MAGNET Storm Water Network Analysis

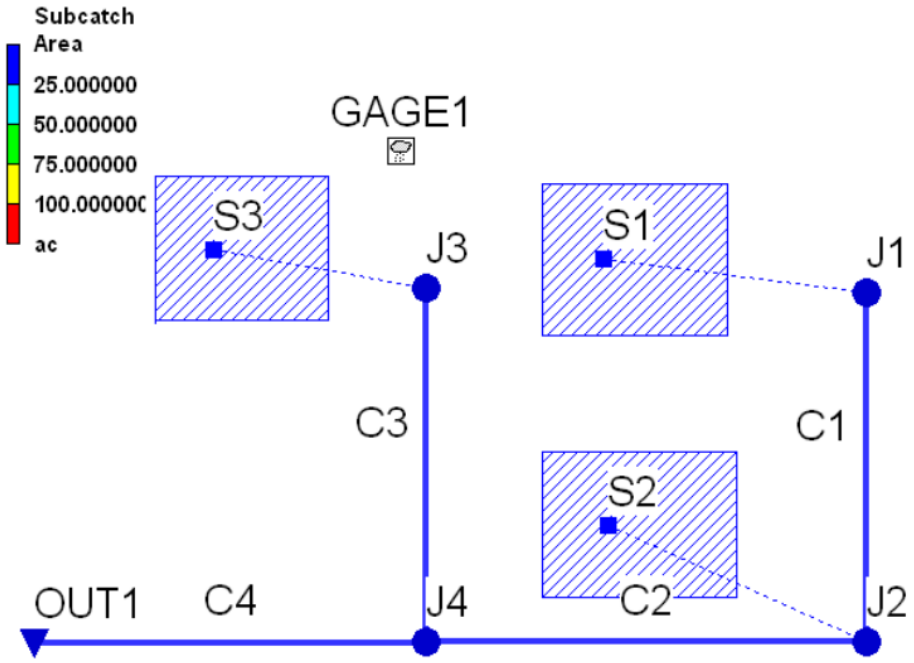
==

Quick Tutorial

MAGNET4WATER

Hydrosimulatics

Based on:
EPASWMM5 Tutorial with Images for SWMM 5.1.014
by Robert Dickinson



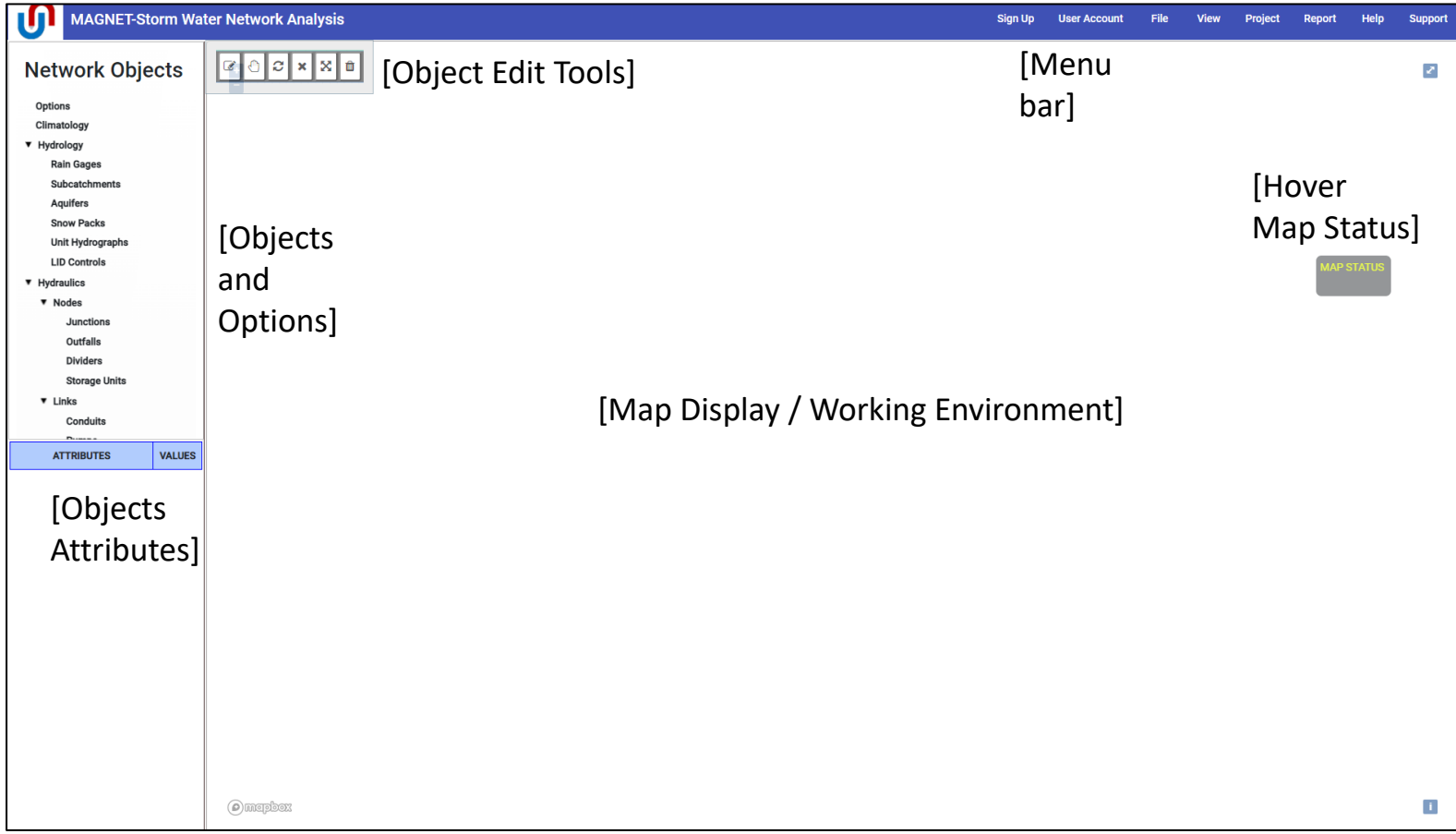
EPASWMM5 Tutorial with Images for SWMM 5.1.014

Browse to:

https://www.magnet4water.net/stormnet/

This site hosts the complete, integrated platform for storm water network modeling, analysis, and visualization.

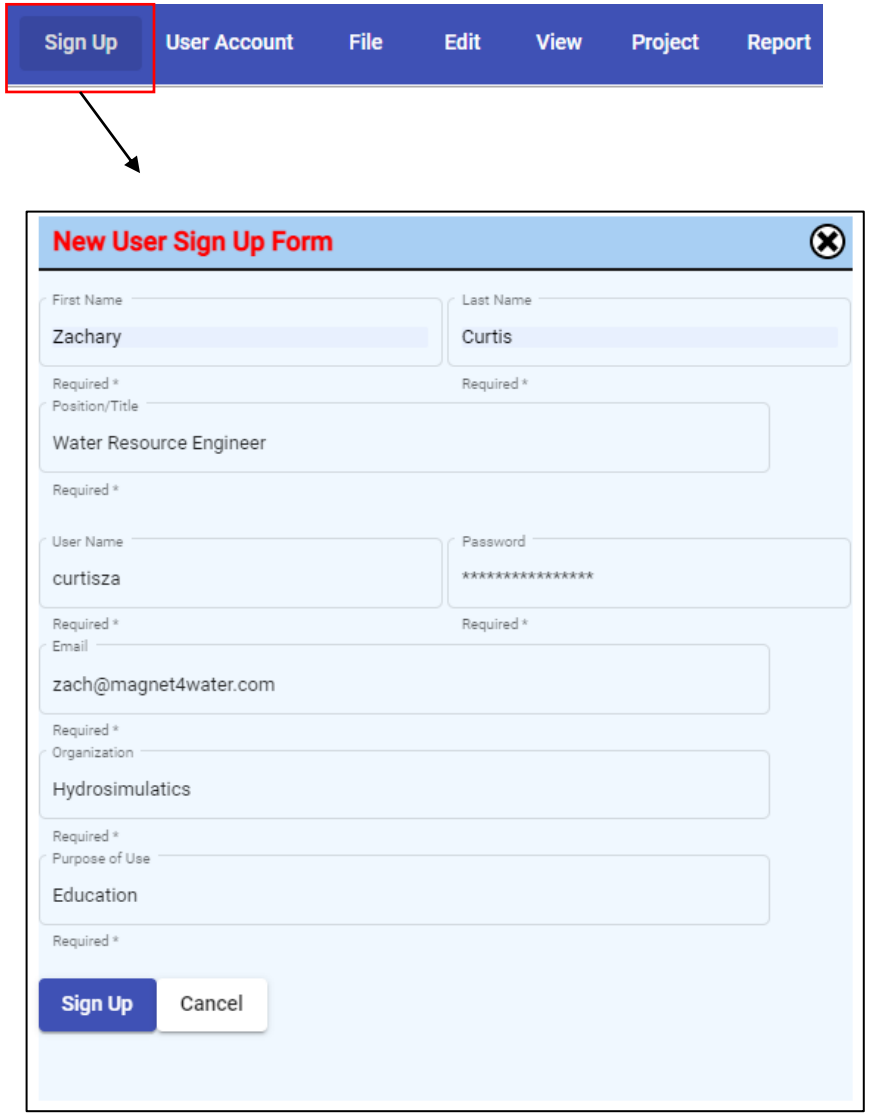
The default interface environment is shown here.



Create an account:

Sign Up [Menu Bar]
> Complete Form

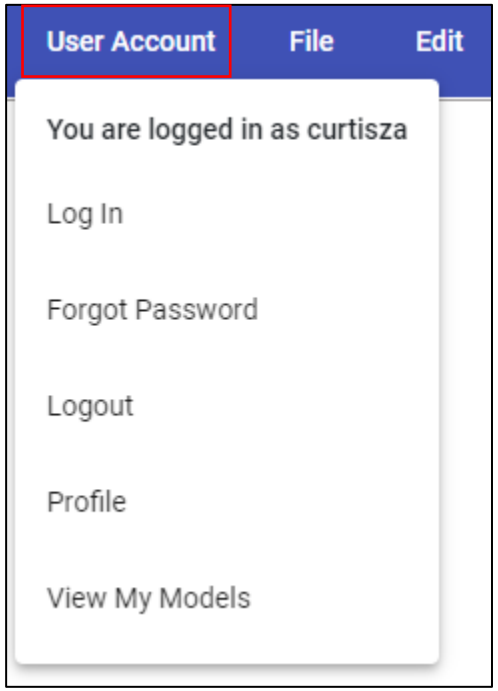
*The email you provide is used for resetting your password should you forget it.



User Account:

User Account [Menu Bar] > …

Use this menu to change request a password reset, update your email, user type, etc., or view and/or download previously executed models for later use.



[Change email, User Type, Password, etc]

[View previously executed models and download model files for later use]

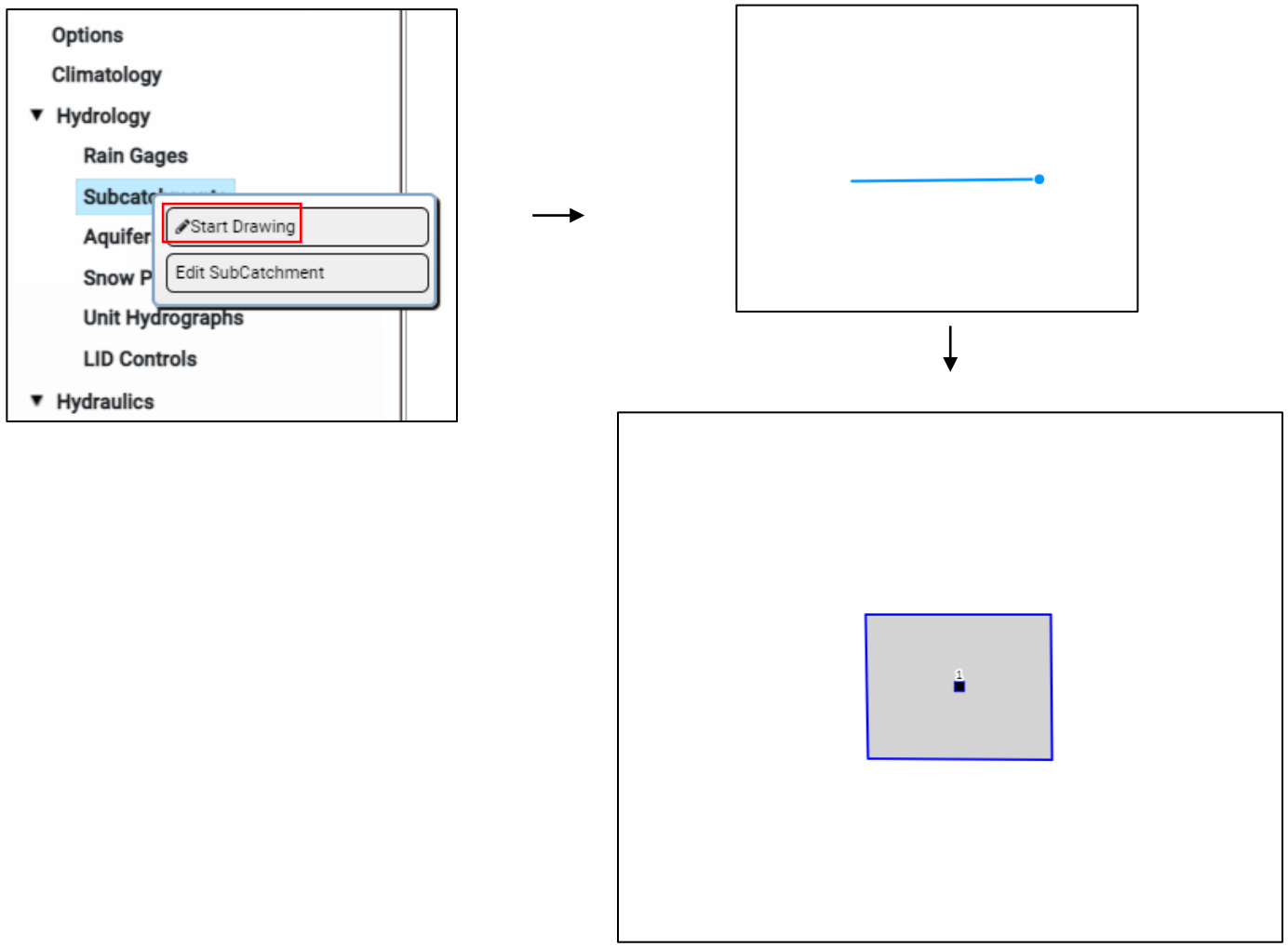
Add Subcatchments:

Hydrology > Subcatchments > Start Drawing

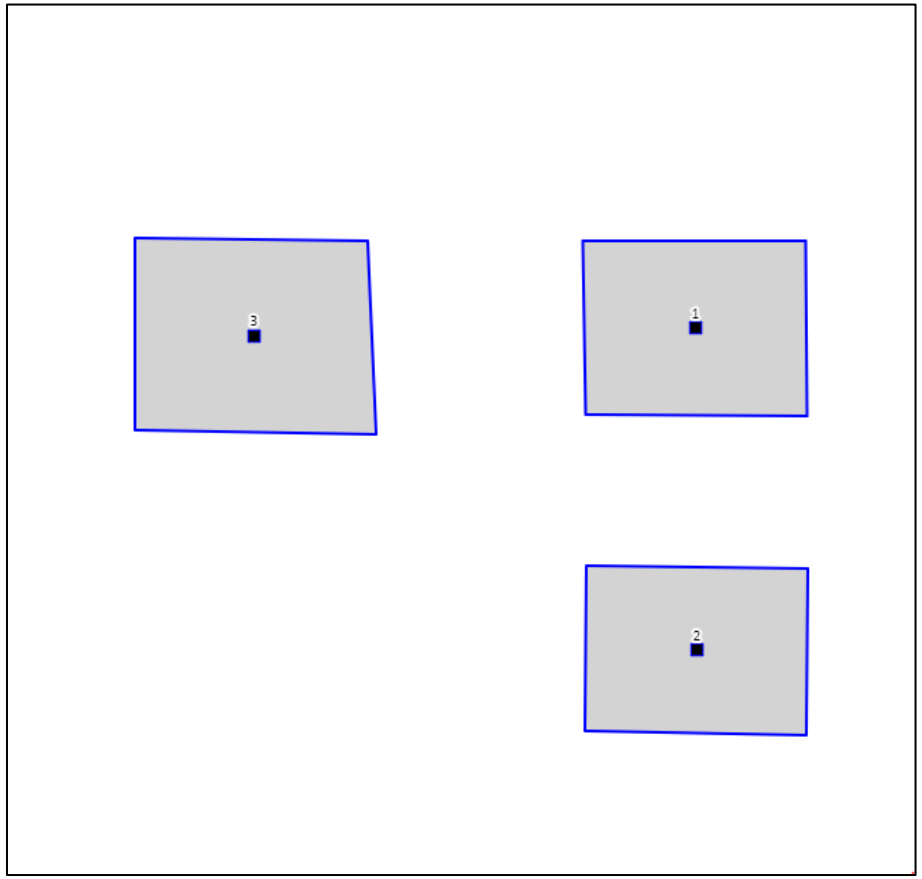
The cursor turns into a blue circle, indicating ‘Drawing Mode’ is active.

Use a single-click to add the first corner (vertex) of the subcatchment to the map display. Use additional single-clicks to add the other corners.

A double-click “closes” the subcatchment.



Repeat the steps on the previous slide to add the 2nd and 3rd subcatchments to the map display.



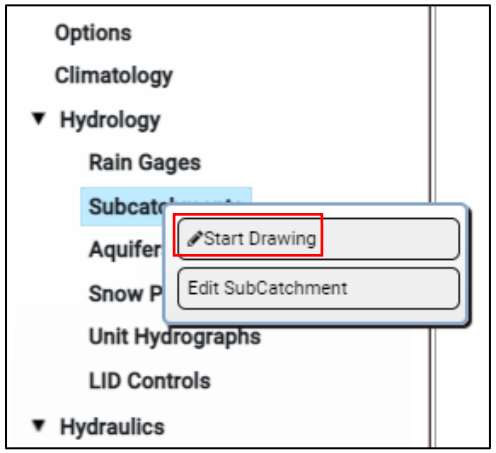
Edit Subcatchments:

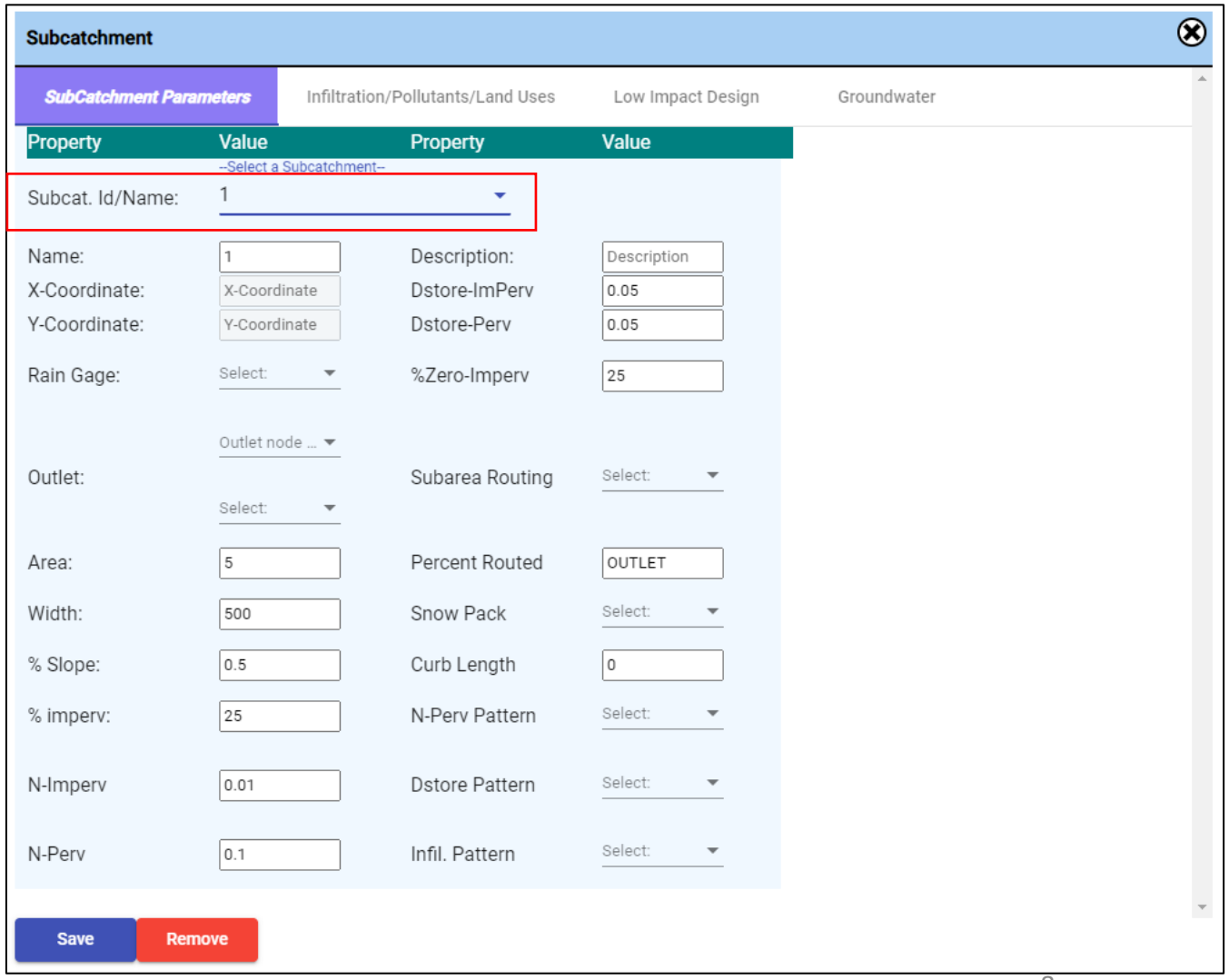
Hydrology > Subcatchments > Edit Subcatchments

*A single-click on any map object also launches its Editor menu.

Select the first subcatchment by its Subcat. ID/Name (1 for this example).

Default values are provided, but you can assign different values (next slide).

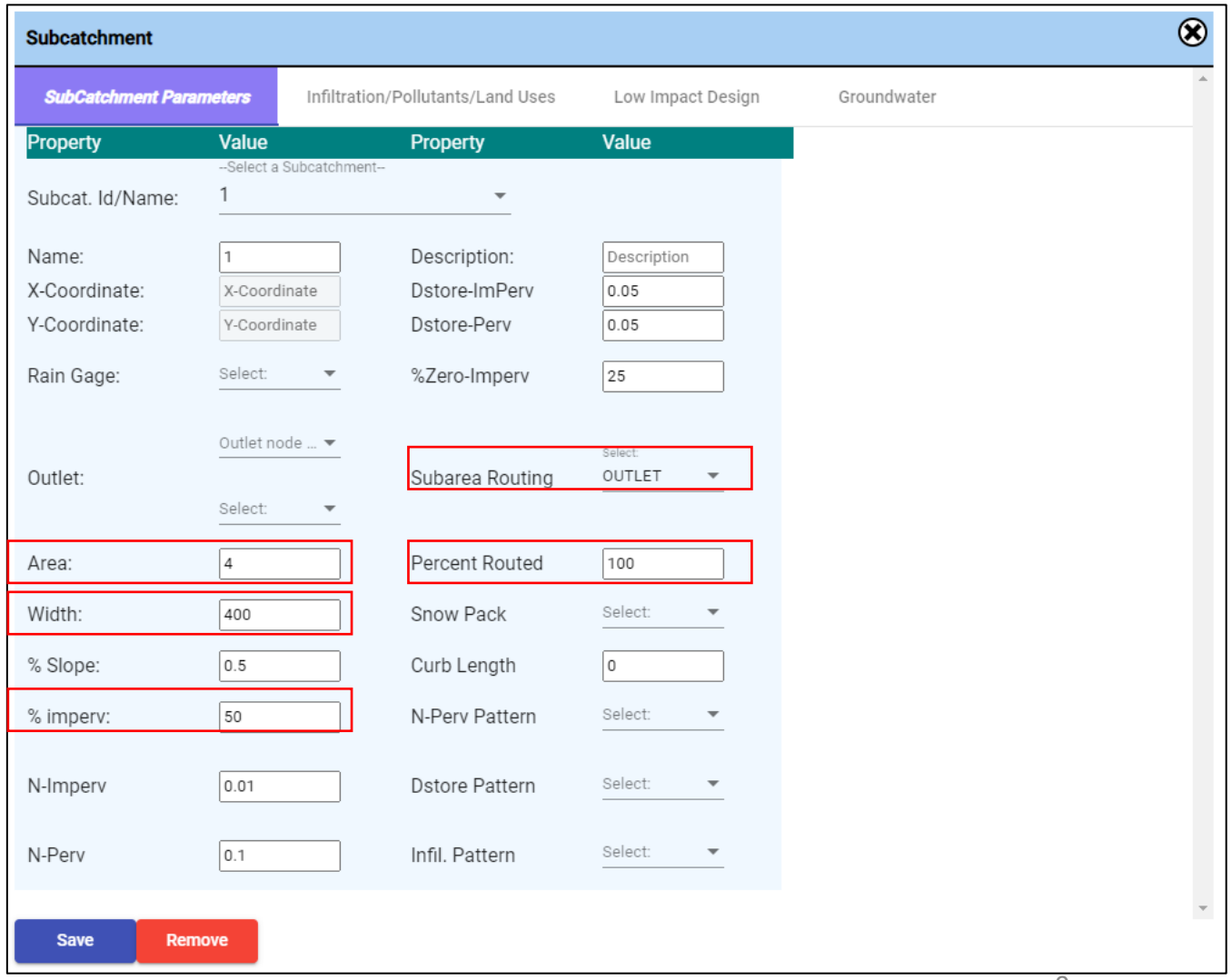




The Area and Width in this subcatchment and all others should be set to 4 and 400, respectively.

The percentage of impervious land cover should be set to 50%.

Make sure the Subarea Routing is set to OUTLET and the Percent Routed is 100.

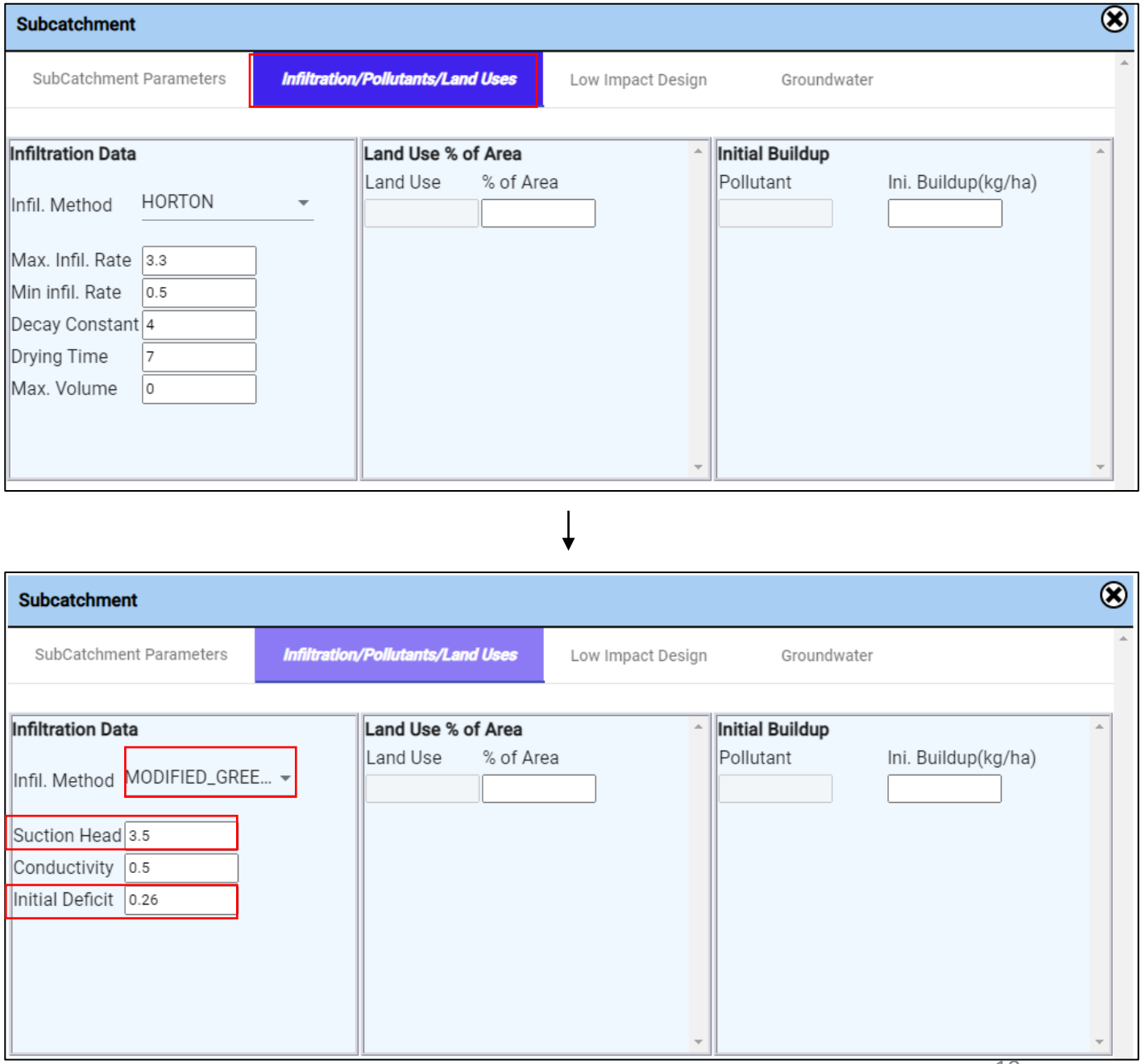


Still within the Subcatchment Editor, navigate to the Infiltration/Pollutants/Land Uses tab.

Select MODIFIED_GREEN_AMPT as the Infil. Method under Infiltration Data.

Assign a Suction head of 3.5, a Conductivity of 0.5, and an Initial Deficit of 0.26

Click ‘Save’ to finalize changes before exiting the Subcatchment Editor (‘X’ button).

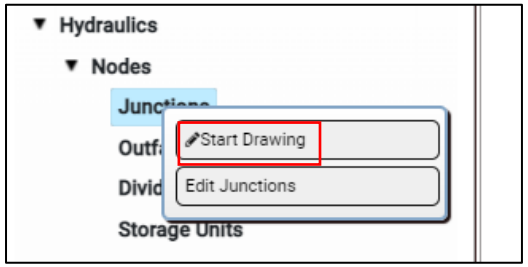


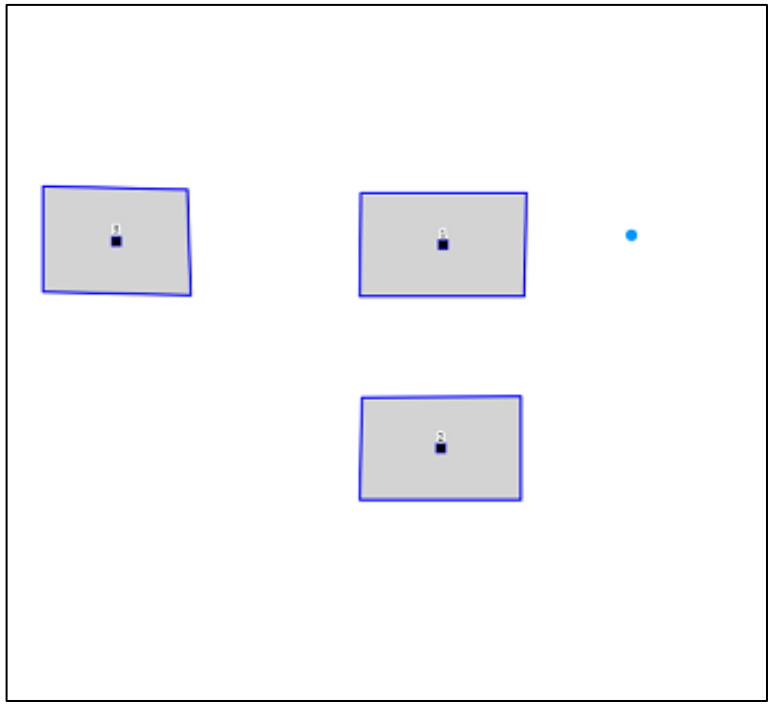
Add Junctions:

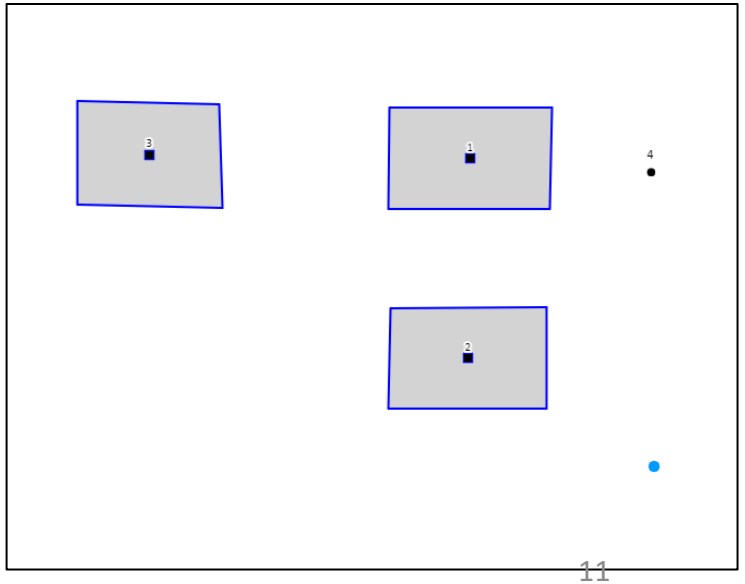
Hydraulics > Nodes > Junctions > Start Drawing

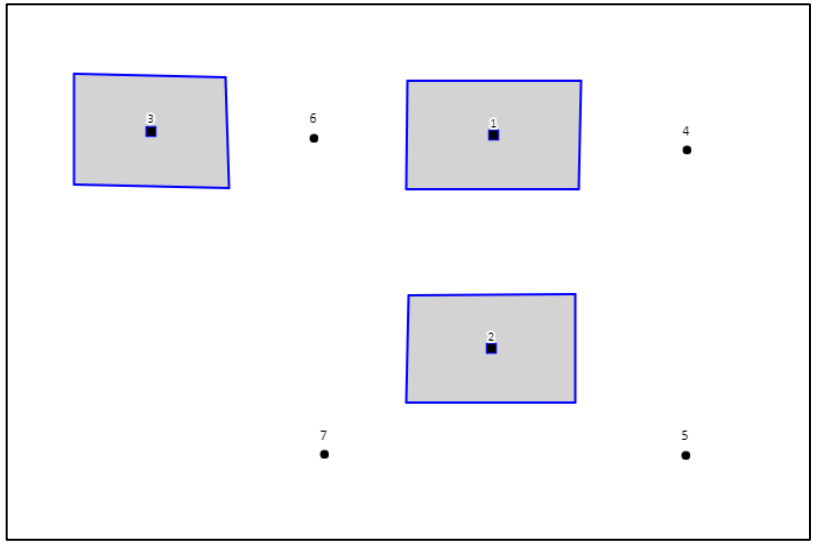
The cursor turns into a blue circle, indicating ‘Drawing Mode’ is active.

Use a single-click to add the first junction to the map display. Then use additional clicks to add the remaining junctions while in Drawing mode.



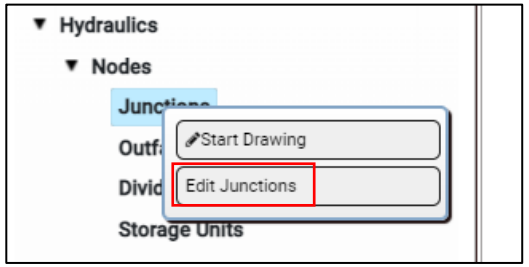


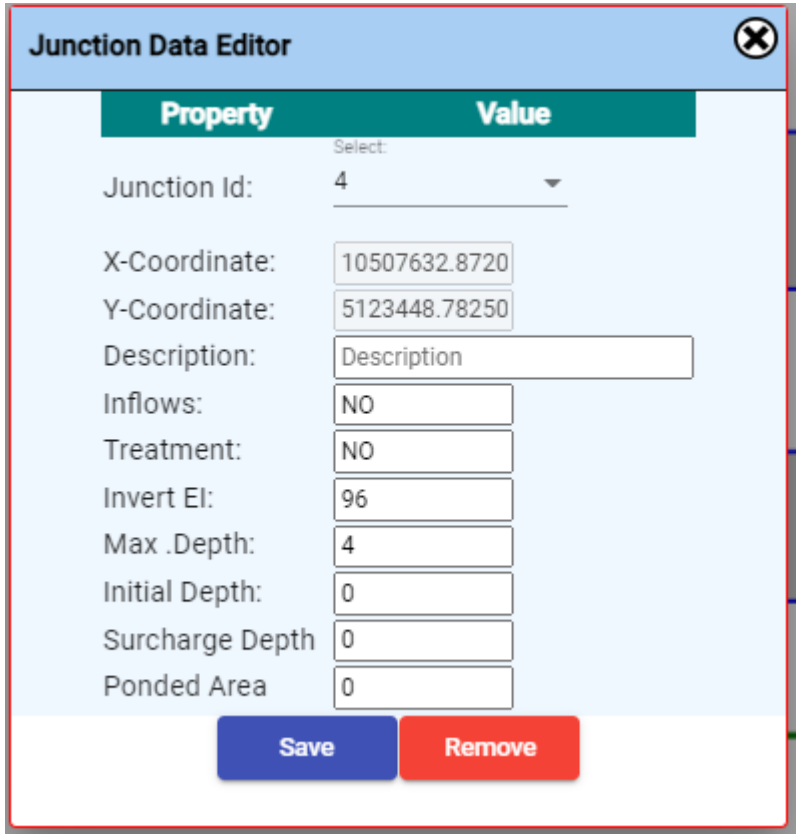




Edit Junctions:

Hydrology > Nodes > Junctions > Edit Junctions





Select the different junctions by their Junction ID (4-7 for this example) for attribute editing.

The invert elevation (Invert El) in Junction 4 should be set to 700ft. The Max Depth should be set to four.

Click ‘Save’ to finalize changes before selecting a new junction for editing: Junction 5, edit attributes, Save; Junction 6, edit, Save; and so on…

After updating and saving all of the junctions, close the Junction Data Editor window (‘X’ button).

Junction	Invert El
4	96
5	90
6	93
7	88

Add Conduits:

Hydraulics > Links > Conduits > Start Drawing

Again, the cursor turns into a blue circle, indicating ‘Drawing Mode’ is active.

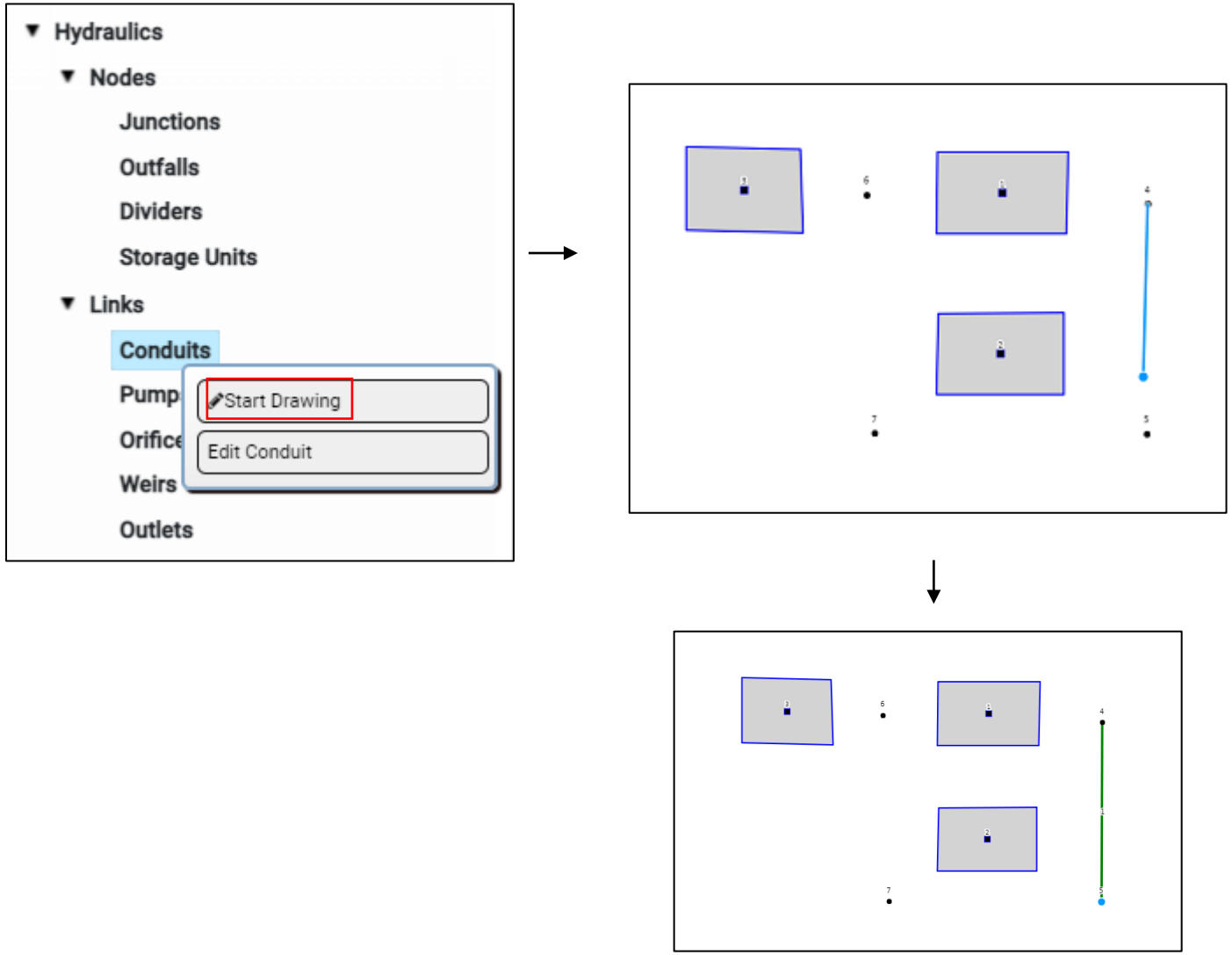
Click once on Junction 4 to snap the conduit’s starting position to it. A blue line will extend from the node as you move the cursor.

Double click on Junction 5 to snap the conduit’s ending position to it.

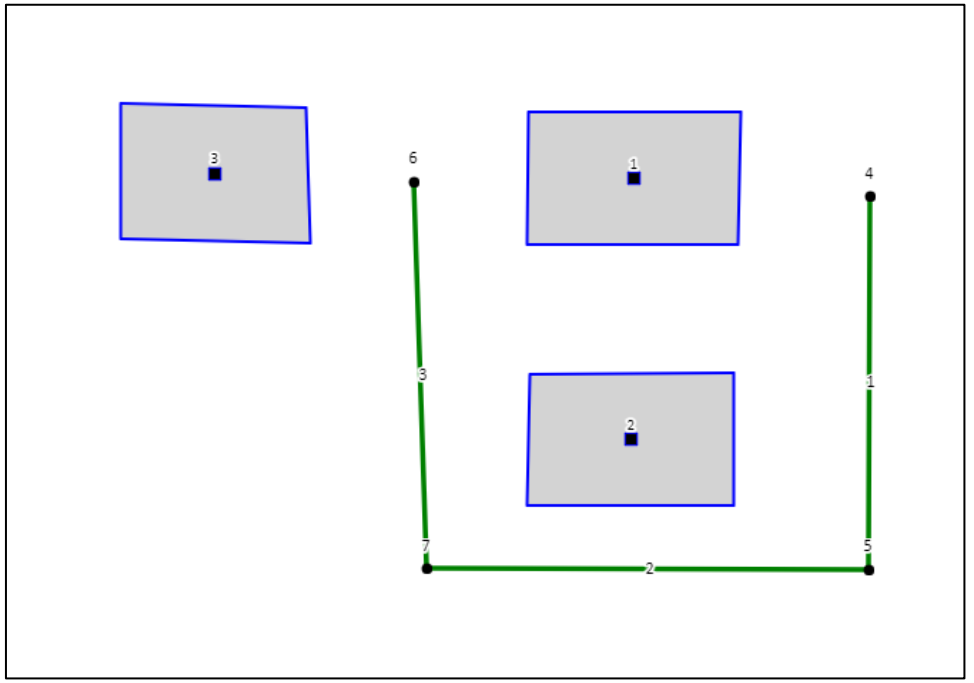
*Once a link is started (i.e., snapped to the starting node), a single click only places a vertex for drawing purposes. The link is still “open” (not snapped to an end node).

**A link must have a starting and ending node. “Bad” pipes are deleted from the map display.

***Conduits should be drawn from the upstream Junction to the downstream Junction (i.e., from higher to lower elevation).



Repeat the steps from the previous slide to add Conduit 2 (connecting Junction 5 and 7) and Conduit 6 (J3 to J7).

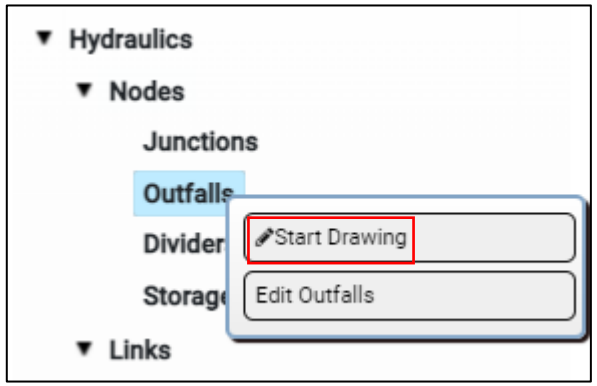


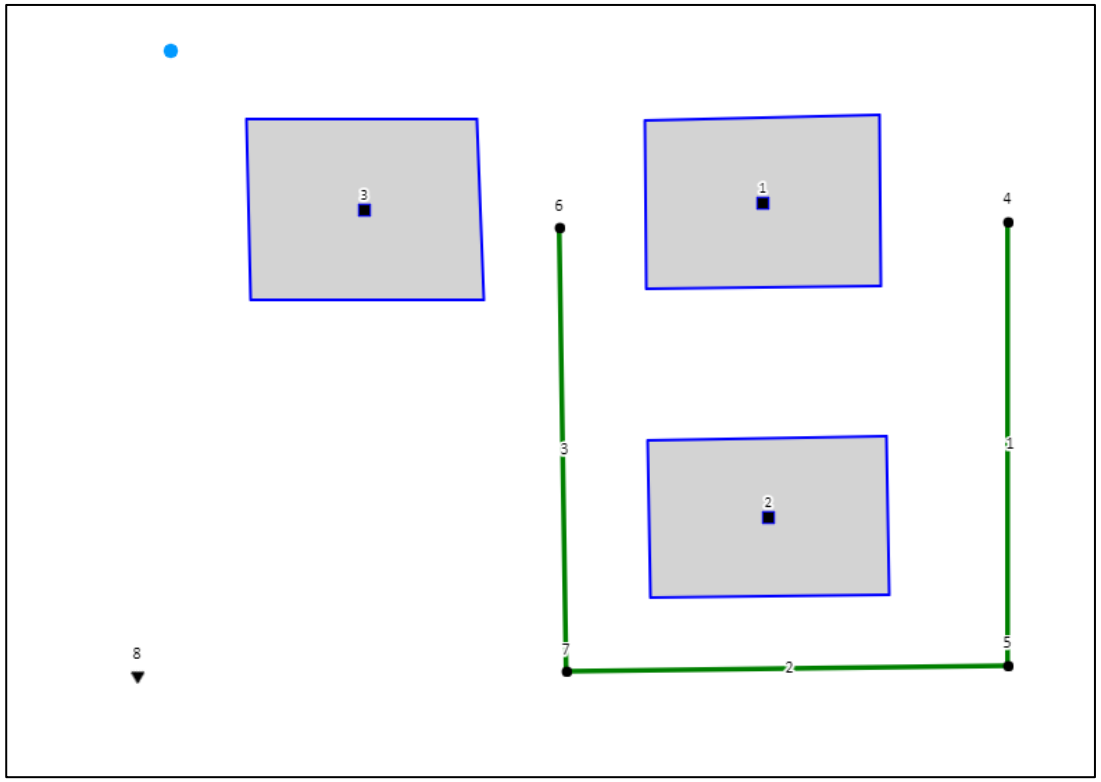
Add Outfall:

Hydrology > Nodes > Outfalls > Start Drawing

The cursor turns into a blue circle, indicating ‘Drawing Mode’ is active.

Use a single-click to add the outfall to the map display.





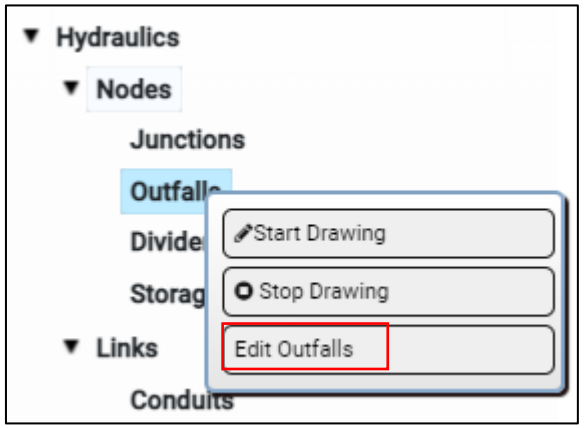
Edit Outfall:

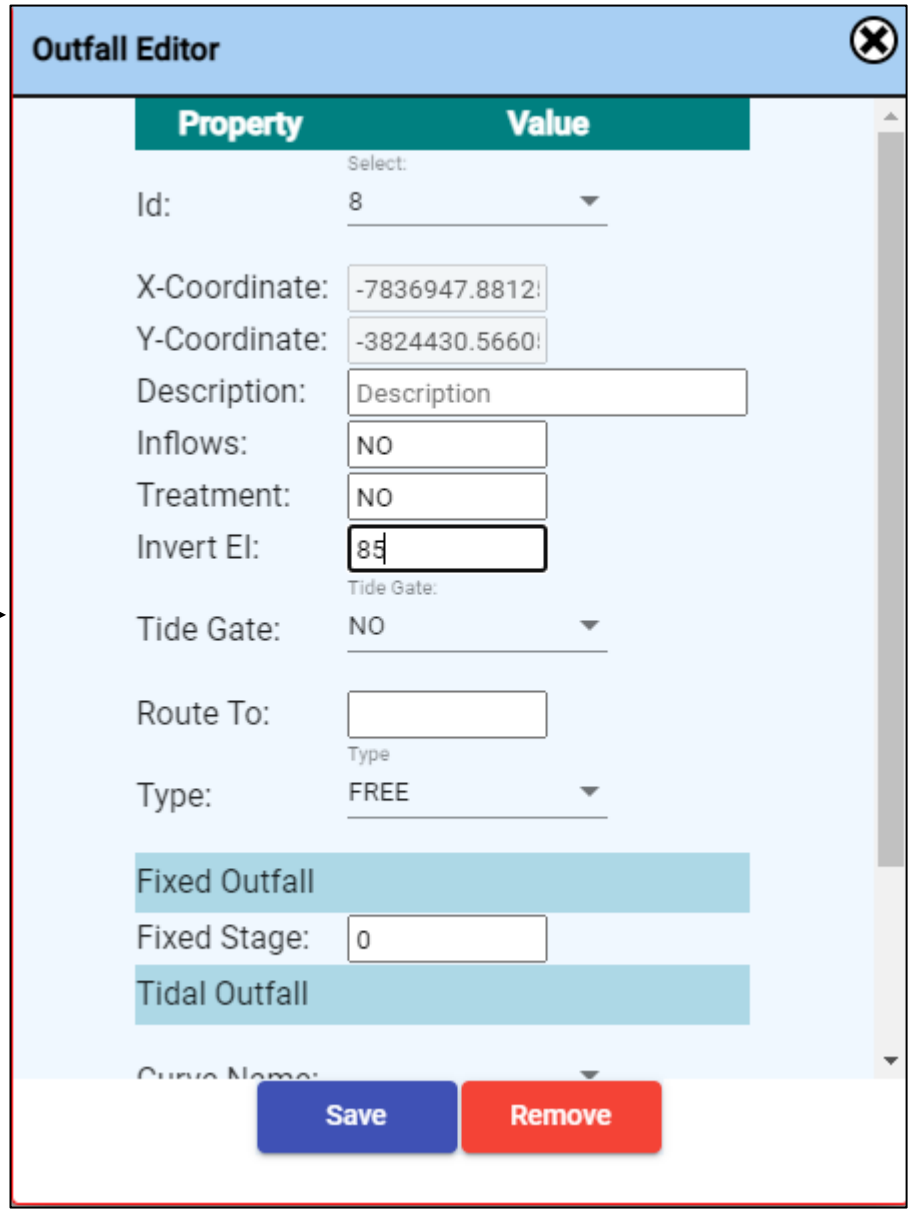
Hydraulics > Nodes > Outfalls > Edit Outfalls

Select the outfall by its ID (8 for this example) for attribute editing.

The invert elevation (Invert El) of the outfall should be 85. All other default values can be used.

Remember to Save before closing the Outfall Editor.

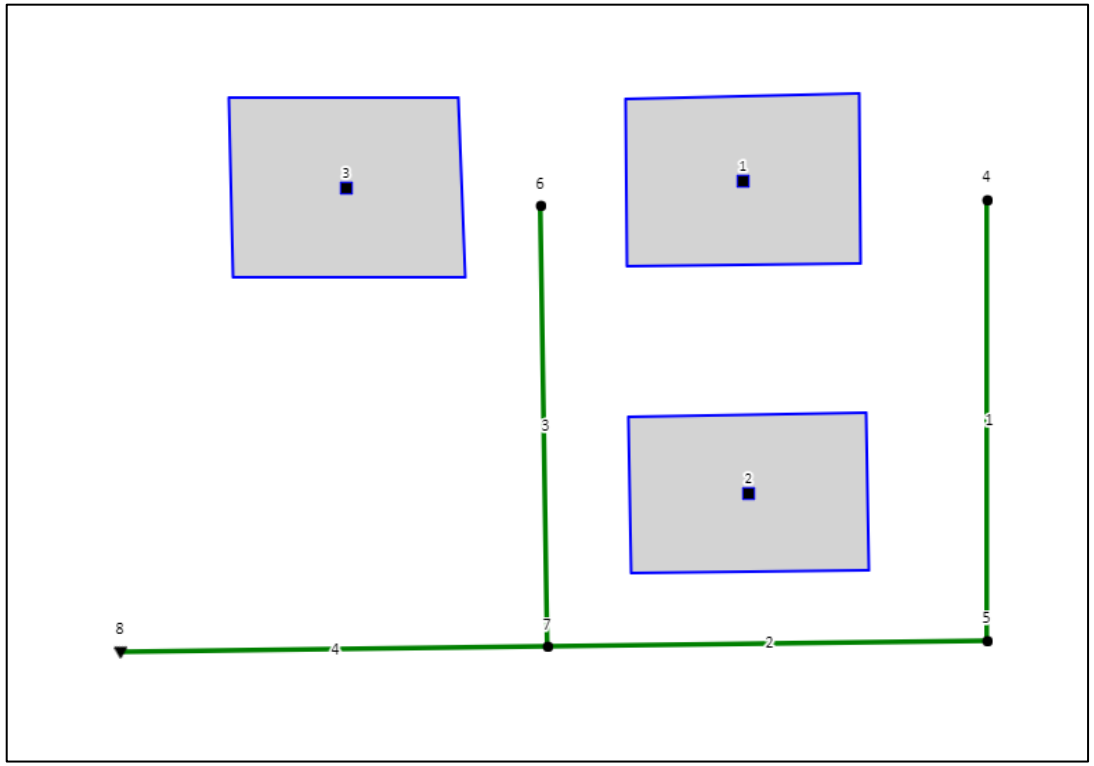




Add Last Conduit:

Hydraulics > Links > Conduits > Start Drawing

Connect Junction 7 to the Outfall (ID=8) using a Conduit (Conduit ID=4).



Edit Conduits:

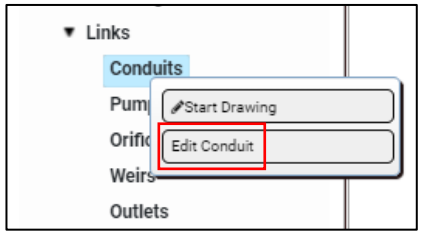
Hydraulics > Links > Conduits > Edit Conduit

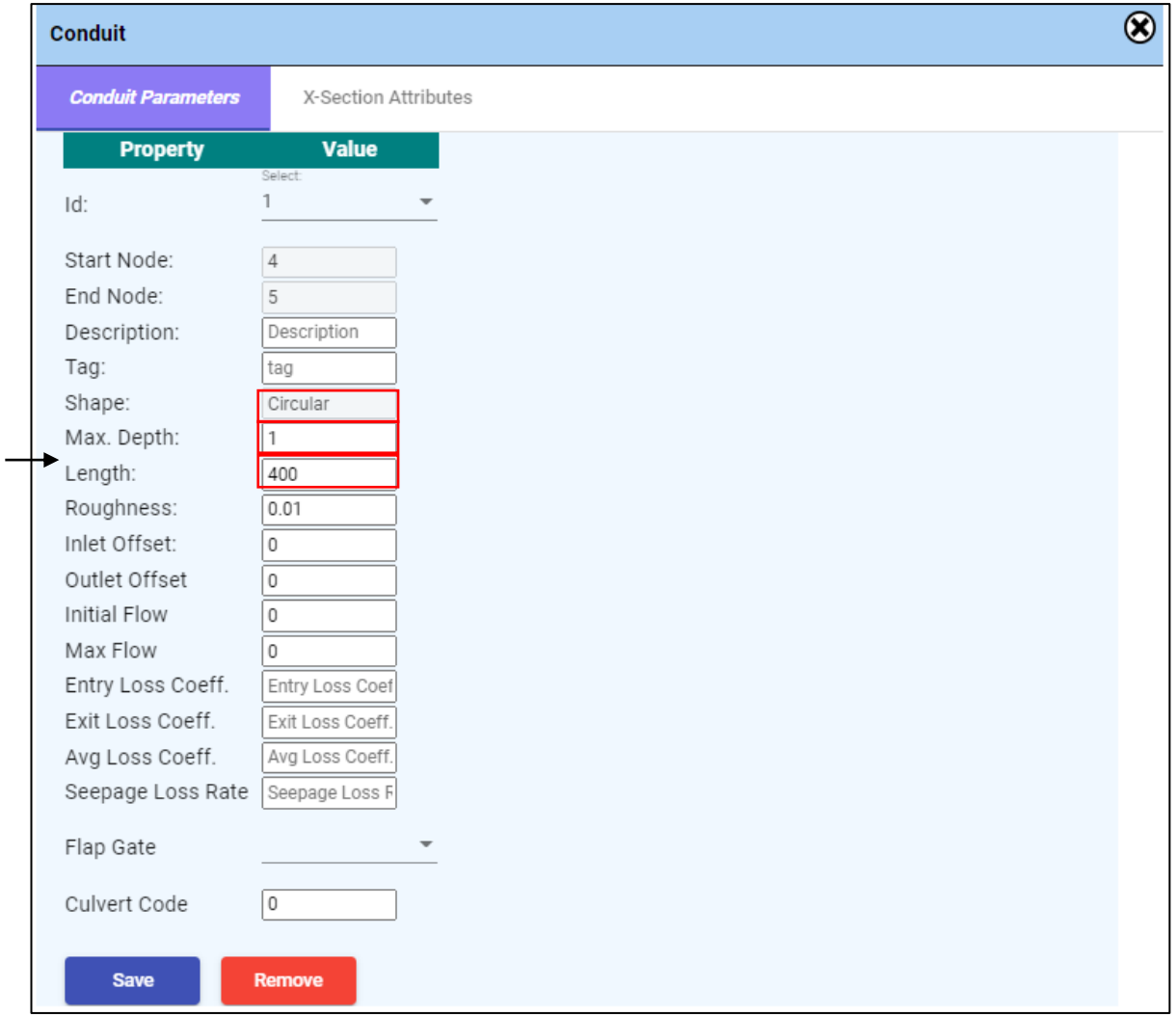
Select the different Conduits by their Conduit ID (1-4) in this example.

Assign a Length of 400 for all Conduits. Assign a Max Depth of 1 for Conduits 1-3, and a Max Depth of 1.5 for Conduit 4,

The Shape of the Conduits is circular (default shape).

Remember to Save before closing the Outfall Editor.



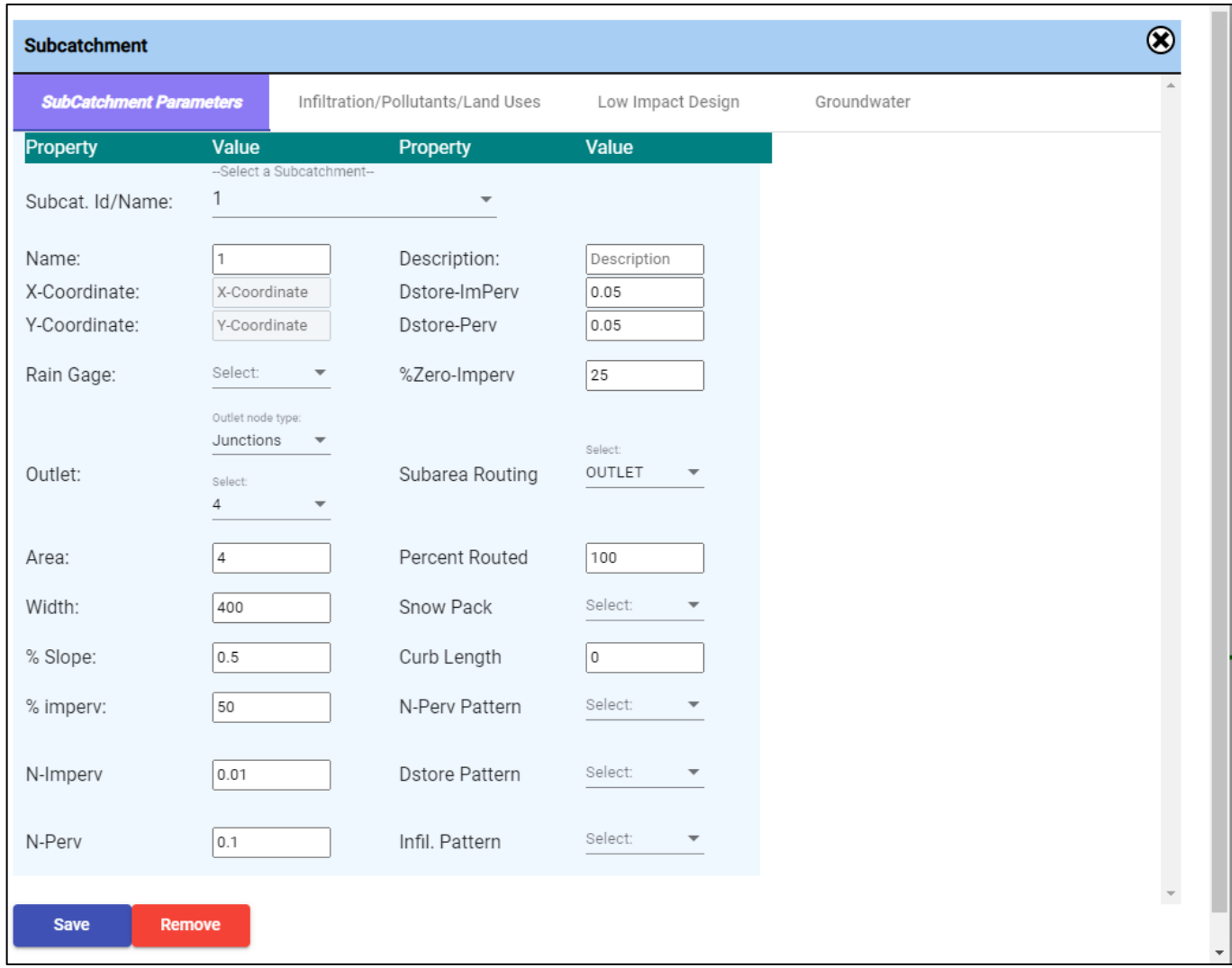


Add Subcatchment Outlets:

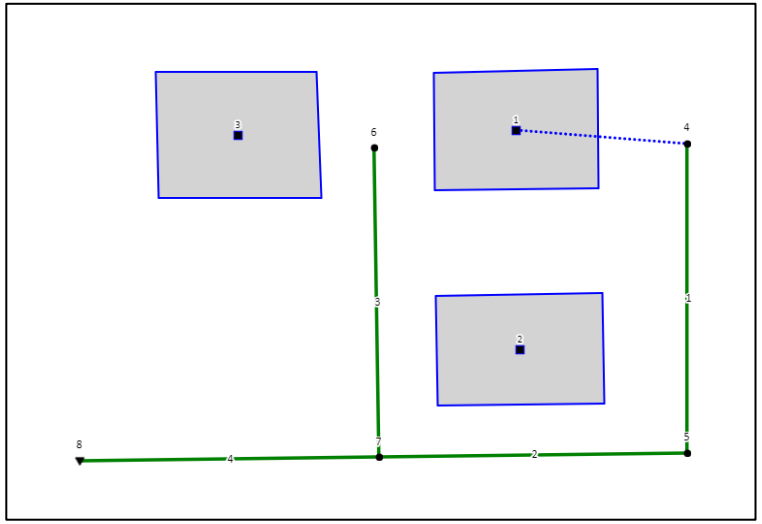
Hydrology > Subcatchments > Edit Subcatchments

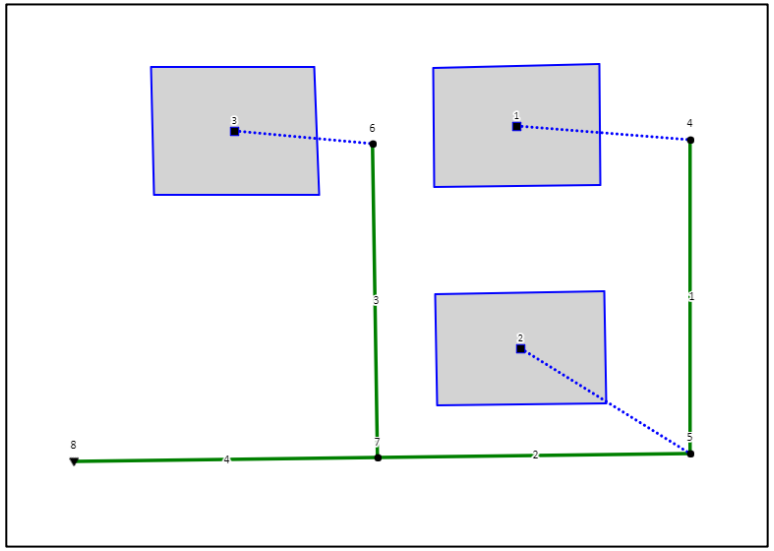
Select Subcatchment 1 and assign Junction as the Outlet node Type. Then select Junction 4 as the Outlet for Subcatchment 1.

Save the changes, then change to Subcatchment 2 to assign Junction 5 as the outlet.
Subcatchment 3 uses Junction 6 as its outlet.



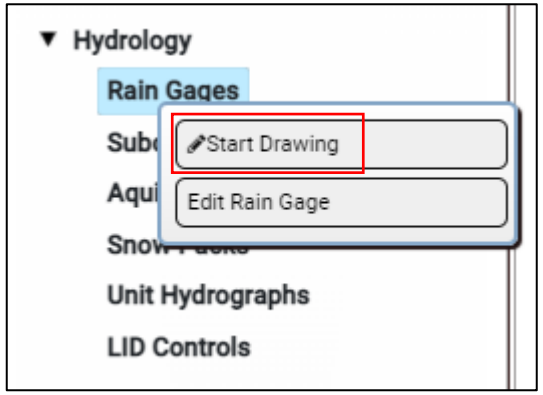
Save the changes, then change to Subcatchment 2 to assign Junction 5 as the outlet. Subcatchment 3 uses Junction 6 as its outlet.

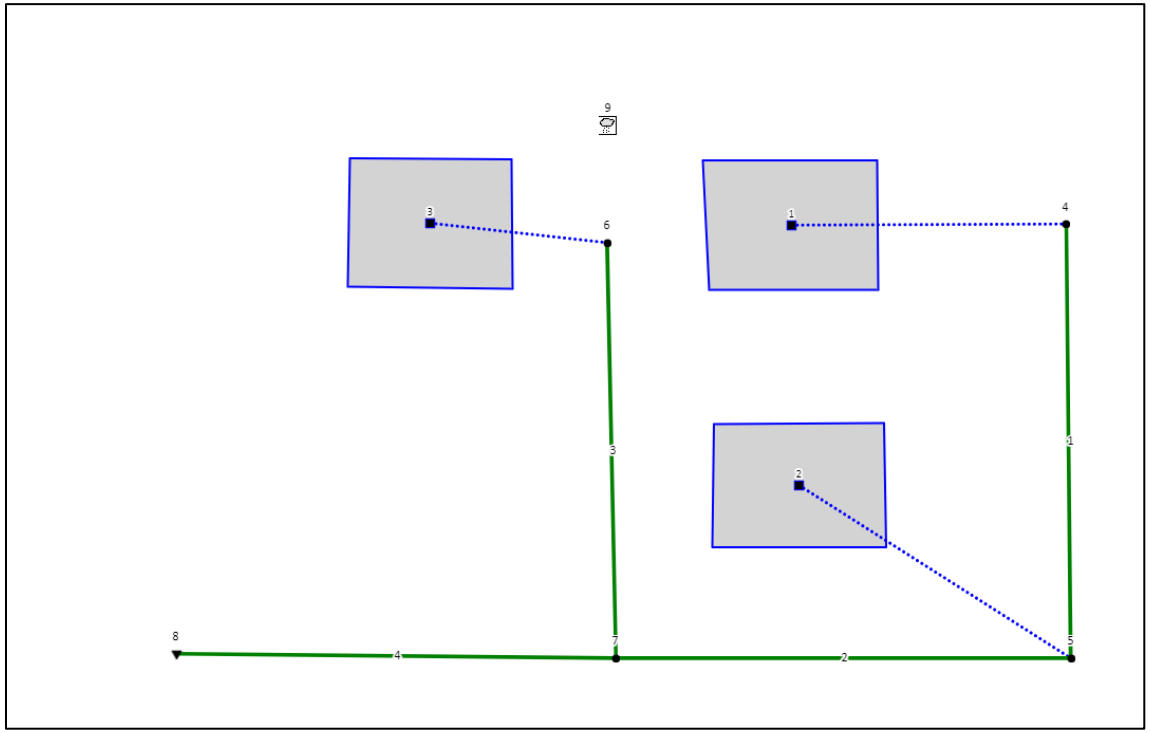




Add Rain Gage:

Hydrology > Rain Gages > Start Drawing





The cursor turns into a blue circle, indicating 'Drawing Mode' is active.

Use a single-click to add the Rain Gage to the map display.

Edit Rain Gage:

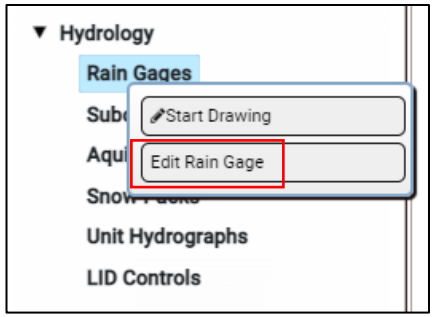
Hydrology > Rain Gages > Edit Rain Gage

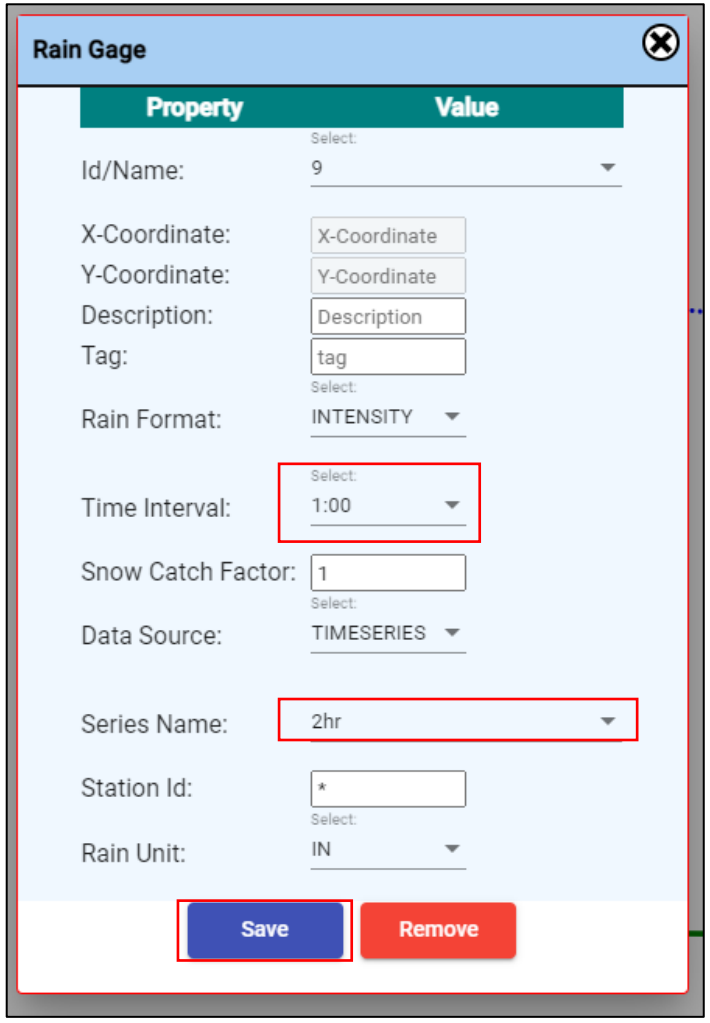
Select the Rain Gage by its ID (9 for this example) for attribute editing.

The Time Interval should be changed to one hour (1:00). Then select the default Series Name (2hr), which you will edit in the next step.

Make sure the Rain Format used is INTENSITY and the Rain Unit is IN (inches).

Remember to Save before closing the Rain Gage Editor.



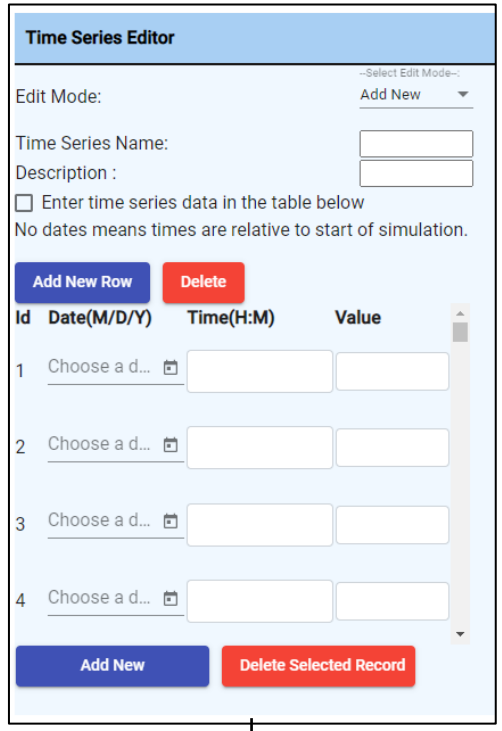


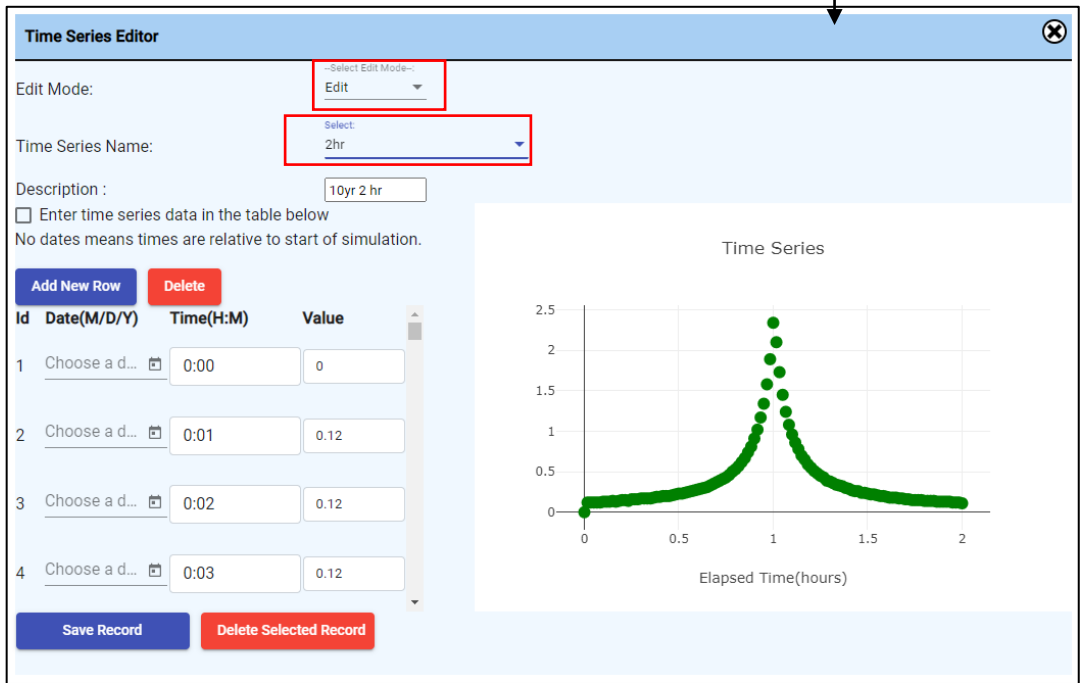
Edit Time Series:

Time Series > Edit Time Series

Select Edit from Edit Mode, then select 2Hr from the Time Series Name drop-down menu.

This is the default Time Series that we will customize / edit for this example.



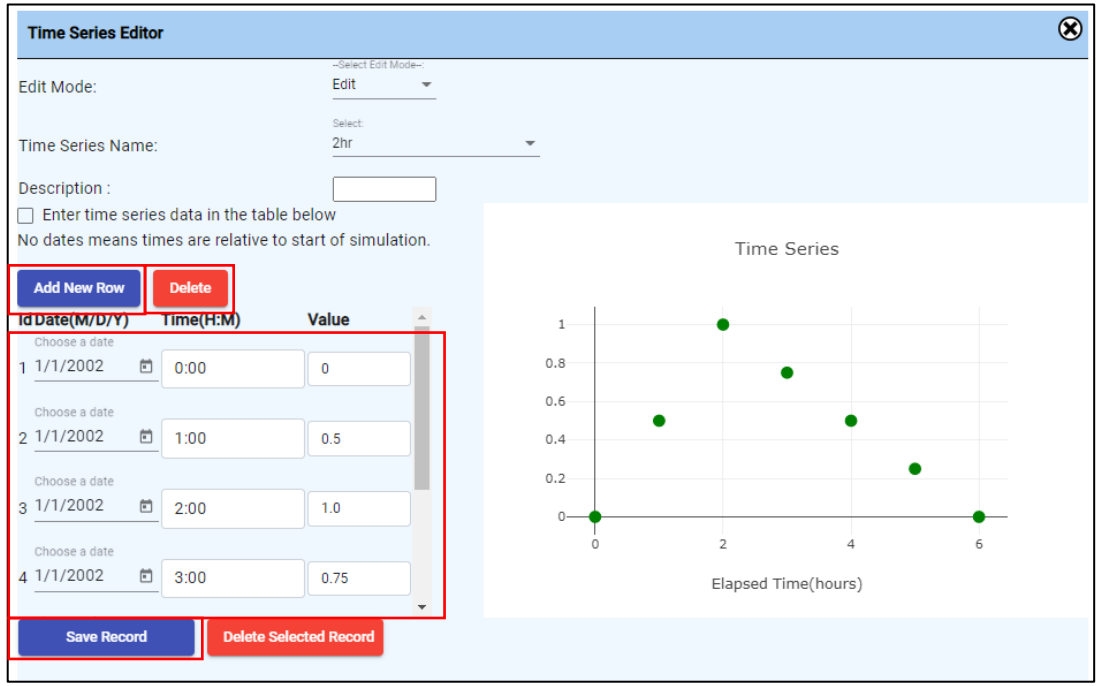


Use the Delete button to remove the existing entries from the Time Series Table. Then use the Add New Row button to create 7 empty entries.

For each entry, use a date of 1/1/2002. Then enter the following data pairs:

Time (H:M)	Value
0:00	0
1:00	0.5
2:00	1.0
3:00	0.75
4:00	0.5
5:00	0.25
6:00	0

Click Save Record to save the changes, then close the Time Series Editor.

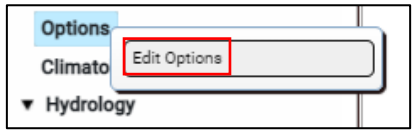


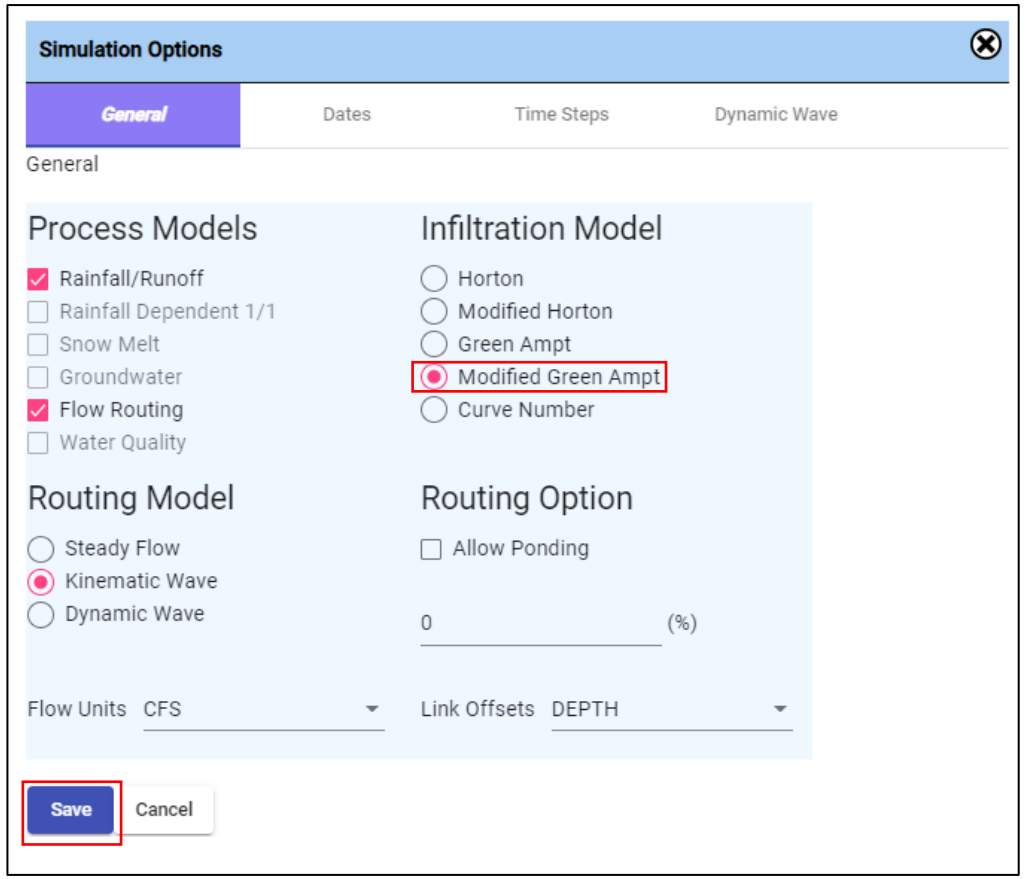
Edit Simulation Options

Options > Edit Options

Make sure the Rainfall/Runoff and Flow Routing Process Models are checked / enabled. The Kinematic Wave Routing Model should be used.

Change the Infiltration Model to Modified Green Ampt, then click Save.

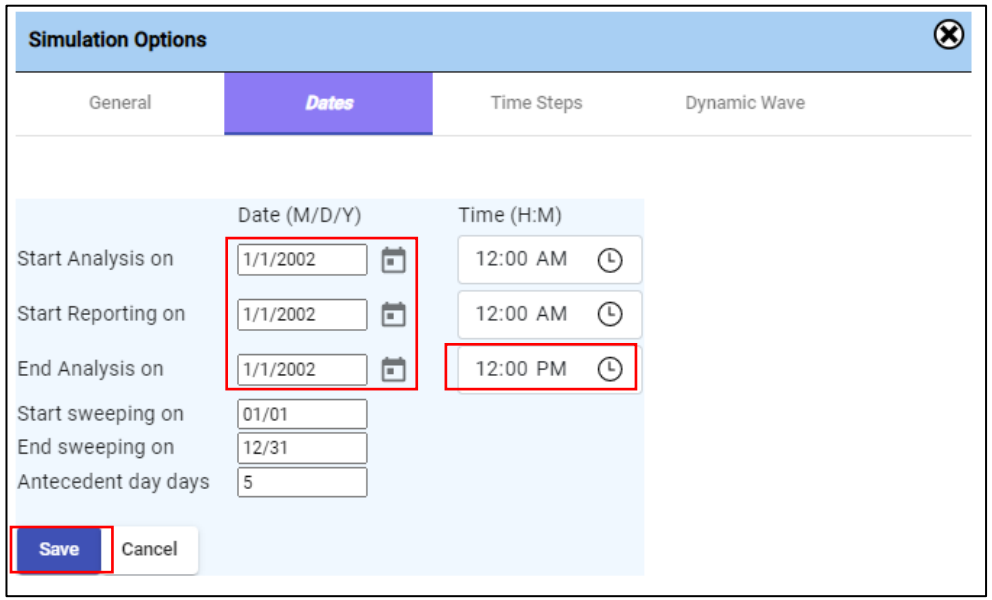




Still within Simulation Options, navigate to the Dates tab, and use 1/1/2002 as the date for Start Analysis on, Start Reporting on, and End Analysis on.

The Time (H:M) for Start Analysis and Start Reporting should be 12:00 AM.

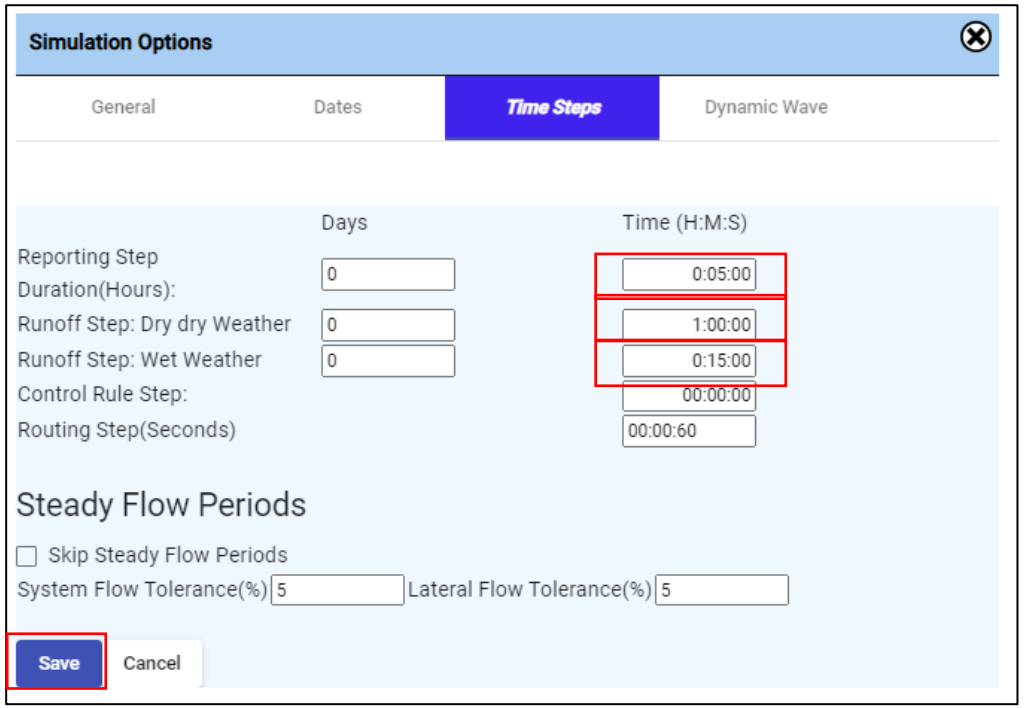
The Time for End Analysis should be 12:00pm (12 hour simulation duration).



Still within Simulation Options, navigate to the Time Steps tab tab, and use a Reporting Step of 5 minutes (0:05:00), and Dry Weather Runoff Step of 1 hour (1:00:00), and a Wet Weather Runoff Step of 15 minutes (0:15:00).

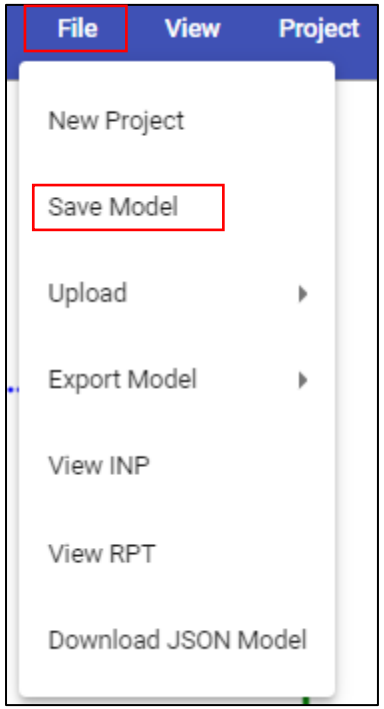
The Time (H:M) for Start Analysis and Start Reporting should be 12:00 AM.

The Time for End Analysis should be 12:00pm (12-hour simulation duration).



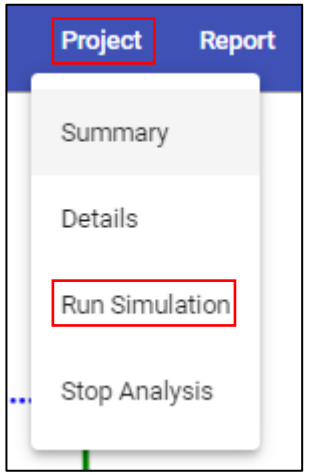
Save Model:

File > Save Model



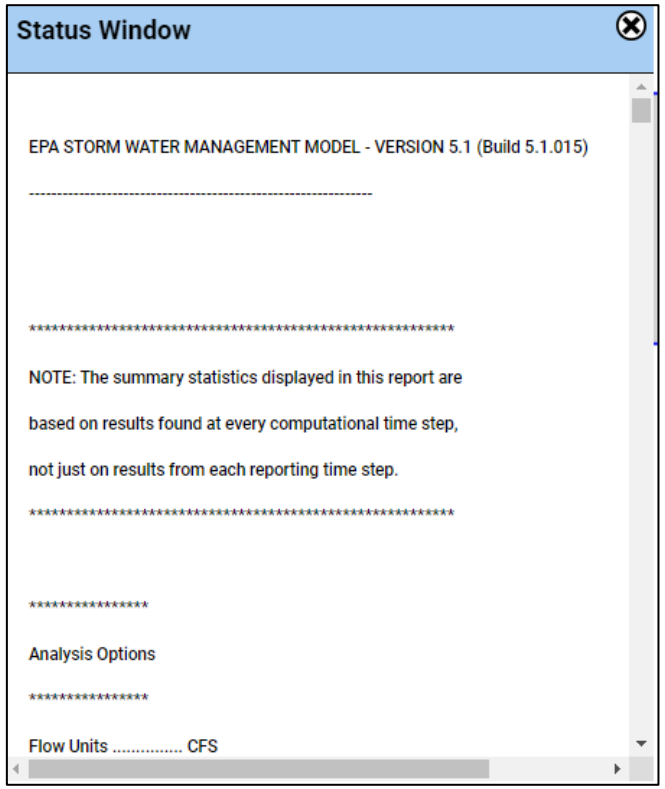
Rim Simulation:

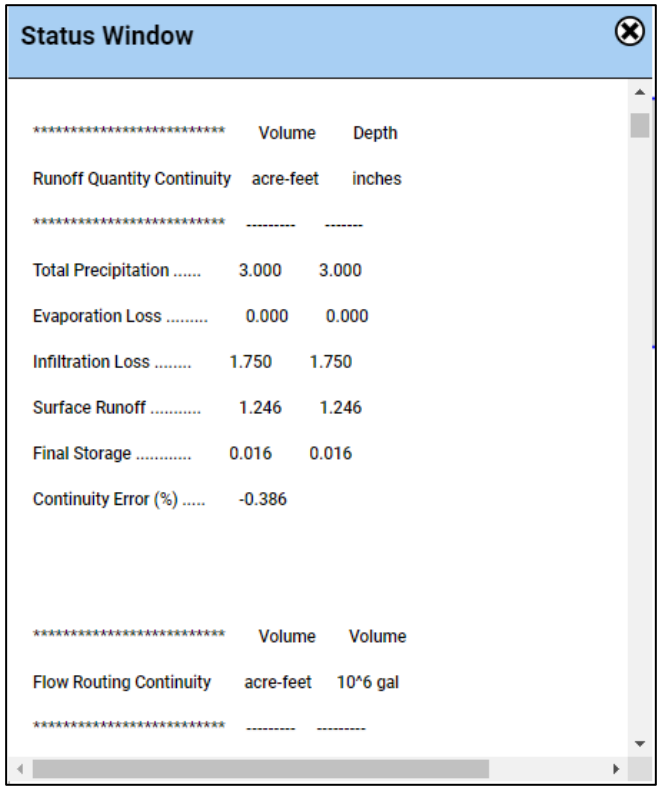
Project > Run Simulation



After the model is executed, the Status Window will appear.

This window indicates if there were any errors found in the model, and if the process models are balanced (water coming in equals water going out), among other information, at different times.





Visualize Results Across the Network:

Go to Browser Window:

View > Map Browser

Select Runoff as the Subcatchment Parameter.

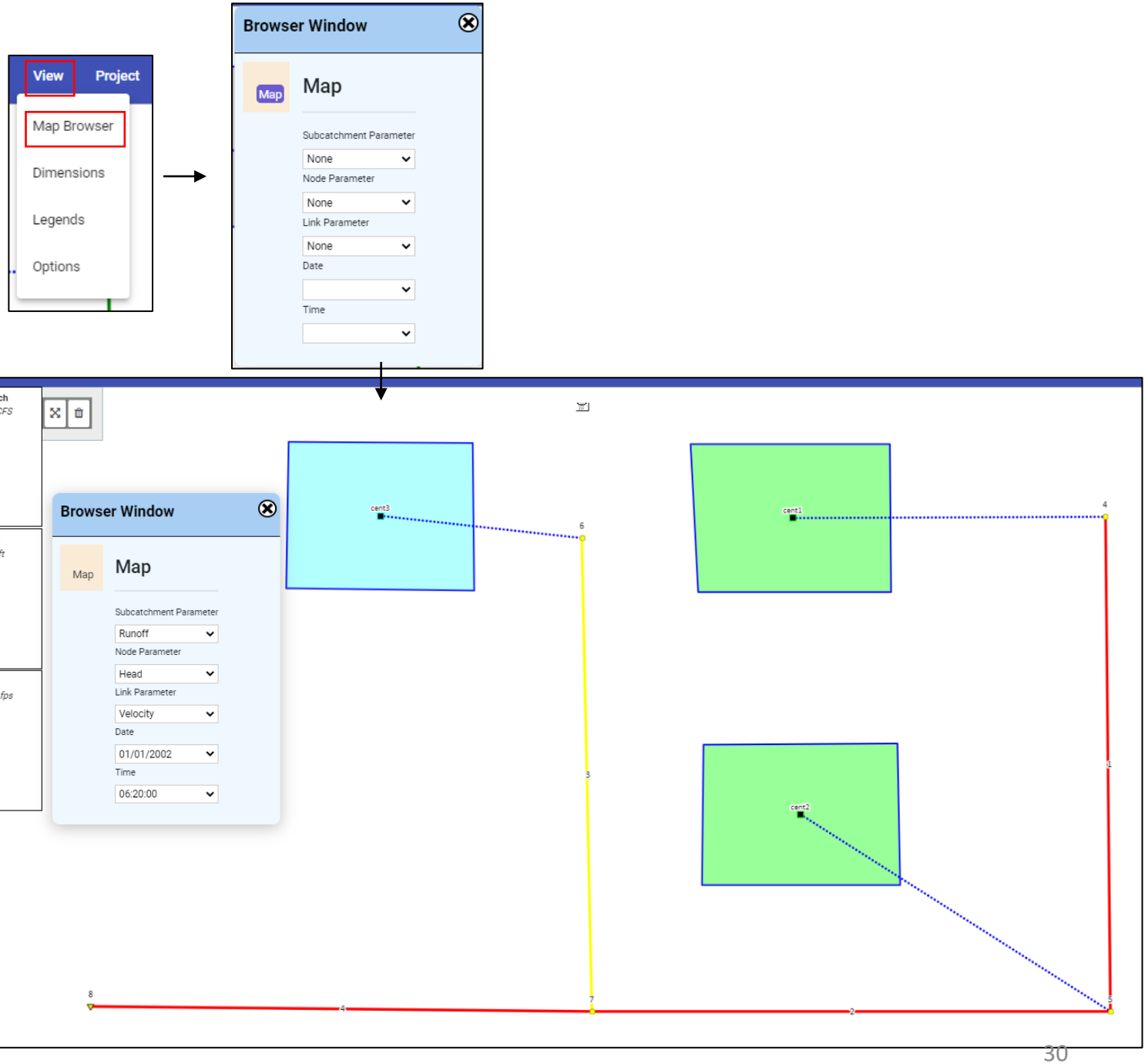
Select Head as the Node Parameter.

Select Velocity as the Link Parameter

Select 01/01/2002 as the Date

Select ‘6:20:00’ as the Time.

The map objects will then be color-coded based on the parameter values at 6 hours and 20 minutes of simulation. Note the legend that automatically appears.

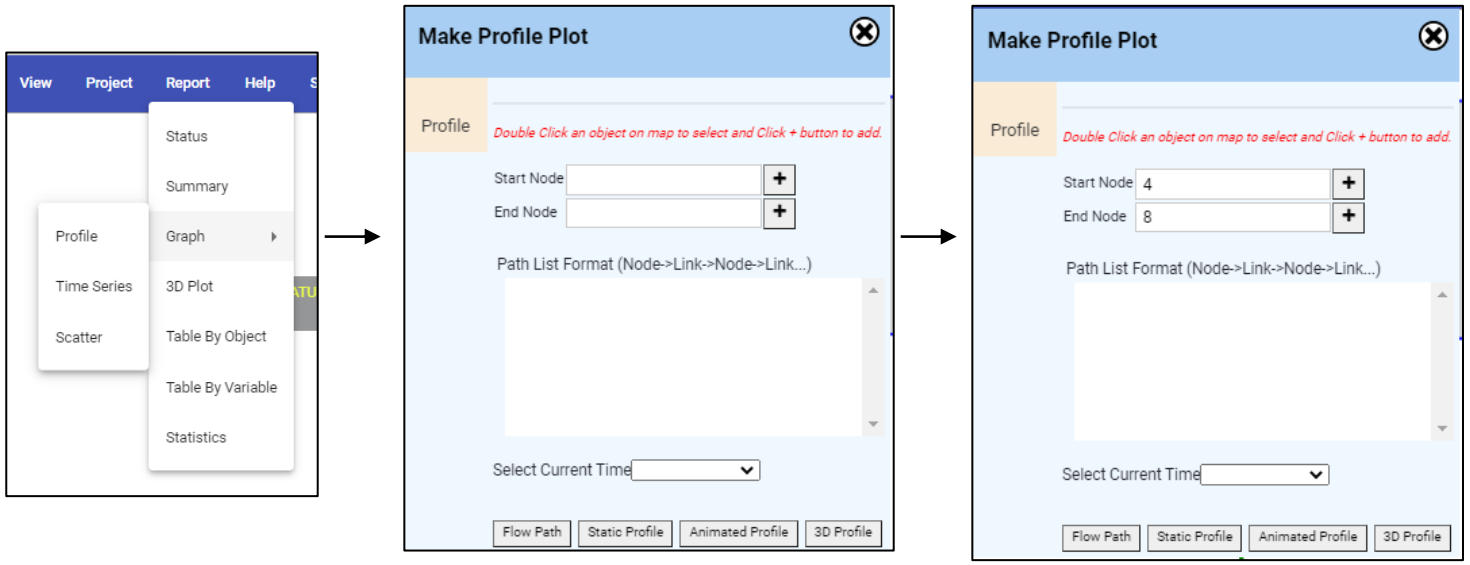


Visualize Results Along a Profile:

Report > Graph > Profile

Double-click on Junction 4 in the map display, then click the + button next to Start Node. This makes Junction 4 the start Node of the profile.

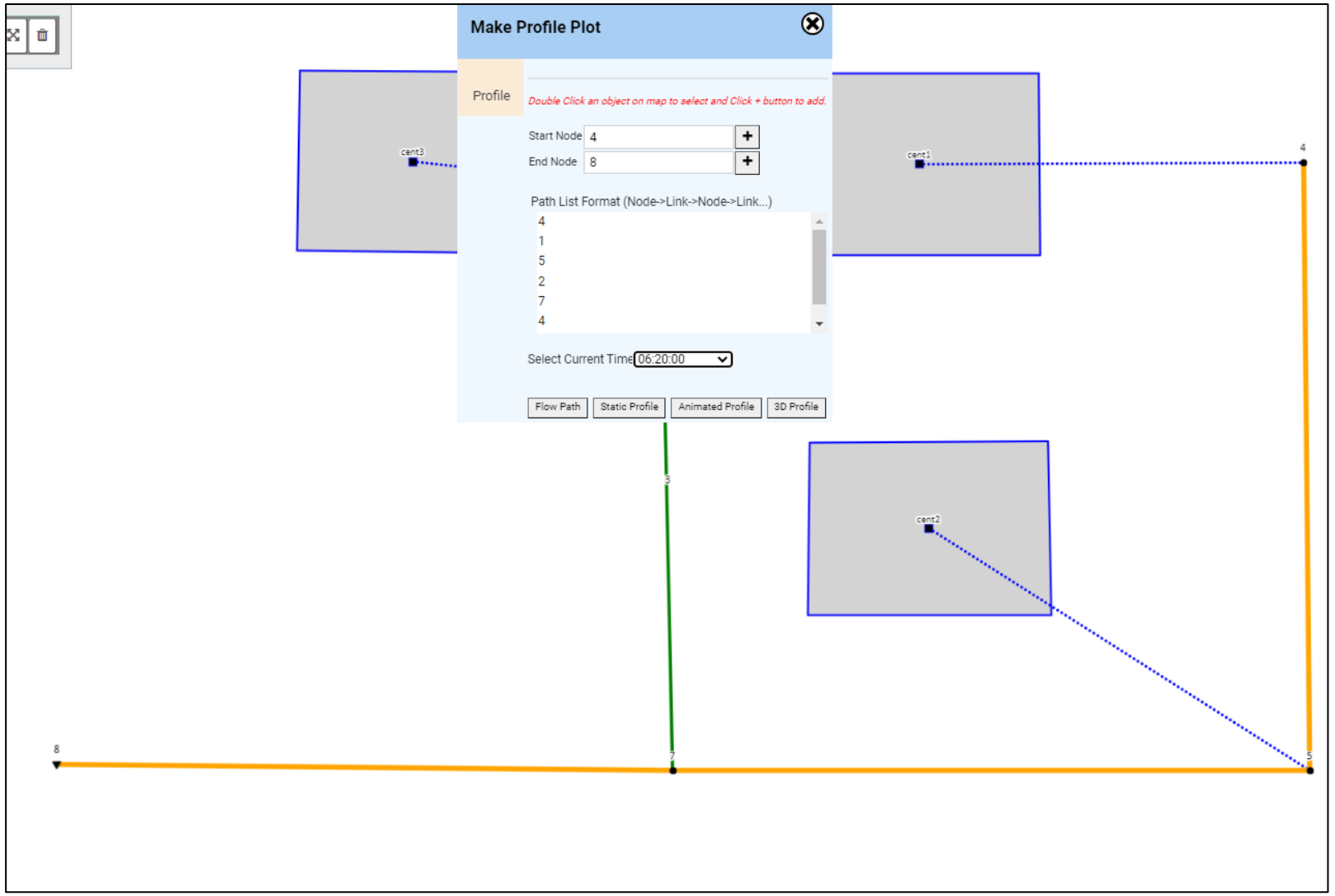
Similarly, double-click on Junction 8 and click the + button next to End Node to make Junction 8 the End Node of the profile.



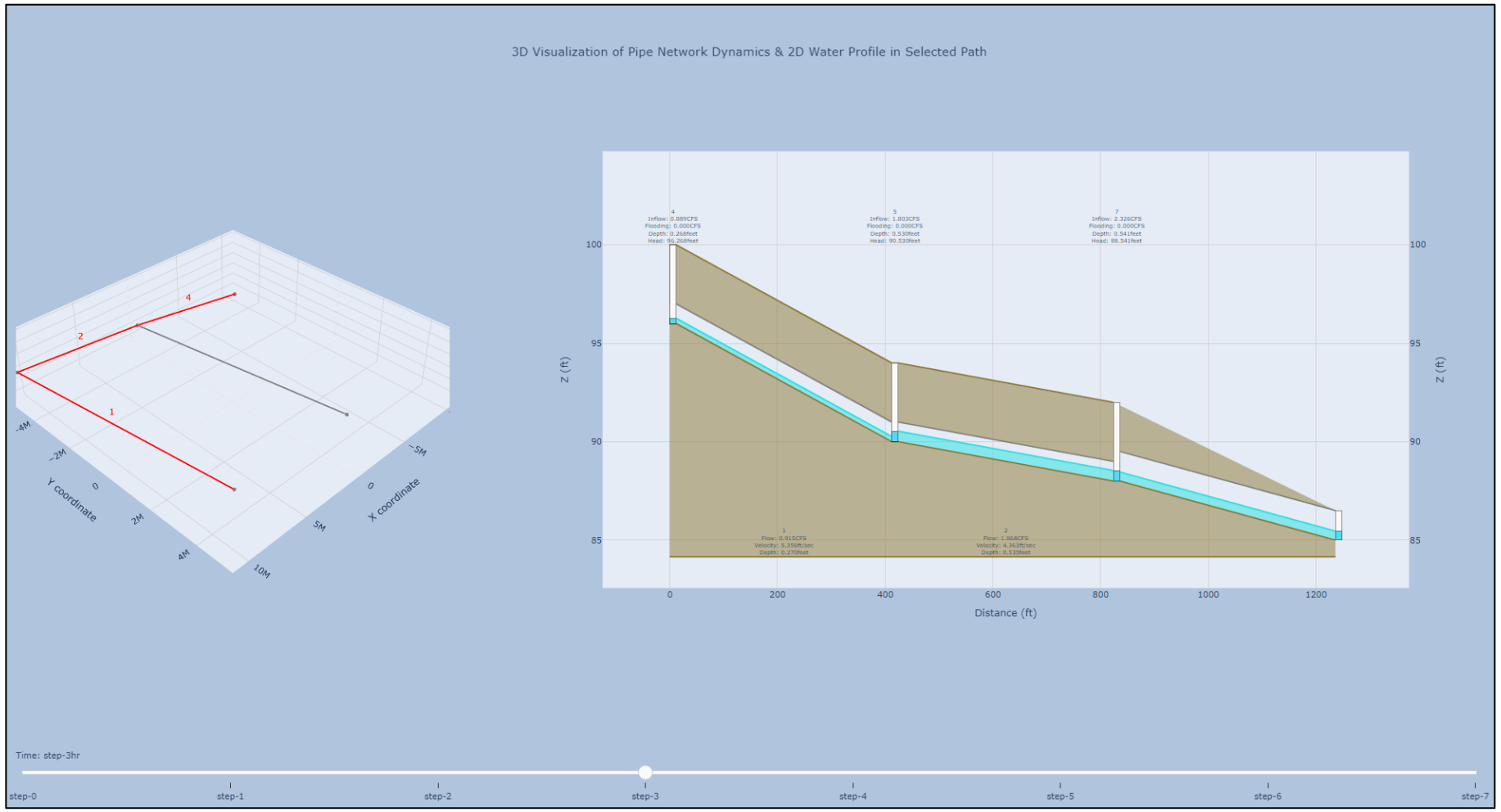
Click the ‘Flow Path’ button. A profile path will automatically be generated along the flow path from the Start Node to the End Path. The path is indicated on the map display as a yellowish-orange line.

The path is also listed in the Path List Format table (Node -> Link -> Node -> Link …).

Click on the ‘3D Profile’ button to launch the 3D profile plots (next slide).



This slide shows the 3D visualization of the Pipe Network and 2D Water Profile along the selected path.



Visualize Results in the Conduits at all Times:

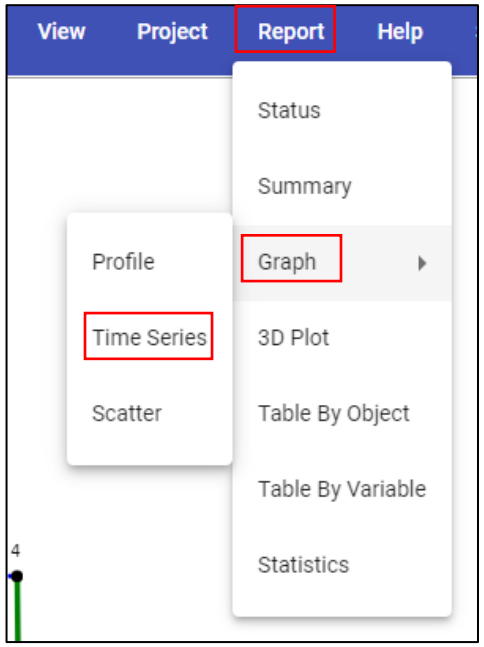
Report > Graph > Time Series

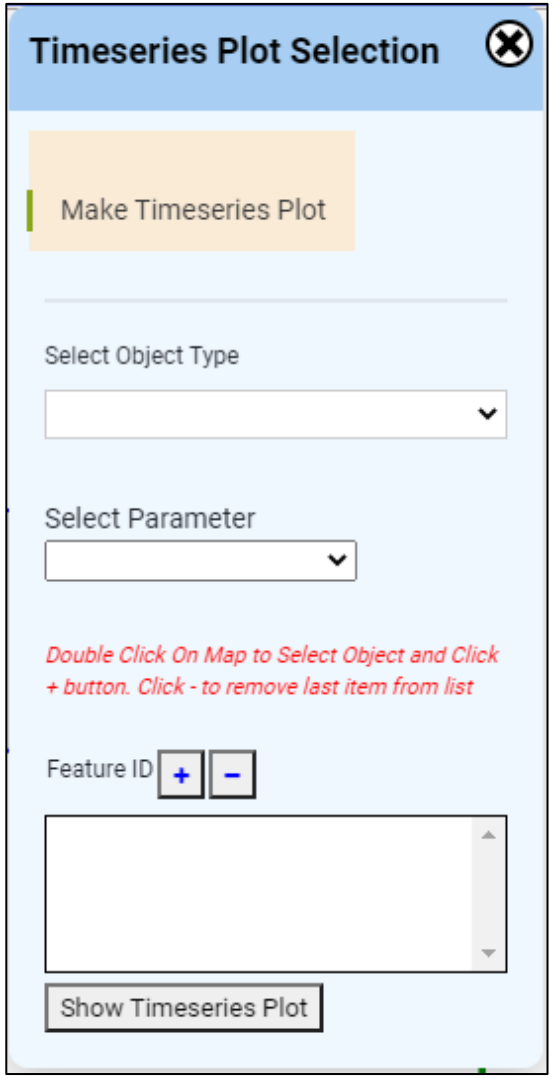
Select Link as the Object Type.

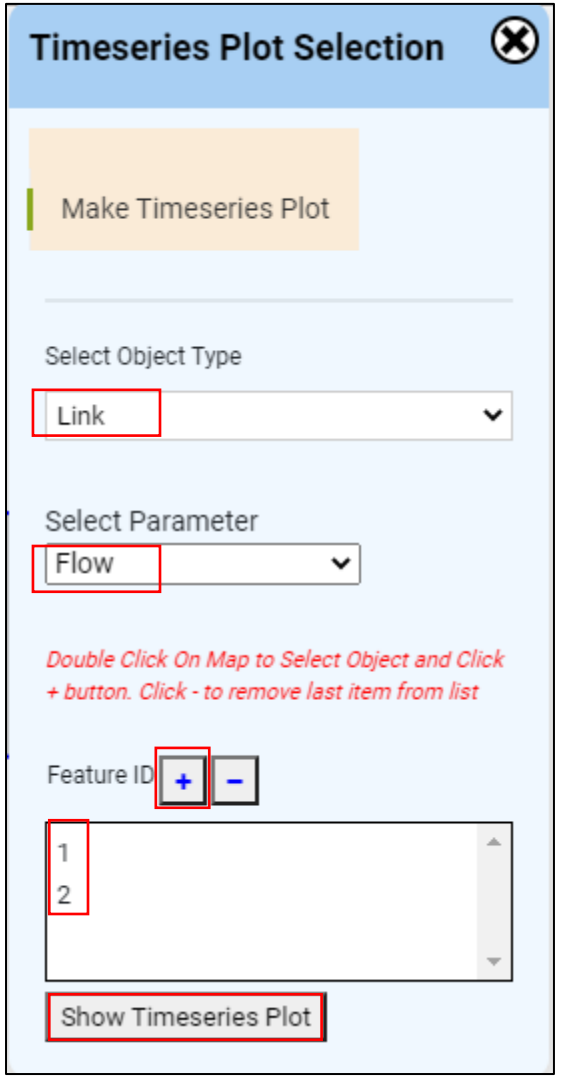
Select Flow as the Parameter.

Double-click on Conduit 1 in the map display, then click the + button next to add the conduit to the Feature ID list. Repeat for Conduit 2.

Click the ‘Show Timeseries Plot’.

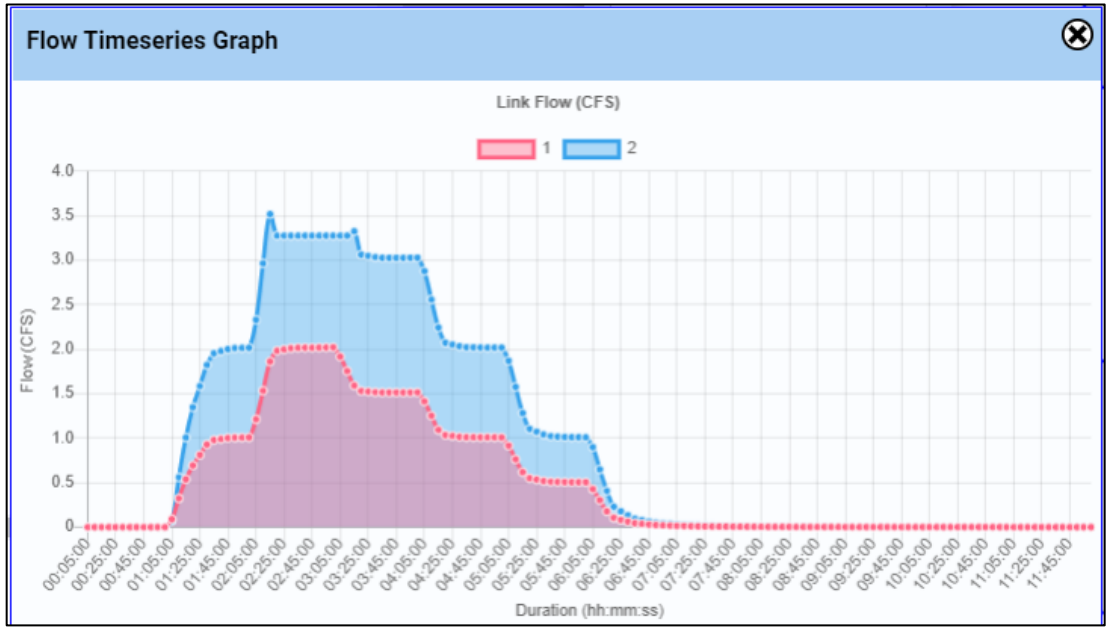






Visualize Results in the Conduits at all Times:

A plot will automatically be generated showuing the flow results in Conduits 1 and 2 over the entire simulation duration.



Saving the Model File:

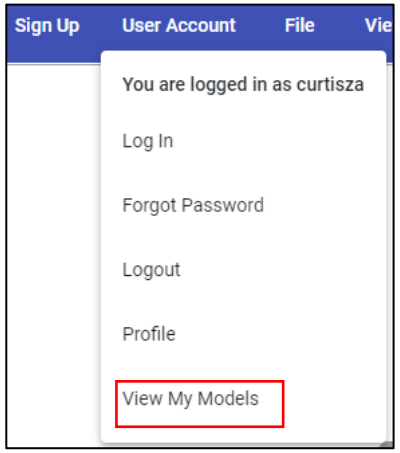
User Account > View My Models

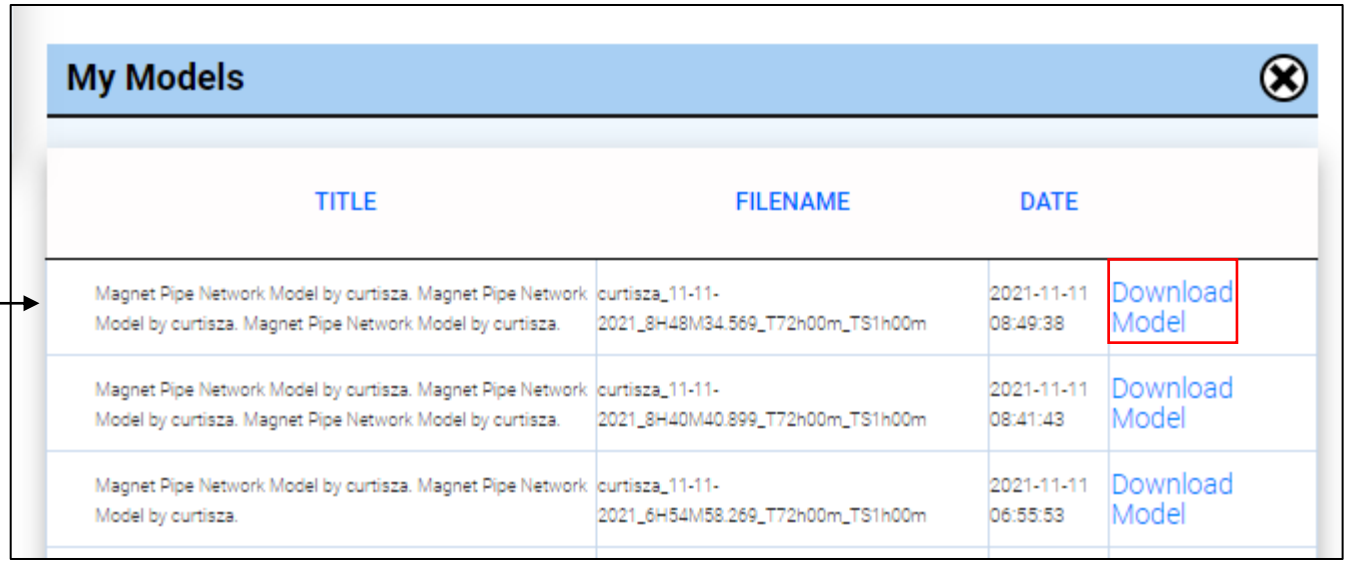
This opens a list (table) of previously executed models linked to your user account.

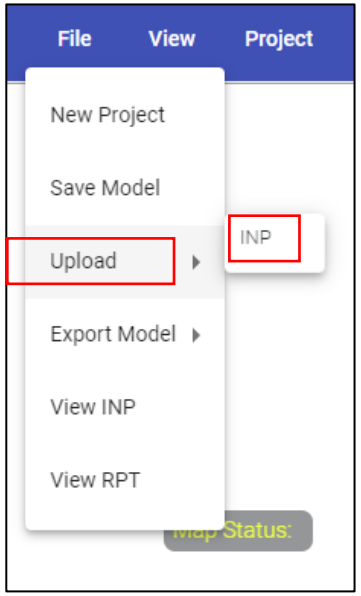
Click ‘Download Model’ to download an.inp (model input) file to your local machine’s Downloads folder.

The .inp file can be uploaded back into MAGNET Pipe Network Analysis with:

File > Upload > INP







Questions / Comments / Issues?

Use: [Menu Bar] > Support > Contact Us

Or directly email:
support@magnet4water.com