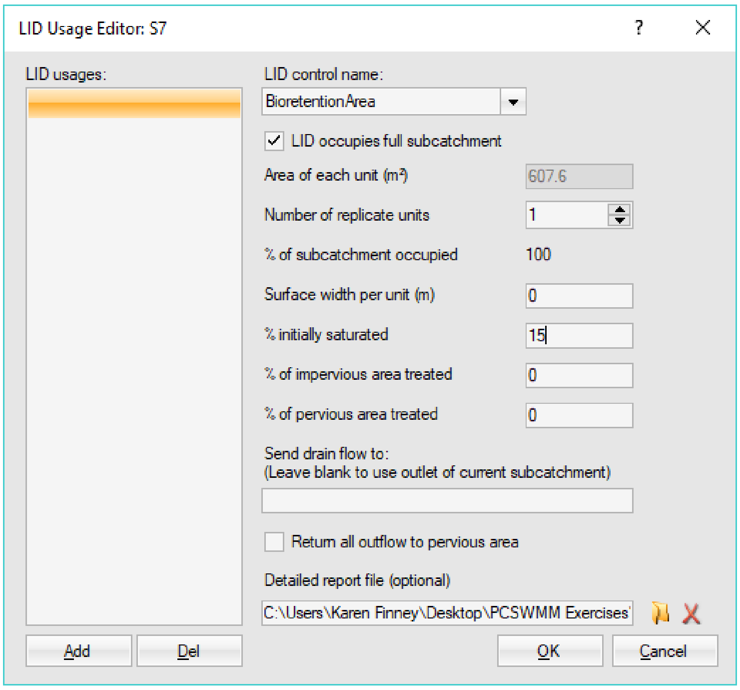


11. Browse to **PCSWMM exercises\K026\Initial** and create a new folder called **LID reports** and double click on the new folder. PCSWMM will name the report file S3 RainwaterHarvesting.txt by default.
12. Click **Save** and **OK** to close the **LID Usage Editor**.

Note: Although we are specifying the location of the LID detailed reports the report files will not be created until after SWMM5 has been run.

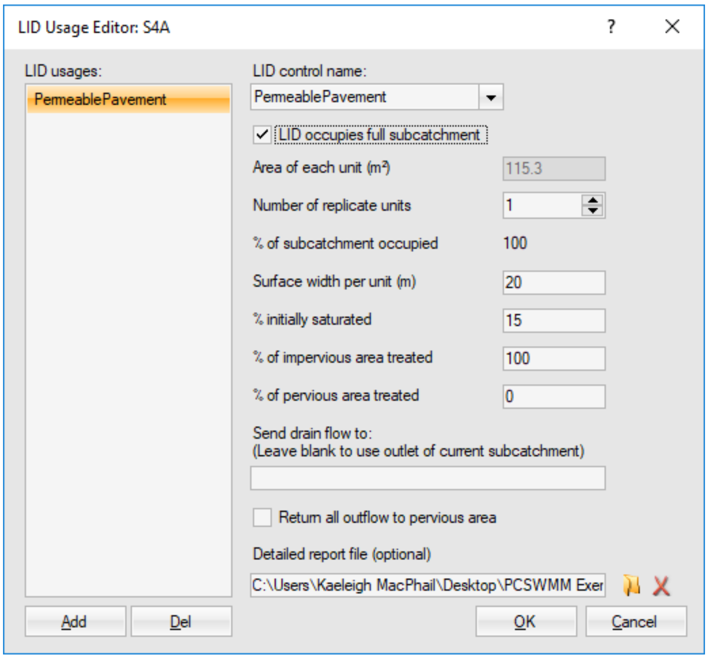
Now let's set up LID properties for subcatchment **S7** or the bioretention area.

13. Select **S7** from the **Subcatchments** layer in the **Map panel**.
14. Click in the **LID Controls** in the **Attributes panel** and click on the **ellipsis** button.
15. Click the **Add** button.
16. Select **BioretentionArea** from the **LID control name** drop-down list.
17. Check **LID occupies full subcatchment**. This will automatically change the area of the LID to be 100% of the subcatchment occupied.
18. Change the **% initially saturated** to **15** (please note the screenshot shown is in SI units, US units will differ).
19. Leave the **% of pervious area treated** as **0**.
20. We need to specify a location to save the detailed LID report for **S7**.
21. In the **LID Usage Editor** click on the **Open** button, browse to **PCSWMM exercises\K026\Initial\LID reports**, PCSWMM will name the report S7 BioretentionArea.txt.
22. Click **Save** to close the **Browse** window and **OK** to close the **LID Usage Editor**.



Now set up the LID properties for subcatchment **S4A** (the permeable pavement area).

23. Select subcatchment **S4A** from the **Subcatchments** layer in the **Map panel**.
24. Click the **LID Controls** in the **Attributes panel** and click on the **ellipsis** button.
25. Click on the **Add** button.
26. Select **PermeablePavement** from the LID control name drop-down list.
27. Check **LID occupies full subcatchment**.
28. Change the **Surface width per unit (ft | m)** to **66 ft |20 m**, the **% initially** saturated to **15** and the **% of impervious area treated** to **100** as shown in the following screenshot (please note the screenshot shown is in SI units, US units will differ).
29. Leave the **% of pervious area treated** as **0**.



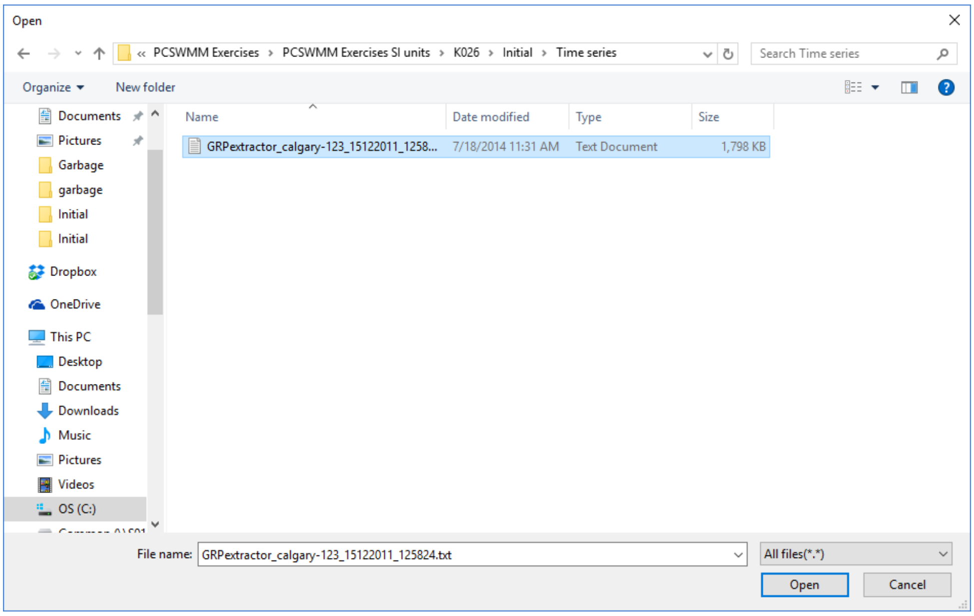
Let's specify a location to save the detailed LID report for S4A.

30. In the **LID Usage Editor** click on the **Open** button, browse to **PCSWMM exercises\K026\Initial\LID reports.**
31. Click **Save** to close the Browse window and **OK** to close the **LID usage editor**.

8.3 Loading continuous rain gage time series data

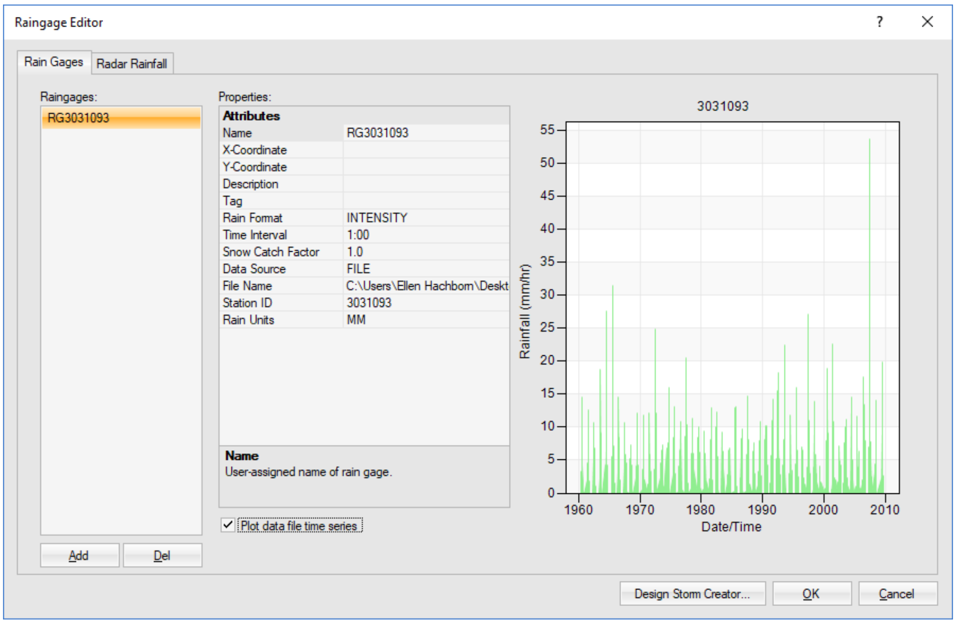
We will now run the model using a continuous long-term time series file. For this example, the units for the rainfall will be left as mm (even for US units) as the original precipitation data is in mm. Let's begin by creating a rain gage.

1. Add a new rain gage named **RG3031093**.
2. Click on the **Rain Gages** item in the **Project panel**. It will appear gray as there are currently no Rain Gages in the model.
3. Click on the **Add** button in the **Raingage Editor** under the **Rain Gages** tab.
4. Change the **Name** to **RG3031093**.
5. Set the **Rain Format** to **INTENSITY** (default) and set the **Time interval** to **1:00** (default).
6. Set the data source to be from a **FILE** called **GRPextractor_calgary-123_15122011_125824.txt** file from the **PCSWMM exercises \K026\Initial\Time series** folder.
7. Under the **Data Source** field, select **FILE.**
8. Click on the **ellipsis**  button in the **File Name** field and navigate to **PCSWMM exercises \K026\Initial\Time series.**
9. In order to see the time series, change the **Filter** from **.dat** to show **All files** in the open dialog.
10. Select **GRPextractor_calgary-123_15122011_125824.txt** and click the **Open** button.

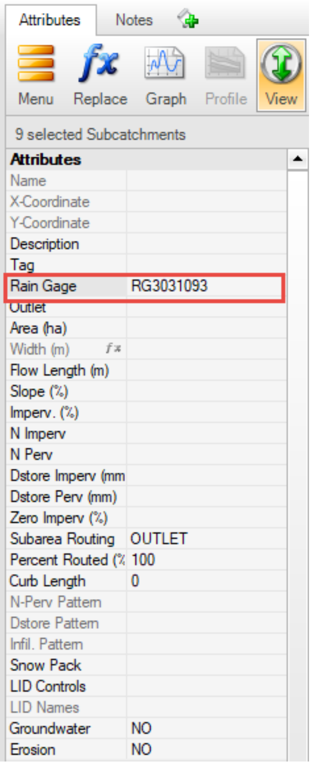


11. In the **Station** ID field type in **3031093**.
12. Set the **Rain Units** to be **MM** (for US units as well).
13. Check **Plot data file time series** to display the time series in the preview window.
14. Click **OK** button to close the **Raingage Editor**.

Note: In this exercise we are using Environment Canada rain gage data directly from the source file.



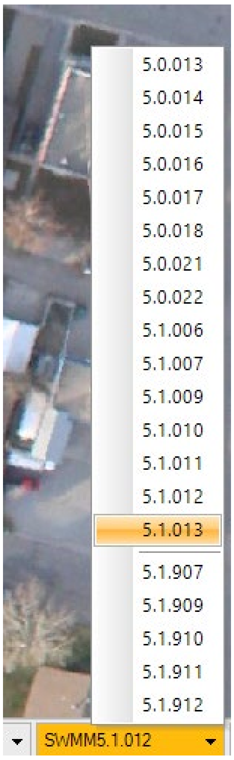
15. Assign the **RG3031093** rain gage to all the subcatchments.
16. Click on the **Subcatchments** from the **Layers panel**.
17. Press **Ctrl** + **A** to select all nine subcatchments.
18. Click on the **Rain Gage** attribute field in the **Attributes panel** and select **RG3031093** from the drop-down list.



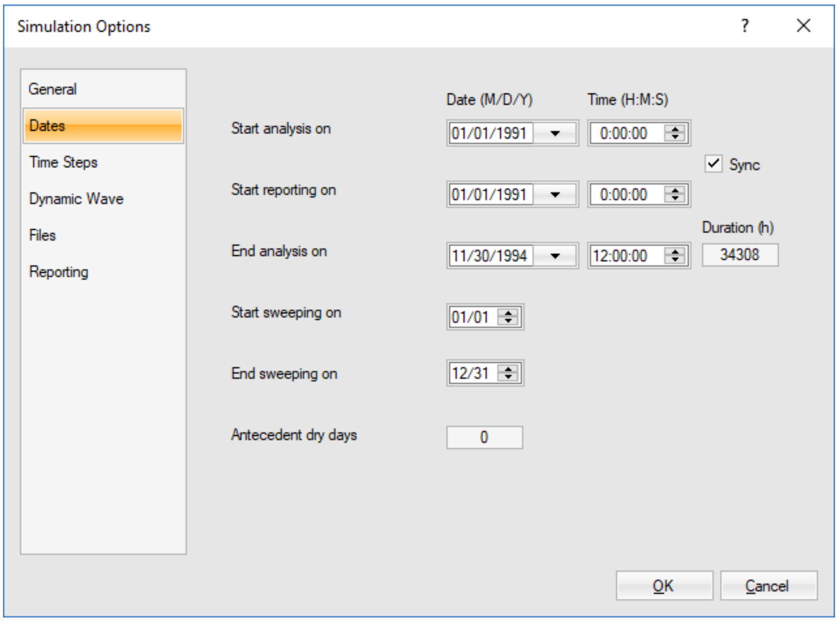
8.4 Running the model

Now we'll set the simulation options to run a 4 year simulation (1991-1994).

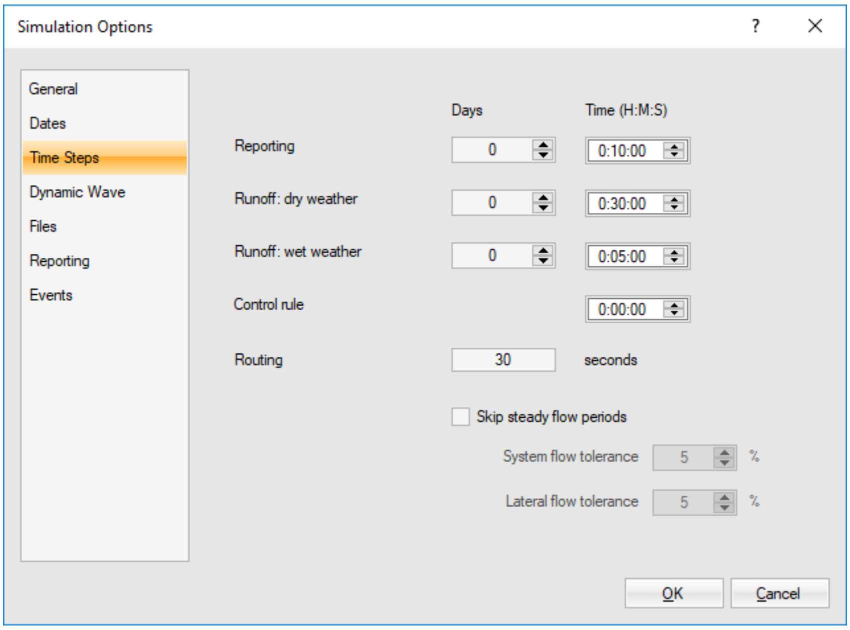
1. Change the SWMM engine to **SWMM5.1.013** by clicking on the SWMM version number box located at the bottom of the interface.



2. Click on the **Simulation options** from the **Project panel**.
3. Click on the **Dates** tab and set the **Start analysis on** to **01/01/1991** and **End analysis on** to **11/30/1994** at **12:00:00**. This can be easily done by first setting the start date and then entering 34308 hours in the **Duration** box.



4. Click on the **Time Steps** tab in the **Simulation Options** editor and change the **Reporting** time step to **10** min, the **Dry weather runoff** time step to **30** min, the **Wet weather runoff** time step to **5** min, and the **Routing** time step to **30** s.



5. Click on the **OK** button to close the **Simulation Options**.
6. Click on the **Run**  button in the **Project panel** to run the model.

8.5 Results comparison

1. Click on the **Status panel** tab to review the result summary report.
2. Click on **LID Results** from the list of sections in the **Status panel**.
3. Review the **LID Performance Summary** and check that the reported losses are reasonable.

The flow volume reduction in each LID considering total inflow and surface outflow can be calculated using the following equation (click view to display equation): . These percentages can be used in the water quality simulations.

$$\%\, Flow\ Volume\ Reduction = \frac{(Total\ Inflow - Surface\ Outflow)}{Total\ Inflow} \times 100$$

4. Click on the **Graph** tab to open the **Graph panel**.
5. Open up all three LID detailed report files from the **PCSWMM exercises\K026\Initial\LID reports folder.**
6. Open **LID report files** by clicking on the **Open** button and navigating to **PCSWMM exercises\K026\Initial\LID reports**.
7. Select all three LID reports by holding **Shift** while clicking on each of them.
8. Click **Open** to open the LID reports.
9. Plot the runoff time series by expanding **Subcatchments > Runoff** and select **S1A** and **S1B**.
10. Click on **Tools** > **Objective Functions**.

Compare the LIDs in terms of peak flows and runoff volumes. Because subcatchment S1A is 100% impervious and S1B is completely pervious (0% impervious), the Runoff from S1B should be significantly less (about half).

11. Click on the **Menu** button in the **Graph panel** and select **Clear Graph**.

Now plot the **Runoff** for **S1B**, **S2** and **S7**. The total runoff treated by **S7** can be calculated by adding the total runoff from **S1B** and **S2** (~48 090 ft^3 | 1361.8 m^3) and subtracting the total runoff from **S7** (~13 066 ft^3 | 370 m^3).

Repeat steps 9 and 10 by plotting the runoff for subcatchments **S4** and **S4A**. In this case **S4A** receives all the runoff from subcatchment S4 (parking lot). You will notice that the runoff from S4 is significantly larger than the runoff coming off of subcatchment S4A as there is permeable pavement treating the runoff.

Plot the **System rainfall** by expanding **System > Rainfall** and select **System.** Notice how the Subcatchment S4A has more runoff than S4 during the larger events. This is because during these events the LID would overtop due to the high intensity rainfall.

8.6 Evaluating LID quantity

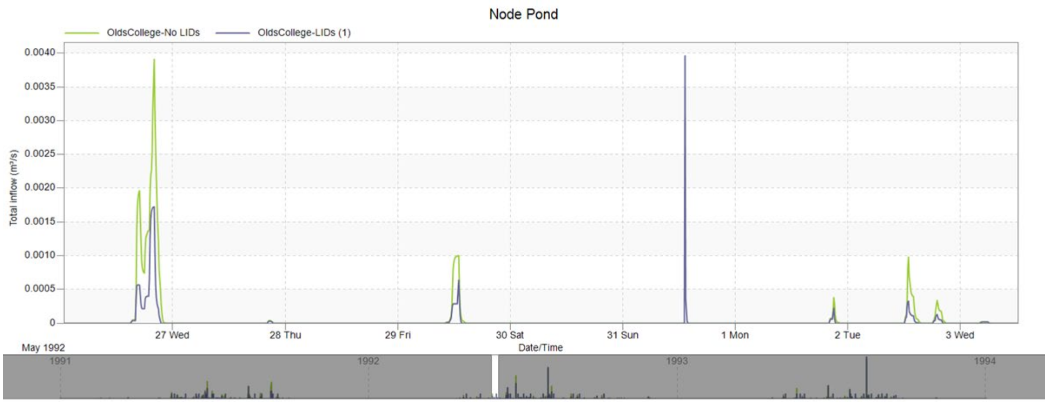
We will now evaluate the effect of LIDs on flow quantity by comparing results with a model scenario for the area without LIDs.

1. Duplicate the current scenario and name it **OldsCollege-No LIDs.**
2. Click on the **Plan** button in the toolbar of the **Project panel**.

3. In the **Scenario Manager,** click on the **Add** button and select **Duplicate Current Project.**
4. Name the project **OldsCollege-NoLIDs**.
5. In the **Description (Title)** box edit the text to read **Olds College water quantity analysis – No LIDs.**
6. Click the **Create** button to close the **Create Scenario** window.
7. Open the new scenario With the **Scenario manager** still open click on the newly created scenario **OldsCollege-NoLIDs** and click **Open**.

Choose to **Save project** if asked.

8. Delete all the LID controls.
9. Click **LID Controls** in the **Project panel**.
10. Select all of the LID controls.
11. Click the **Del** button, then click **OK**.
12. When asked whether to remove all references to deleted LID controls from subcatchments, select **Yes**. Now this scenario model does not have any LIDs.
13. Click on the **Run** button in the **Project panel** to run the model.
14. In the **Time series manager** (in the Graph panel) expand the **Nodes>Total inflow>PondOF** and examine the flow coming into the pond. Observe the results.
15. Click on the **Scenarios** button, check both scenarios and then **Compare Scenarios** to display the results of both runs.



Confirm the reduction in flow with LIDs by zooming in to individual events. Also observe additional flow occurring only with LIDs. Check the pond source by plotting subcatchment S3 runoff (Rain barrels release water after a 48 hour delay).

1. Expand **Subcatchments>Runoff>S3**
2. We can also compare model results by using the **Summary/Comparison** tool.
3. In the **Map panel**, click on the **Tools** button and select **Summary/Comparison**.
4. In the **Project Summary/Comparison** window, click the **Show Scenarios** button. Check the boxes next to the **OldsCollege-LIDs (1)** and **OldsCollege_No_LIDs** scenarios and click **Refresh**.
5. Under **Summary Tables,** compare the the scenarios by selecting **Runoff quantity continuity, Results statistics, Subcatchment attributes** etc.

www.hydrosoft.co.kr

하이드로소프트

Telephone: 031.8017.8033

support@hydrosoft.co.kr

www.hydrosoft.co.kr

Follow Us