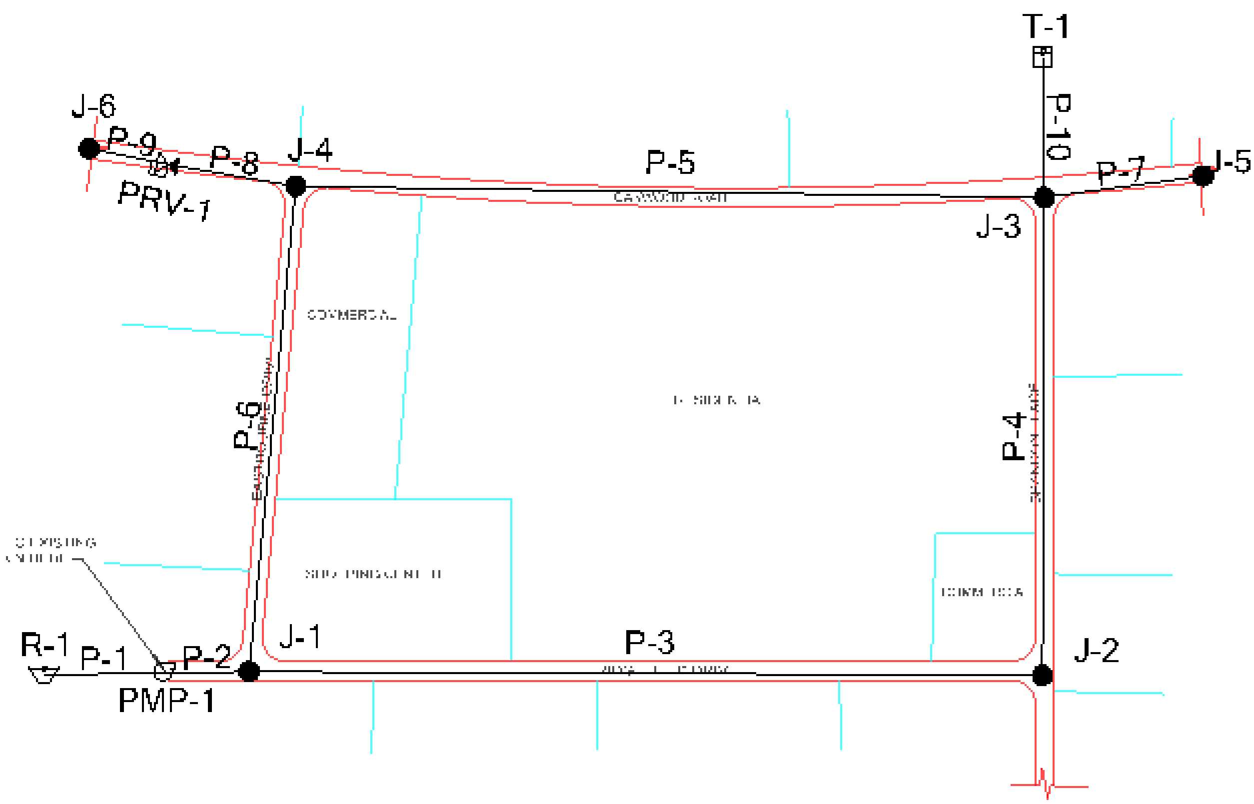
WaterGEMS
Quick Start Lessons

Building a Network and Performing a Steady-State Analysis

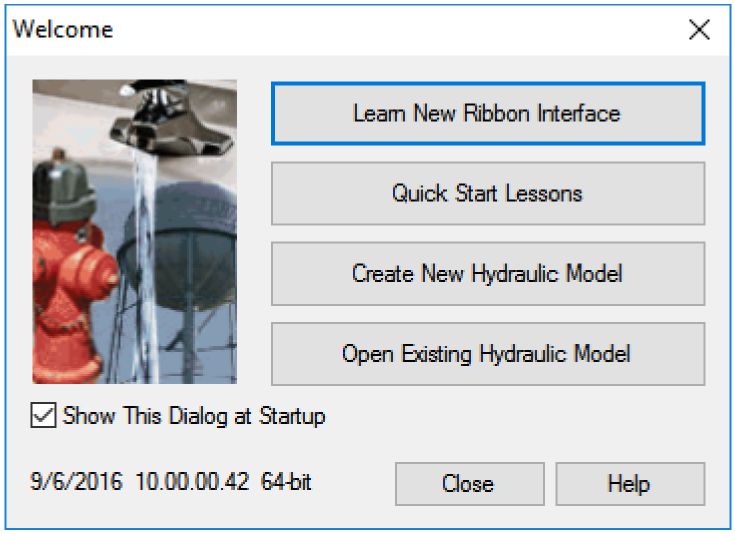
In this lesson, you will draw a schematic model of a simple pipe network connected to a larger water distribution system. You will use a scaled background drawing to assist you in drawing the pipe network, and assign user-defined lengths to four pipes. As you draw a schematic of a pipe network, you will enter pipe lengths, while the software automatically assigns labels to each pipe and node. If building a scaled model of a pipe network, as you draw elements, the software will automatically manage pipe lengths based on position of pipes bends, start nodes, and stop nodes, and allow customized labeling formats.

To simulate the connection to the larger water distribution network, you will draw a reservoir connected to a pump, and enter a pump curve. The pump curve will be developed from 3 data points: the static pressure of the distribution system at the connection point (converted to head), partial hydrant flow and pressure (converted to head), and full hydrant flow and pressure (converted to head). This method simulates the larger water distribution system's ability to deliver a range of flows and related pressures to the new pipe network with a single pump curve. Representing the network connection in this way is an approximation of actual conditions and may not be representative for all operational conditions. Care should be taken to ensure all applicable operating conditions are addressed.

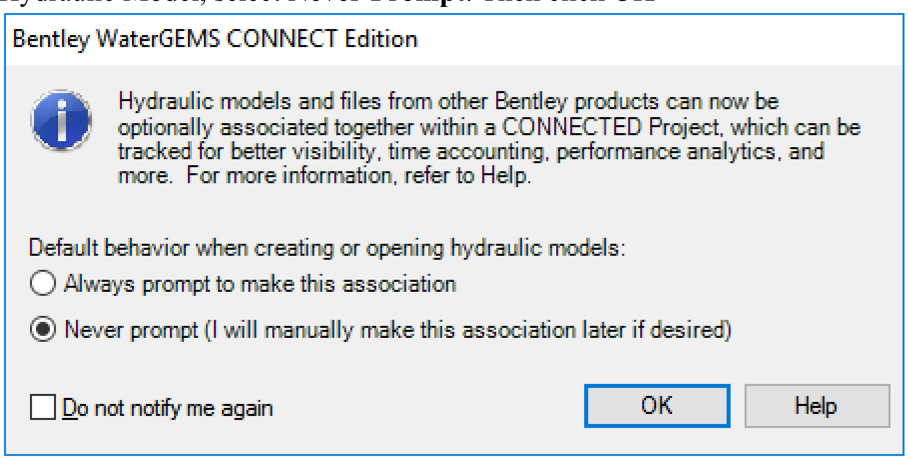


Step 1: Create a New Hydraulic Model File

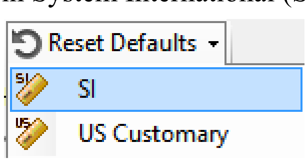
1. Start WaterGEMS from **Start** > **All Programs** > **Bentley**.
2. On the Welcome dialog click **Create New Hydraulic Model**; otherwise select **File** > **New**.



3. If you receive a prompt about associating this file with a CONNECTED Hydraulic Model, select **Never Prompt**. Then click OK

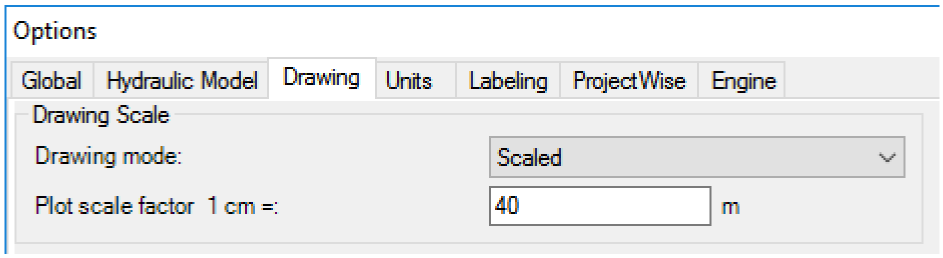


4. Choose **Tools** > **More** > **Options**, Click the **Units** tab. Since you will be working in System International (SI) units, click the Reset Defaults button and select **SI**.



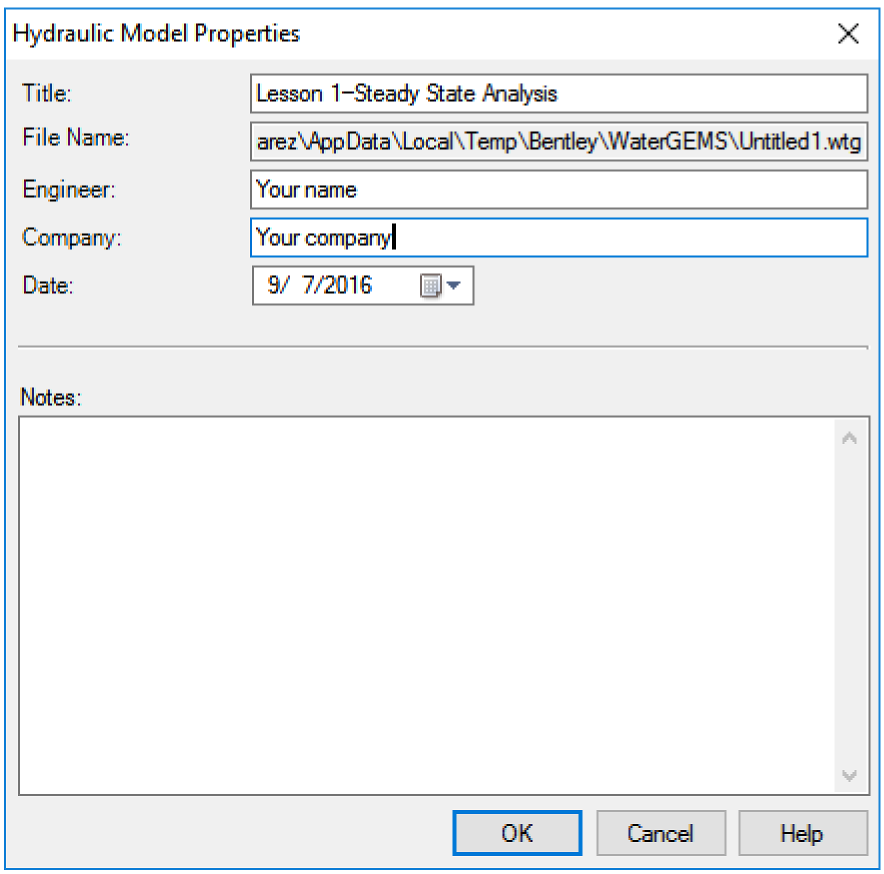
5. The first six WaterGEMS Quick Start Lessons use SI units. If you would like to set the default unit systems for new Hydraulic Models to be SI, select **SI** from the drop down menu also.

6. Click the Drawing tab to make sure Drawing Mode is set to Scaled.
Set the Plot Scale Factor 1 cm = **40m**



7. Click OK.

8. Set up the Hydraulic Model. Choose **File > Hydraulic Model Properties** and name the Hydraulic Model **Lesson 1—Steady State Analysis** and click OK.

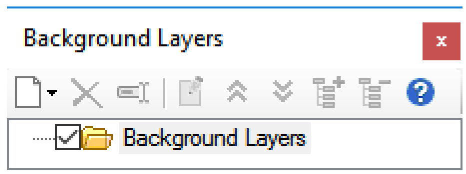


9. Choose **File > Save As**.

10. Enter the file name **MyLesson1** for your model, and click **Save**.

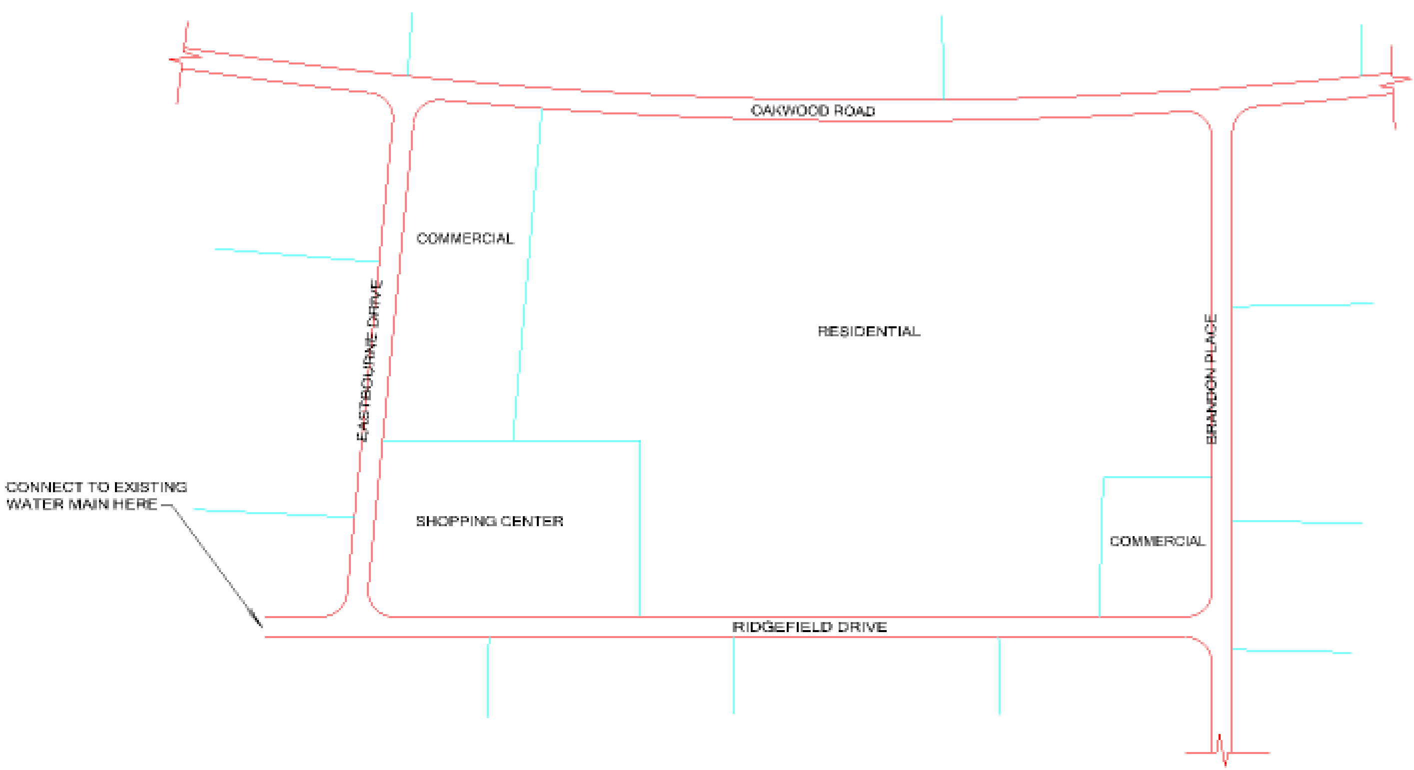
Step 2: Lay out the Network

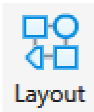
1. You will be using a background file to lay out the pipe network.
2. Select **View > Backgrounds** to open the *Background Layers* manager. The Background Layers Manager may already be docked in the lower left corner of your screen, its default position.



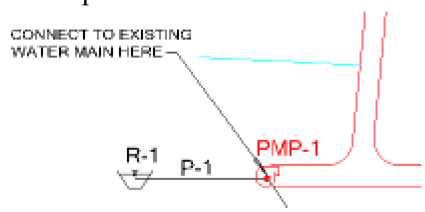
3. Right click on the *Background Layers* folder and select **New > File**.
4. On the *Select Background* dialog, browse to **Lesson1.dxf** in the Lessons folder and click *Open*.
5. Click *OK* on the *DXF Properties* dialog to accept the defaults.

6. Click the *Zoom Extents* button on the top of the drawing area to zoom to the background, or select **View > Zoom Extents**.
7. Your drawing should now look like this:

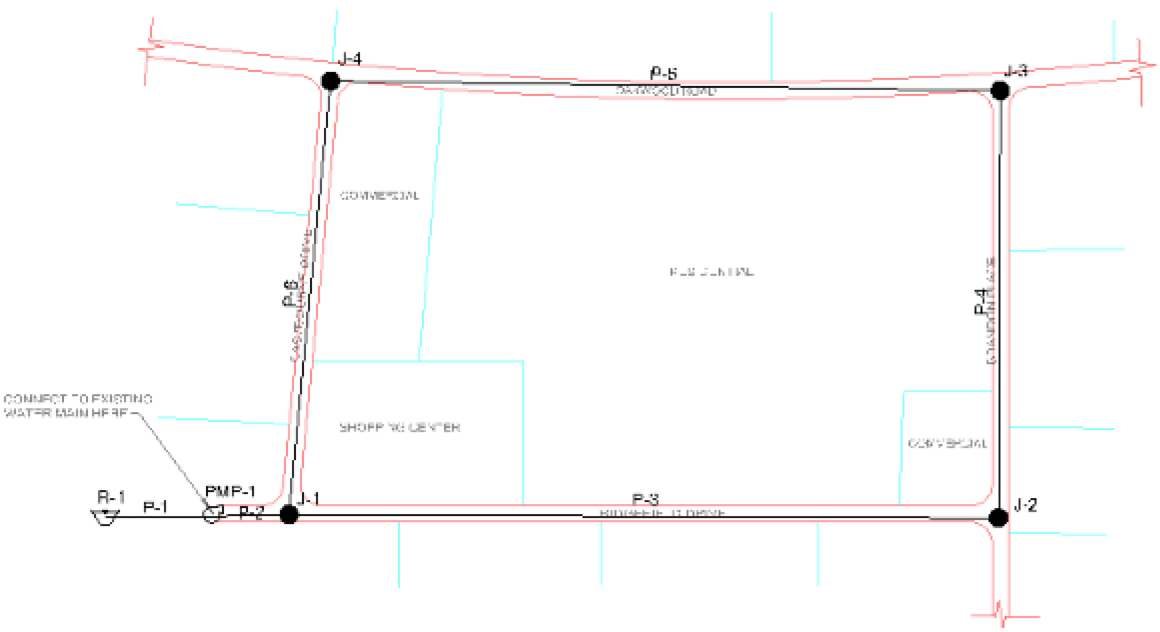




8. Select the **Layout** tool from the **Home** tab. Follow the instructions below to draw the network (see figure on page 2). Lay out the pipe network in the order shown. Any variations can be corrected by right clicking on that element and modifying the Label in the Properties Manager.
9. Move the cursor to the drawing pane, right click, and select **Reservoir** from the menu
10. Click to place R-1 to the left of the existing water main connection.
11. Move the cursor to the location of pump P-1. Right-click and select Pump from the shortcut menu. Click to place it.

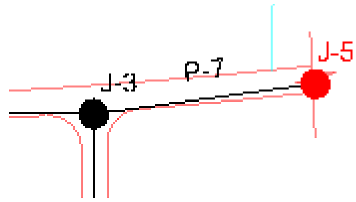


12. Right click to select Junction from the menu and click to place J-1.
13. Click to place junctions J-2, J-3, and J-4.P-3, P-4, P-5 will automatically be drawn
14. Click on J-1 to draw P-6.
15. Right-click and choose Done from the menu.

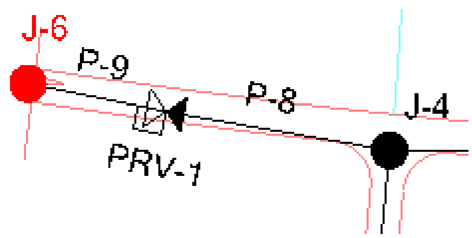


16. Select the **Layout** tool again from the Home tab.
17. Click on J-3. Move the cursor to the location of J-5, and click. This will insert J-5 & P-7.

18. Right-click and select **Done**.

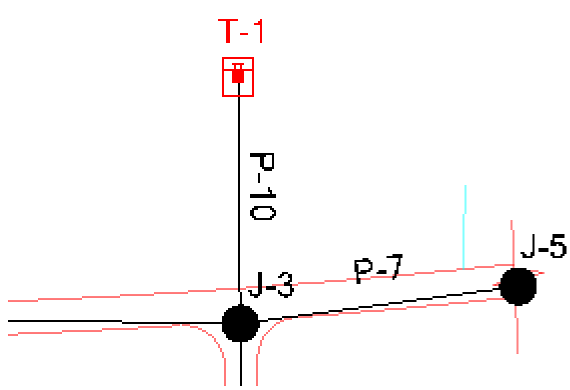


19. Select the **Layout** tool again from the Home tab.
20. Click on J-4. Move the cursor to the left towards the location of the PRV. Right click to select **PRV** from the list of elements.
21. Click to insert PRV-1 in the appropriate location. This will insert PRV-1 & P-8. Right click to select **Junction** from the list of elements.
22. Click to place junction J-6 in the appropriate location. This will insert J-6 & P-9. Right click and select **Done.**

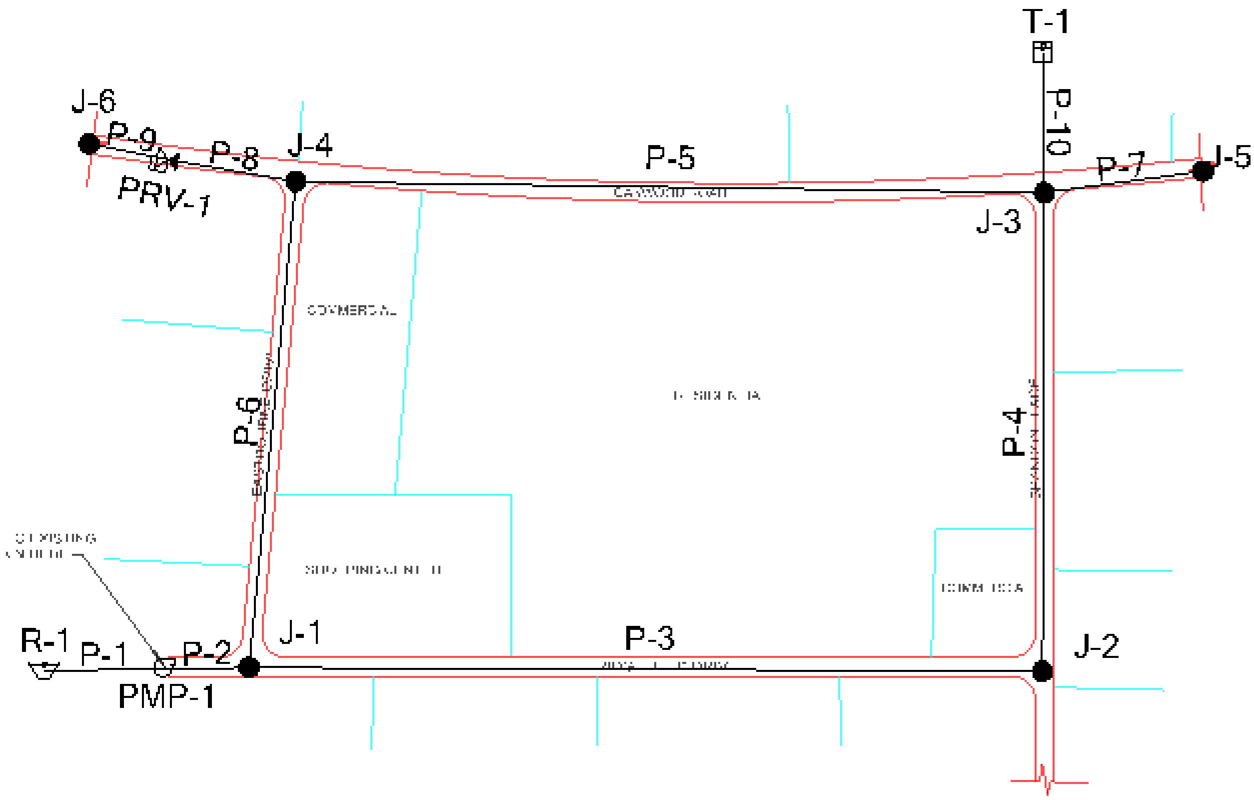


Be sure to lay out the pipes in numerical order (P-7 through P-9), so that their labels correspond to the labels in the diagram.

23. Select the **Layout** tool again from the Home tab.
24. Click on J-3. Move the cursor upwards towards the location of the Tank. Right click to select **Tank** from the list of elements.
25. Click to insert the tank, **T-1**. Pipe **P-10** should connect the tank to the network if you laid out the elements in the correct order.



26. Your model should now look like this:



27. Save the model by clicking **Save**  or choose **File > Save**.

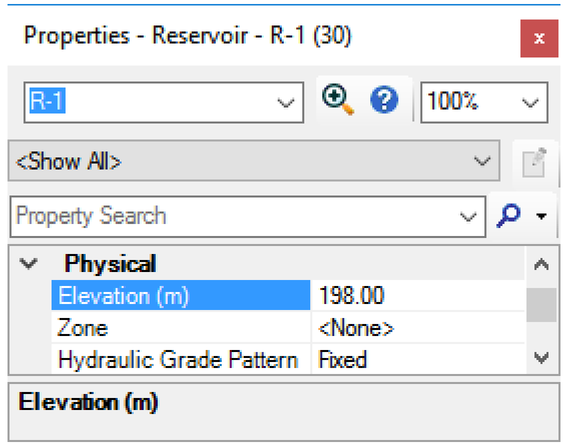
Step 3: Enter and modify data

- **Property Editor**—You can use the Select tool and double-click an element to bring up its Property Editor. In AutoCAD, click the element once with the Select tool to open the element's editor.
- **FlexTables**—You can click on FlexTables to bring up dynamic tables that allow you to edit and display the model data in a tabular format. You can edit the data as you would in a spreadsheet.
- **User Data Extensions**—The User Data Extensions feature allows you to import and export element data directly from XML files.
- **Alternative Editors**—Alternatives are used to enter data for different "What If?" situations used in Scenario Management.

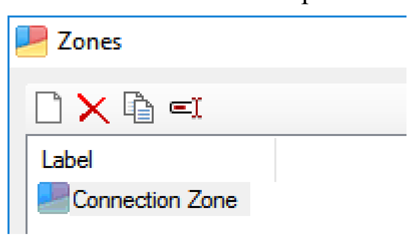
Entering Data through Property Editor

To access an element's property editor in WaterGEMS, double-click the element.

1. Open the **Reservoir Editor** for reservoir R-1 by double-clicking on R-1.

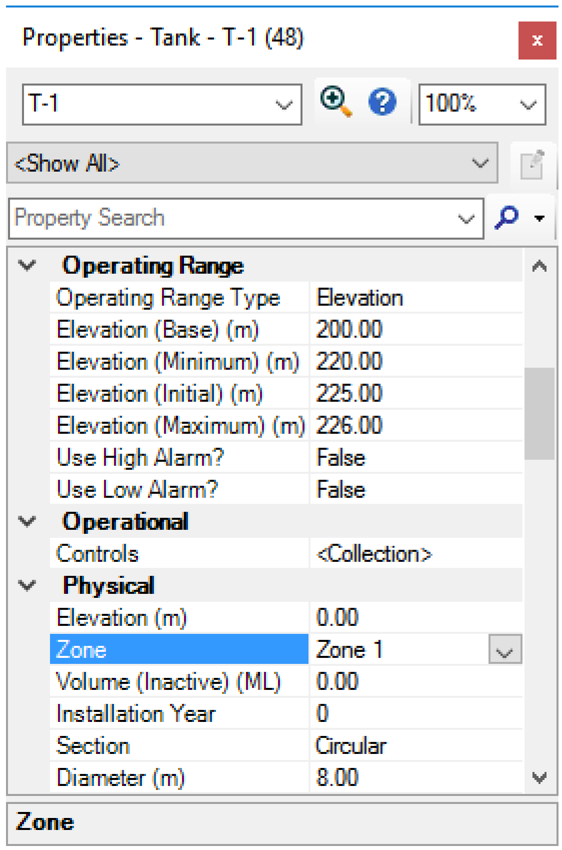


2. Enter the Elevation as 198.
3. Set Zone to **Connection Zone**.
   a. Click the menu to Edit Zones which will open the Zones Manager.
   b. Click **New**.
   c. Enter a label for the new pressure zone called Connection Zone.



   d. Click Close.
   e. Select the zone you just created from the Zone menu.
   f. Close the Reservoir Property Editor.
4. Double click on tank **T-1** and enter the following:
   Elevation (Base) = 200
   Elevation (Minimum) = 220
   Elevation (Initial) = 225
   Elevation (Maximum) = 226
   Diameter (m) = 8
   Section = Circular

Set the Zone to Zone 1(You will need to create Zone-1 in the Zone Manager as described above.)

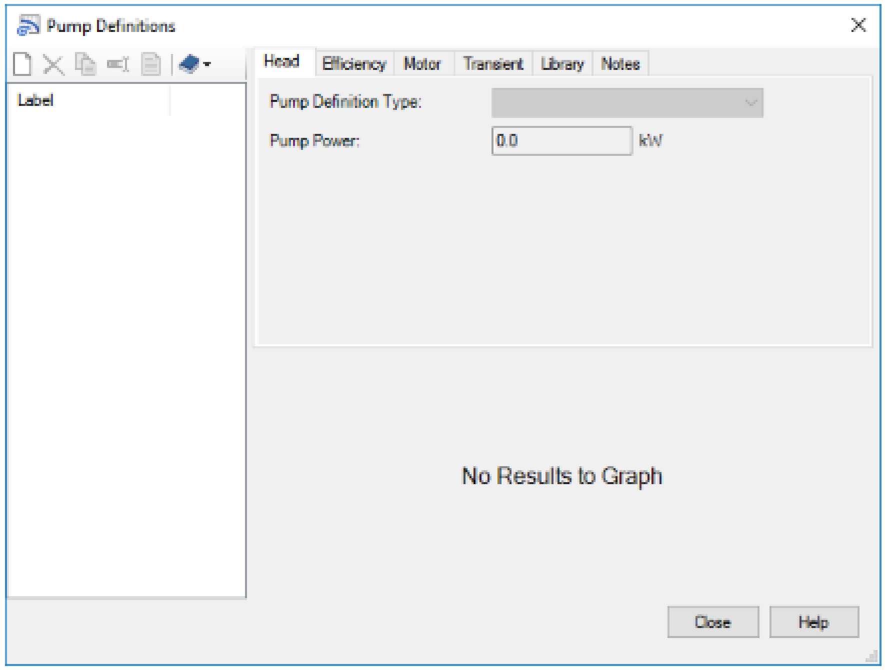


5. Close the Tank Property editor.

6. Double click on pump **PMP-1**.

   a. Enter **193 m** for the Elevation.

   b. Click in the Pump Definition field and click on Edit Pump Definitions from the drop-down list to open the Pump Definitions manager.

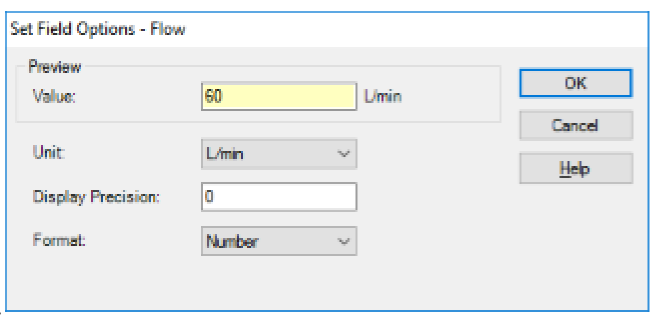


   c. Click **New** to create a new pump definition. Name it **PMP-1**.

   d. Select Standard (3 Point) from the Pump Definition Type menu.

   e. Right click on **Flow** to open the Units and Formatting menu.

   f. Click on it and then in the Set Field Options dialog, set the Units to **L/min**



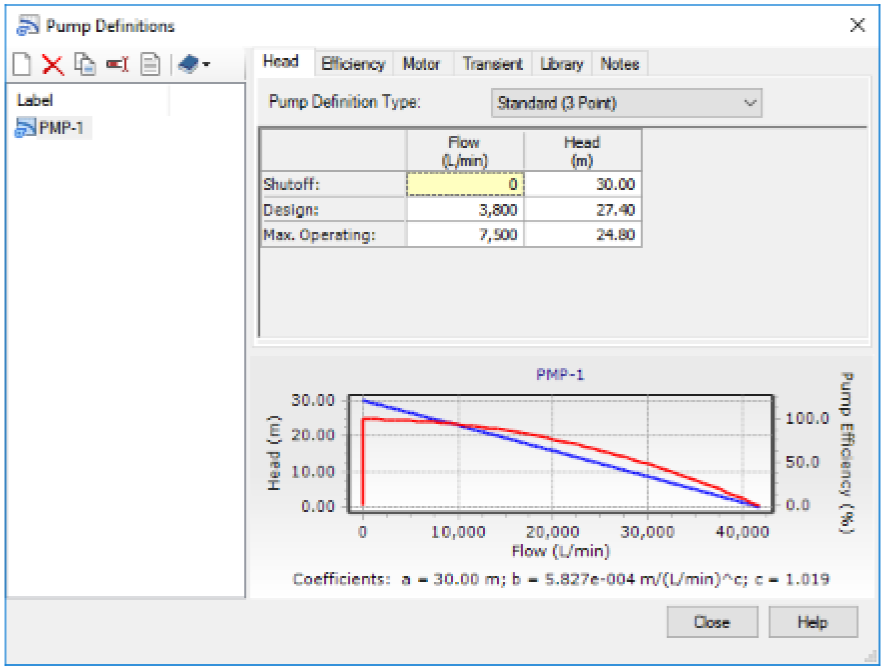
.

   g. Click OK.

h. Enter the following information:

	Flow (L/min)	Head (m)
Shutoff:	0	30.00
Design:	3,800	27.40
Max. Operating:	7,500	24.80

i. The Pump Definitions Manager should look like this:



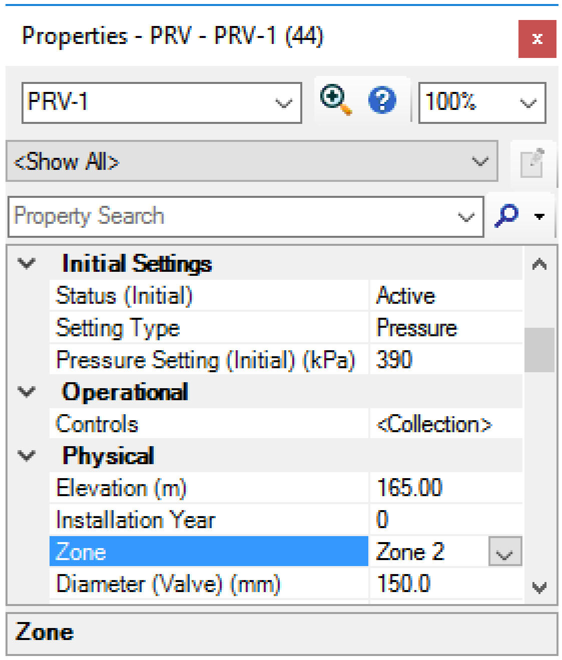
j. Click Close.

k. In the Property Editor, select **PMP-1** from the Pump Definition menu.

l. Close the Pump's Property Editor

7. Double click valve **PRV-1** in the drawing. Enter in the following data:
Status (Initial) = Active
Setting Type= Pressure
Pressure Setting (Initial)= 390 kPa
Elevation =165 m
Diameter (Valve) = 150 mm

8. Create Zone 2 and set the valve's Zone field to Zone-2.

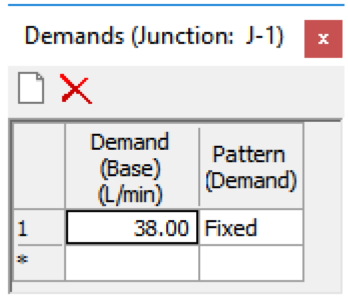


Close the PRV Property Editor.

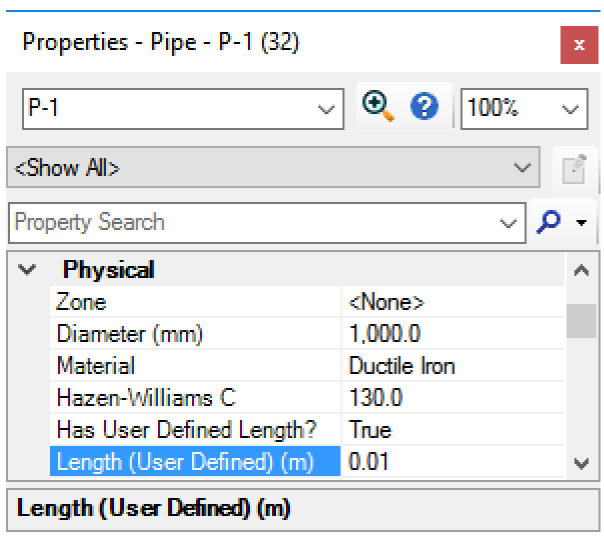
Enter the following elevation, demand, and zone data for J-1 through J-6 through the Junction Property Editor. Leave all other fields set to their default values

Junction	Ground Elevation (m)	Zone	Demand (l/min)
J-1	184	Zone-1	38
J-2	185	Zone-1	31
J-3	184	Zone-1	34
J-4	183	Zone-1	38
J-5	185.5	Zone-1	350
J-6	165	Zone-2	356

9. Click the ellipsis in the Demand Collection field to open the Demands manager enter each value for the Demand Base Flow (L/min). You may need to right-click on Demand (Base) column > Units and Formatting… > and set Unit to L/min.



10. Close the Junction Property Editor.
11. Specify user-defined lengths for pipes P-1, P-7, P-8, P-9 and P-10.
    a. Double-click pipe **P-1** to open the Pipe Property Editor.
    b. Set **Has User Defined Length?** to True. Then, enter a value of **0.01 m** in the Length field. Since you are using the reservoir and pump to simulate the connection to the main distribution system, you want headloss through this pipe to be negligible. Therefore, the length is very small and the diameter will be large.
    c. Enter **1000 mm** as the diameter of P1.



    d. Change the lengths (but not the diameters) of pipes P-7 through P-10 using the following user-defined lengths:
    P-7 = Length (User Defined): 400 m

P-8 = Length (User Defined): 500 m
P-9 = Length (User Defined): 31 m
P-10 = Length (User Defined): 100 m

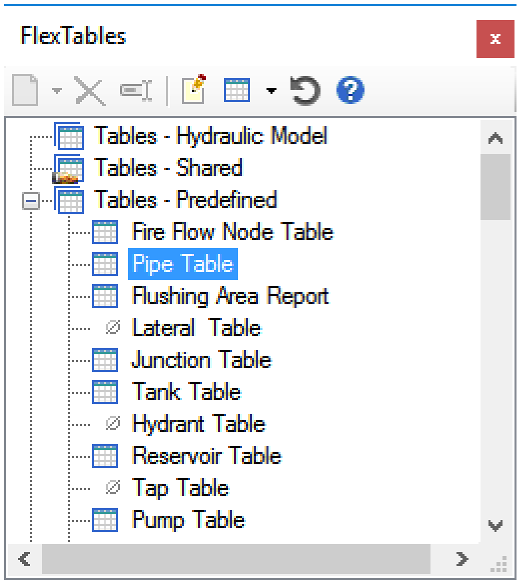
e. Close the Pipe Property Editor.

Step 4: Entering Data through FlexTables

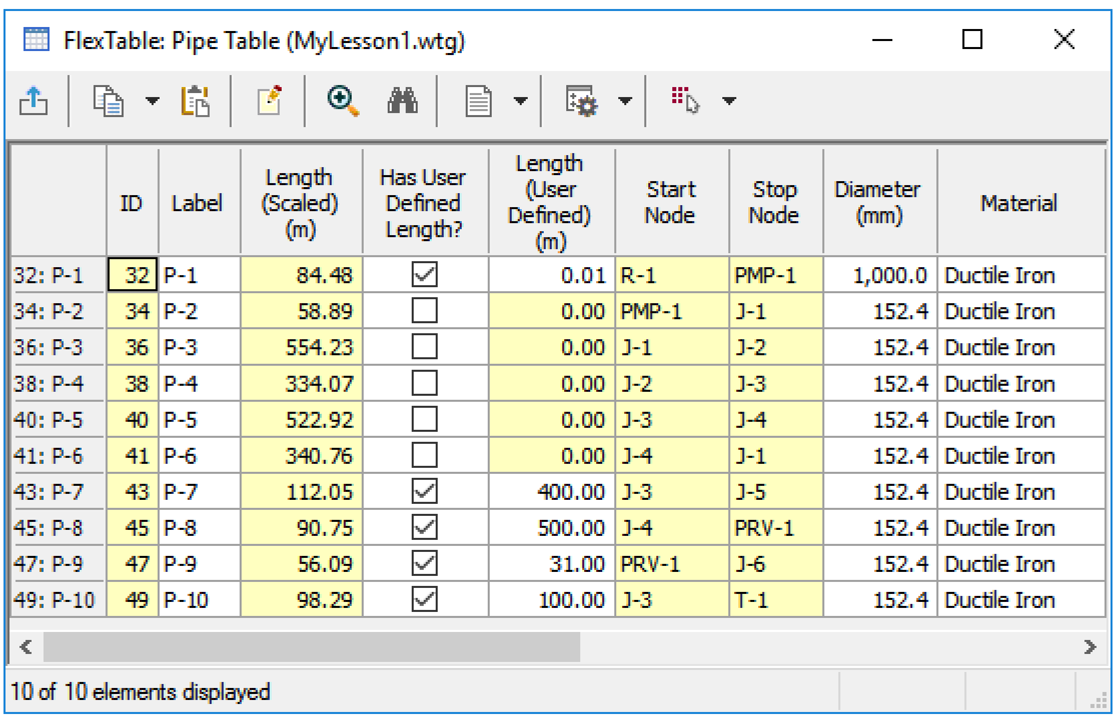
It is often more convenient to enter data for similar elements in tabular form, rather than to individually open the Property Editor for an element, enter the data, and then select the next element. Using FlexTables, you can enter the data as you would enter data into a spreadsheet.

To use FlexTables

1. Click the FlexTables button on the Home tab or choose **View > FlexTables**.



2. Double-click Pipe Table. Fields that are white can be edited, yellow fields can not. The Flextable columns have been re-ordered in the figure below.

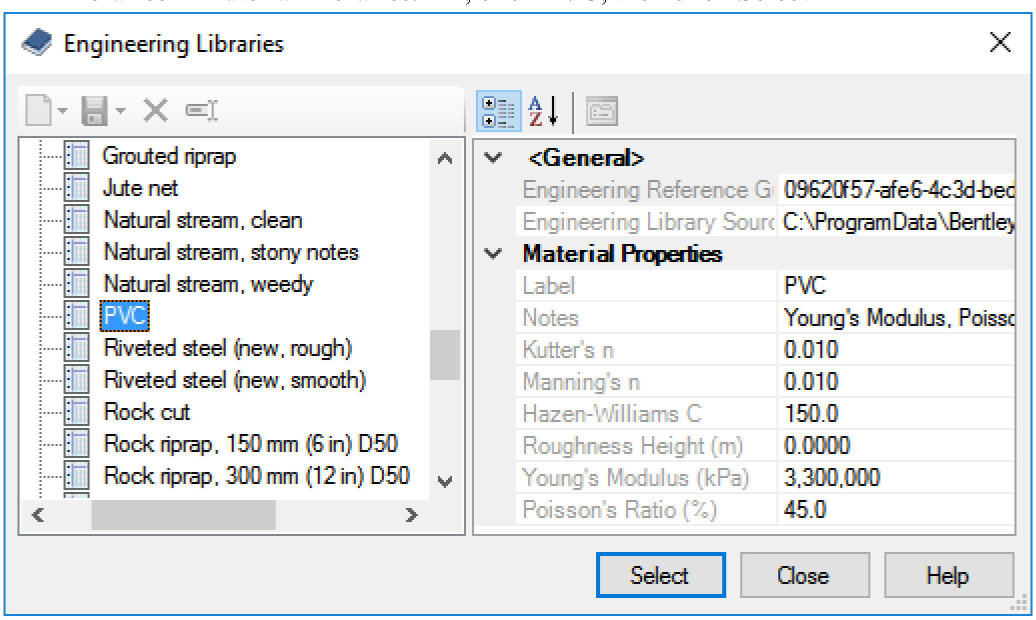


	ID	Label	Length (Scaled) (m)	Has User Defined Length?	Length (User Defined) (m)	Start Node	Stop Node	Diameter (mm)	Material
32: P-1	32	P-1	84.48	☑	0.01	R-1	PMP-1	1,000.0	Ductile Iron
34: P-2	34	P-2	58.89	☐	0.00	PMP-1	J-1	152.4	Ductile Iron
36: P-3	36	P-3	554.23	☐	0.00	J-1	J-2	152.4	Ductile Iron
38: P-4	38	P-4	334.07	☐	0.00	J-2	J-3	152.4	Ductile Iron
40: P-5	40	P-5	522.92	☐	0.00	J-3	J-4	152.4	Ductile Iron
41: P-6	41	P-6	340.76	☐	0.00	J-4	J-1	152.4	Ductile Iron
43: P-7	43	P-7	112.05	☑	400.00	J-3	J-5	152.4	Ductile Iron
45: P-8	45	P-8	90.75	☑	500.00	J-4	PRV-1	152.4	Ductile Iron
47: P-9	47	P-9	56.09	☑	31.00	PRV-1	J-6	152.4	Ductile Iron
49: P-10	49	P-10	98.29	☑	100.00	J-3	T-1	152.4	Ductile Iron

3. Enter the pipe diameters as shown in this table:

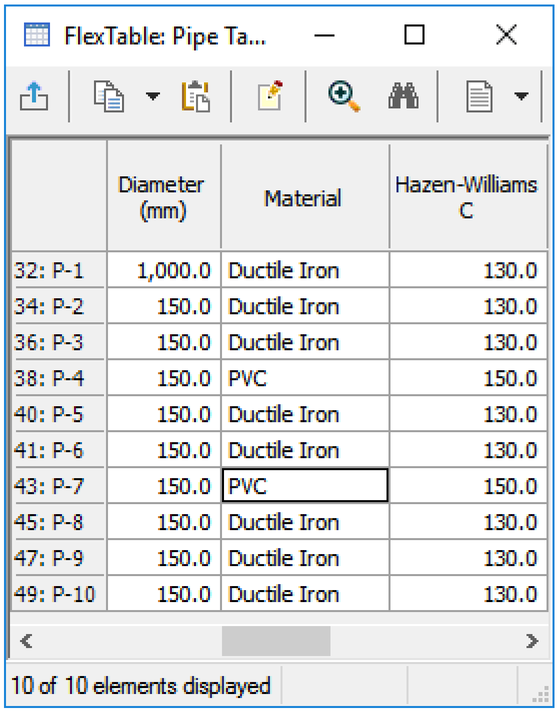
Pipe	Material	Diameter (mm)
P-1	Ductile Iron	1000
P-2	Ductile Iron	150
P-3	Ductile Iron	150
P-4	PVC	150
P-5	Ductile Iron	150
P-6	Ductile Iron	150
P-7	PVC	150
P-8	Ductile Iron	150
P-9	Ductile Iron	150
P-10	Ductile Iron	150

4. To enter the pipe material type for P-4 & P-7, click the appropriate pipe material cell, then click the ellipsis to open the Engineering Libraries. Expand Material Libraries > Material Libraries.xml, click PVC, then click Select



Notice that the C values for the pipes will be automatically assigned to preset values based on the material; however, these values could be modified if a different coefficient were required.

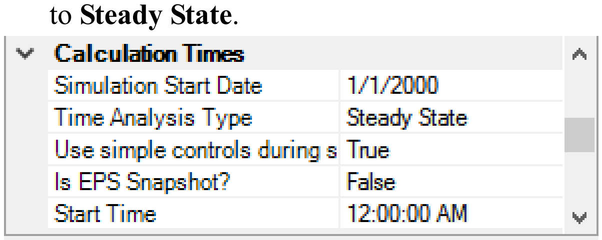
5. Leave other pipe data set to default values. Close the Pipe Flex Table and Property Editor when finished.



	Diameter (mm)	Material	Hazen-Williams C
32: P-1	1,000.0	Ductile Iron	130.0
34: P-2	150.0	Ductile Iron	130.0
36: P-3	150.0	Ductile Iron	130.0
38: P-4	150.0	PVC	150.0
40: P-5	150.0	Ductile Iron	130.0
41: P-6	150.0	Ductile Iron	130.0
43: P-7	150.0	PVC	150.0
45: P-8	150.0	Ductile Iron	130.0
47: P-9	150.0	Ductile Iron	130.0
49: P-10	150.0	Ductile Iron	130.0

Step 5: Run a Steady-State Analysis

1. Click **Analysis > Options** to open the Calculation Options manager.
2. Double-click **Base Calculation Options** under the Steady-State/EPS Solver heading to open the Property Editor. Make sure that the Time Analysis Type is set to **Steady State**.



Calculation Times	
Simulation Start Date	1/1/2000
Time Analysis Type	Steady State
Use simple controls during s	True
Is EPS Snapshot?	False
Start Time	12:00:00 AM

3. Close the Property Editor and the Calculation Options manager..
4. Click **Analysis > Validate**, then click Ok if no problems are found.

5. Click Compute  on the Home or Analysis tabs to analyze the model.
6. When calculations are completed, the Calculation Summary will open. A blue light is an informational message, a green light indicates no warnings or issues, a yellow light indicates warnings, and a red light indicates issues.
7. Close the Calculation Summary dialog.
8. Save your file

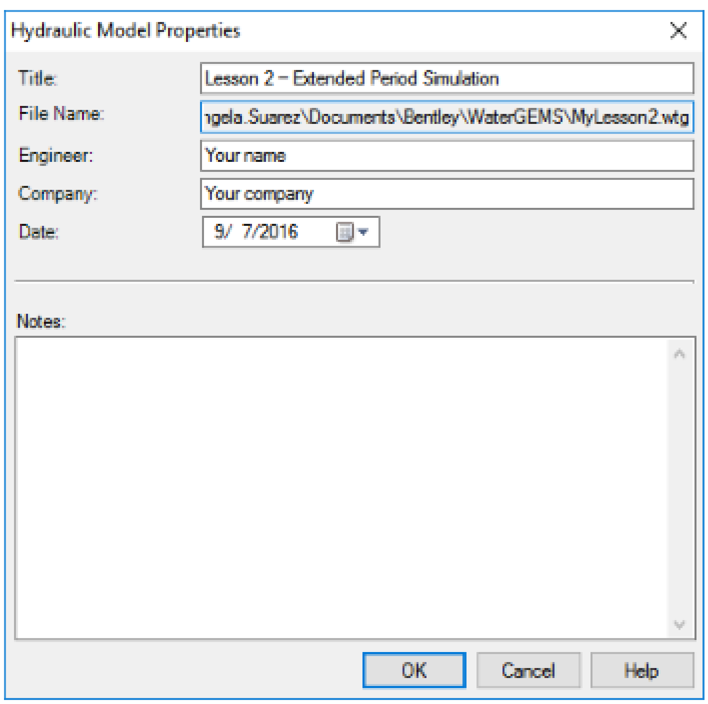
Extended Period Simulation

This lesson will illustrate how to model a water distribution system over time using the extended period simulation (EPS) calculation engine and by adding demand patterns to junctions. An EPS can be conducted for any duration you specify. System conditions are computed over the given duration at a specified time increment. Some of the types of system behaviors that can be analyzed using an EPS model include how tank levels fluctuate, when pumps are running, whether valves are open or closed, and how demands change throughout the day.

Opening the existing Hydraulic Model

This lesson is based on the Hydraulic Model created in Building a Network and Performing a Steady-State Analysis. If you have not completed it, then open the Hydraulic Model **Lesson2.wtg** from the Lessons directory. If you completed Lesson 1, then you can use the **MyLesson1** file you created.

1. Open **MyLesson1.wtg**. (or **Lesson2.wtg**)
2. Click **OK** to the message dialog that comes up about CONNECTED Projects
3. After you have opened the file, choose **File > Save As**.
4. Enter the filename **MyLesson2** and click **Save**.
5. Choose **File** > **Hydraulic Model Properties**, and change the Title to **Lesson 2—Extended Period Simulation**.



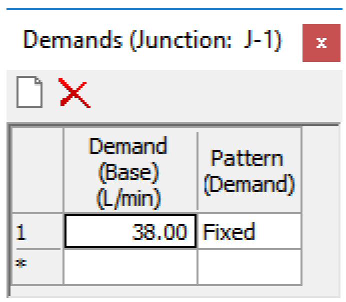
6. Click OK.

Step 1: Creating Demand Patterns

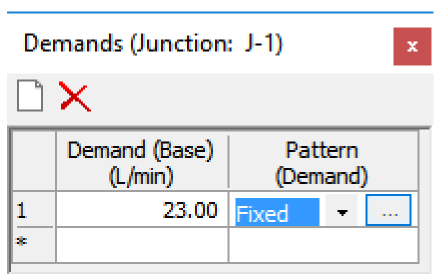
Water demand in a distribution system fluctuates over time. For example, residential water use on a typical weekday is higher than average in the morning before people leave for work, and is usually highest in the evening when residents are home and preparing dinner, washing clothes, etc. This variation in consumption over time can be modeled using demand patterns. Demand patterns are multipliers that vary with time and are applied to a given base demand, most typically the average daily demand.

In this lesson, you will be dividing the single fixed demands for each junction node in Lesson 1 into two individual demands with different demand patterns. One demand pattern will be created for residential use, and another for commercial use. You will enter demand patterns at the junction nodes through the junction's property editor.

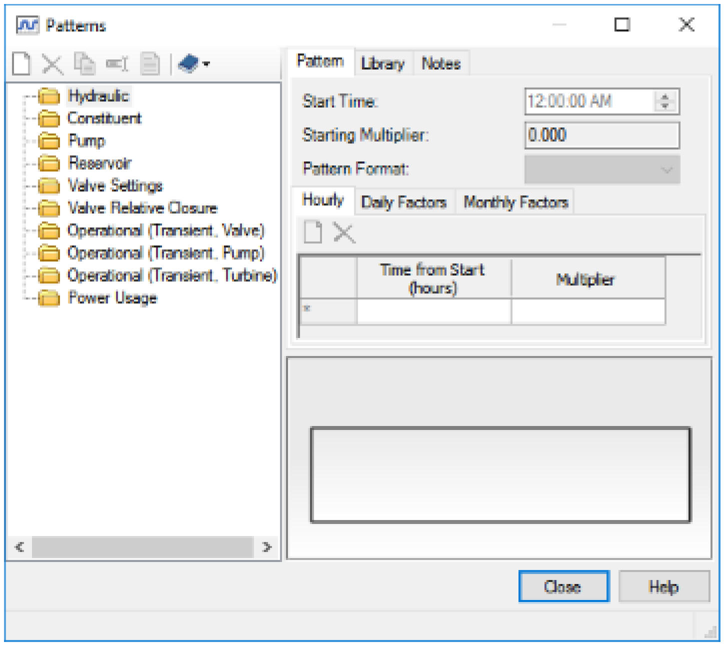
1. Open the Property Editor for Junction J-1 (double-click junction J-1) and click the ellipsis in the Demand Collection field to open the Demands box.



2. By default, the demand pattern is set to Fixed. Enter **23** l/min for Flow. (If field already has a number from previous lesson, type over it.)



3. Click in the Pattern (Demand) field and click the ellipsis to open the Patterns manager.



4. Highlight the *Hydraulic* folder and click New to create a hydraulic pattern.

   a. Rename the new pattern **Residential**.

   b. Leave the Start Time **12:00:00 AM**.

   c. Enter **0.5** as the Starting Multiplier.

   d. In the Pattern Format menu select **Stepwise**.

      The resulting demand pattern will have multipliers that remain constant until the next pattern time increment is reached.

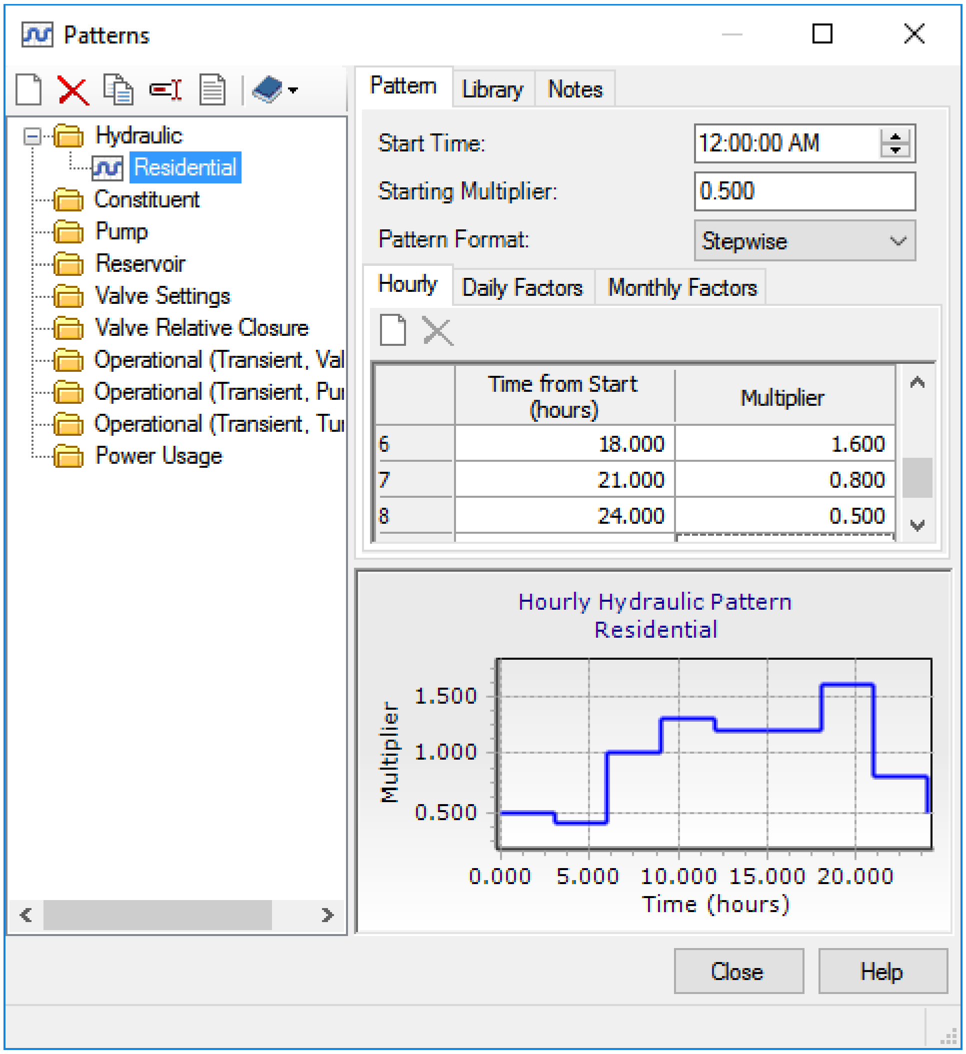
      Note that the multiplier for the last time given (24 hrs.) must be the same as the Starting Multiplier (0.5). These values are equal because the demand curve represents a complete cycle, with the last point the same as the first.

   e. Under the Hourly tab, enter the following times and multipliers:

Time from Start	Multiplier
3	.4
6	1
9	1.3
12	1.2

15	1.2
18	1.6
21	.8
24	.5

f. The Residential Pattern should look like the following:



5. Highlight the Hydraulic folder again and click New to create a new pattern for commercial demands.

   a. Rename the new pattern **Commercial**.

   b. Leave the Start Time **12:00:00 AM**.

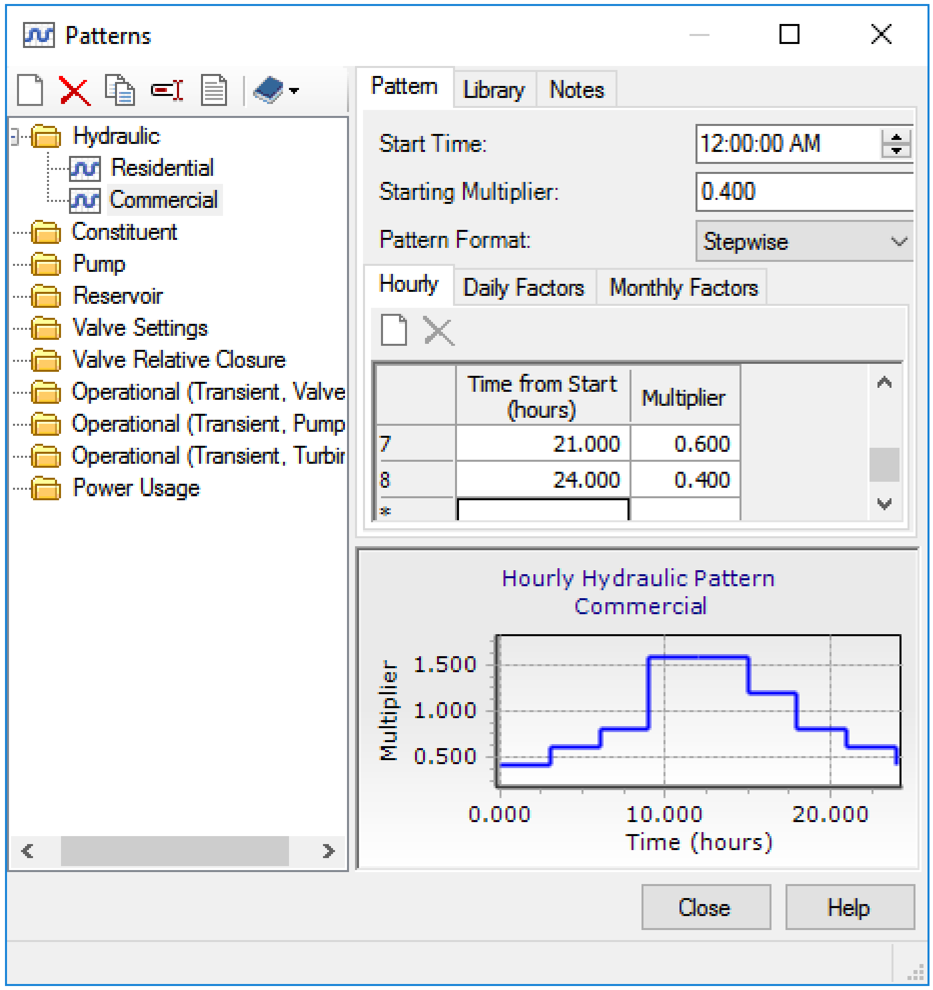
   c. Enter **0.4** as the Starting Multiplier.

d. In the Pattern Format menu select **Stepwise**.

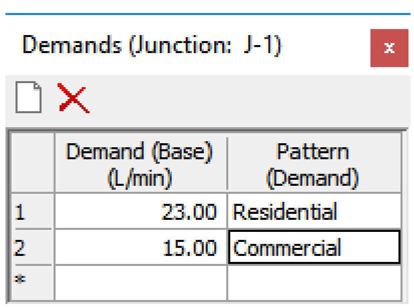
e. Under the Hourly tab, enter the following times and multipliers:

Time from Start	Multiplier
3	.6
6	.8
9	1.6
12	1.6
15	1.2
18	.8
21	.6
24	.4

f. The Commercial Pattern should look like the following:



6. Click Close.
7. Back in the Pattern field, for J-1 select **Residential** from the menu.
8. In the second row, enter a flow of **15 l/min** and select **Commercial** as the pattern for this row.

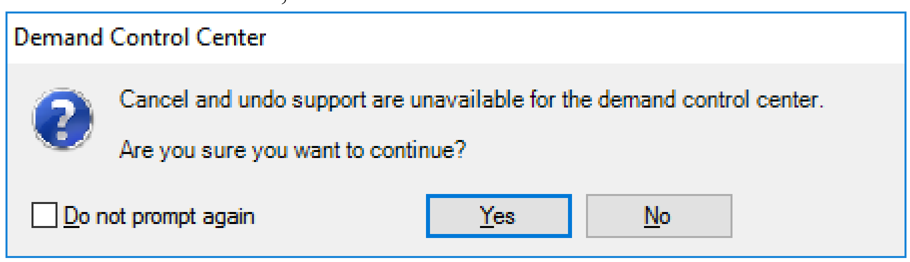


9. Close the Demands dialog box.

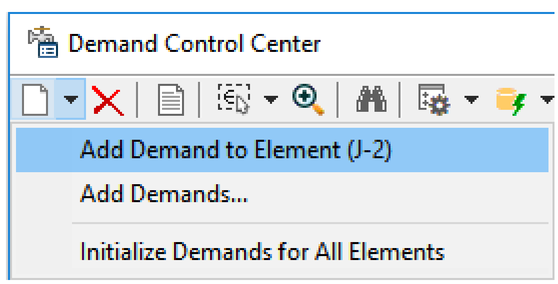
10. Close the J-1 Property Editor.
11. Enter the following demand data for the remainder junctions using the **Residential** and **Commercial** demand patterns already created.

Junction	Residential Demand (l/min)	Commercial Demand (l/min)
J-2	23	8
J-3	23	11
J-4	23	15
J-5	350	N/A
J-6	280	76

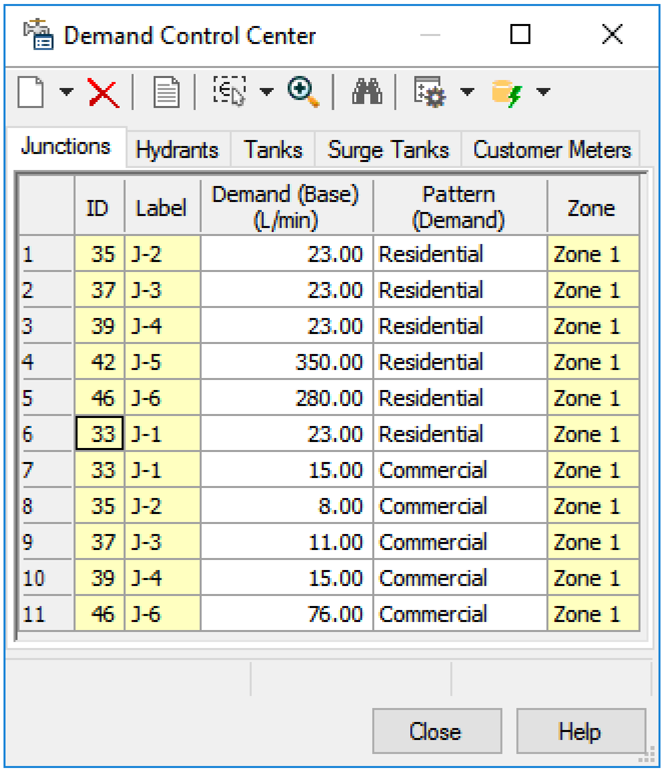
12. You can easily enter this data by using the Demand Control Center in **Components > Demand Center > Demand Control Center**.
13. If you receive a message about cancel and undo not being available for the Demand Control Center, click **Yes** to continue.



14. Enter the demand and corresponding pattern for each of the junctions. To enter a second demand for a junction, select the desired junction's label in the table and click the down arrow next to New and select **Add Demand to Element (J-#)** as needed.



15. When you are done, the Demand Control Center will look like this:



	ID	Label	Demand (Base) (L/min)	Pattern (Demand)	Zone
1	35	J-2	23.00	Residential	Zone 1
2	37	J-3	23.00	Residential	Zone 1
3	39	J-4	23.00	Residential	Zone 1
4	42	J-5	350.00	Residential	Zone 1
5	46	J-6	280.00	Residential	Zone 1
6	33	J-1	23.00	Residential	Zone 1
7	33	J-1	15.00	Commercial	Zone 1
8	35	J-2	8.00	Commercial	Zone 1
9	37	J-3	11.00	Commercial	Zone 1
10	39	J-4	15.00	Commercial	Zone 1
11	46	J-6	76.00	Commercial	Zone 1

16. Now you will set up an additional demand pattern to simulate a three-hour fire at node J-6.

    a. Highlight the label **J-6** and click **New > Add Demand to Element (J-6)**

    b. Enter a Demand (Base) of **2000 l/min**.

    c. Click the Pattern column and select the ellipsis button to open the Patterns Manager.

    d. Highlight the *Hydraulic* folder, then click New to create a new pattern.

    e. Rename the new pattern **3-Hour Fire**

    f. Leave the Start Time **12:00:00 AM**

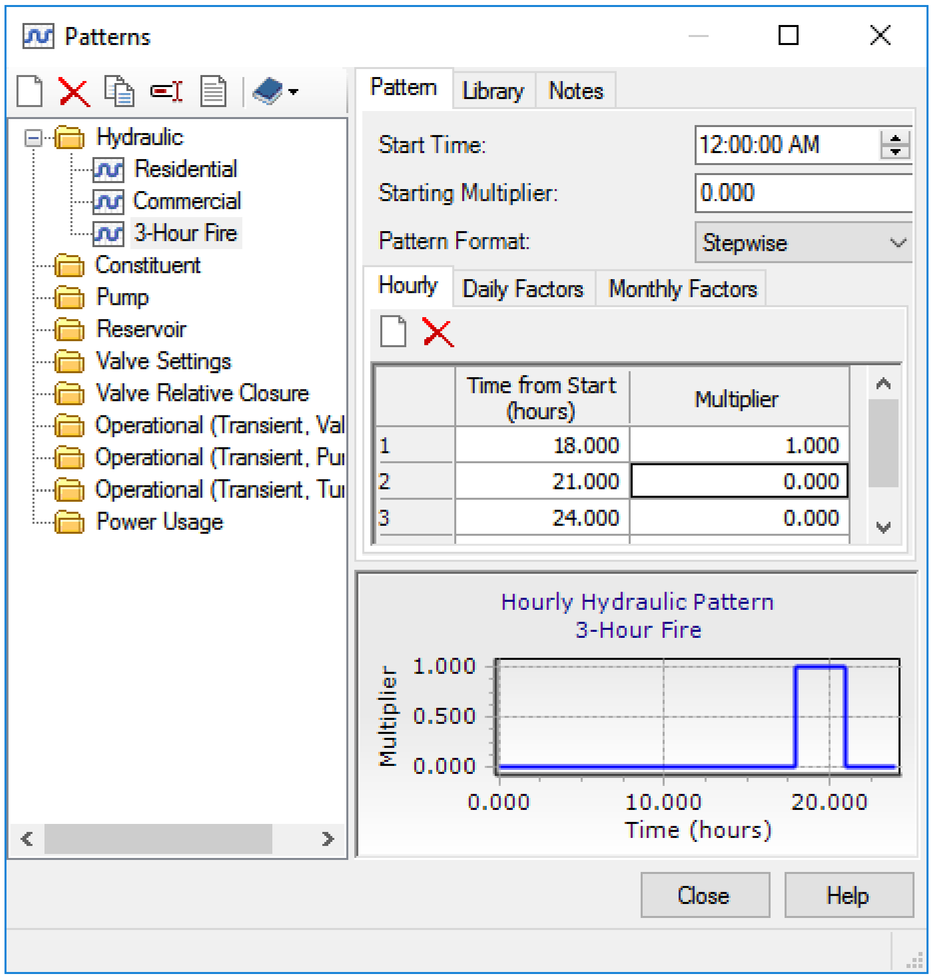
    g. Enter 0.00 as the Starting Multiplier.

    h. Select the Stepwise format.

i. Under the Hourly tab, enter the following times and multipliers:

Time from Start	Multiplier
18	1
21	0
24	0

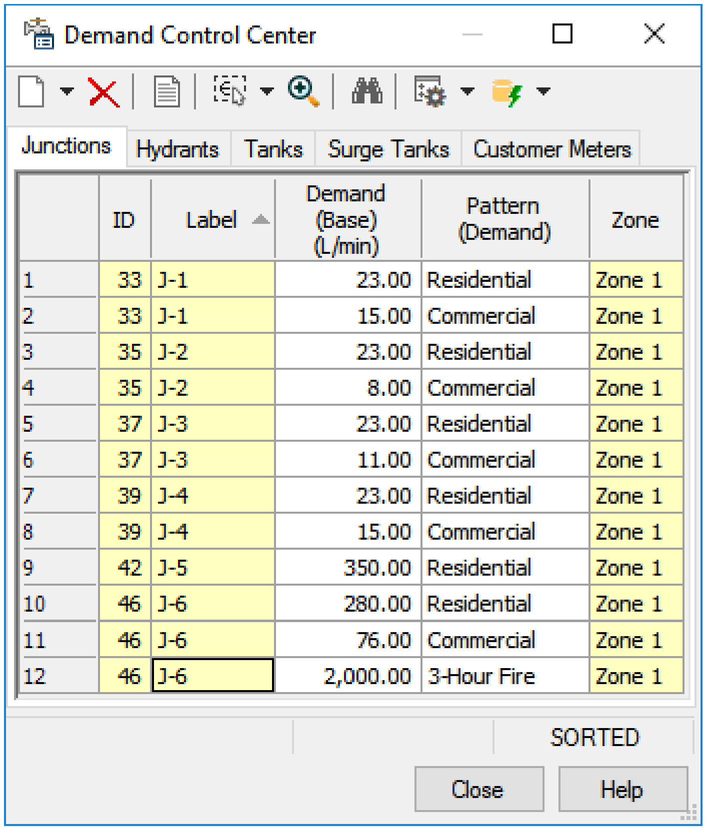
j. After you have filled in the table, look at the Graph in the lower section of the Patterns manager.



The value of the multiplier is zero, except for the period between 18 and 21 hours, when it is 1.0. Since the input the demand as 2000 l/min., the result will be a 2000 l/min. fire flow at junction J-6 between hours 18 and 21.

k. Click Close.

17. Select the new pattern, **3-Hour Fire**, from the Pattern (Demand) selection box.

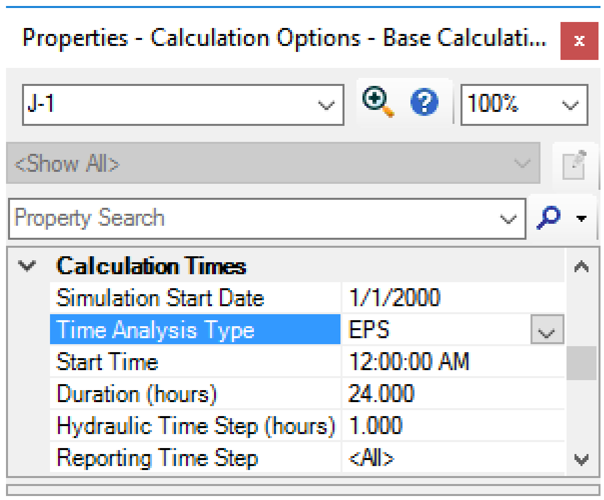


	ID	Label	Demand (Base) (L/min)	Pattern (Demand)	Zone
1	33	J-1	23.00	Residential	Zone 1
2	33	J-1	15.00	Commercial	Zone 1
3	35	J-2	23.00	Residential	Zone 1
4	35	J-2	8.00	Commercial	Zone 1
5	37	J-3	23.00	Residential	Zone 1
6	37	J-3	11.00	Commercial	Zone 1
7	39	J-4	23.00	Residential	Zone 1
8	39	J-4	15.00	Commercial	Zone 1
9	42	J-5	350.00	Residential	Zone 1
10	46	J-6	280.00	Residential	Zone 1
11	46	J-6	76.00	Commercial	Zone 1
12	46	J-6	2,000.00	3-Hour Fire	Zone 1

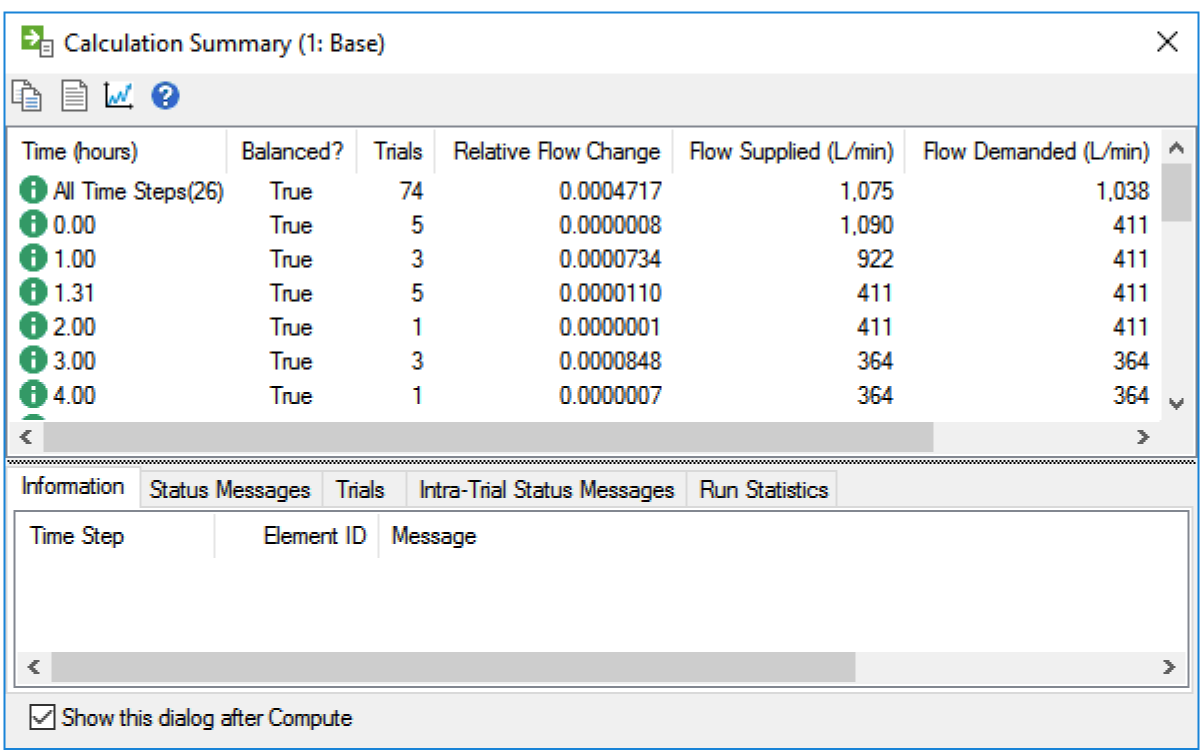
18. Close the Demand Control Center.

Step 2: Running an Extended Period Simulation (EPS)

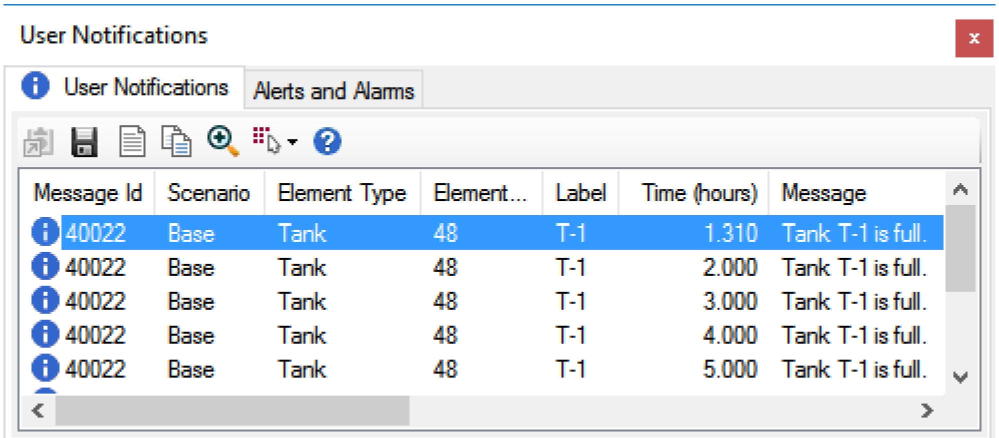
1. Go to **Analysis** > **Options** to open the Calculation Options dialog.
2. Double-click **Base Calculation Options** under *Steady State/EPS Solver* to open the property editor and select **EPS** from the Time Analysis Type menu



3. Click **Analysis** > **Validate**, then click Ok if no problems are found.
4. Click **Compute** on the Home or Analysis tabs to analyze the model.
5. The Calculation Summary opens.



6. Close the Calculation Summary.
7. When there are errors or warnings then the User Notifications dialog box opens. A blue light is an informational message, a yellow light indicates warnings, and a red light indicates issues.



8. Close the User Notifications dialog box and other open dialogs.
9. Click Save or choose File > Save to save the Hydraulic Model.

Scenario Management

One of the many Hydraulic Model tools in Bentley WaterGEMS is Scenario Management. Scenarios allow you to calculate multiple “What If?” situations in a single file. You may wish to try several designs and compare the results, or analyze an existing system using several different demand alternatives and compare the resulting system pressures.

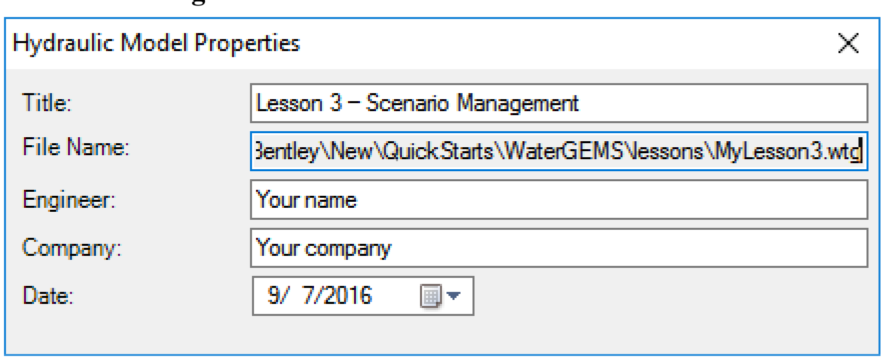
A scenario is a set of Alternatives, while alternatives are groups of actual model data. Scenarios and alternatives are based on a parent/child relationship where a child scenario or alternative inherits data from the parent scenario or alternative.

In Lessons 1 and 2, you constructed the water distribution network, defined the characteristics of the various elements, entered demands and demand patterns, and performed steady-state and extended period simulations. In this lesson, you will set up the scenarios needed to test four “What If?” situations for our water distribution system. These “What If?” situations will involve changing demands and pipe sizes. At the end of the lesson, you will compare all of the results using the Scenario Comparison tool.

Opening the existing Hydraulic Model

This lesson is based on the Hydraulic Model created in **Extended Period Simulations**. If you have not completed it, then open the Hydraulic Model **Lesson3.wtg** from the Lessons folder. If you completed Lesson 2, then you can use the **MyLesson2** file you created.

1. Open **MyLesson2.wtg** (or **Lesson3.wtg**)
2. Click **OK** to the message dialog that comes up about CONNECTED Projects.
3. After you have opened the file, choose **File** > **Save As**.
4. Enter the filename **MyLesson3** and click **Save**.
5. Choose **File** > **Hydraulic Model Properties**, and change the Title to **Lesson 3—Scenario Management**.

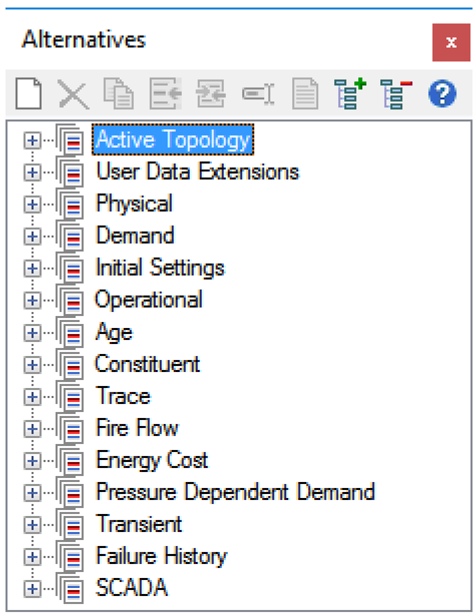


6. Click OK.

Step 1: Create a New Alternative

First, you need to set up the required data sets, or alternatives. An alternative is a group of data that describes a specific part of the model.

There are fifteen alternative types:

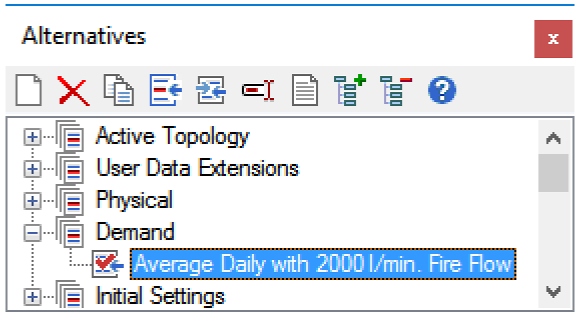


In this example, you need to set up different physical and demand alternatives for each design trial you want to evaluate. Each alternative will contain different pipe sizes or demand data.

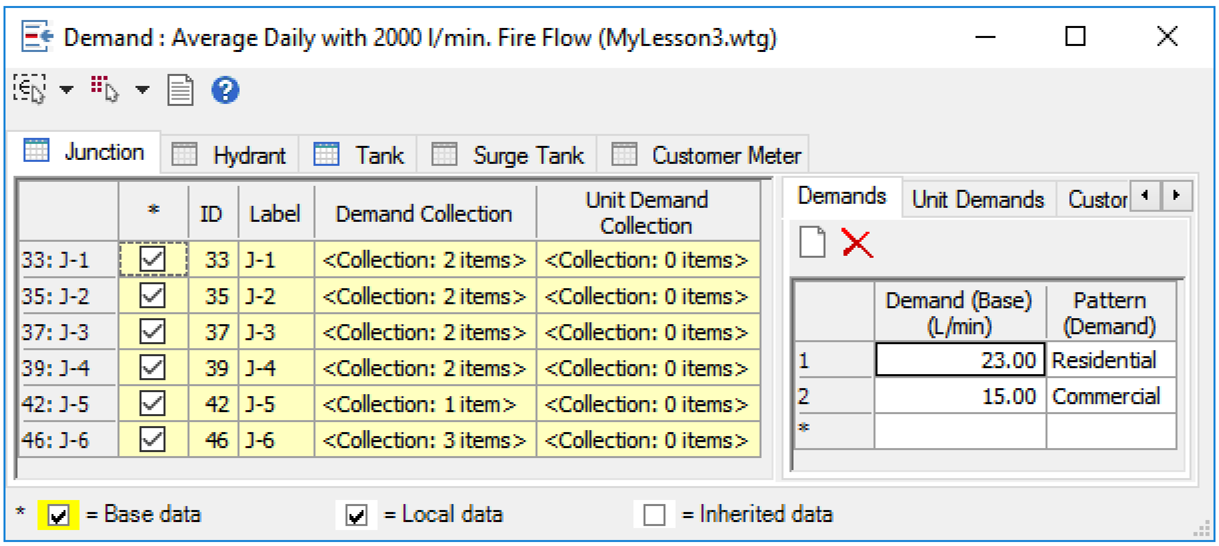
In Bentley WaterGEMS, you create families of alternatives from base alternatives. Base alternatives are alternatives that do not inherit data from any other alternative. Child alternatives can be created from the base alternative. A Child alternative inherits the characteristics of its parent, but specific data can be overridden to be local to the child. A child alternative can, in turn, be the parent of another alternative.

1. Choose **Analysis > Alternatives**.
2. Expand the Demand alternative. The Base Demand alternative contains the demands for the current distribution system.

3. Change the default demand name.
   a. Select **Base Demand**, then right click to rename it.
   b. Enter the new name, **Average Daily with 2000 l/min. Fire Flow**.



   c. Double-click on the alternative to open the Demand Alternative manager.

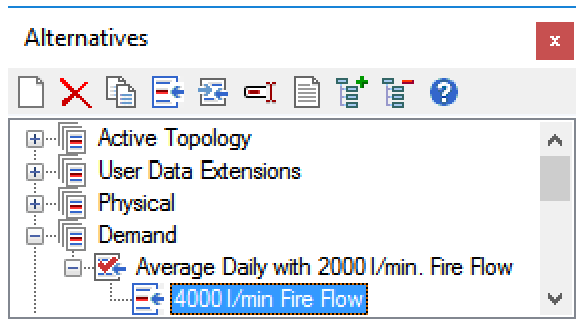


4. Close the Demand Alternative Manager.

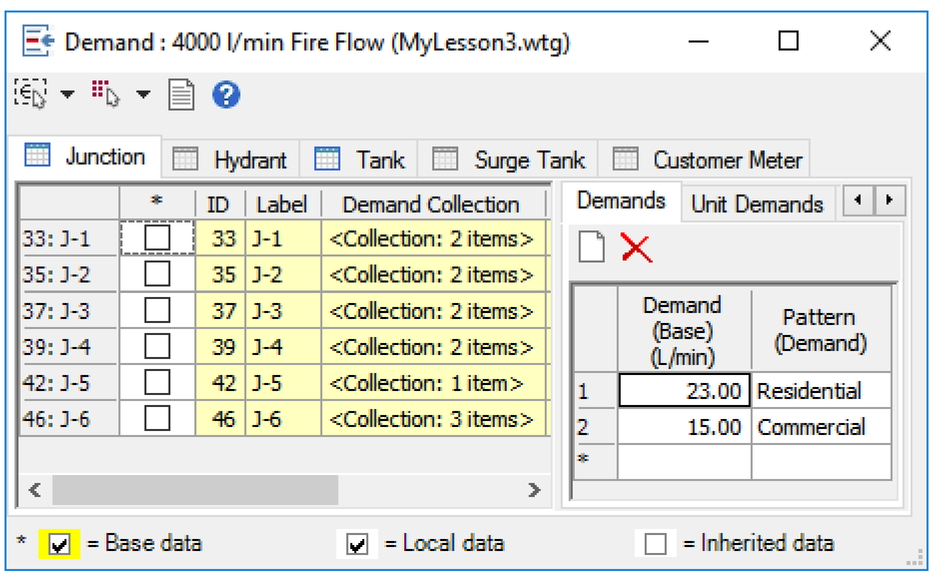
5. Now you should add a child of the **Average Daily with 2000 l/min. Fire Flow** alternative, because the new alternative will inherit most data. Then, you can locally change the data that you want to modify. You will modify the existing demand data by increasing the fire flow component at node J-6 from 2000 l/min. to **4000 l/min**.

   a. In the Alternatives manager, right-click the **Average Daily with 2000 l/min. Fire Flow** alternative, then select **New > Child Alternative**.

   b. Highlight the new alternative and click Rename. Enter a label of **4000 l/min Fire Flow** for the new Alternative.

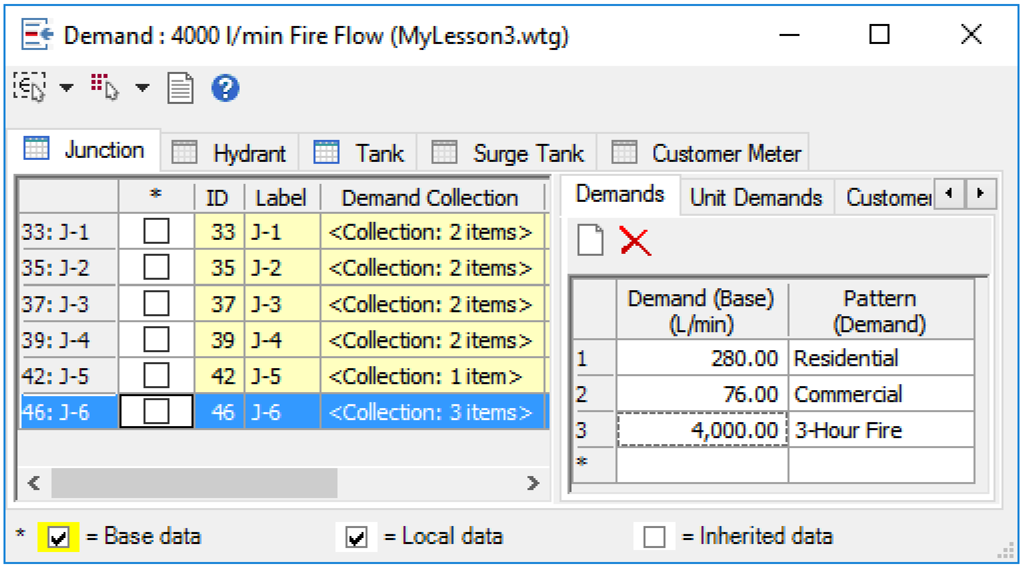


   c. Double-click to open the Demand Alternatives editor for the new alternative which shows the data that was inherited from the parent alternative (Empty check box next to ID)



If you change any data, the check box will become selected because that record is now local to this alternative and not inherited from the parent.

6. Click in the Demand Collection column for node **J-6**. Change the 2000 l/min. fire demand to **4000 l/min**.



7. **Close** to exit the Demand Alternative Editor.

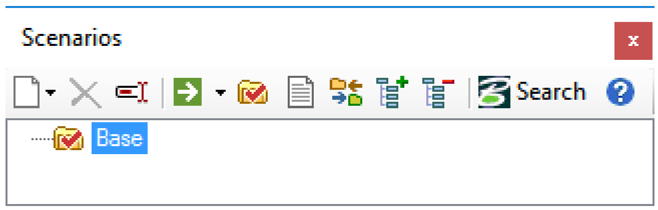
8. **Close** the Alternatives Manager

Step 2: To create and edit Scenarios

Alternatives are the building blocks of a scenario. A scenario is a set of one of each of the types of alternatives, plus all of the calculation information needed to solve a model.

Just as there are base, parent, and child alternatives, there are also base, parent, and child scenarios. The difference is that instead of inheriting model data, scenarios inherit sets of alternatives. To change the new scenario, change one or more of the new scenario's alternatives. For this lesson, you will create a new scenario for each different set of conditions you need to evaluate.

1. Choose **Analysis > Scenarios** to open the Scenario Manager.



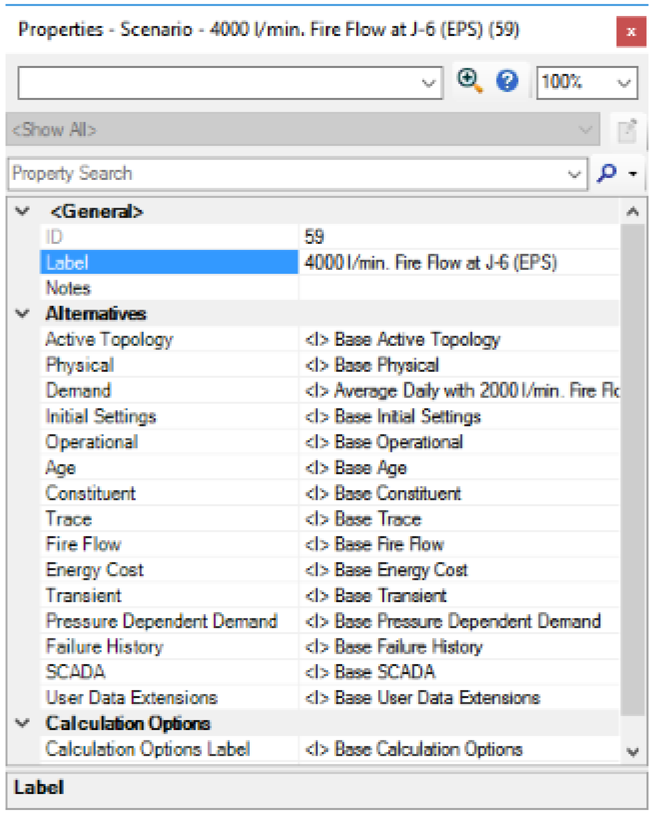
   There is always a default Base Scenario that is composed of the base alternatives. Initially, only the Base is available, because you have not created any new scenarios.

2. Rename the Base Scenario to **2000 l/min., 3-hour Fire Flow at J-6 (EPS)**.

3. Create a child scenario from the existing base scenario to incorporate the new demand alternative.

    a. Right-click on the **2000 l/min., 3-hour Fire Flow at J-6 (EPS)** scenario and select **New > Child Scenario.**

    b. Enter a scenario name of **4000 l/min. Fire Flow at J-6 (EPS)**. Double-click the scenario to open the Property Editor for the scenario.

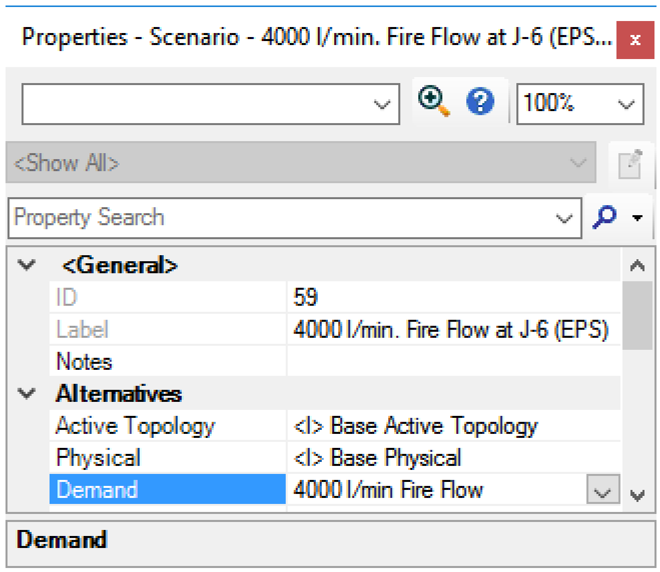


The new scenario lists the alternatives as inherited from the base scenario.

4. Your new Child Scenario initially consists of the same alternatives as its parent scenario. Set the Demand Alternative to the new alternative you created, **4000 l/ min. Fire Flow**.

   a. Click in the **Demand Alternative** field

   b. From the menu, select the **4000 l/min. Fire Flow** alternative.

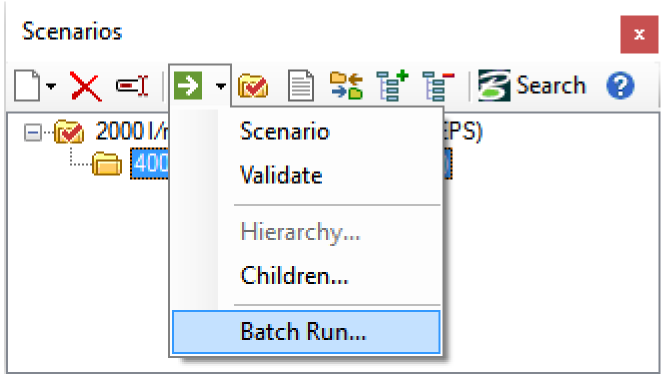
   

   Now the demand alternative of the new child Scenario is no longer inherited from the parent scenario, but is local to this scenario.

   c. Close the Property Editor.

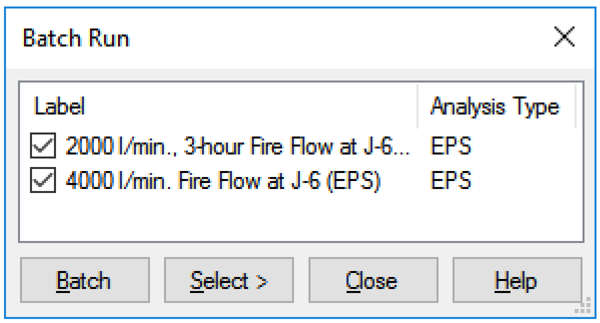
Step 3: To calculate both of the scenarios using the Batch Run tool

1. In the Scenario Manager, click the down arrow next to **Compute** and Select **Batch Run**



.

2. Select both check boxes next to the scenario names in the Batch Run dialog.



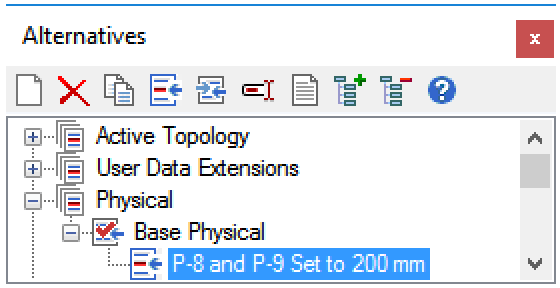
3. Click **Batch**.
4. Click **Yes** at the prompt to run the batch for two scenarios.
5. After computing finishes, click **OK**.

Step 4: To create a Physical Alternative

You need to further examine what is going on in the system as a result of the fire flow, and find solutions to any problems that might have arisen in the network as a result. You can review output tables to quickly see what the pressures and velocities are within the system, and create new alternatives and scenarios to capture your modifications. Notice that in the ***4000 l/min. Fire Flow at J-6*** scenario, pressure at J-6 drops below vapor pressure for the entire duration of the fire flow (18-21hrs) indicating that such high flow could not be supplied. We will try to increase pipe sizes for P-8 and P-9 to 200 mm.

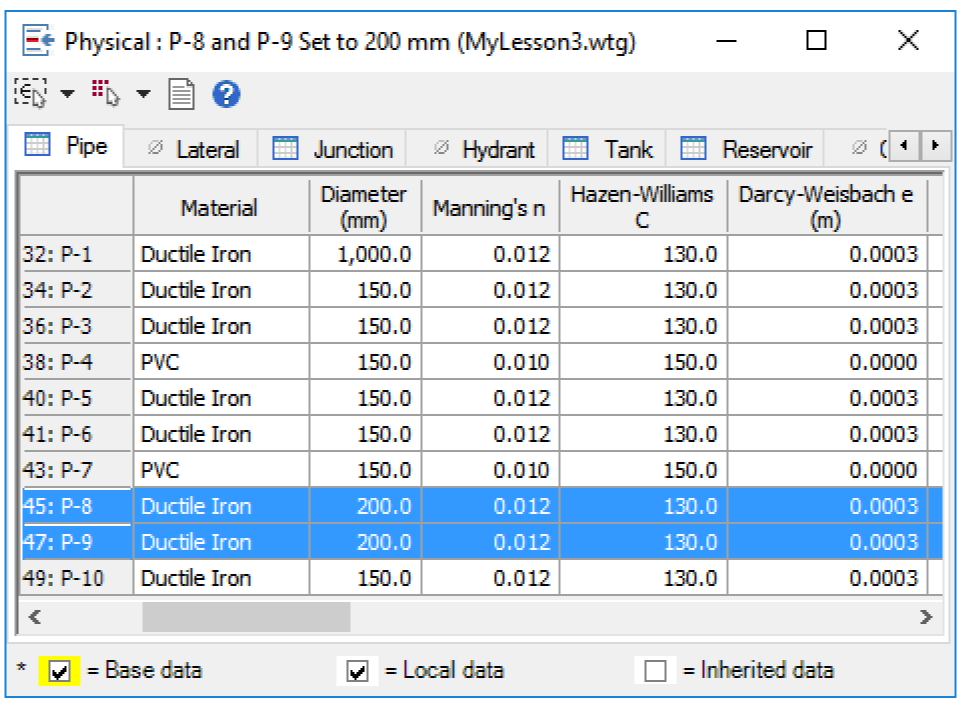
1. A new physical alternative will be created to contain the new pipe sizes. Click **Analysis > Alternatives**. Under Physical, highlight **Base Physical.** Right click and select **New > Child Alternative.**

1. Rename the new Child Alternative **P-8 and P-9 Set to 200 mm**.



1. Double-click the newly created physical alternative to open the Physical alternative editor.

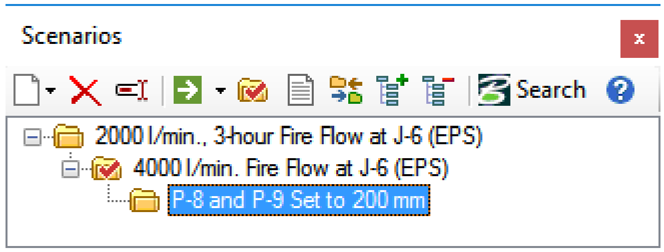
2. In the Pipe tab for this Alternative, change the diameter for pipes P-8 and P-9 to 200 mm.



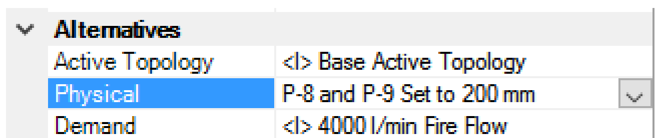
	Material	Diameter (mm)	Manning's n	Hazen-Williams C	Darcy-Weisbach e (m)
32: P-1	Ductile Iron	1,000.0	0.012	130.0	0.0003
34: P-2	Ductile Iron	150.0	0.012	130.0	0.0003
36: P-3	Ductile Iron	150.0	0.012	130.0	0.0003
38: P-4	PVC	150.0	0.010	150.0	0.0000
40: P-5	Ductile Iron	150.0	0.012	130.0	0.0003
41: P-6	Ductile Iron	150.0	0.012	130.0	0.0003
43: P-7	PVC	150.0	0.010	150.0	0.0000
45: P-8	Ductile Iron	200.0	0.012	130.0	0.0003
47: P-9	Ductile Iron	200.0	0.012	130.0	0.0003
49: P-10	Ductile Iron	150.0	0.012	130.0	0.0003

3. Close the alternative editor dialog.

4. Create a new scenario having the new physical alternative with the pipe sizes for P8 and P-9 increased to 200 mm.
5. Choose **Analysis > Scenarios**.
   d. Select **4000 l/min. Fire Flow at J-6 (EPS)** in the list of Scenarios.
   e. Right click to select **New > Child Scenario.**
   f. Name the new Scenario **P-8 and P-9 Set to 200 mm**.



   g. Double click scenario **P-8 and P-9 Set to 200 mm** to open the Property Editor for the scenario. Click Physical and select the **P-8 and P-9 Set to 200 mm** alternative.



   h. Close the Property Editor to return to the Scenarios Manager.
   i. Highlight the **P-8 and P-9 Set to 200 mm** scenario and right click to **Make Current**. A red check mark on the scenario’s icon indicate that this is the current scenario.
   j. Click **Compute** to run the Scenario.
6. Close the open boxes and save the model.

Reporting Results

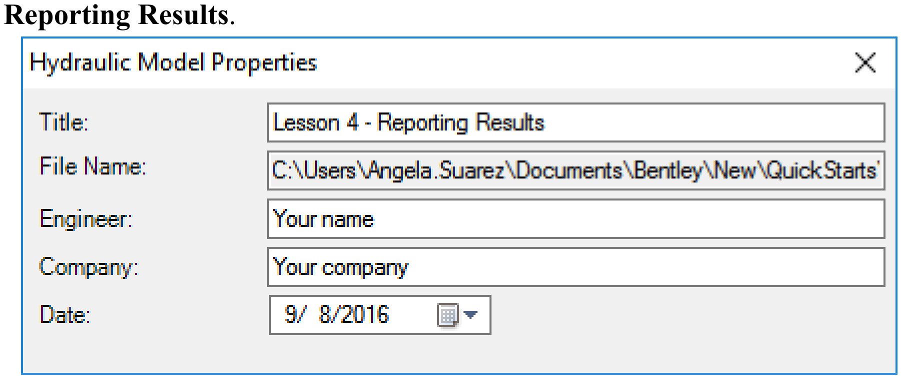
An important feature in all water distribution modeling software is the ability to present results clearly. This lesson outlines several of Bentley WaterGEMS reporting features, including:

- **Element Tables (FlexTables)**, for viewing, editing, and presentation of selected data and elements in a tabular format.
- **Profiles**, to graphically show, in a profile view, how a selected attribute, such as hydraulic grade, varies along an interconnected series of pipes.
- **Contouring**, to show how a selected attribute, such as pressure, varies throughout the distribution system.
- **Element Annotation**, for dynamic presentation of the values of user-selected variables in the plan view.
- **Color Coding**, which assigns colors based on ranges of values to elements in the plan view. Color coding is useful in performing quick diagnostics on the network.
- **Reports**, which display and print information on any or all elements in the system.

For this lesson, you will use the file from the **Scenario Management** lesson, saved as **MyLesson3.wtg** in the Lessons folder. If you did not complete this lesson, you may use the file **Lesson4.wtg**.

To open the existing Hydraulic Model

1. Open **MyLesson3.wtg (**or **Lesson4.wtg)**.
2. Click **OK** to the message dialog that comes up about CONNECTED Projects.
3. Select **File** > **Save As**.
4. Enter the filename **MyLesson4**, and click **Save**.
5. Select **File** > **Hydraulic Model Properties**, and change the Title to **Lesson 4 - Reporting Results**.



FlexTables

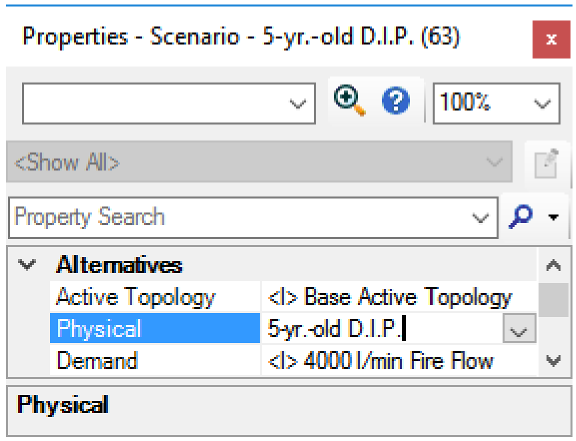
When data must be entered for a large number of elements, clicking each element and entering the data can be time consuming. FlexTable fields can be changed using the global edit tool, or filtered to display only the desired elements. Values that are entered into the table will be automatically updated in the model. The tables can also be customized to contain only the desired data. Columns can be added or removed, and the order in which they appear can be rearranged.

FlexTables are dynamic tables of input values and calculated results. White columns are editable input values, and yellow columns are non-editable calculated values. When data is entered into a table directly, the values in the model will be automatically updated. These tables can be printed or copied into a spreadsheet program.

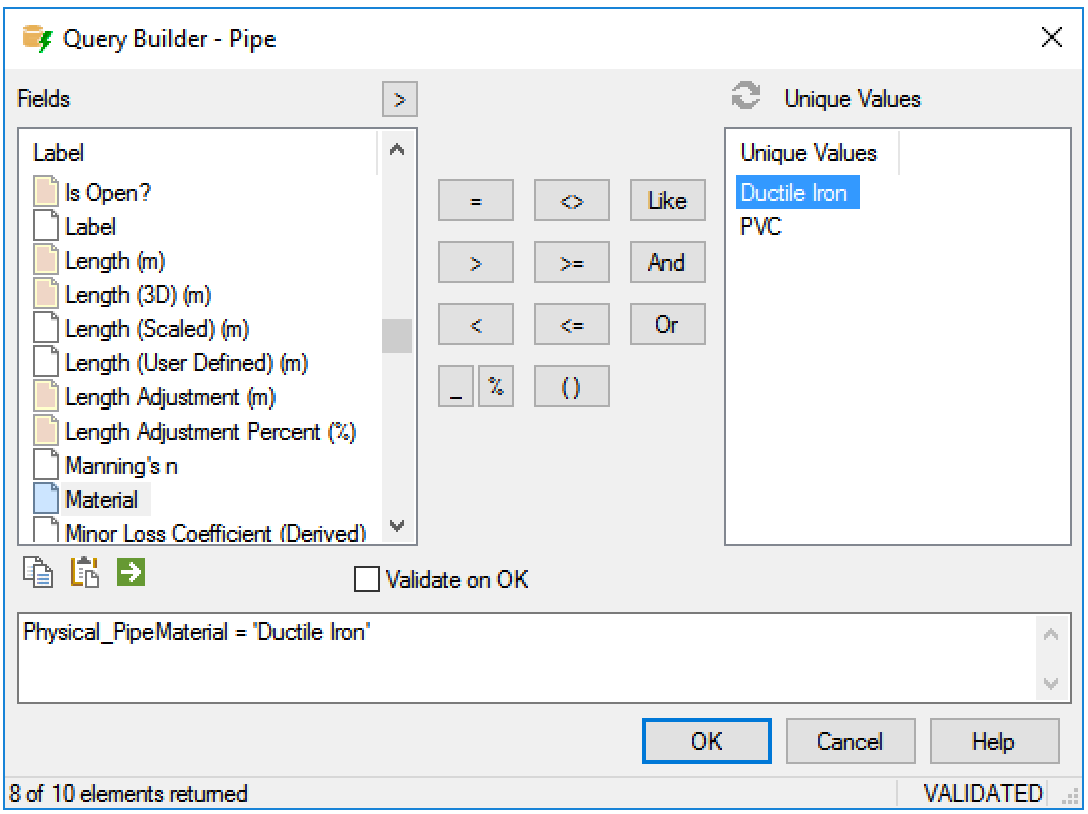
Global Edit and Filtering are very useful tools. For example, if you decide to evaluate how the network might operate in five years. Assume that the C factor for 5-year old ductile iron pipe reduces from 130 to 120. It would be repetitive to go through and edit the pipe roughness through the individual pipe Property Editors, particularly when dealing with a large system. Instead, you will use the filter tool in this example to filter out the PVC pipes, and then use global edit tool to change the pipe roughness on the ductile iron pipes only.

To use Global Edit and Filtering

1. Set up a new Alternative and Scenario to capture the changes to the C values.
    a. Click **Analysis** > **Alternatives**. Highlight the **P-8 and P-9 Set to 200 mm** Physical Alternative and click New Child Alternative.
    b. Rename the new Alternative **5-yr.-old D.I.P.**
    c. Close the Alternatives manager.
    d. Choose **Analysis > Scenarios**.
    e. Select the **P-8 and P-9 Set to 200 mm** scenario.
    f. Right click and select New > Child Scenario.
    g. Rename the new scenario **5-yr.-old D.I.P.**
    h. Double-click the new scenario to open the Property Editor. Change the Physical alternative to 5-yr.-old D.I.P.
    i. Close the scenario manager.



2. Choose **View > FlexTables > Pipe Table**.
3. Right-click the Material column and choose **Filter > Custom** from the menu.
4. The query builder opens.



a. Double-click on **Material.**

b. Click the = equal sign.

c. Click  to select the Unique Values for Material

d. Double-click **Ductile Iron**.

e. Click Apply  , then Click OK.

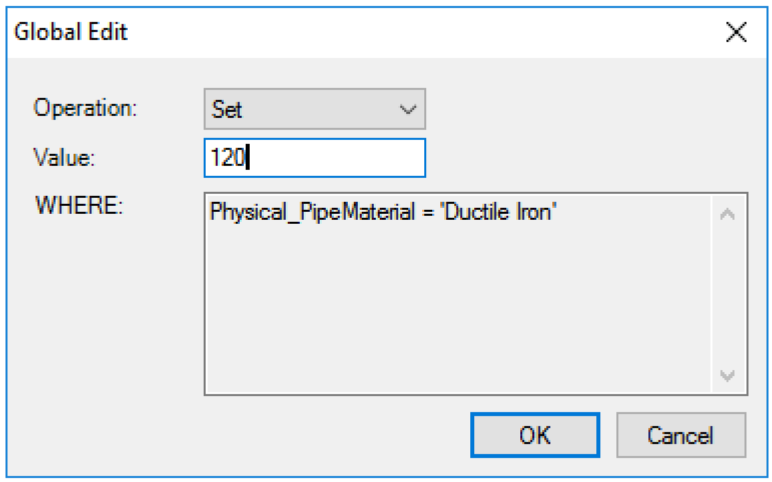
f. Click OK to exit the query builder.

5. Use the Global Edit tool to modify all of the roughness values in the table.

   a. Right-click the Hazen-Williams C column and select **Global Edit**.

   b. Select **Set** from the Operation list.

   c. Enter **120** into the Global Edit field.

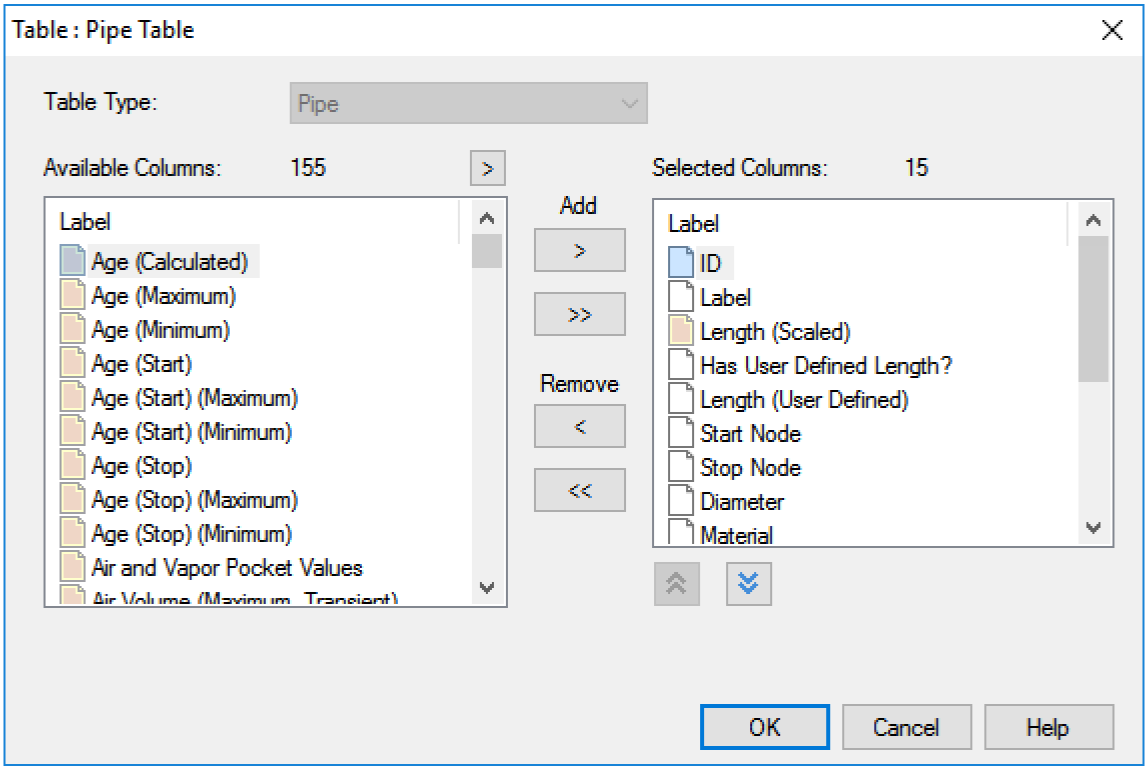


   d. Click **OK**. All of the values are now set to 120.

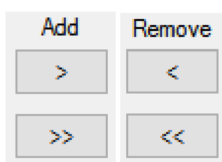
6. To deactivate the filter, right-click on any column header and click **Filter** > **Reset** from the menu. Click **Yes** to reset the filter.

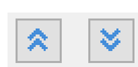
7. You may also wish to edit a table by adding or removing columns.

   a. Click Edit  to open the table editor.



   b. Scroll through the list on the left to view the types of data available for placement in the table. You can select an item to add or remove from the table.

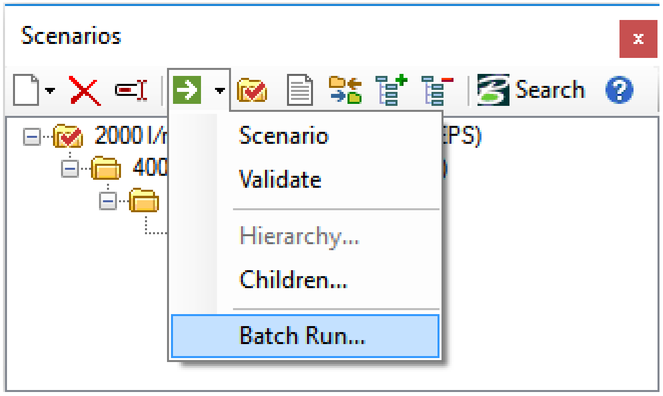


   c. You can adjust the order in which the columns will be displayed by using the arrows below Selected Columns .

   d. Click Ok to save your changes or Cancel to exit the table without making changes.

8. Close the table.

9. Choose **Analysis** > **Scenarios** > **Compute Scenario** > **Batch Run**.



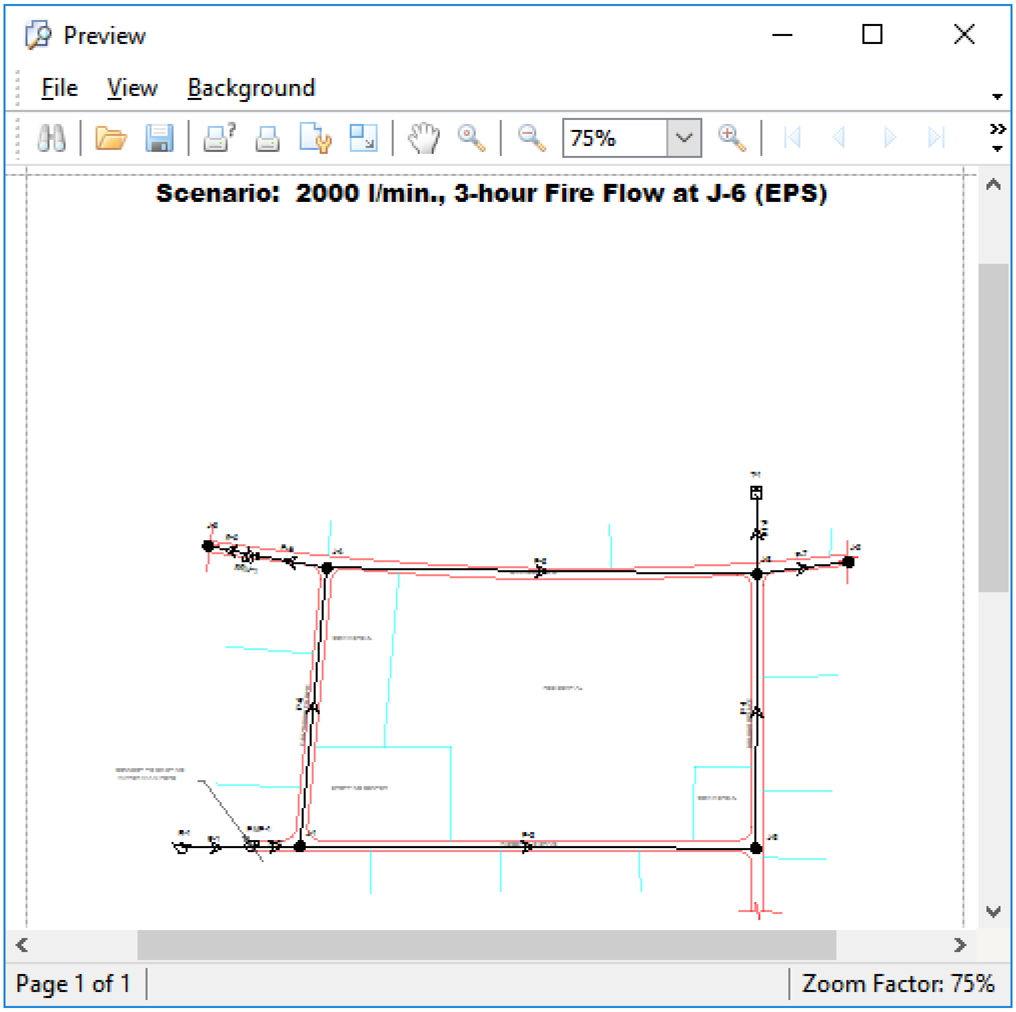
10. Check **5-yr.-old D.I.P.**, and then click Batch.
11. Close the table when you are finished.

Create a Print Preview and Profile

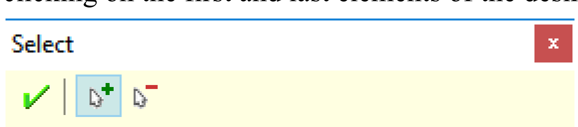
1. To create a print preview of the distribution system, choose **File** > **Print** > **Print Preview.** Leave the Settings as “Fit To Page”.

   This option will create a preview of the entire system regardless of what the screen shows.

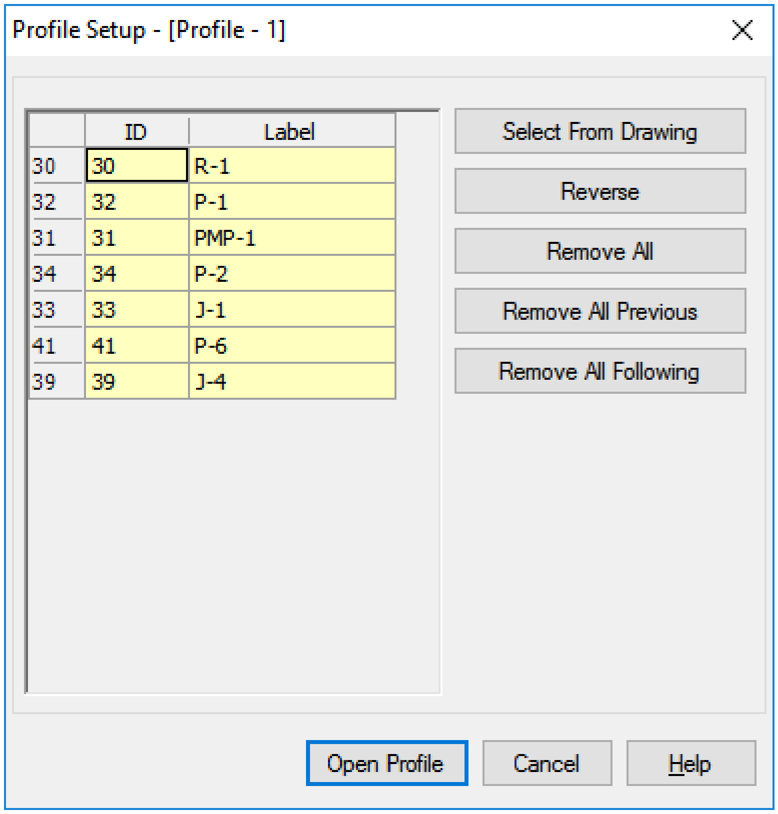
The print preview opens in a separate window, which can then be printed or copied to the clipboard.



2. Close the print preview window.
3. To create a profile view, choose **View** > **Profiles**. This activates the Profiles manager.
4. Click New to open the Profile Setup manager, and then click **Select From Drawing** to choose the elements to profile.
5. The dialog box closes and **Select** opens. Choose a few elements (or a path by clicking on the first and last elements of the desired path) and click Done  .

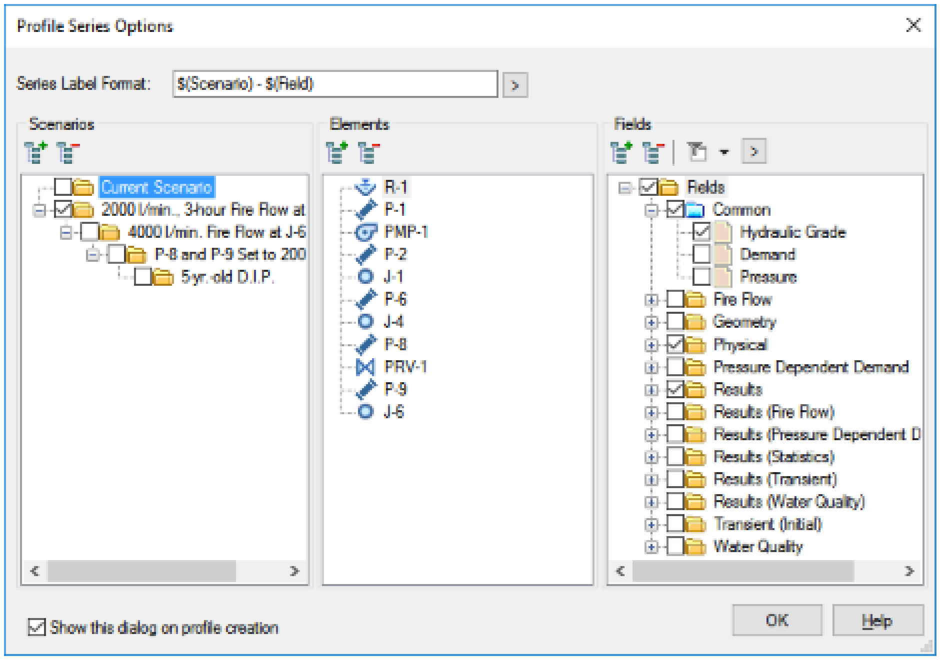


6. The Profile Setup opens with the selected elements appearing, in order, in the list.

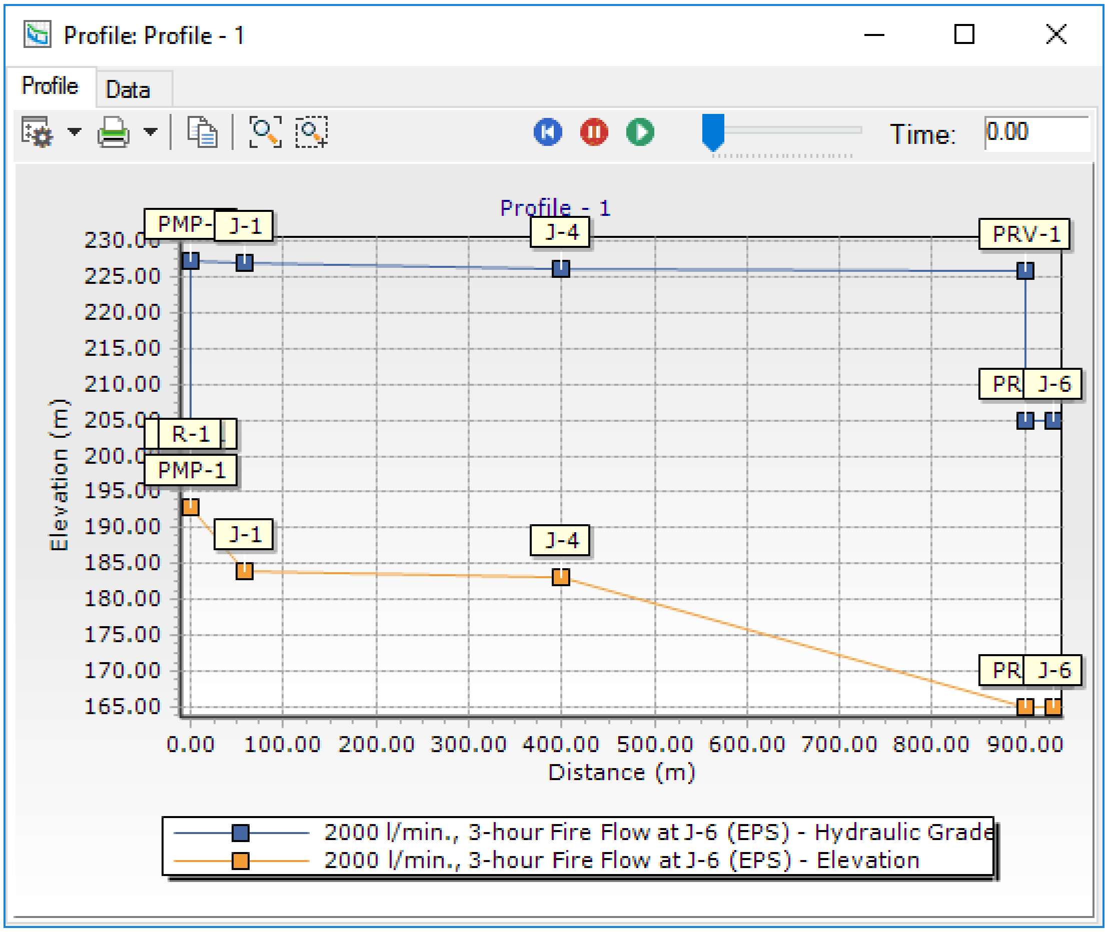


Click Open Profile to view the profile.

7. Profile Series Options opens so you can select which attributes to profile. The default fields are **Elevation** and **Hydraulic Grade**. Leave those defaults and click **OK** to Open the profile.



8. After you create the profile, you can make adjustments to its appearance by clicking on Series Options 



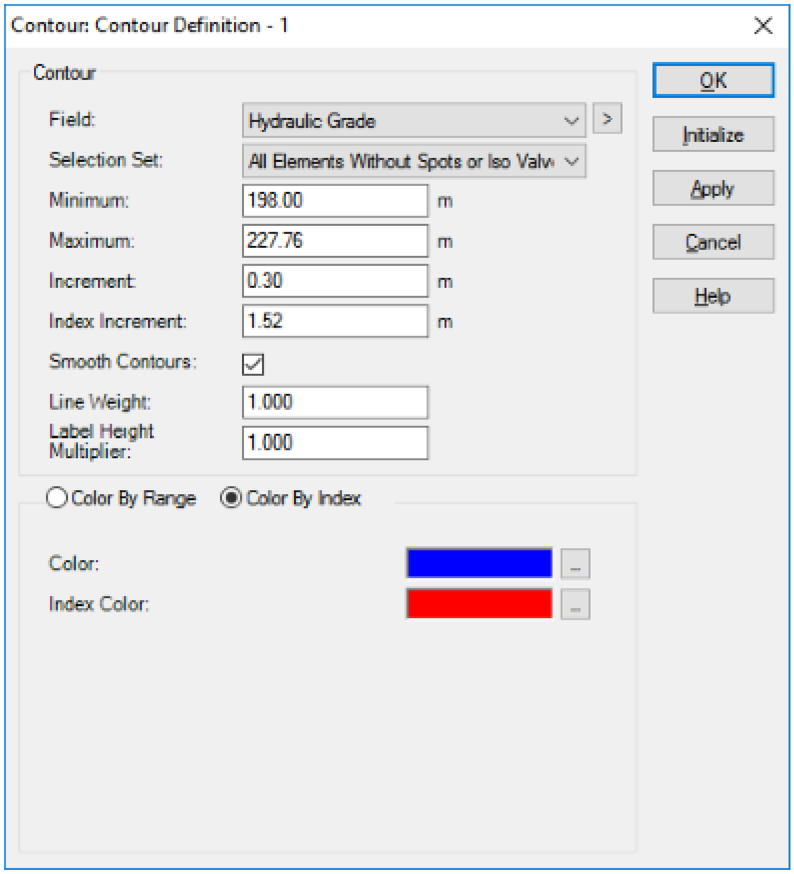
9. The profile graph can be printed or copied to the clipboard.
10. Close the Profile window.
11. Close the Profile manager.

To create a contour

The contouring feature in Bentley WaterGEMS enables you to generate contours for reporting attributes such as elevation, pressure, and hydraulic grade. You can specify the contour interval, as well as color code the contours by index values or ranges of values. In this lesson, you will contour based on hydraulic grade elevations.

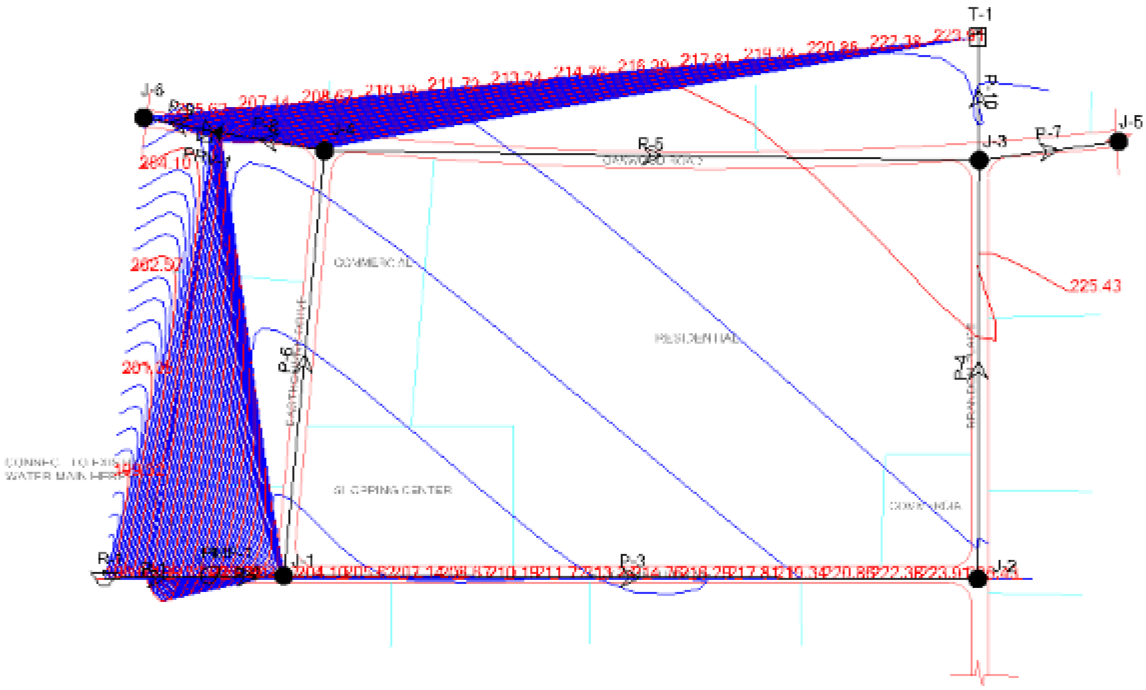
1. Choose **View** > **Contours** .
2. Click **New** in the Contour Manager.

3. Choose **Hydraulic Grade** from the Field menu.
4. Leave the default Selection Set.
5. Click **Initialize** and select **Full Range** to update the Minimum and Maximum HGL elevations.
6. Make sure **Color by Index** is selected
7. Select Smooth Contours to improve the overall appearance of the drawing.

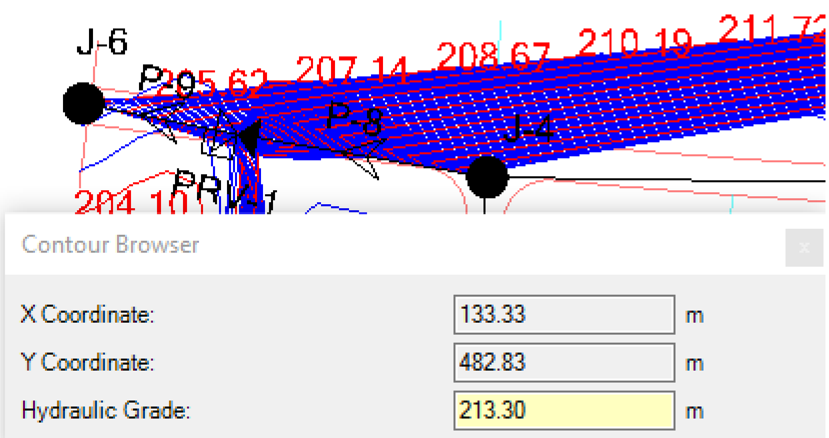


8. Click OK.

9. View the results in the drawing pane.



10. Click on the Contour Browser button  to display the calculated Hydraulic Grade as you move the cursor around in the drawing pane.



11. Close the Contour Browser.

12. Un-check **Contour Definition -1** in the Contours Manager.

13. Close the Contours Manager

Element Symbology

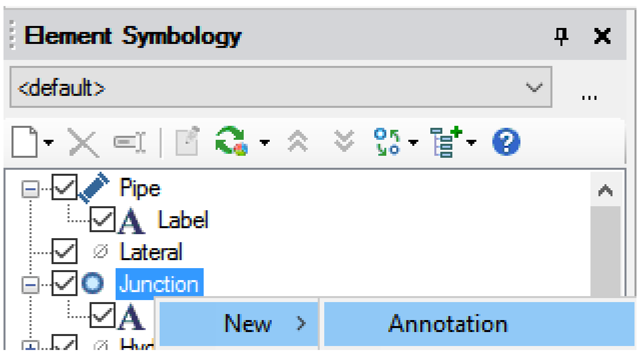
When you want to label network attributes use the Annotation feature. With it, you can control which values are displayed, how they are labeled, and how units are expressed.

Select **View > Symbology** to open the Element Symbology manager if you have closed it previously. If you are using the default workspace configuration, the Element Symbology manager is located directly on the left side of the drawing pane.

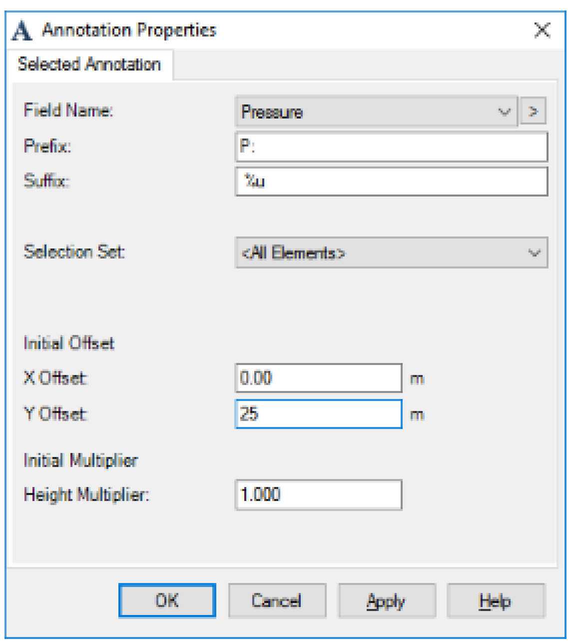
Annotate Pressure at junctions

14. In the Element Symbology Manager, highlight **Junction**. Right click to select **New** > **Annotation**

.



15. Select **Pressure** as the Field Name to annotate. Set the Prefix to **P:** and the **Y Offset to 25 m**. Leave everything else as is.



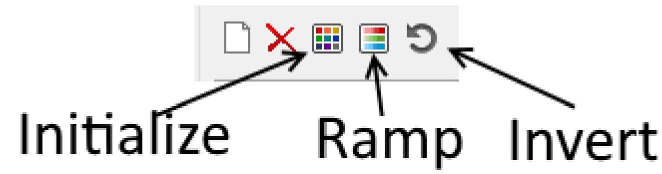
16. Click Apply.
17. The drawing will now display all of the annotations. You can try changing the properties of an element and recalculating. The annotations will update automatically to reflect any changes in the system.
18. If the annotation is crowded, you can click and drag the annotation to move it.
19. Click OK.

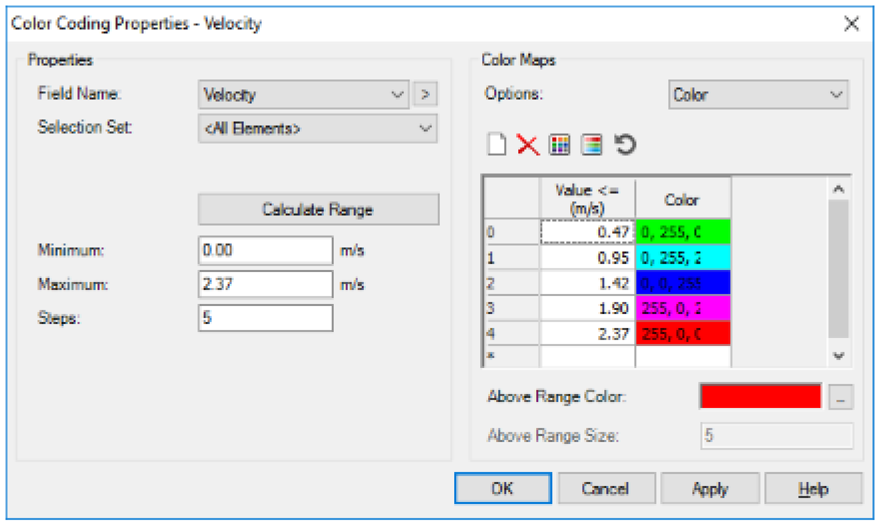
Color Coding

Choose **View > Symbology** and click the element to create the New Color Coding. The Color Coding dialog box allows you to set the color coding for links, nodes, or both. You will color code by velocity (link attribute) and pressure (node attribute) in this example.

1. In the Element Symbology Manager, highlight **Pipe**. Right click to select **New** > **Color Coding**
   a. Select **Velocity** from the Field Name menu.
   b. **Calculate Range > Full Range** so the program can calculate the full range of velocity values for all pipes in this scenario. The minimum and maximum fields should be populated. Leave the default Steps value to be **5**.
   c. Set Options to **Color**.

d. Click **Initialize** so the program selects the color coding ranges automatically based on the Minimum, Maximum and Steps values.





e. Click **OK** to generate the Color Coding.

2. You can add a legend to the drawing. Right-click on the Velocity color coding and select Insert Legend from the menu. You can move the legend in the drawing by clicking the mouse and dragging the legend.

1. In the Element Symbology Manager, highlight **Junction**. Right click to select **New** > **Color Coding**

   a. Select **Pressure** from the Field Name menu.

   b. **Calculate Range > Full Range**. Leave the default Steps value to be **5**.

   c. Set Options to **Color and Size**

   d. Click **Initialize**

   e. Click **OK** to generate the Pressure Color coding.

2. Go to **Home > Times** and then click the **Play** button to observe how the pressure and velocity change, particularly during hours 18-21.

3. Close any open dialog boxes.

4. Save the model.

Automated Fire Flow Analysis

One of the primary goals of a water distribution system is to provide adequate capacity to fight fires. Bentley WaterGEMS automated fire flow analysis can be used to determine if the system can meet the fire flow demands while maintaining minimum pressure constraints. Fire flows can be computed for all nodes in the system, or you can create a selection set consisting of specific nodes where you wish to test available flow.

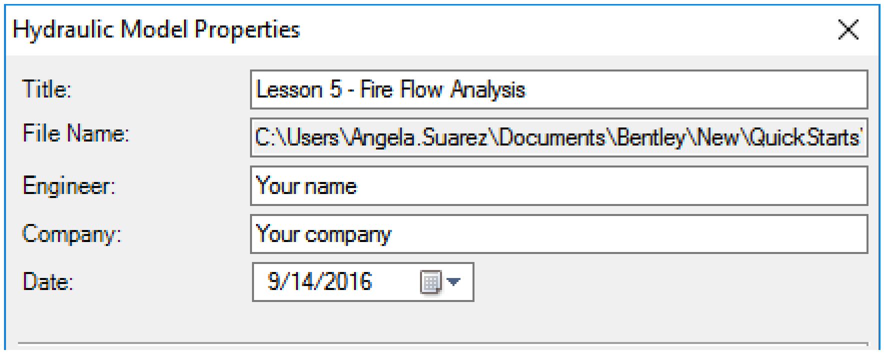
Fire flows are computed at each node by iteratively assigning demands and computing system pressures. The model assigns the fire flow demand to a node and checks the model, checking to see if all pressure and velocity constraints are met at that demand. If a constraint is not met, the flow is reduced until the constraint is just met; if all constraints are exceeded, the fire flow is increased until the constraint is barely met within a tolerance. The analysis automatically rechecks the system pressures if a constraint is violated. Iterations continue until the constraints are met, or until the maximum number of iterations is reached.

The purpose of this example is to walk you through the steps to create, calculate, and analyze a fire-flow scenario. This lesson again uses the distribution system from the previous lessons.

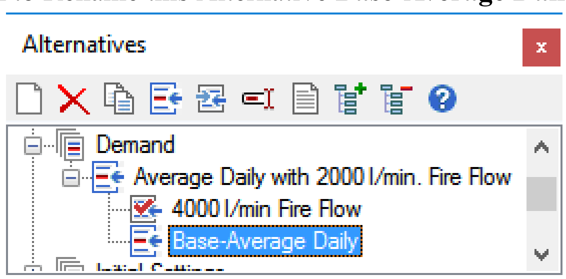
Step 1: Inputting Fire Flow Data

1. Start WaterGEMS and open the **Lesson5.wtg** file, found in the Lessons folder.
   Or
   if you have previously completed the Reporting Results lesson, you can use your **MyLesson4** file.
2. Click **OK** to the message dialog that comes up about CONNECTED Projects.
3. Choose **File > Save As** and save as **MyLesson5**.

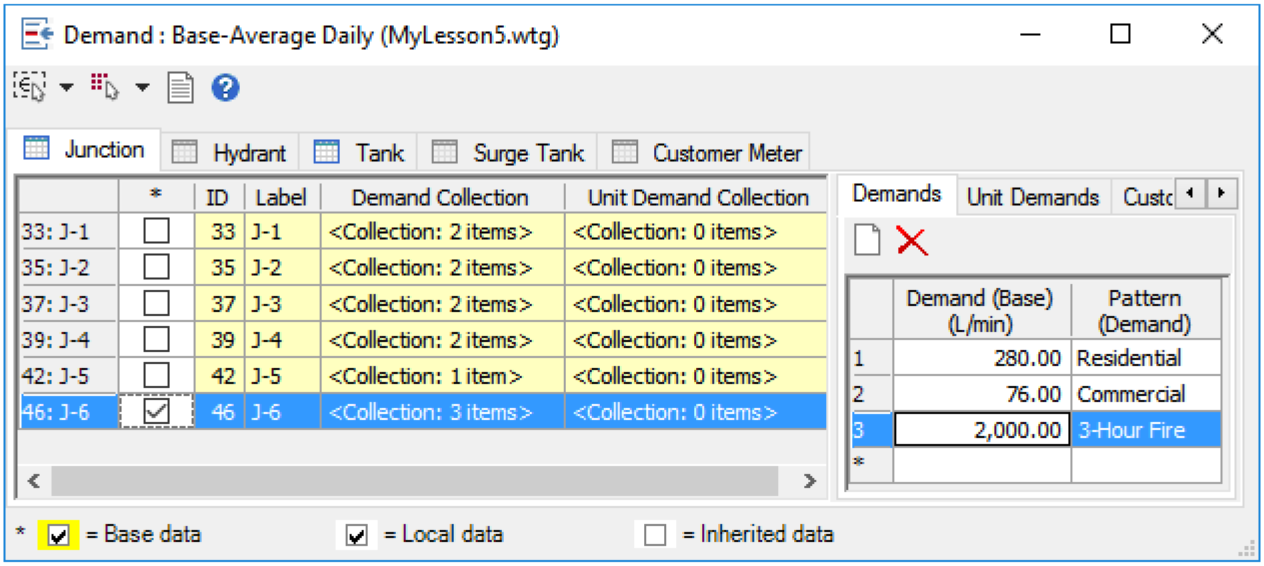
4. Choose File > Hydraulic Model Properties and name the title of the Hydraulic Model **Lesson 5—Fire Flow Analysis**.



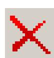
5. Click OK.
6. Previously, you ran an analysis with a fire flow at node J-6 by manually adding a large demand to the individual node. Before running the automated fire flow analysis, you will create a new Demand Alternative, removing that demand. In the U.S., fire flows are generally added to max day demands.
   a. Choose Analysis > Alternatives.
   b. Expand the Demand Alternative and select **Average Daily with 2000 l/min Fire Flow**. Right-click and select **New > Child Alternative**.
   c. Right click to Rename this Alternative **Base-Average Daily**.



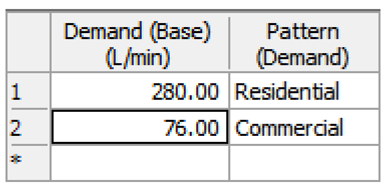
d. Double-click to open the **Base-Average Daily** alternative and put a check in the box for J-6.



e. In the Demands tab, select the row with 2,000 Flow and 3-Hour Fire and click



to delete it.

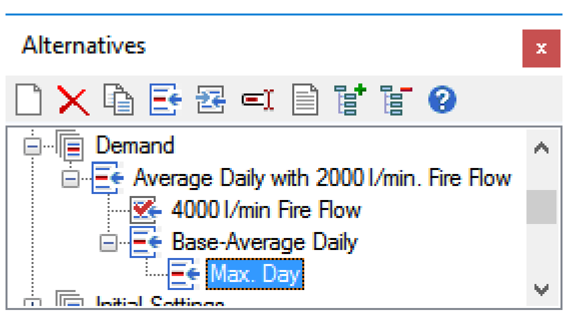


f. Click Close to exit the Demand Alternative.

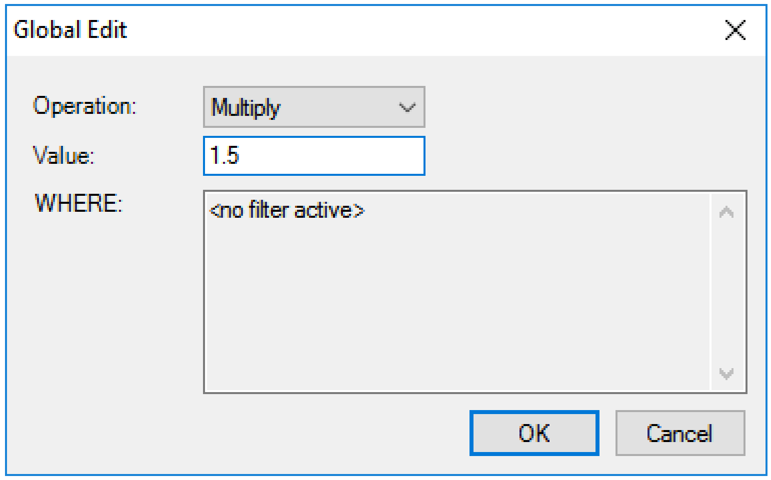
7. You are going to analyze the fire flows by adding to the Maximum Day Demands, which are 1.5 times the Average Day Demands.

a. Right-click on **Base-Average Daily** then select **New > Child Alternative**.

b. Right click to Rename this Alternative **Max. Day**.



c. Double click to open the **Max. Day** alternative.Highlight the **J-1** row. Right-click the **Demand (Base)** column and select Global Edit. Set the Operation to multiply, and enter a value of 1.5.



d. Click **OK**.

e. Repeat steps c and d for J-2 through J-6.

f. Click **Close** to exit the Demand Alternative.

8. Select the **Fire Flow** alternative and expand to select the **Base-Fire Flow** Alternative.

9. Double click to set up the Base-Fire Flow Alternative.

   a. In the F**ire Flow (Needed)** field, enter **3000 l/min**.

   b. In the **Fire Flow (Upper Limit)** field enter **6000 l/min**.

   c. **Apply Fire Flows By** should be set to **Adding to Baseline Demand.**

   This selection means that when WaterGEMS performs the analysis, the fire flow will be added to any demands already assigned to the junction. Alternatively, you could have selected to replace these demands, so that the fire flow would represent the total demand at the node.

   d. Pressure Constraints Pressure (Residual Lower Limit) and Pressure (Zone Lower Limit) should be set to **150 kPa**.

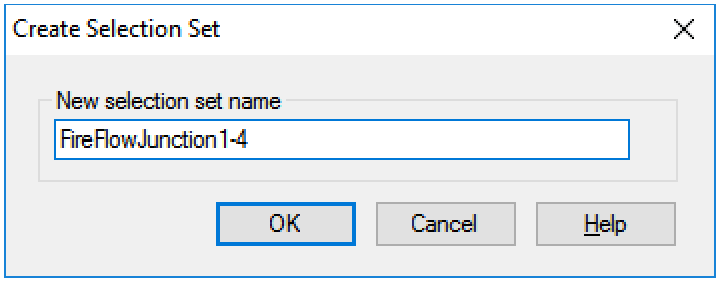
   e. Leave the check box for **Use Minimum System Pressure Constraint** cleared, so that the minimum pressure will only be checked for the zone a particular node is in.

   If you had multiple zones within your project and wanted to insure that a minimum system-wide pressure constraint was met, you could check the **Use Minimum System Pressure Constraint** box and enter it in the box provided. This box is grayed out until the check box is activated.

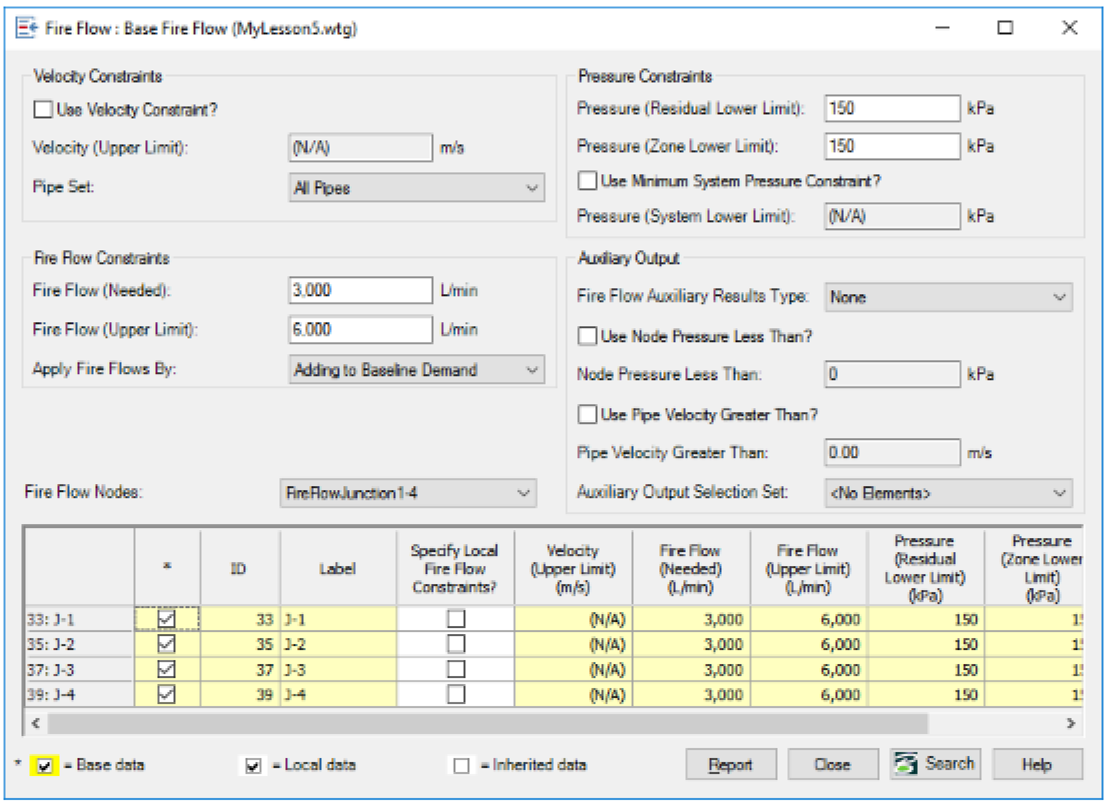
f. Create a selection set to choose from the Fire Flow Nodes drop-down menu. For this example, a fire flow analysis is only needed for the junctions at the four street corners in our drawing.

g. The Fire Flow Alternative manager can remain open. In the drawing and while pressing the <**Shift**> key, click nodes **J-1, J-2, J-3**, and **J-4.**

h. Right-click and select **Create Selection Set**, then name the set **FireFlow-Junction1-4** and click **OK**.



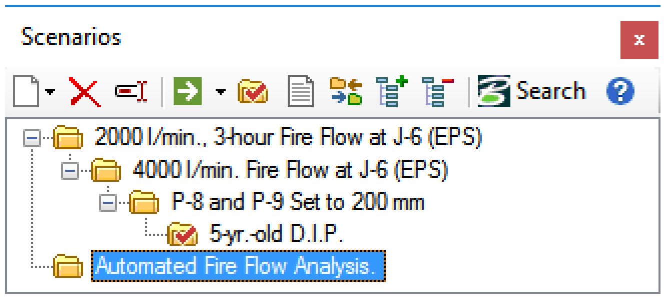
i. In the Fire Flow Alternative manager, select FireFlowJunction1-4 from the Fire Flow Nodes drop-down menu.



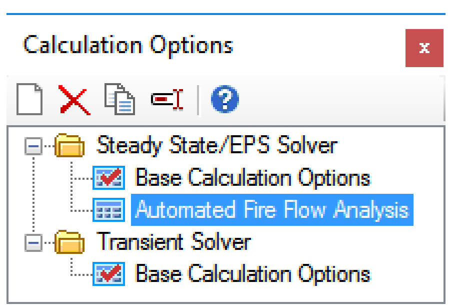
10. Click Close to exit the Fire Flow Alternative manager.

Step 2: Calculating a Fire Flow Analysis

1. Click **Analysis** > **Scenarios**.
2. In the Scenarios dialog, click **New** > **Base Scenario.**
3. Name the new Scenario **Automated Fire Flow Analysis.**

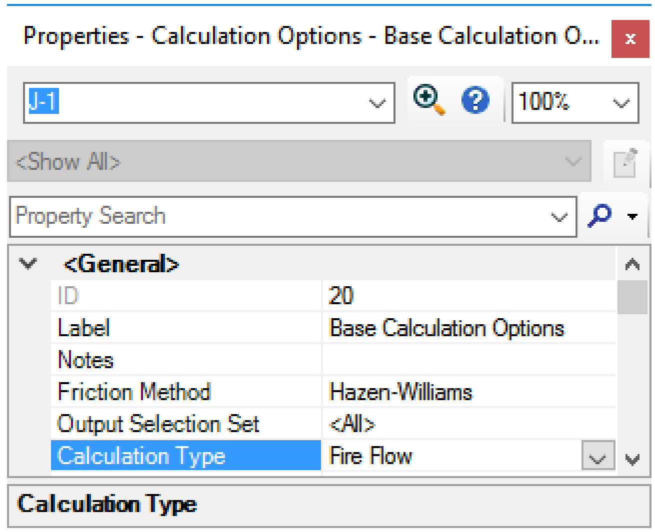


4. Double-click the **Automated Fire Flow Analysis** scenario to open the Property Editor.
   a. Change the Physical Alternative to **P-8 and P-9 Set to 200 mm.**
   b. Change the Demand to **Max. Day** and leave all other Alternatives set to their defaults.
   c. Close the properties dialog
5. In the Scenarios manager, make **Automated Fire Flow Analysis** the current scenario by highlighting it and clicking the Make Current button 
6. Click **Analysis** > **Options**.
7. In the Calculation Options dialog, Select **Steady State/EPS Solver** and click the **New** button. Rename the new option **Automated Fire Flow Analysis**.



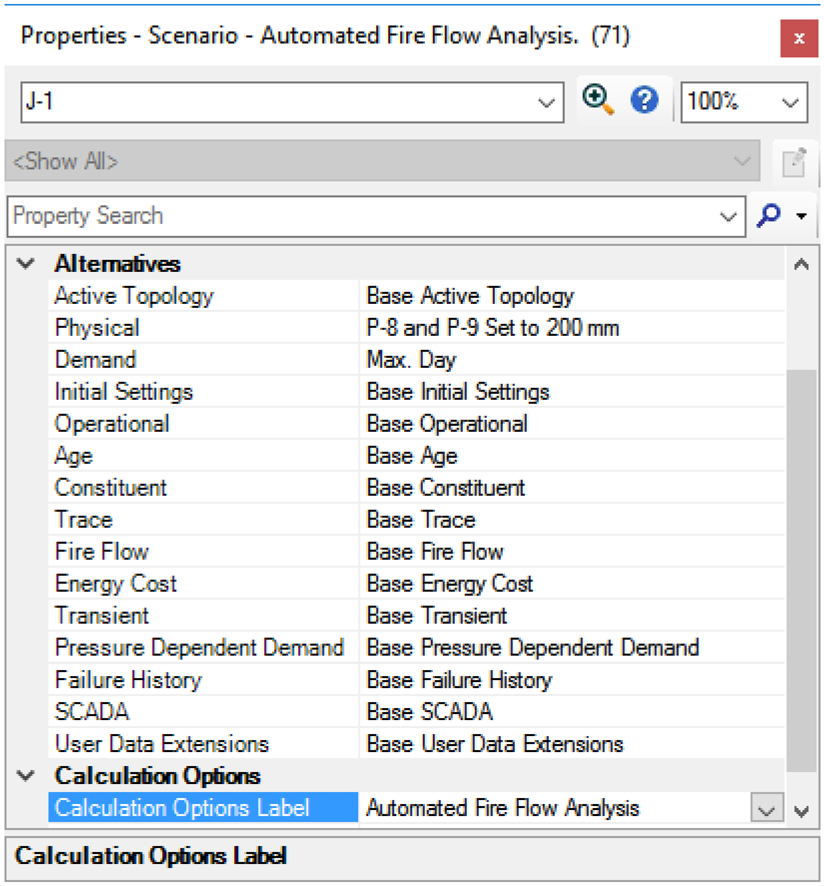
8. Double click **Automated Fire Flow Analysis** to open the Property Editor.

9. Change the **Calculation Type** to **Fire Flow**. Close the Calculation Options dialog.



10. Choose **Analysis > Scenarios**.
11. Double click **Automated Fire Flow Analysis** Scenario to open the Property Editor.

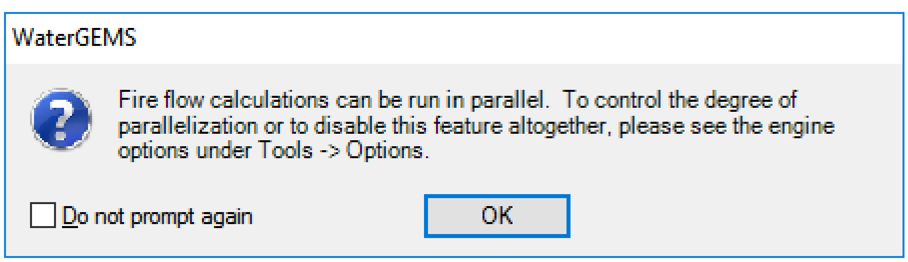
d. Change the Calculation Options to **Automated Fire Flow Analysis**.



e. Close the Property Editor.

f. Run the **Automated Fire Flow** Scenario by selecting **Compute** on the Home or Analysis tabs, or by clicking the Compute button in the Scenario manager.

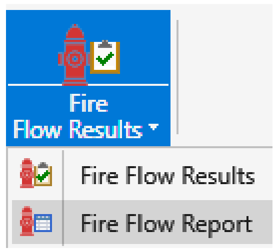
g. If you receive a prompt about running Fire Flow Calculations in parallel, click OK. This option enables the user to choose (or not) hyperthreading in an attempt to make the fire flow runs faster..



When the calculation is complete, Close the Calculation Summary.

Step 3: Viewing Fire Flow Results

1. Make sure that **Automated Fire Flow Analysis** is selected in the Scenario list box.
2. Click **Analysis > Fire Flow Results > Fire Flow Report**

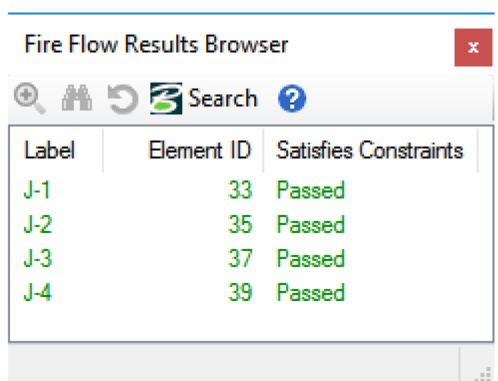
   

   In the Satisfies Fire Flow Constraints column, all of the boxes are checked except for the nodes that you did not analyze, because the specified needed flow of 3000 l/min. was available and minimum pressures were exceeded.

   For nodes J-1 and J-3, pressures were computed for the Fire Flow Upper Limit of 6000 l/min. because none of the node pressures ever dropped below specified minimum pressures and no velocity constraint was specified.

   Nodes J-2 and J-4 reached their minimum residual pressures at flows slightly below the maximum of 6000 l/min.

   The report contains the Minimum System Pressure (excluding the current node being flowed) and its location.
3. Click **Analysis > Fire Flow Results > Fire Flow Results**

The Fire Flow Results Browser reports whether each of the fire flow nodes satisfies constraints or not. It also allows you to jump to fire flow nodes and display the results of fire flow analysis at the highlighted node if you have specified Auxiliary Output options in the Fire Flow Alternative settings.

When you are finished reviewing the report, **Close** the Fire Flow Report and save your file.

Note: Another good way to review an automated fire flow analysis is to use color coding. If you have a situation where no nodes meet the pressure constraints for the needed fire flow, you can color code these nodes in the plan view for easy identification.

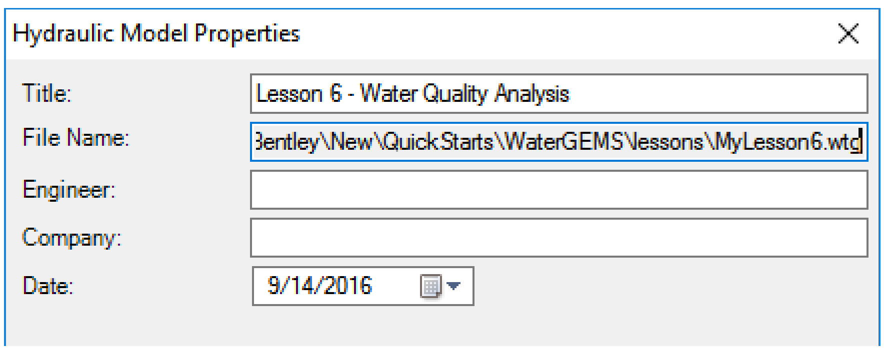
Water Quality Analysis

In conjunction with Extended Period simulations, Bentley WaterGEMS is capable of performing a water quality analysis to compute water age, constituent concentration, or percentage of water from a given node (trace analysis). Using these features, you can look at factors such as residence time in tanks, chlorine residuals throughout the system, and which tank or reservoir is the primary water source for different areas in your system.

This lesson uses the file called **Lesson6.wtg** located in the Lessons folder.

To open the existing lesson

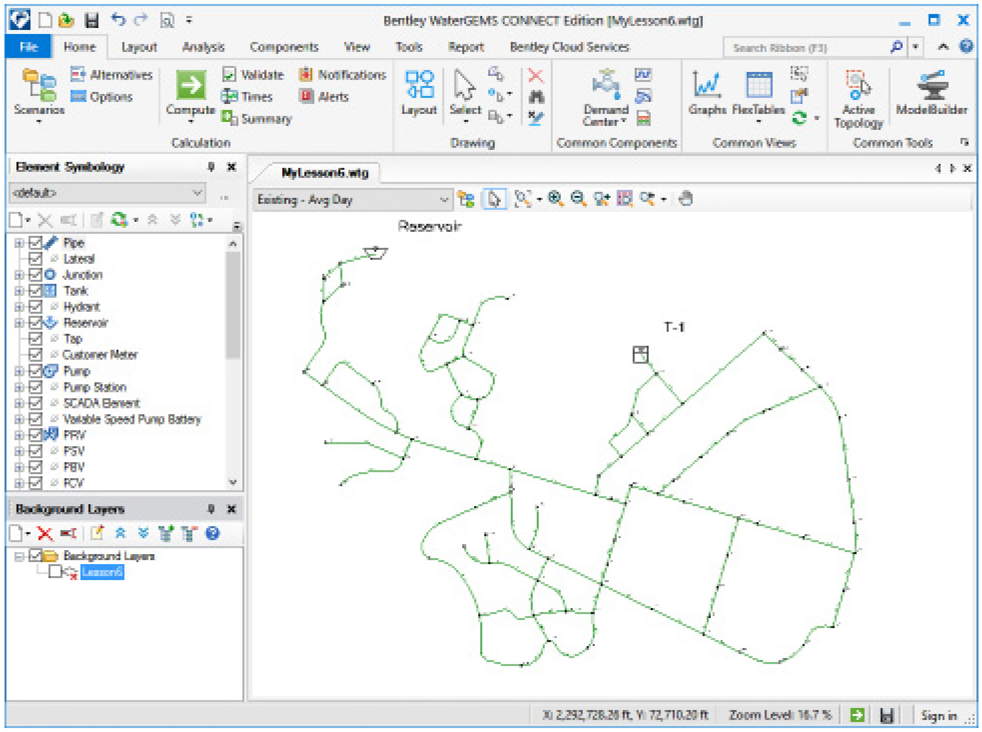
1. Open **Lesson6.wtg**.
2. Click **OK** to the message dialog that comes up about CONNECTED Projects.
3. Click **File > Save As**. Enter the filename **MyLesson6** and click **Save**.
4. Choose **File** > **Hydraulic Model Properties**, and change the Hydraulic Model Title to **Lesson 6—Water Quality Analysis**.



5. Click OK.

The water distribution system has already been set up for you. It has one reservoir and one tank. The system serves primarily residential areas, with some commercial water use as well. There are two pumps connected to the reservoir. However, under normal conditions, only one pump will be in use. A background drawing has been included for reference.

If you would like to turn off the .DXF background, clear the background check box in the Background Layers pane.(View > Backgrounds)

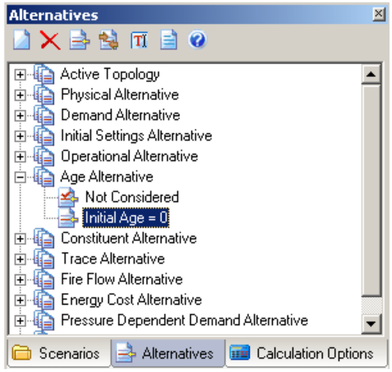


Step 1: Computing Water Age

You will begin by running an age analysis for water in the system, assuming an initial age of 0 for all nodes. The water from the reservoir will be an infinite supply of new water, so the age of water elsewhere in the system will be a reflection of time from the start of the run and how long ago the water left the reservoir. The analysis will be run for a 2-week period (336 hours). Based on this model size, 2 weeks should be more than adequate to capture the water age in the network. If the network was much larger, you may need to use a longer duration to ensure maximum water age is captured.

1. Choose **Analysis > Alternatives** .
2. Select **Age** Alternative and click **New** to create a new age alternative.

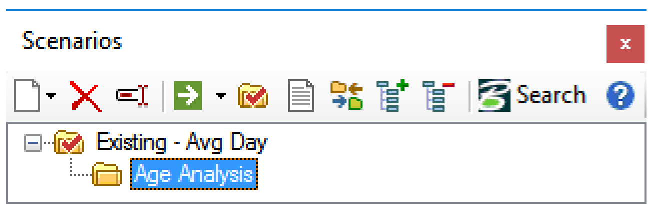
3. Name the new alternative **Initial Age = 0**. Since you are assuming an initial age of 0 everywhere in the system, you do not need to enter any initial ages.



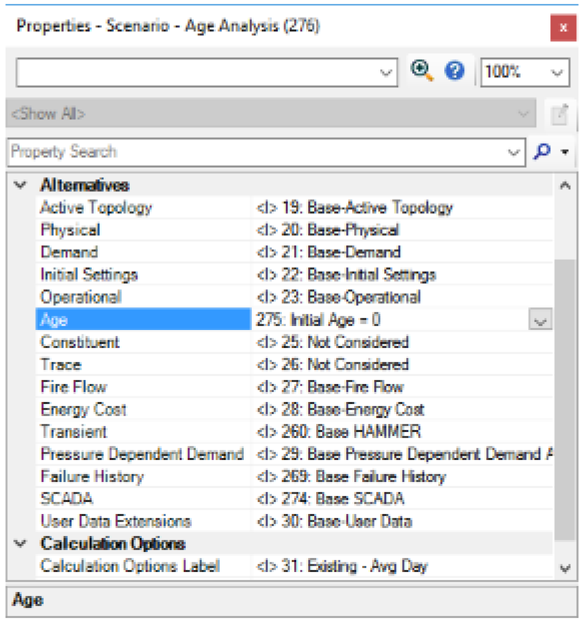
4. Next, set up a new Scenario to run an Extended Period Simulation incorporating the new Alternative.

   a. Click **Analysis** > **Scenarios**; note that the **Existing - Avg Day** scenario already exists

   b. Highlight the **Existing - Avg Da**y scenario and click **New** > **Child Scenario** and rename it **Age Analysis**.



c. Double-click on the new scenario to open the Property Editor. In the Age Alternative field select **Initial Age = 0**, from the drop-down menu.



d. Close the Property Editor.

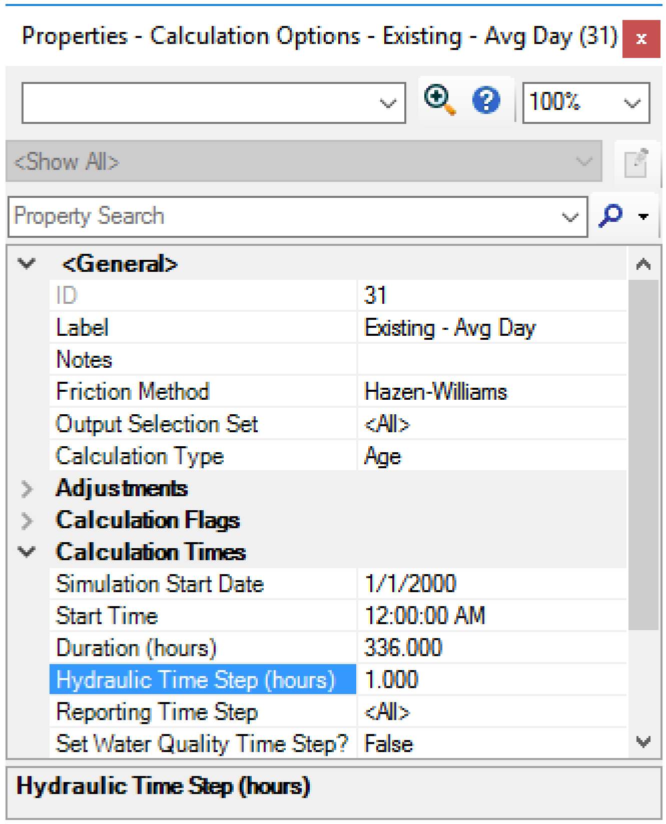
e. Click **Analysis** > **Options** and double click **Existing - Avg Day** to view the settings for this Scenario. **Extended Period Analysis (EPS)** should already be selected as the **Time Analysis Type**.

f. Set the **Calculation Type** to **Age**

g. Enter a Start Time of **12:00:00 AM**.

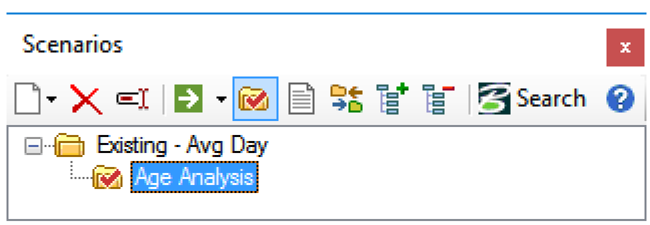
h. Set a Duration of **336** hours.

i. Set a Hydraulic Time Step of **1** hour.



j. Close the Property Editor and Calculations Options.

5. Click **Analysis** > **Scenarios** and make **Age Analysis** current by right-clicking it and selecting **Make Current**.



6. Click Compute  and then close the Calculation Summary and Scenarios Manager.

7. Select **View** > **Symbology** to open the Element Symbology manager. If you are using the default workspace configuration, the Element Symbology manager is located directly on the left side of the drawing pane.

8. Right-click on **Pipe** and select **New** > **Color Coding**

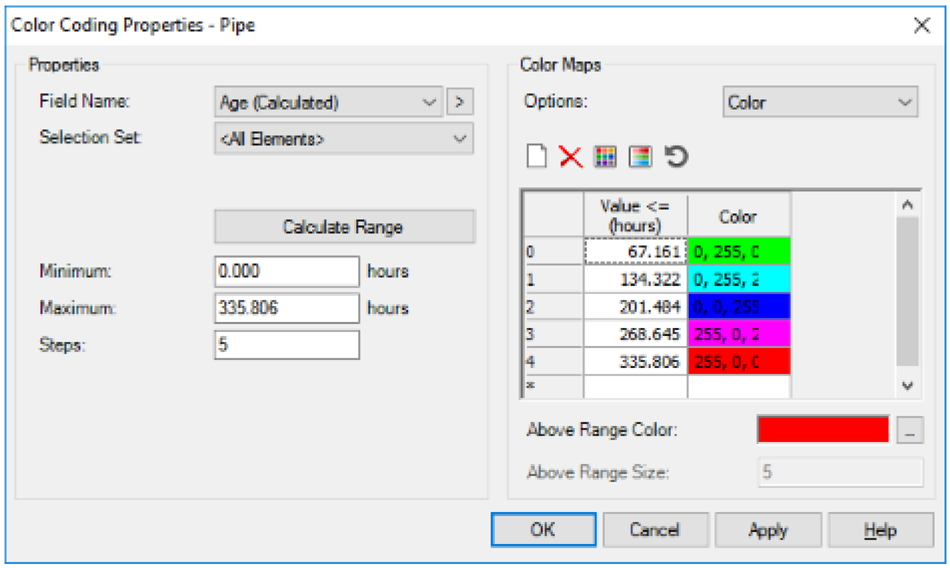
9. Select **Age (Calculated)** as the **Field Name**.

10. Click Calculate Range and Select **Full Range.**.

11. Click **Initialize**  to set up a default color scheme. Accept this default scheme.

    If you get a message about Bentley WaterGEMS being unable to determine the limits for mapping, make sure that Age Analysis is selected in the Scenario drop-down list, in the toolbar.

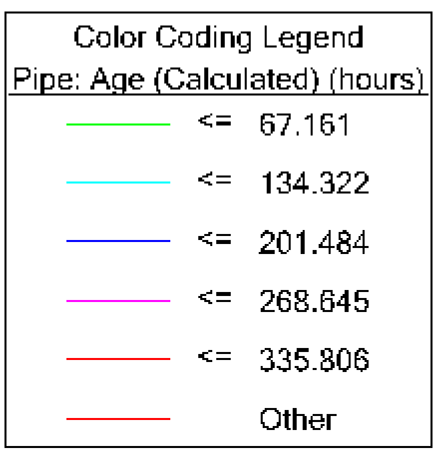
12. Click **Apply**.



13. Click **OK**.

14. In the Element Symbology manager, right-click on **Age (Calculated)** and click **Insert Legend.**

15. Click in the drawing to place the legend

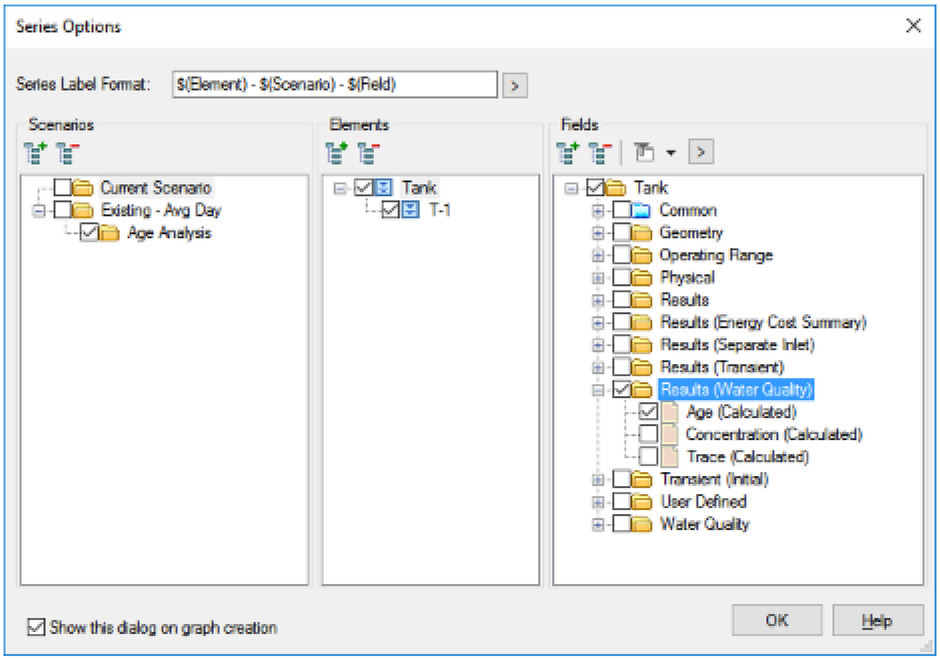


16. Go to **Analysis > Times** and click the play button to see how water ages in the system.

17. A good way to check if your network has had sufficient time to reach an equilibrium point is to look at **Age vs. Time graphs** for your elements.

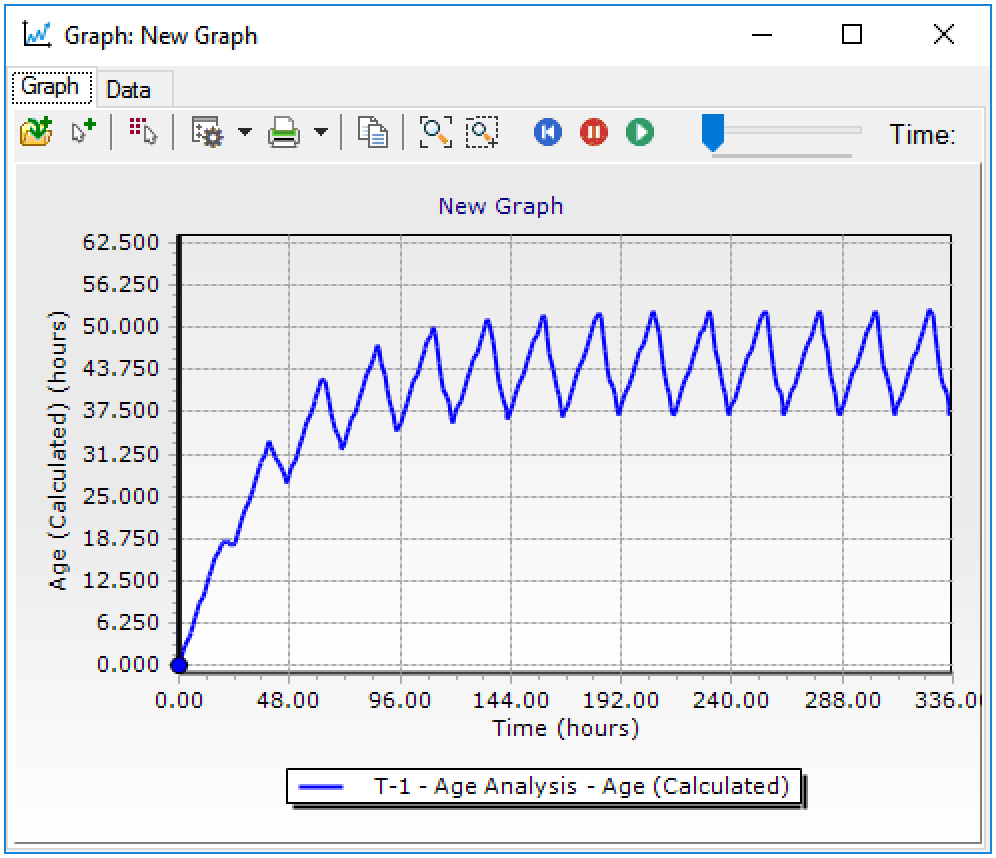
    a. Right-click on **Tank T-1** and select **Graph**

    b. In the Series Option dialog make sure that **Age Analysis** is checked in the Scenarios column, **Tank** and **T-1** are checked in the Elements column, and **Results (Water Quality)** and **Age (Calculated)** are checked in the Fields column. Uncheck all other boxes.



    c. Click **OK**.
    From the graph, you can see that once a repeating pattern is reached, the age of the water fluctuates between approximately 37 and 52 hours in 24-hour

periods. Looking at these equilibrium ranges for various nodes can help guide you in setting up initial water age values in subsequent runs.



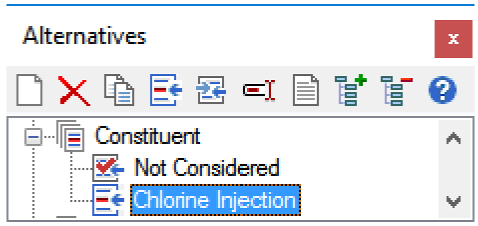
d. Close all open windows.

Step 2: Analyzing Constituent Concentrations

In this portion of the lesson, you will model the change of chlorine residuals in the system over time. Bentley WaterGEMS stores information on constituent characteristics in a file called a constituent library. It is recommended to determine actual constituent characteristics from lab and or field testing to use in a water quality model. You will add information for chlorine to this library, set up initial concentrations in the system, and run the simulation.

1. Choose **Analysis > Alternatives**.
2. Click the **Constituent** Alternative and click **New**.

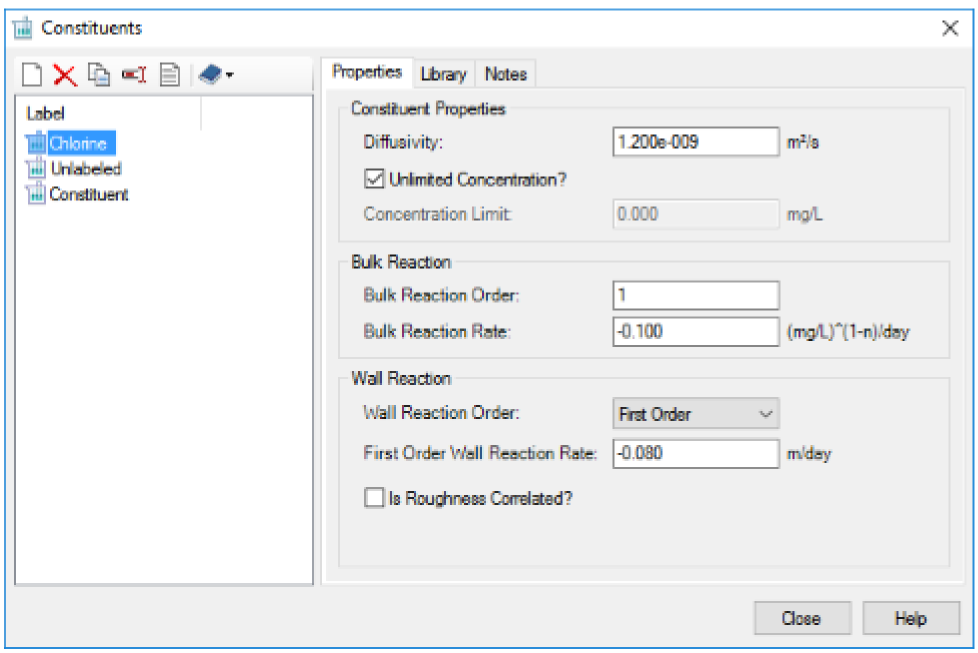
3. Name the new alternative **Chlorine Injection** and double-click to open.



4. Click the **Ellipsis (…)** next to the Constituent drop-down menu to open the Constituents manager.
5. Make sure that the **Chlorine** Label is highlighted, and enter the data below in that dialog box.

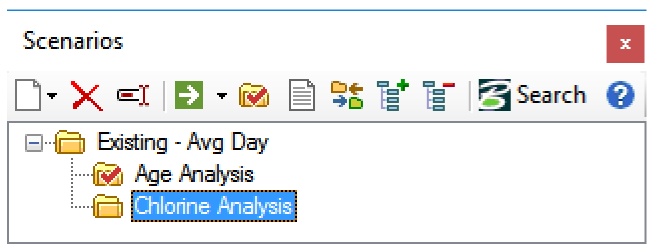
Label	**Chlorine**
Diffusivity	1.2e-9 m2/s
First Order Wall Reaction Rate	-0.08 m/day
Bulk Reaction Rate	-0.10 (mg/L)^(1-n)/day

6. Leave the **Unlimited Concentration** check box selected, and click **OK**.

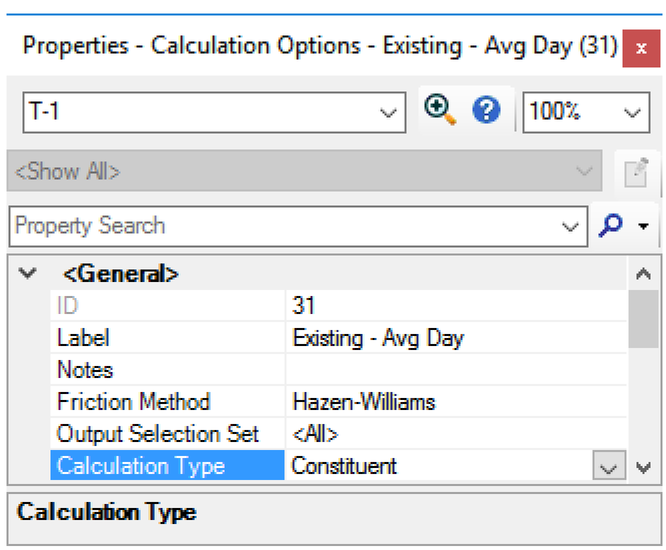


7. Click **Close** to exit the Constituent Library. You should now be back in the Constituent Alternative Editor.
8. Select **Chlorine** from the Constituent list box.

9. On the same **Chlorine Injection** editor window, go to the **PRV** and **Pump** tabs and set the **Concentration (Initial)** for each to **1 mg/l.**
10. Click the **Junction** tab, and initialize the chlorine concentrations by entering a value of **1 mg/l** at each junction node. (Right-click the column heading and use Global Edit to set the **Concentration (Initial)** fields.)
11. In the Reservoir tab, enter a **Concentration (Initial)** value of **2.0 mg/l** for the reservoir.
12. Set the tank's **Concentration (Initial)** to **0.5 mg/l**.
13. Close the **Editor** and the **Alternatives Manager**.
14. Now, open the Scenario Manager (**Analysis > Scenarios)** and set up a new Scenario in order to run the Constituent Analysis.
    a. Create a new Child off of the **Existing - Avg Day** Scenario by highlighting it and clicking **New > Child Scenario**.
    b. Enter **Chlorine Analysis** as the new scenario name.



    c. Double click the newly created **Chlorine Analysis** scenario to open its Property Editor. In the **Constituent Alternative** dropdown, select **Chlorine Injection.**
15. Click **Analysis > Options.** Double-click **Existing - Avg Day** to open the Property Editor and set the Calculation Type field to **Constituent**.

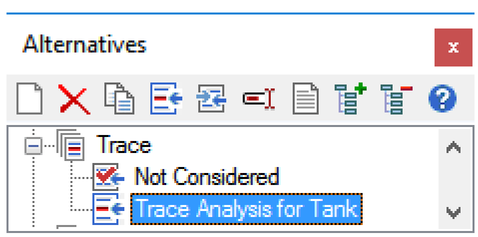


16. **Close** the Property Editor and Calculations Options manager.
17. Make **Chlorine Analysis** the current scenario
18. Run the **Chlorine Analysis** Scenario by selecting **Compute** on the Home or Analysis tabs, or by clicking the **Compute** button in the Scenario manager.
19. Close the Calculation Summary and Scenarios Manager.
20. Set up color coding for the pipes. This time, color code by C**oncentration (Calculated)**. Scroll through the time steps **(Analysis** > **Times)** to view how the concentrations change throughout the network. When you look at your results using color coding, tables, and graphs, try to discover what better initial values for chlorine concentration might be.

Step 3: Performing a Trace Analysis

A trace analysis determines the percentage of water at all nodes and links in the system from a specific source node (the trace node). In systems with more than one source, it is common to perform multiple trace analyses using the various source nodes as the trace nodes in successive analyses. For this analysis, you will perform a trace analysis to determine the percentages of water coming from the tank.

1. Select **Analysis** > **Alternatives**.
2. Click the **Trace** alternative to highlight it.
3. Click **New**.
4. Name the new alternative **Trace Analysis for Tank**, and double-click it to open the alternative editor.



5. In the Trace Element list box, select the tank, **T-1**. (click the ellipsis button to select it from the drawing).
6. Close the editor.
7. Close the Alternatives Manager.

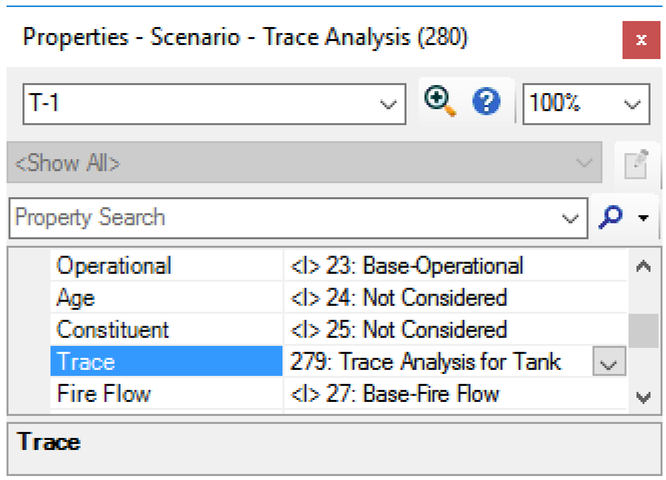
8. Next, set up a new scenario to run an Extended Period Simulation incorporating the new alternative.

    a. Select **Analysis > Scenarios**.

    b. Create a new child for the **Existing - Avg Day** Scenario by highlighting it and clicking **New > Child Scenario**.

    c. Enter **Trace Analysis** as the new scenario name.

    d. Double-click the new scenario to open the Property Editor. Change the **Trace** Alternative to **Trace Analysis for Tank**.



    e. Close the Property Editor.

    f. Click **Analysis > Options**. Double-click **Existing - Avg Day** and change the Calculation Type to **Trace**.

    g. Close the Property Editor and Calculations manager.

9. Make **Trace Analysis** the current scenario (**Analysis > Scenarios).**

10. Run the **Trace Analysis** Scenario by selecting **Compute** on the Home or Analysis tabs, or by clicking the **Compute** button in the Scenario manager.

11. Close the Calculation Summary and Scenarios manager.

12. Create pipe color coding to show the trace by going to **Element Symbology > Pipe > New > Color Coding**. Color code by **Trace (Calculated)**. Make sure **Trace** is the only attribute being color coded (Age and Constituent color coding should be turned off). As you scroll through the time periods, notice how the colors spread outward from the tank during periods when the tank is draining, and recede when the tank begins to fill. For more information on reporting features, Reporting Results.

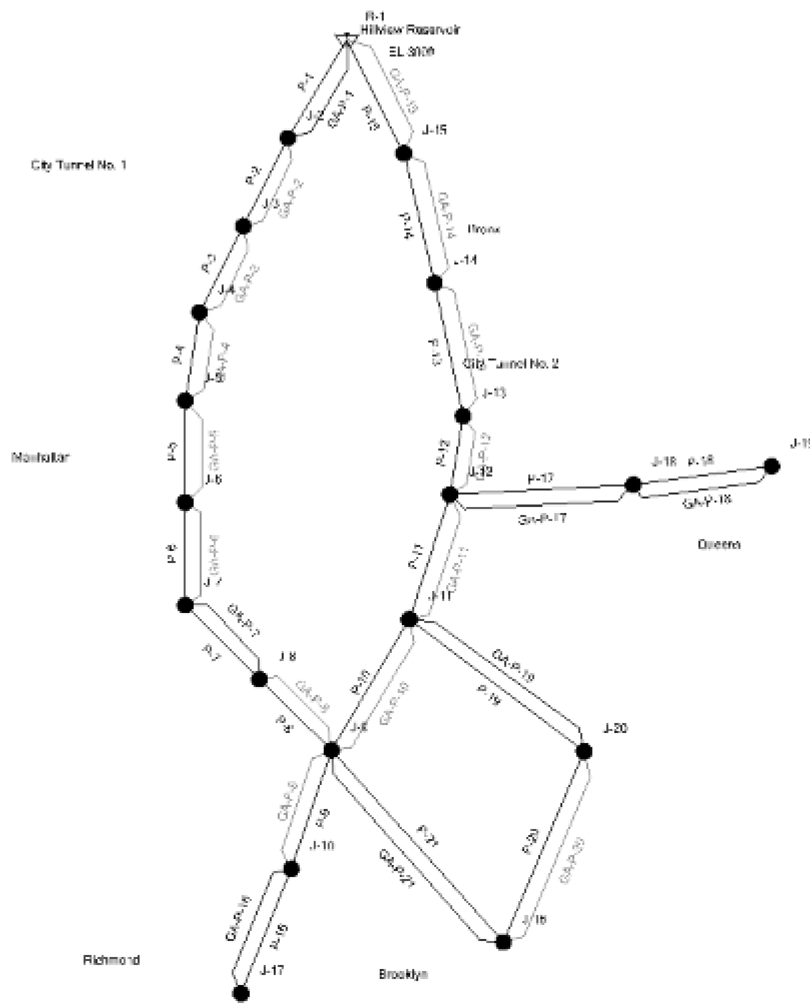
13. Close the open dialog boxes and save this model.

Darwin Designer to Optimize the Setup of a Pipe Network

In this lesson, you use Darwin Designer to optimize the setup of a pipe network.

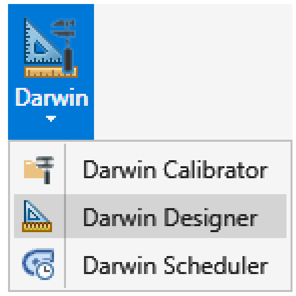
Step 1: Creating the Darwin Designer Optimization

1. In Bentley WaterGEMS choose **File > Open**.
2. Browse to the Lessons\Designer folder and open **DesignerSample1.wtg.**

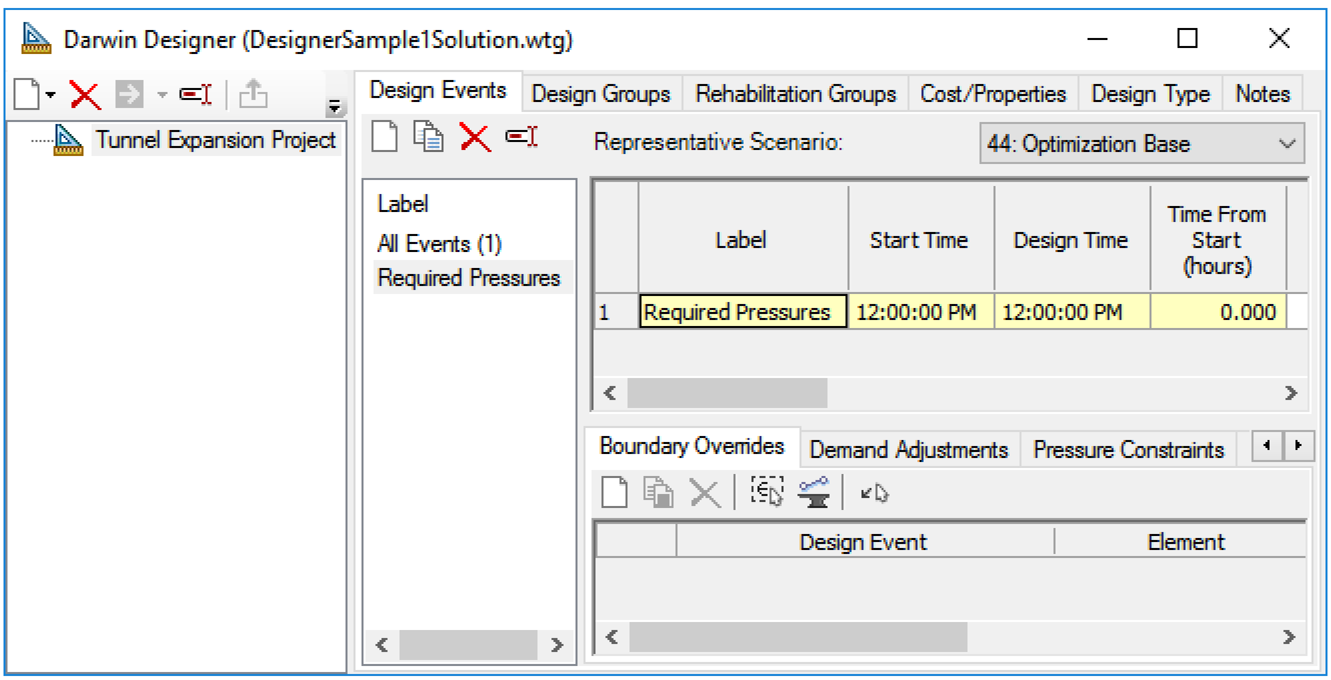


3. Click **OK** to the message dialog that comes up about CONNECTED Projects.
4. Go to **File > Save As** and name the file **DesignerSample1Solution**

5. Click **Analysis > Darwin > Darwin Designer**.

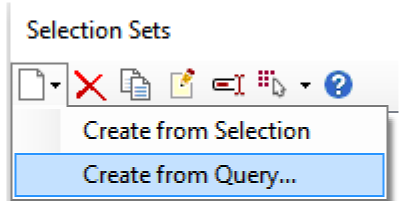


6. In the Darwin Designer window, click **New > New Designer Study**.
7. Highlight the new design and click the Rename button. Enter **Tunnel Expansion Project**.
8. In the **Design Events tab** Select **Optimization Base** as the representative scenario in the drop-down list.
9. In the **Design Event** tab, click **New**.
10. Name the design event **Required Pressures**, and click **OK**.

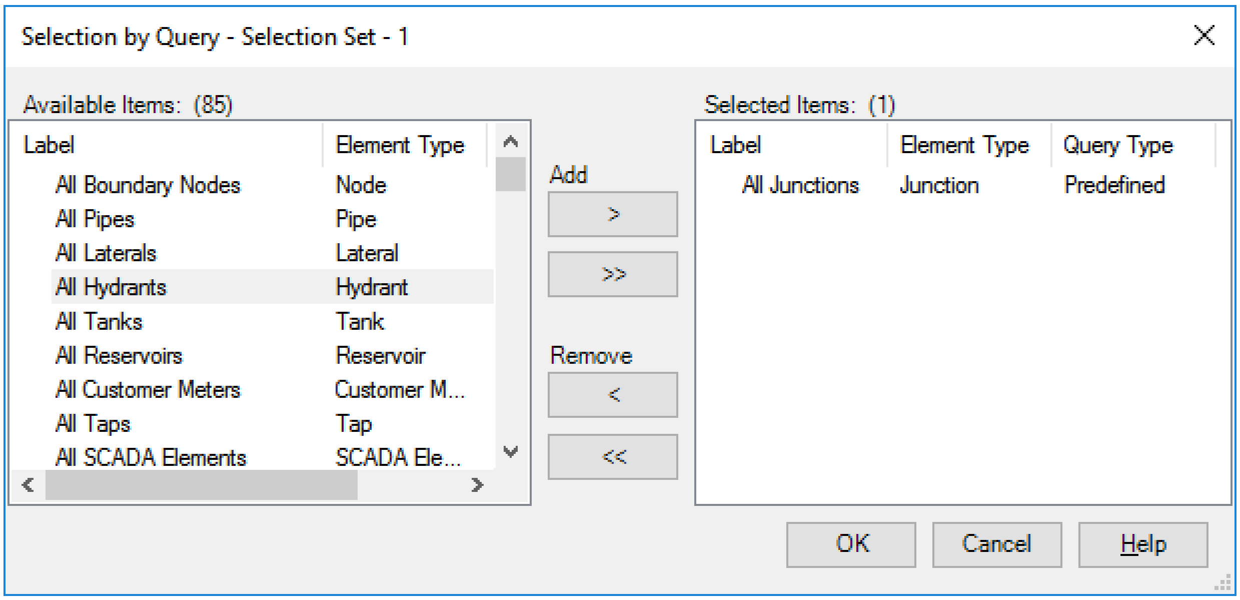


11. Set pressure constraints for all junctions.
    a. First, minimize the Darwin Designer dialog.
    b. Create a new selection set containing all of the junctions in the model. Click **View > Selection Sets**.

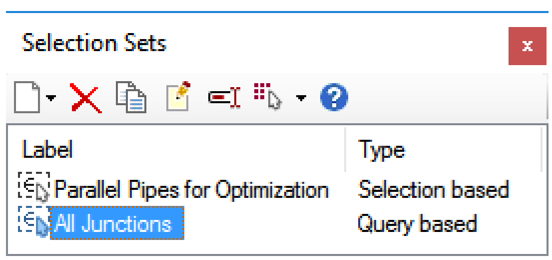
c. Click New > Create From Query.



d. Double-click the **All Junctions** query, then click **OK**.



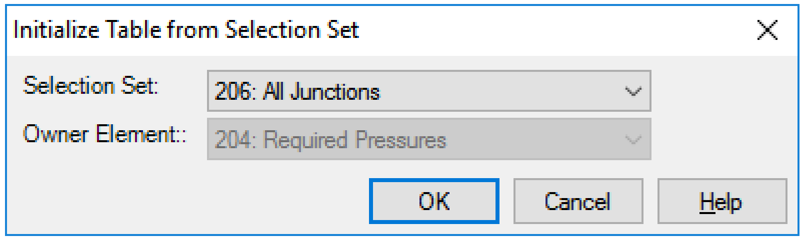
e. Rename the new selection set **All Junctions**.



f. Back in **Darwin Designer**, click the **Pressure Constraint**s tab.

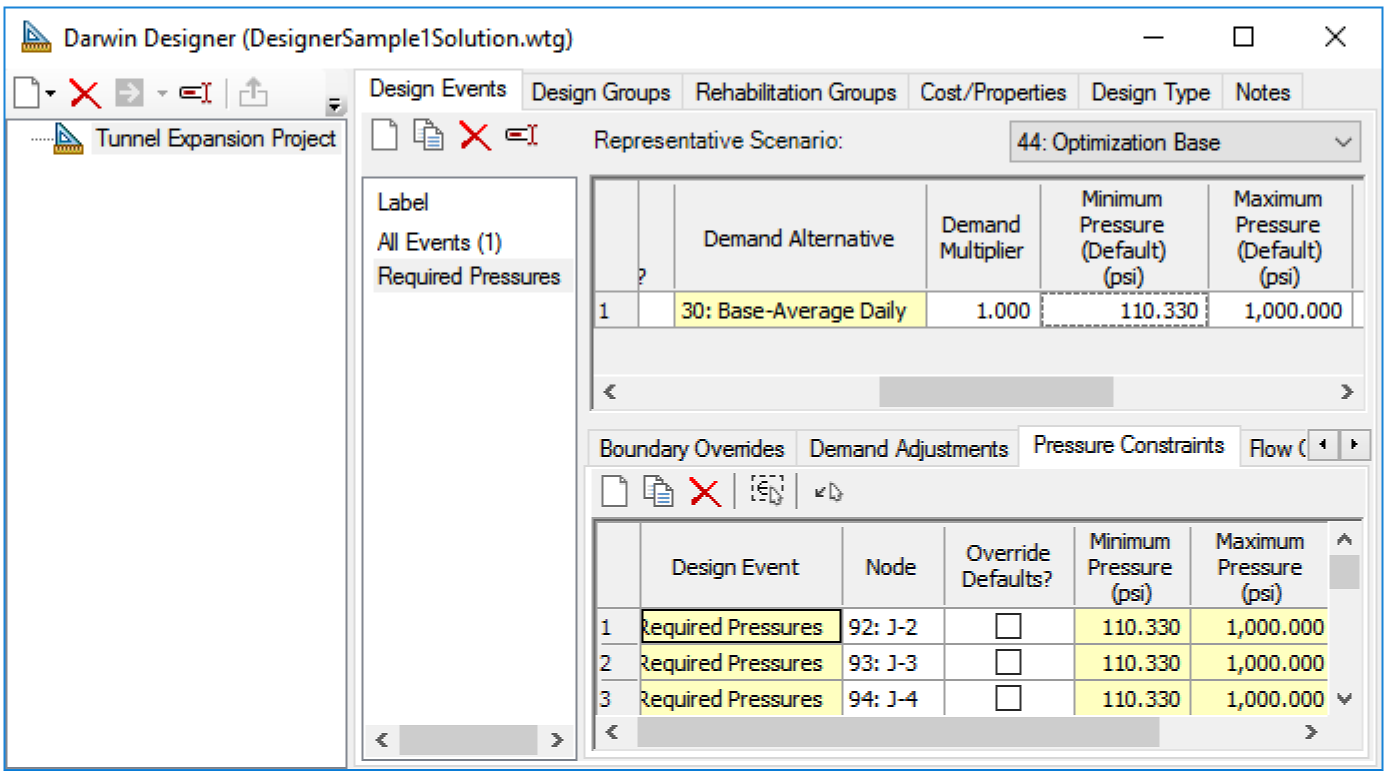
g. Click the **Initialize Table from Selection Set**  button.

h. Select **All Junctions** from the Selection Set drop-down list, then click **OK**.

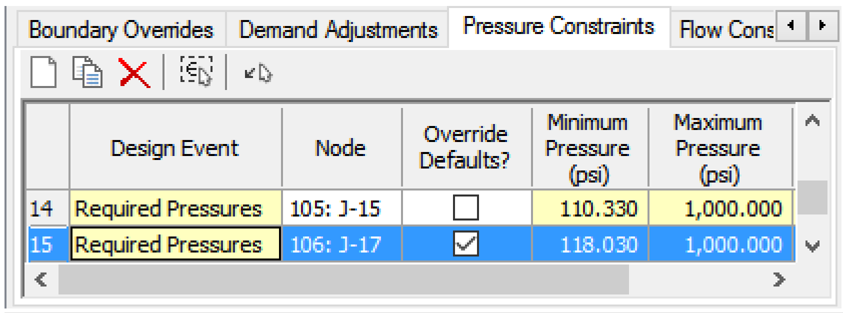


i. In the table in the upper right of the Designer dialog, set the **Minimum Pressure (Default)** value to **110.33 psi** (HGL = 255 ft.).

j. In the table in the upper right of the Designer dialog, set the **Maximum Pressure (Default)** value to **1000 psi**. For this example, maximum pressure is not a consideration, so if you set it to a high value it won't affect the calculations.

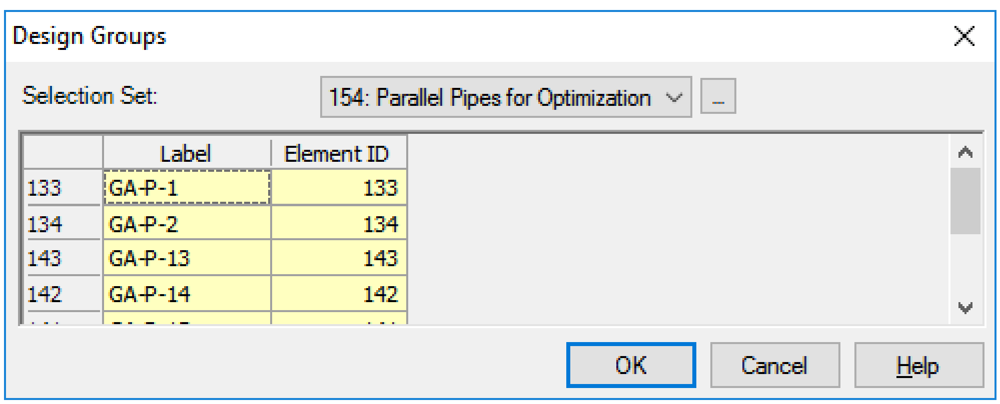


12. Customize junction J-17 to require a minimum pressure of **118.03 psi**.
    a. In the Pressure Constraints area, scroll so you can see junction **J-17**.
    b. Select the **Override Defaults?** check box.
    c. Type a minimum pressure of **118.03 psi**., maximum pressure of **1000 psi**



13. Click the **Design Groups** tab.

14. Click **Select Elements for Design Group** button  . This button lets you automatically create one design group for each pipe in the network or for a particular set of pipes.
    a. In the Selection Sets drop-down list, select **Parallel Pipes for Optimization**. This highlights a selection set containing a specific subset of the pipes in your network.

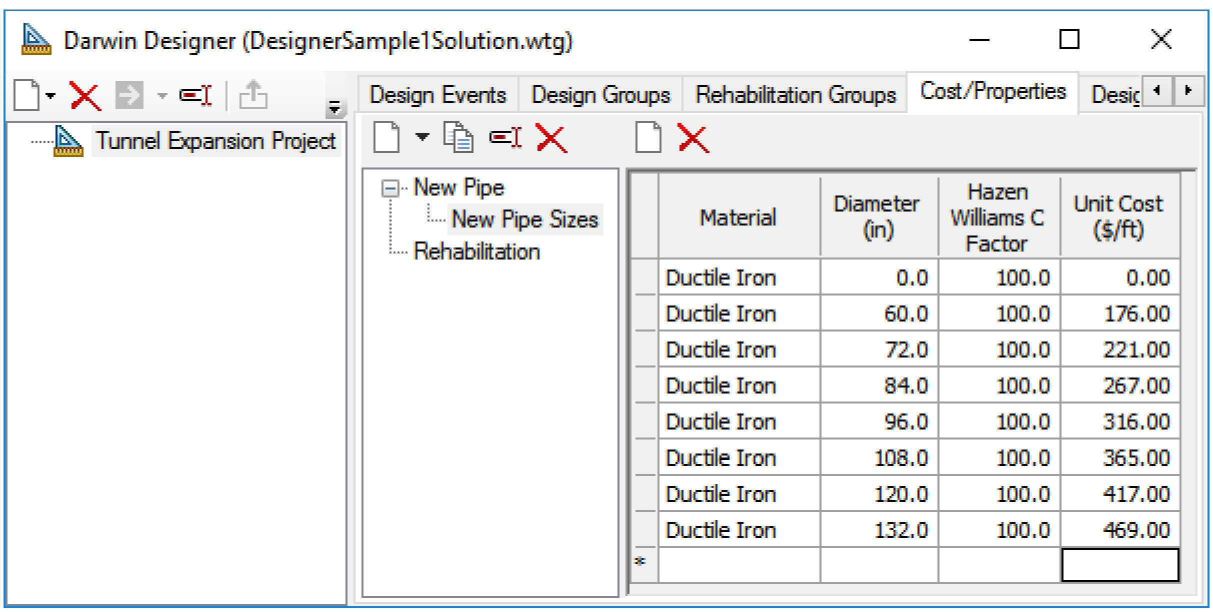


    b. Click **OK.**

15. Add an option group for your optimization.
    a. Click the **Cost/Properties** tab.
    b. Highlight **New Pipe** in the tree-view.
    c. Click **New** > **Design Option Groups**.
    d. Name the new table **New Pipe Sizes**

e. Type the following pipe material, size, roughness coefficient, and cost:

Material	**Diameter (in.)**	**Hazen Williams C Factor**	**Cost ($/ft)**
Ductile Iron	0	100	0.00
Ductile Iron	60	100	176.00
Ductile Iron	72	100	221.00
Ductile Iron	84	100	267.00
Ductile Iron	96	100	316.00
Ductile Iron	108	100	365.00
Ductile Iron	120	100	417.00
Ductile Iron	132	100	469.00

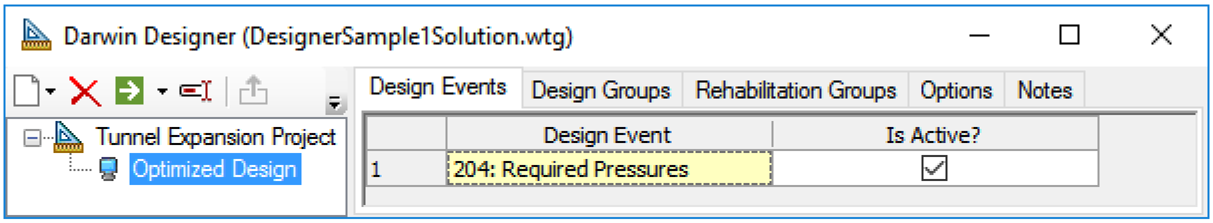


16. Create a new optimized design run.
    a. In the **Designs** tree-view, right-click **Tunnel Expansion Project** and select **New Optimized Design Run**.

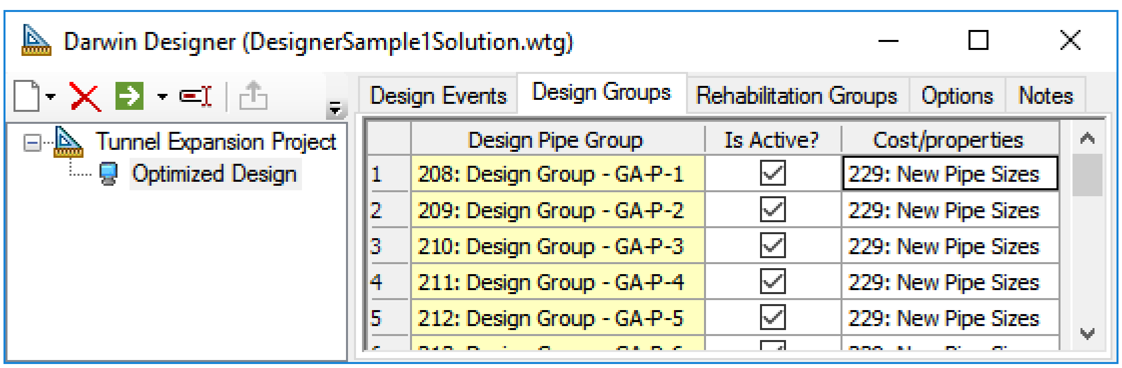
       Or, click the **New** button and select **New Optimized Design Run**.

    b. Name the design run **Optimized Design**.

17. Select the design event you want to use, **Required Pressures**, by clicking the **Active** check box.



18. Click the **Design Groups** tab.
    a. Make sure the **Is Active?** check boxes for all of the design groups are checked.
    b. Right-click the **Cost/Properties** column heading.
    c. Select **Global Edit**.
    d. Choose **New Pipe Sizes** as the option group you want to use and click OK. .



19. Click the **Options** tab.

a. Set the GA Parameters as follows (most of these are the default settings, with the exception of Random Seed and Penalty Factor):

GA Parameter	Value
Maximum Era Number	6
Era Generation Number	150
Population Size	50
Cut Probability	1.7
Splice Probability	60.0
Mutation Probability	1.5
Random Seed	0.4
Penalty Factor	25,000,000.000

b. Set the Stopping Criteria as follows:

Stopping Criteria	Value
Maximum Trials	50,000
Non Improvement Generations	200

c. Set the Top Solutions, Solutions to Keep to **3**. This sets how many results will be available as results (see Step 2: Viewing Results later in the lesson).

20. Click **Compute** to calculate the optimized design.

While the calculation proceeds, Bentley WaterGEMS displays the Darwin Designer Run Progress dialog box, which displays the following information:

- **Fitness**—In this case, you were calculating based on cost. So, the best fitness is the least costly solution that the GA (Genetic Algorithm) found.
- **Cost ($)**—The lowest cost found by the calculation displays here.
- **Benefit**—Measured pressure improvement in the network. This is 0 because the lesson only considers cost and not pressure benefit.
- **Violation**—The largest violation of established pressure and flow boundaries, such as maximum or minimum pressures, displays here. If there were a violation, you would use the results area Pressure and/or Flow tabs (in the results pane of the main Darwin Designer window) to look for the actual violations.

- **Generations**—The maximum value for generations is determined by the Maximum Era Number and Era Generation Number you set in the Options > GA Parameters. The actual number of generations that get calculated depend on the Options > Stopping Criteria you set.
- **Trials**—The maximum value for trials is determined by what you set in Options > Stopping Criteria. Note that you can set a number larger than (Maximum Era Number)*(Era Generation Number)*(Population Size), but calculations beyond that number (for this example, the value is 45,000) are less likely to produce significant improvements.
  Also, note that the Messages tab might report you exceeded the maximum number of trials. This is usually because Darwin Designer must complete all of the generations before ending a trial, so it is possible that completing generations will cause a few excess trials to be calculated.

21. After the calculation is finished, click **Close** to close the Darwin Designer Run

Step 2: Viewing Results

After you calculate the optimized design results display. You can review results and look for violations of parameters.

1. From the hierarchy pane, you can click on the Solutions folder or any of the individual solutions for more detail. Select the solution you want to see: **Solution 1**.

   You can click the Graph button to view the solutions plotted; each solution is color coded; use the color code as a key when viewing graphs.

   Solutions are ranked by fitness, with Solution 1 being the best.

2. In the Solutions tab, if you scroll down, you can see there are seven pipes that changed from the default. These are the pipes that Darwin added to the scenario to provide the optimal solution:

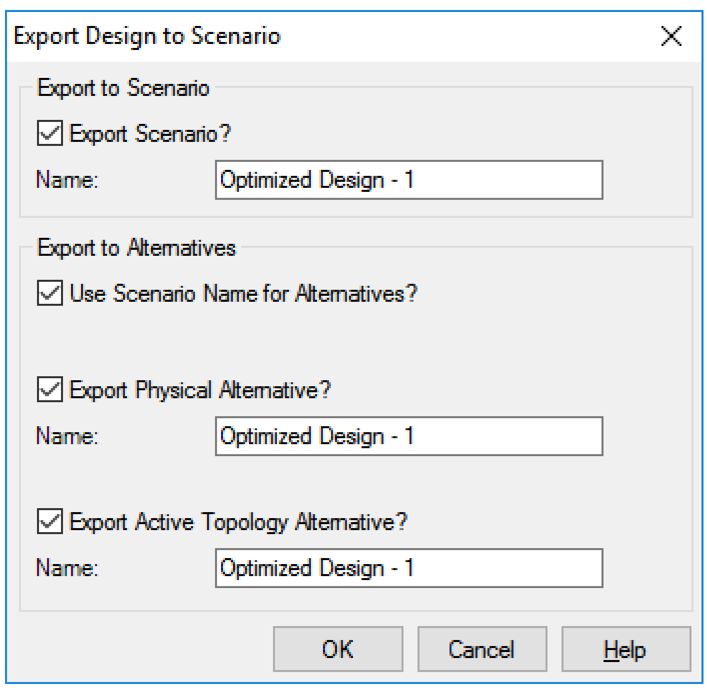
Pipe	Material	ızen-Williar C	Diameter (in)	Cost ($)
139: GA-P-7	Ductile Iron	100.0	120.0	4,003,200.0
148: GA-P-16	Ductile Iron	100.0	96.0	8,342,400.0
149: GA-P-17	Ductile Iron	100.0	96.0	9,859,200.0
150: GA-P-18	Ductile Iron	100.0	84.0	6,408,000.0
135: GA-P-3	Ductile Iron	100.0	72.0	1,613,300.0
151: GA-P-19	Ductile Iron	100.0	72.0	3,182,400.0
153: GA-P-21	Ductile Iron	100.0	60.0	4,646,400.0

3. The Rehabilitation Groups and Flow results under the Simulated Results tab are empty because this lesson does not use those.
4. Click the **Pressure** results under the Simulated Results tab. This displays the maximum and minimum pressure constraints you set on the junctions and the actual pressures calculated by Darwin Designer.

Step 3: Using Results

After you calculate the optimized design results display. You can use the results to create graphs and reports.

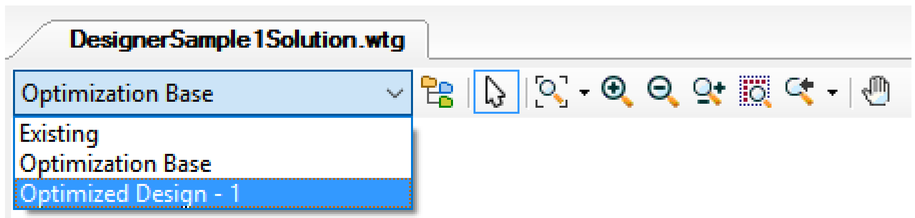
1. Solution 1 clearly provides the least expensive solution. Export the solution to Bentley WaterGEMS so you can use it.
   a. Select **Solution 1** in hierarchy under the Solutions folder.
   b. Click the **Export to Scenario** button  . The Export Design to Scenario dialog box opens.
   c. Select all check boxes to export to the various alternatives.
   d. Name the scenarios you want to export, such as **Optimized Design - 1**. The name you choose must be unique; there cannot already exist a scenario with the same name.

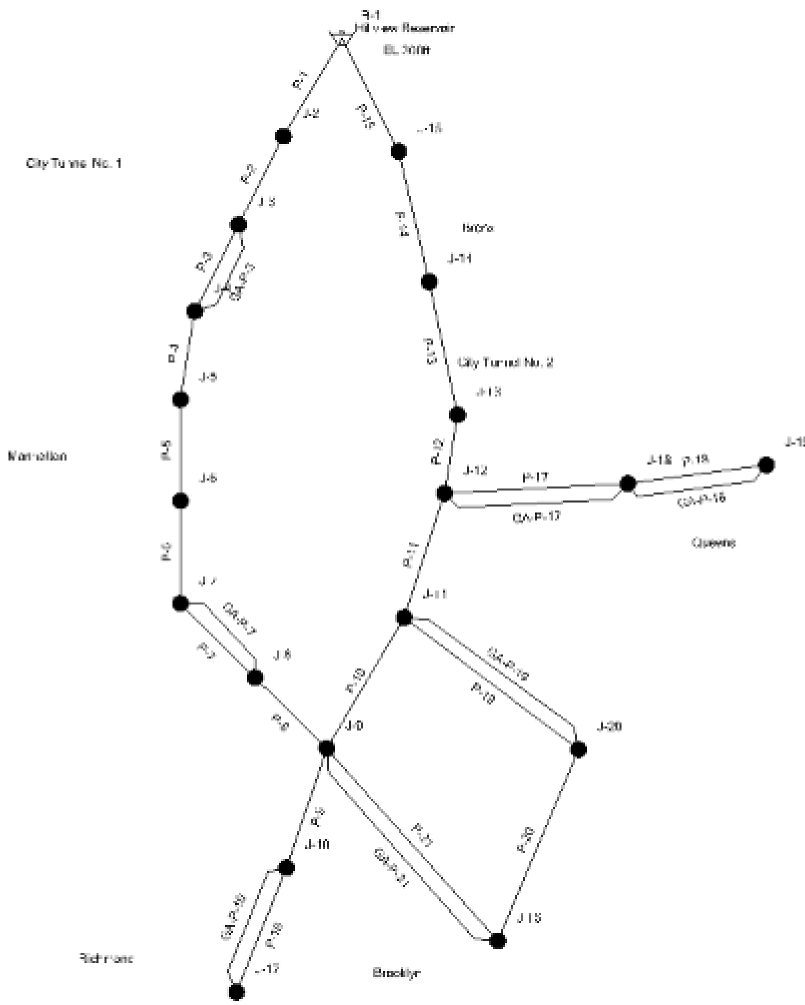


   e. Click **OK**.

2. **Close** Darwin Designer.

3. In Bentley WaterGEMS, select the scenario you exported from the **Scenario** drop-down list. Notice the parallel pipes that have been added to the base network. These are the pipes that meet the optimized design calculated by Darwin Designer.





Darwin Designer to Optimize a Pipe Network

In this lesson, you use Darwin Designer to optimize the setup of a pipe network.

There are three scenarios:

- **Existing System** representing current system conditions
- **Future Condition** representing the system expansion layout
- **Optimization Base** representing the scenario that Designer will optimize.

There are two design goals:

- New pipes to be sized are pipes 54, 68, 70, 72, 74, 76.
- Old pipes need to be rehabilitated by applying possible actions including cleaning pipe, relining pipe, and leaving the pipe as it is (no action or do nothing to a pipe).

The design criteria is:

- Minimum pressure of 45 psi at all demand junctions
- Maximum pressure of 100 psi at all demand junction
- Filling each tank to or above the initial tank level

Getting Started

1. Browse to your Lessons\Designer folder. Open **DesignerSample2.wtg**.
2. Click **OK** to the message dialog that comes up about CONNECTED Projects.
3. Go to **File** > **Save As.** Name the file **DesignerSample2Solution**
4. If needed, select **Existing System** from the Scenario drop-down list. This displays the current network.

   Notice that the Existing scenario comprises two types of pipe:

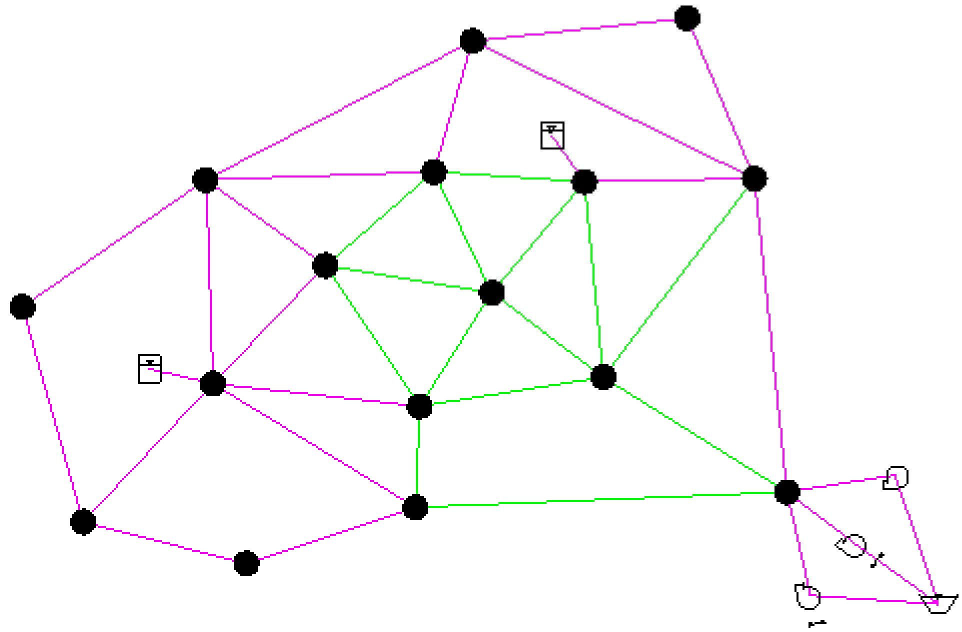
   – In green, there are older pipes, perhaps representing an old downtown section

   – In purple, there are newer pipes, perhaps representing newer additions to the water supply network.

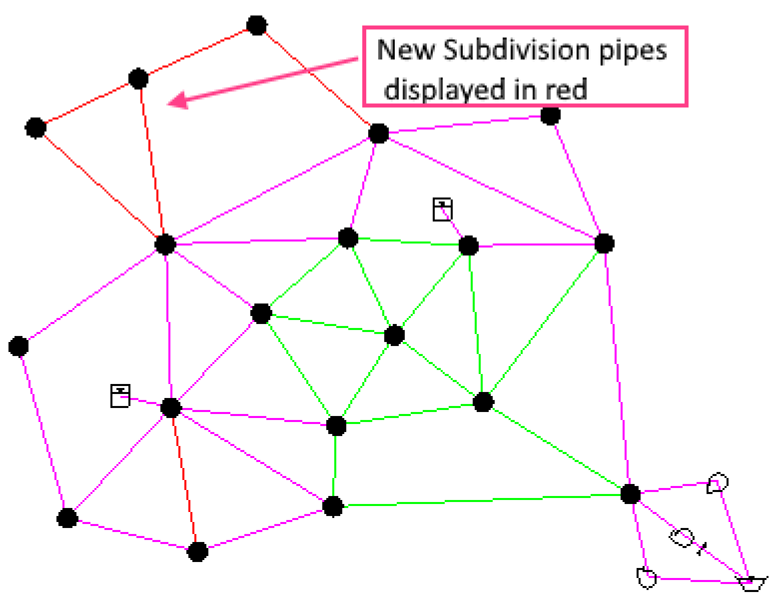
   Note: The Future conditions portion of the model may display in gray for you. You can control visibility of inactive elements by toggling the "Display inactive topology" setting under Tools > More > Options > Global

5. Click **Compute** to calculate the system pressures and tank levels for the Existing Condition.

If you want, you can inspect the pressures and tank volumes, but the purpose for calculating this condition was for a tank level comparison between the Existing and Future Condition scenarios in a later step.



6. Close the Calculation Summary and User Notifications windows.
7. Select the **Future Condition** from the Scenario drop-down list. If needed, click **Zoom Extents** to view the entire network in the window.

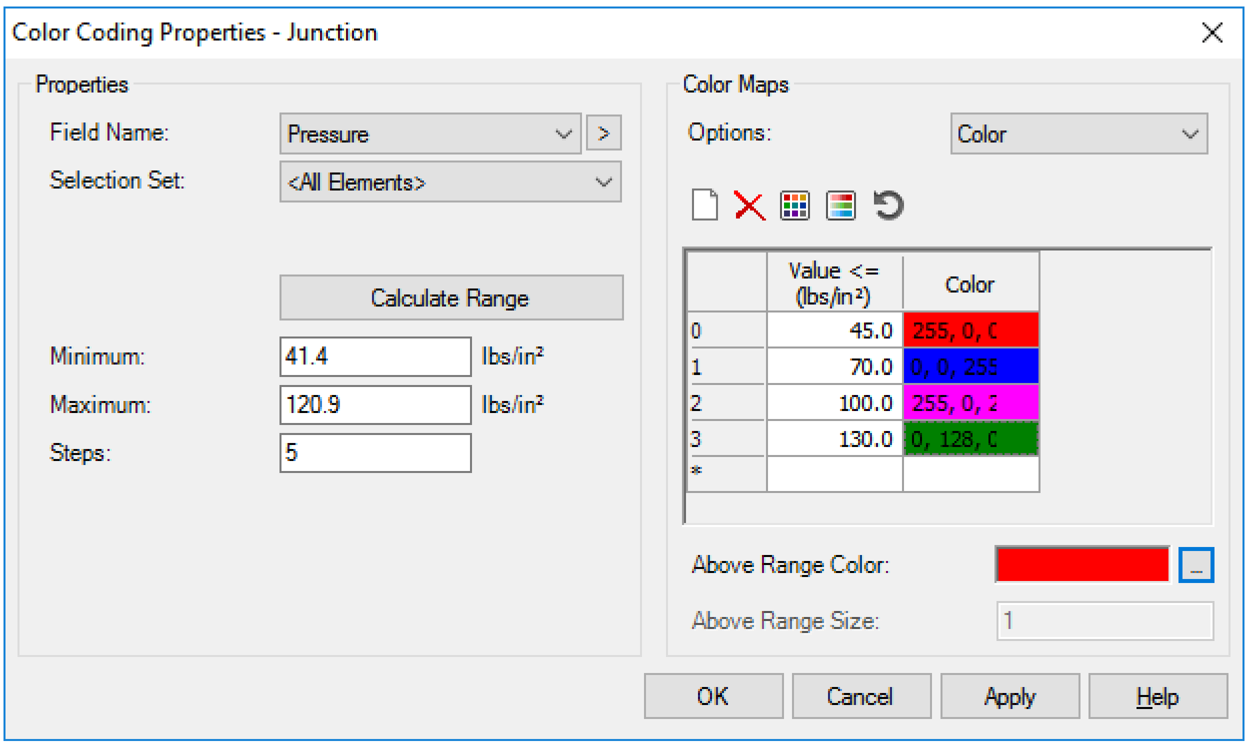


8. Click **Compute** to calculate the system pressures and tank levels for the Future Condition.
9. Close the Calculation Summary and User Notifications windows.
10. Review the pressure at junctions using color coding.
    a. Click **View > Symbology**. By default, Symbology is docked on the left hand side of the drawing pane. Right-click on **Junction** in the list and select **New > Color Coding.** The Color Coding dialog box opens.
    b. Set the **Field Name** to **Pressure**.
    c. Click the **Calculate Range** button and select **Quick Range**. This gives you a quick glance at the pressure ranges for this scenario.

d. In the Color Maps section, click the **New** button. Set the **Value** <= for this first row to **45 psi** and the **Color** to **Red**. Create 3 more rows using the following data:

Value <=	Color
70	Blue
100	Magenta
130	Green

e. The Color Coding dialog should now look like this:



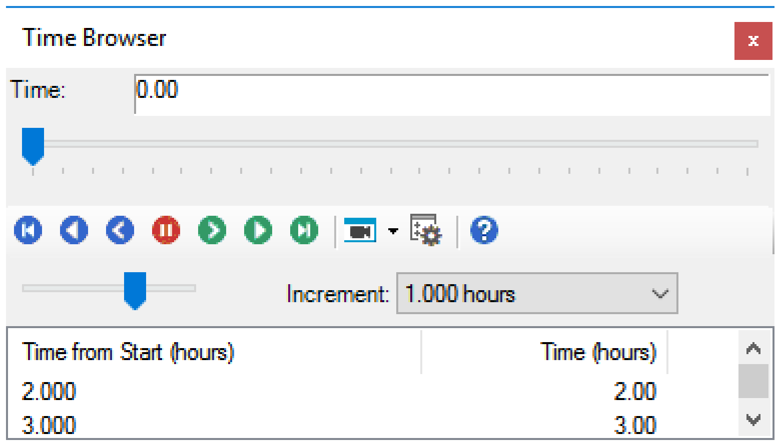
Click OK to apply the color coding and close the dialog.

For this lesson, one objective is to keep the junction pressures above **45psi**. So, when you run the animation, watch for red junctions which indicate unacceptably low pressure.

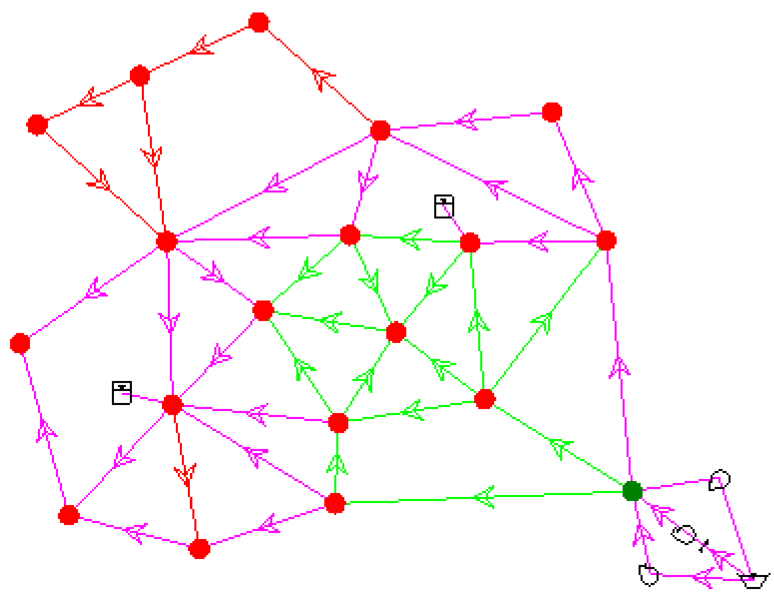
11. Run an animation to see what happens in the network over the course of 24 hours.

    a. Click **Analysis** > **Times** .

    b. Click **Play** to run the animation.



    c. Notice, at hour 6 there is a low pressure junction and by hour 12, most of the junctions are showing a low pressure.

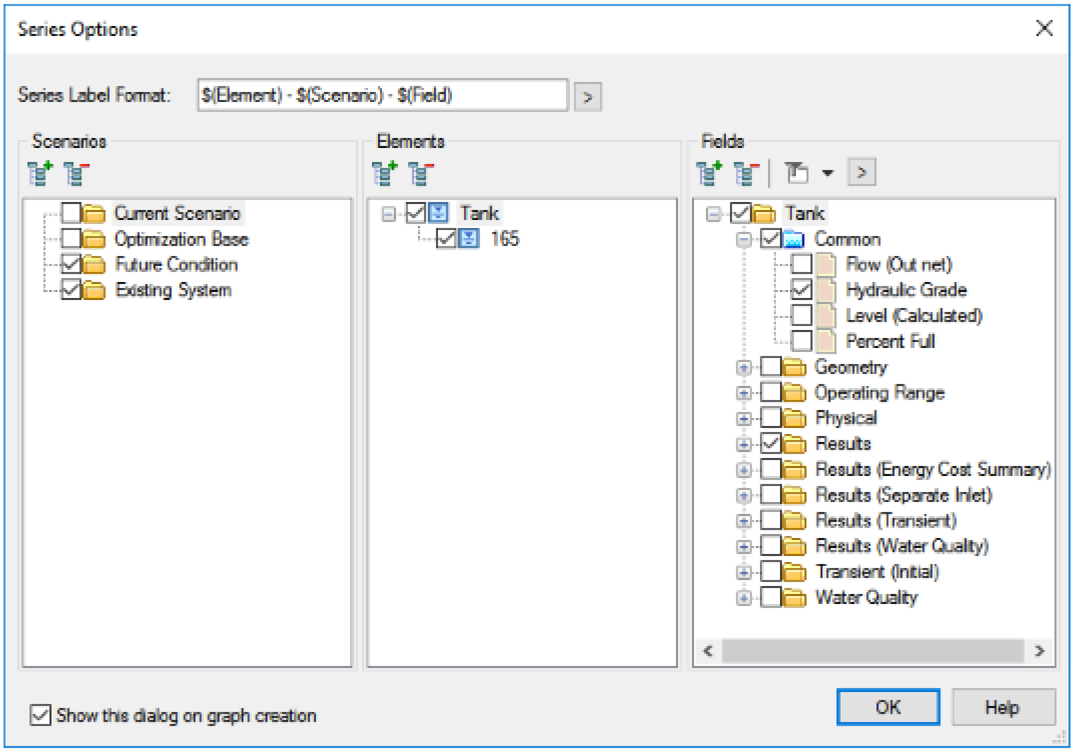


12. Use Graphs to check the levels of the tanks.

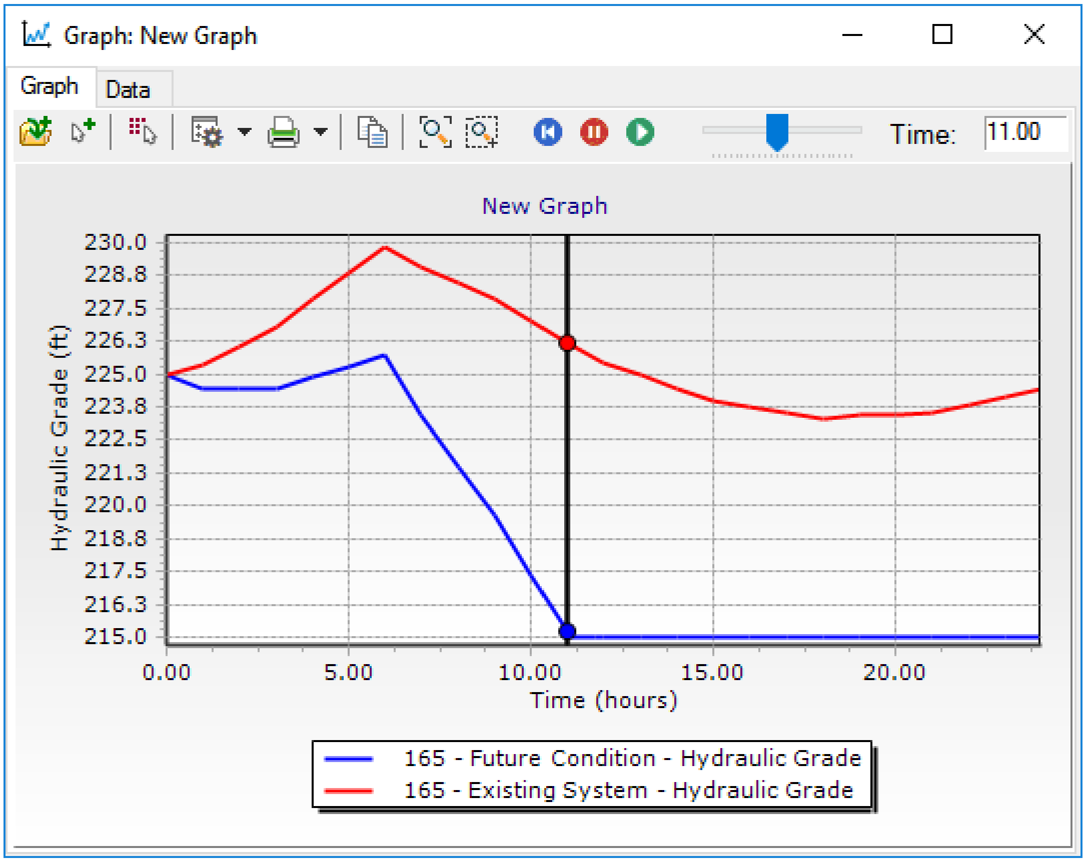
    a. Right-click the tank labeled **165** and select **Graph**.

    b. We want the graph to show the water levels for tank 165 in the Existing scenario and also the Future Condition scenario. In the Series Options dialog, check the box for **Existing System** in the Scenarios list pane.

c. In the Fields list pane, make sure that either **Hydraulic Grade** or **Level (Calculated)** is selected.



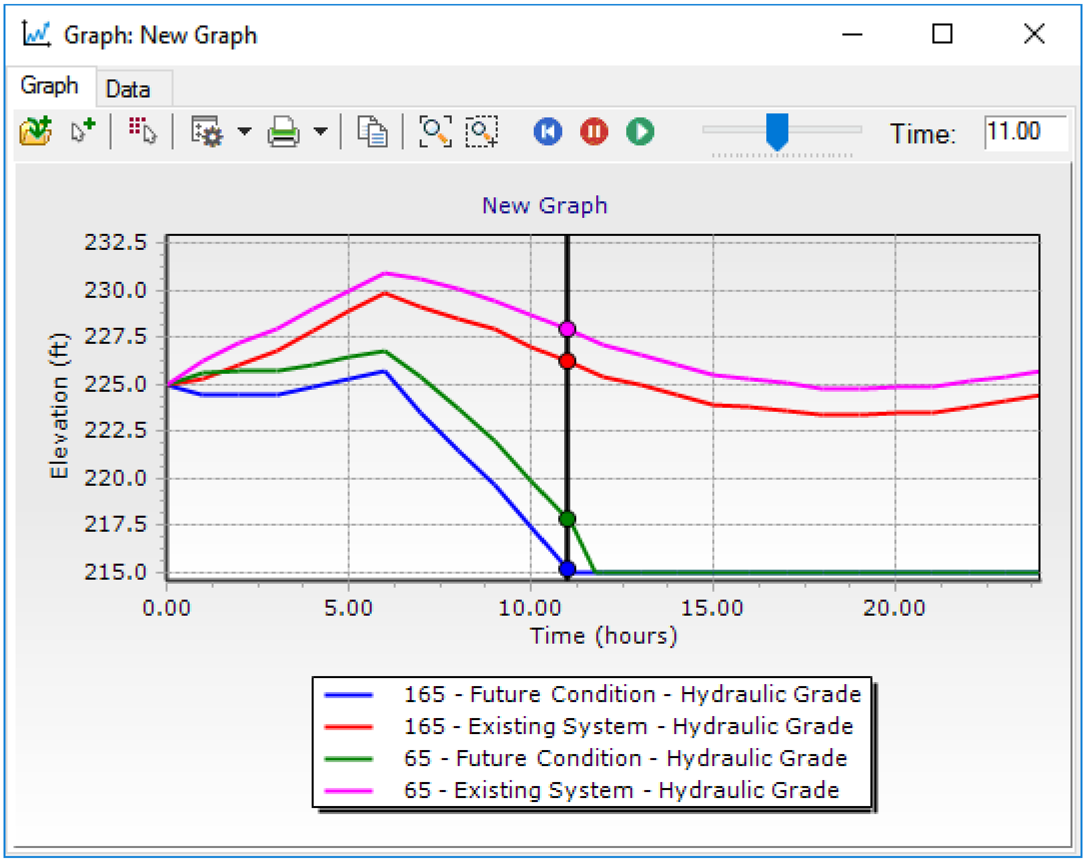
d. Click **OK**



e. Notice that by hour 11, Tank 165 is empty and does not refill.

f. Click the **Add to Graph** button  .

g. In the drawing pane, click tank **65** then right-click and select the green check-mark **Done.**



h. Notice that by hour 12, Tank 65 is also empty.

i. Close the graph window.

13. You need to use Darwin Designer and some analysis in Bentley WaterGEMS to change the existing pipe network to:

- Keep junction pressures above 45psi
- Keep the two water tanks filled

Set Up for Darwin Designer

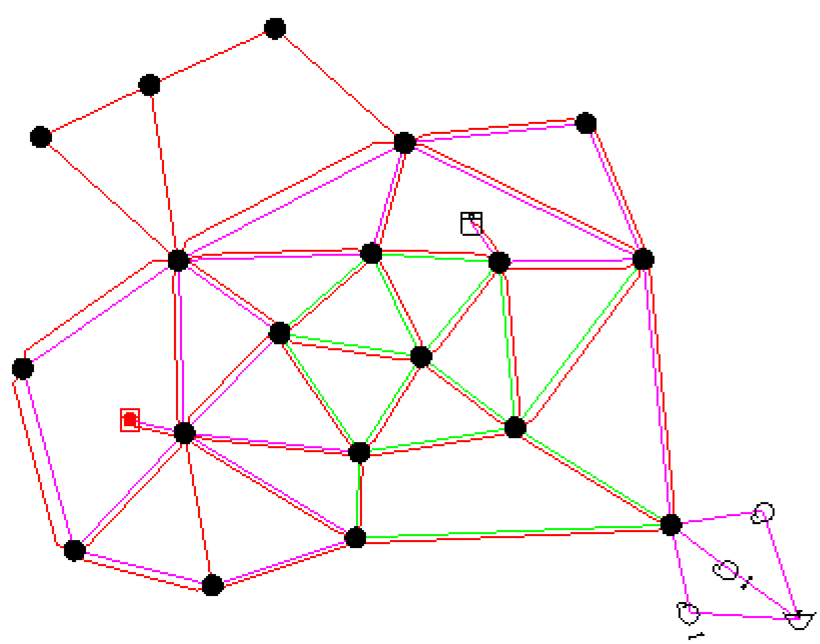
With Darwin Designer, you need to consider two ways of accomplishing a cost-effective design: create new or parallel pipes and rehabilitate existing pipes. Clearly, the new subdivision will get new pipes. And, as you can design an appropriate size for these new pipes, there is no need for parallel pipes and there are no existing pipes on which to perform rehabilitation.

With that in mind, you would create a parallel pipe option for all existing pipes. This parallel pipe option should include a variety of sizes so Darwin Designer has flexibility to choose the most efficient size. Additionally, the pipe sizes must include a 0 diameter, which lets Darwin Designer calculate the efficiency of the system with the pipe absent (without installing the parallel pipe). There are four options in this tutorial for existing pipe:

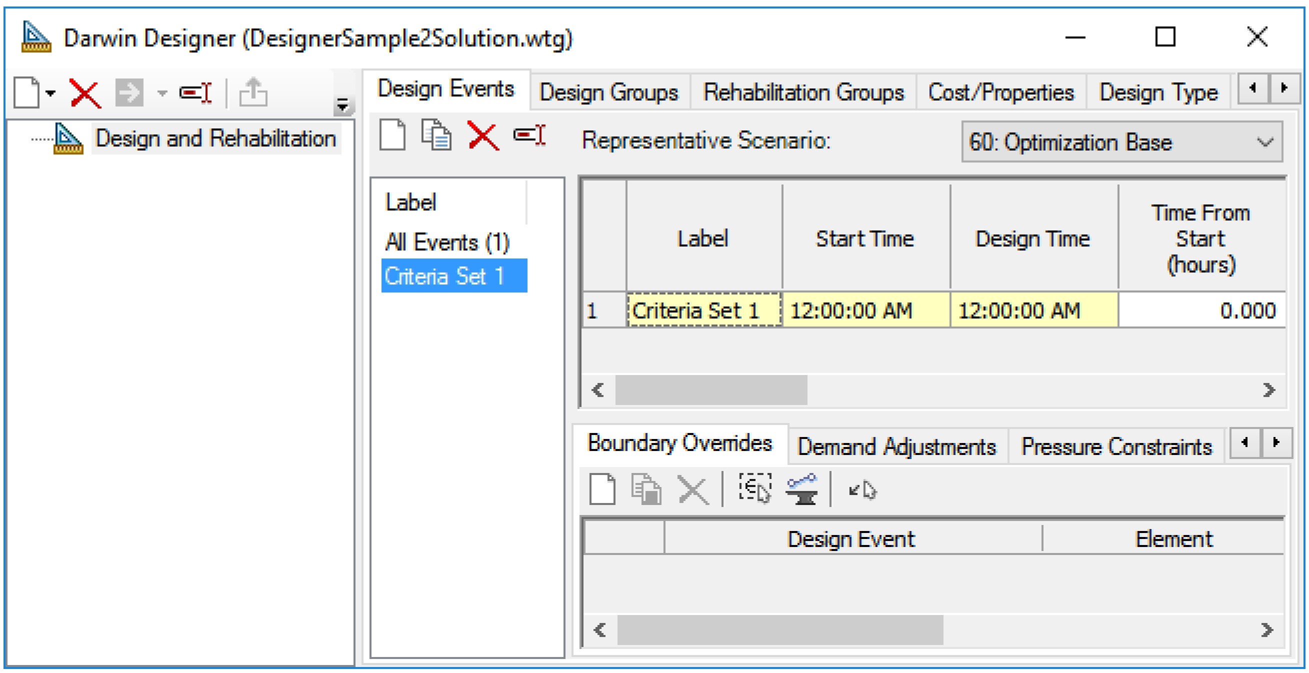
- Install parallel pipe
- Clean existing pipe
- Reline existing pipe
- Take no action

1. Select **Optimization Base** from the Scenario drop-down list.

   This is the future network set up for Darwin Designer optimization. Notice that parallel pipes have been added next to all the existing pipes. All new pipes—parallel and new ones for the subdivision—are colored red.



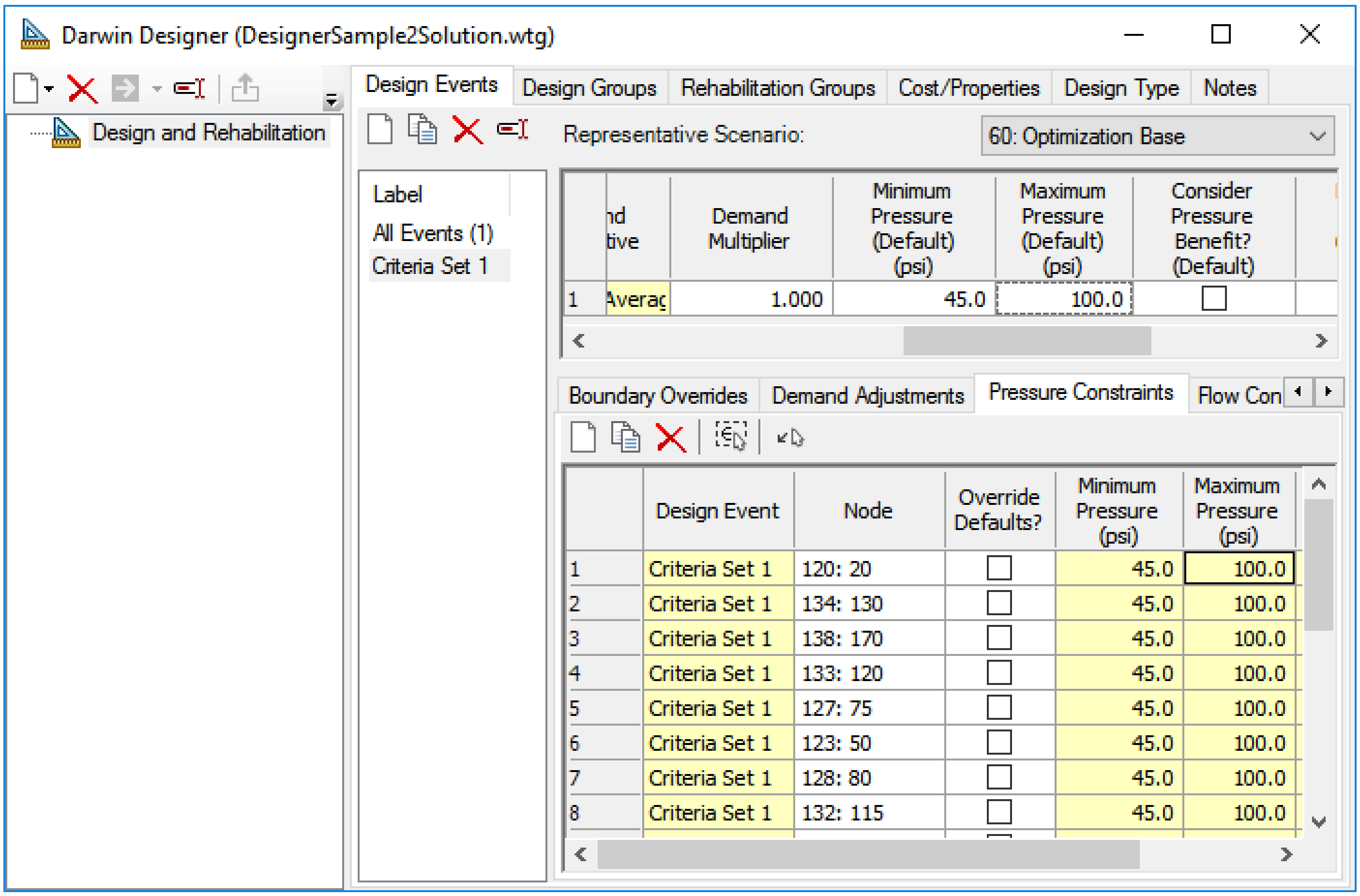
2. Click **Analysis** > **Darwin** > **Darwin Designer.**
3. Create a new designer study, called **Design and Rehabilitation.**
   a. Click the **New** button and select **New Designer Study**.
   b. Rename the study **Design and Rehabilitation.**
4. If needed, select **Optimization Base** from the Representative Scenario drop-down list.
5. Create a new design event, called **Criteria Set 1**.
   a. In the **Design Events** tab click **New**.
   b. Highlight the new design event and click **Rename**.
   c. Enter the name **Criteria Set 1** and click **OK**.



6. Set up the Design Event.
   a. Scroll to the right and set the default minimum and maximum pressure constraints:
      - **Minimum Pressure (Default) to 45 psi**
      - **Maximum Pressure (Default) to 100 psi.**
   b. Click the **Pressure Constraints** tab at the bottom.
   c. Click the **Select From Drawing** button  .

d. In the **Select** toolbar, click the **Query** button  and select **Network > All Junctions.** Then select **Done** .

e. Note that the Pressure Constraints table now contains entries for each junction in the model.



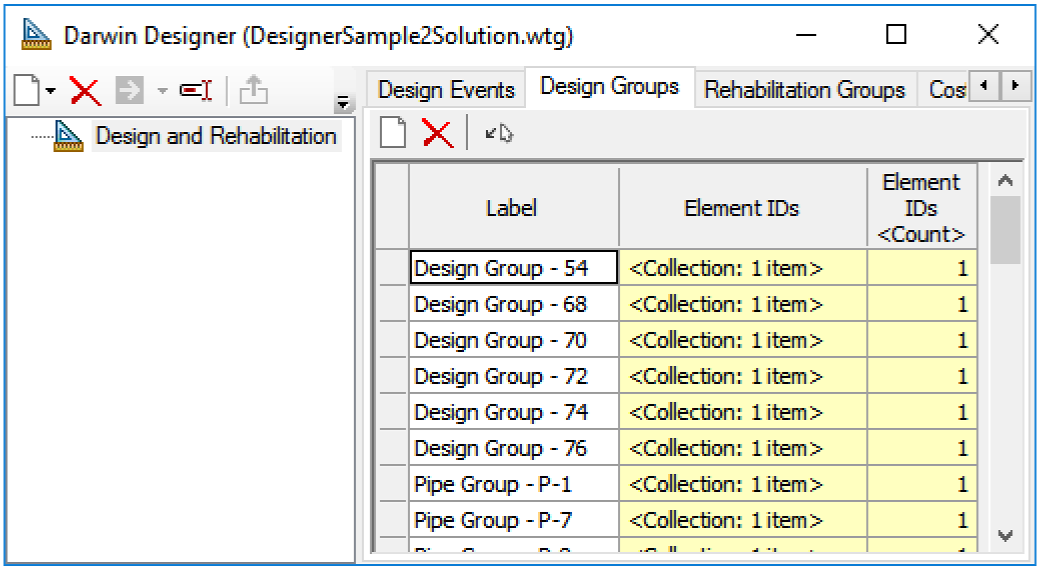
7. Click the **Design Groups** tab.

8. Click **New** to create design groups. You need to create design groups for all new or potentially new pipes, which include:

   – All pipes labeled in the model with a **P** (these are parallel pipes)

– All new pipes: 54, 68, 70, 72, 74, 76

Do not include existing pipes in any of these groups, because these need to be in a rehabilitation group.



9. Click the **Rehabilitation Groups** tab. Create rehabilitation groups containing pipes grouped as follows:

– 4, 8, 30, 32, 34 36

– 2, 10, 12, 14, 16, 18, 20, 22, 24, 26, 28, 48

– 6, 78

– 38, 40, 42, 66

– 44, 46, 50, 58, 62, 80

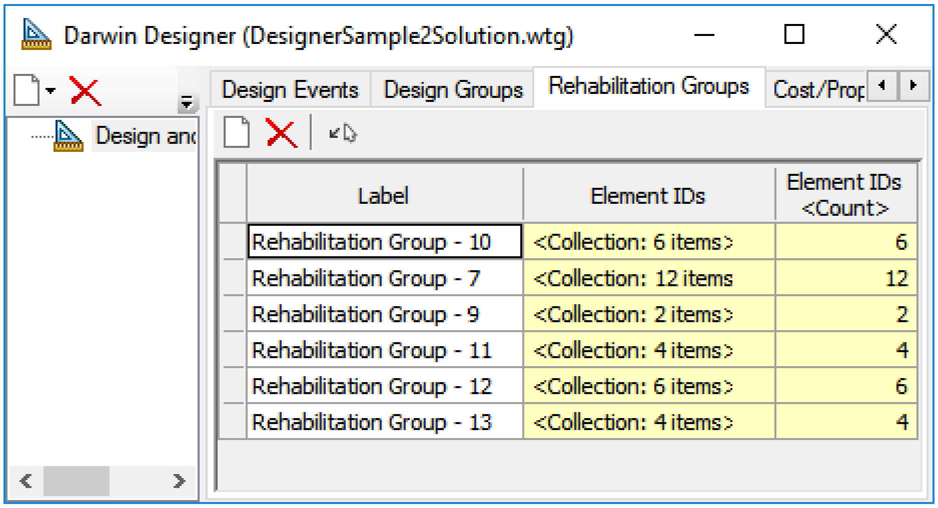
– 52, 56, 60, 64

You might consider grouping pipes based on size or age. To create a rehabilitation group:

a. Click **New**.

b. If desired, rename the Rehabilitation group and click **OK**.

c. Click the **Select Elements for Demand Group** button to choose the pipes you want to include in the group.



10. Click the **Cost/Properties** tab. Create two design option groups and one rehabilitation option group..

    a. Click **New > Design Option Groups** to create a new table.

    b. Rename the table **Design Cost Table - 1.**

    c. Enter the data below into the table. The first table contains a pipe diameter of 0. All parallel pipes will use this option group. Including a diameter of 0 lets Darwin Designer consider not adding a parallel pipe if that pipe is not needed for the optimal solution.

Material	Diameter (in.)	Hazen Williams C Factor	Unit Cost ($/ft.)
Aluminum	6	130	12.80
Aluminum	8	130	17.80
Aluminum	10	130	22.50
Aluminum	12	130	29.20
Aluminum	14	130	36.20
Aluminum	16	130	43.60
Aluminum	18	130	51.50

Material	Diameter (in.)	Hazen Williams C Factor	Unit Cost ($/ft.)
Aluminum	20	130	60.10
Aluminum	24	130	77.00
Aluminum	30	130	105.50
Aluminum	0	130	0.00

d. Create a second design costs table. (You can duplicate the table you just created and delete the row for 0 diameter.) This table is the same as the first one except it does not have a pipe diameter of 0 and is used for new pipes. New pipes must have a minimum diameter because their existence is a requirement, unlike the parallel pipes.

Material	Diameter (in.)	Hazen Williams C Factor	Unit Cost ($/ft.)
Aluminum	6	130	12.80
Aluminum	8	130	17.80
Aluminum	10	130	22.50
Aluminum	12	130	29.20
Aluminum	14	130	36.20
Aluminum	16	130	43.60
Aluminum	18	130	51.50
Aluminum	20	130	60.10
Aluminum	24	130	77.00
Aluminum	30	130	105.50

11. Create a single rehabilitation option groups table containing three actions: Clean, Relining, and Do Nothing. A do-nothing action is necessary so Darwin Designer can consider not rehabilitating some pipes. Each of these actions must reference three functions, one for each column in the table.

12. Click **New > Rehabilitation Option Groups** to create a new rehabilitation option table.
    a. Rename the table **Rehab Cost Table - 1**.
    b. Type the name of an action you want to create, such as **Clean**.
    c. Click the cell under Pre-Rehabilitation Diameter Vs. Post-Rehabilitation Diameter and click the **Ellipsis (…)** button to create a new function. The Rehabilitation Functions manager opens.
    d. Click **New > New Pre-Rehabilitation Diameter Vs. Post-Rehabilitation Diameter Function**.
    e. Name the function, **Function - 0**.
    f. Enter your diameter data (inside pipe diameter) into the table on the right side of the dialog. We recommend you include all the diameters of pipe in the table. (If you do not, Darwin Designer will use interpolation to calculate the diameters you do not include.) In this case, the function does not change the diameter of any pipes.

Pre-Rehabilitation Diameter (in.)	Post-Rehabilitation Diameter (in.)
6	6
8	8
10	10
12	12
14	14
16	16
18	18
20	20

13. In the Rehabilitation Function Manager, click **New > Pre-Rehabilitation Vs. Post-Rehabilitation Unit Cost**.
    a. Rename it **Function - 1**.
    b. Enter the data for pipe diameter and unit cost as follows:

Diameter (in.)	Unit Cost($/ft.)
6	17.00
8	17.00
10	17.00
12	17.00
14	18.20
16	19.80
18	21.60
20	23.50
30	25.50

14. In the Rehabilitation Functions manager, click **New** > **Pre-Rehabilitation Diameter Vs. Post-Rehabilitation Roughness Function**.
    a. Rename it **Function - 2**, and click **OK**.
    b. Enter the data for pipe diameter and roughness as follows:.

Diameter (in.)	Unit Cost($/ft.)
6	130
8	130
10	130
12	130
14	130
16	130
18	130
20	130

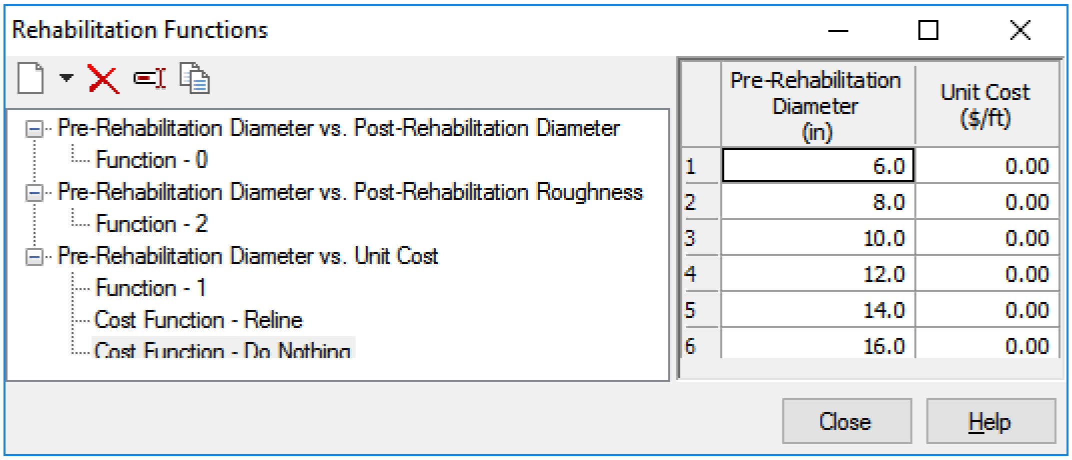
15. Create another Function called **Cost Function - Reline**. This is the cost for relining pipes. Use these values:

Diameter (in.)	Unit Cost ($/ft.)
6	26.20
8	27.80
10	34.10
12	41.40
14	50.20
16	58.50
18	66.20
20	76.80
24	109.20
30	142.50

16. Create a final function called **Cost Function - Do Nothing**. This function is required if you need Darwin Designer to consider not rehabilitating an existing pipe as an option.

Diameter (in.)	Unit Cost ($/ft.)
6	0.00
8	0.00
10	0.00
12	0.00
14	0.00
16	0.00
18	0.00
20	0.00
24	0.00
30	0.00

17. The Rehabilitation Functions manager should now look like this:



18. Click **Close** to close the Rehabilitation Functions manager.

19. For the **Action: Clean**:

    a. Set the Pre-Rehabilitation Diameter vs. Post-Rehabilitation Diameter Function to **Function - 0**.

    b. Set the Pre-Rehabilitation Diameter vs. Unit Cost Function to **Function 1**.

    c. Set the Pre-Rehabilitation Diameter vs. Post-Rehabilitation Roughness Function to **Function 2**.

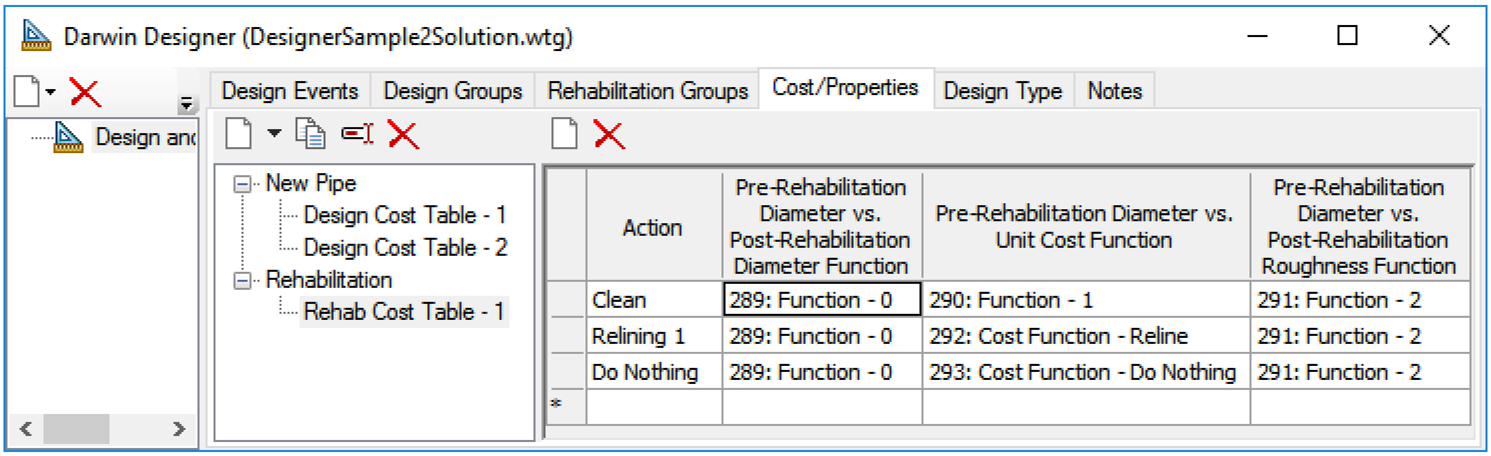
20. Type a new Action, called **Relining 1**.

    a. Set the Pre-Rehabilitation Diameter Vs. Post-Rehabilitation Diameter Function to **Function - 0**.

    b. Set the Pre-Rehabilitation Diameter Vs. Unit Cost Function to **Cost Function - Reline**.

    c. Set the Pre-Rehabilitation Diameter Vs. Post-Rehabilitation Roughness Function to **Function - 2**.

21. Type a new Action called **Do Nothing**.
    a. Set the Pre-Rehabilitation Diameter Vs. Post-Rehabilitation Diameter Function to **Function - 0**.
    b. Set the Pre-Rehabilitation Diameter Vs. Unit Cost Function to **Cost Function - Do Nothing**.
    c. Set the Pre-Rehabilitation Diameter Vs. Post-Rehabilitation Roughness Function to **Function - 2**.

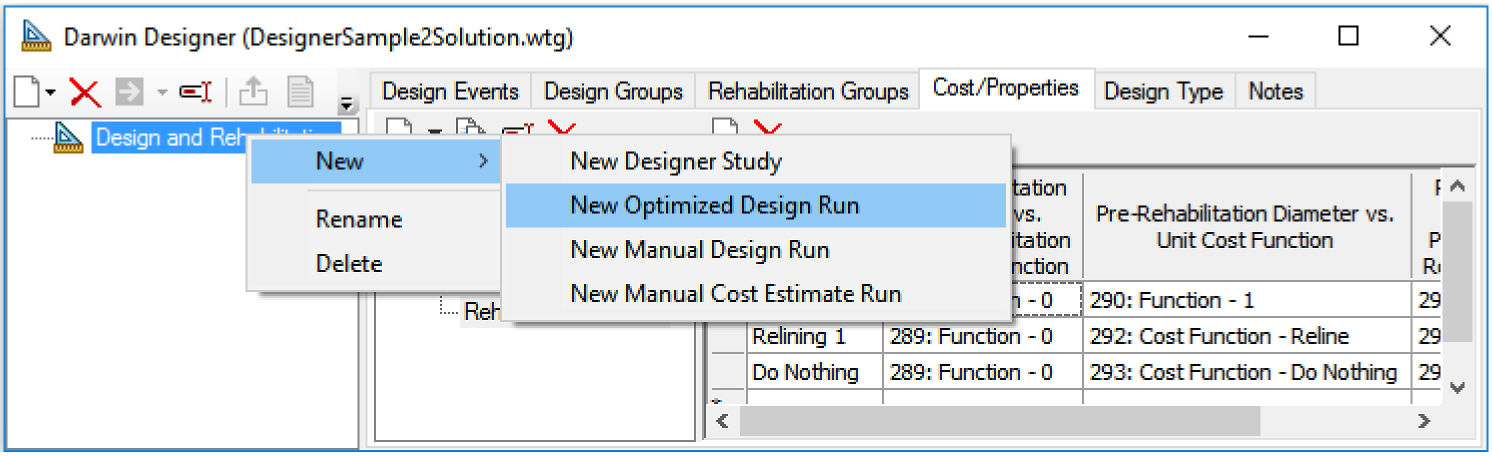


22. Click the **Design Type** tab to set the genetic algorithm parameters. Set the Objective Type to **Minimize Cost**. You are not considering any benefits to increasing system flow or pressure.

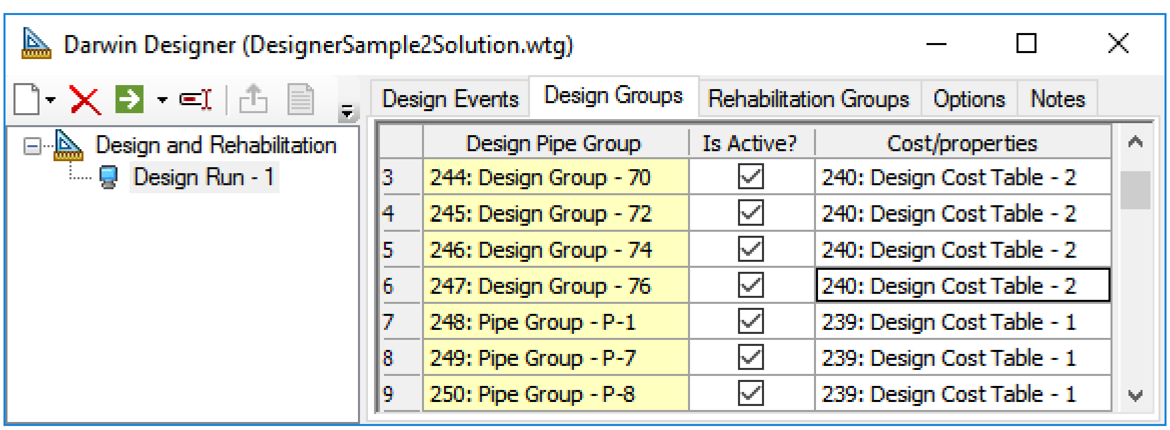
Create the Optimized Design Run

The design run uses your setup and applies it to the network.

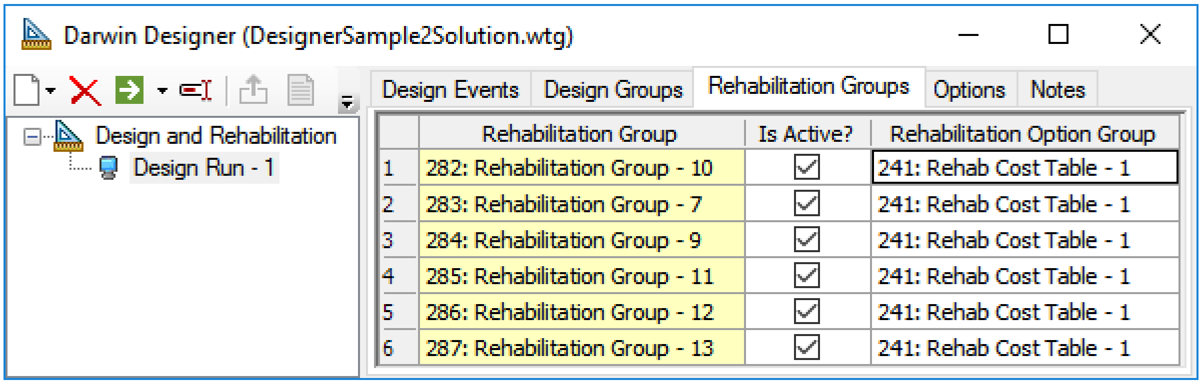
1. Right-click the **Design and Rehabilitation** design run in the left pane, and select **New > New Optimized Design Run**.



2. Name the optimized design run as **Design Run -1**.
3. In the Design Events tab, select the **Is Active?** check box for the Design Event Name Criteria Set 1. This enables the selected design event for the current run.
4. Click the **Design Groups** tab.
5. Make sure the **Is Active?** check box is checked for all of the design groups.
6. Select the design option group used by your design groups.
   a. All groups containing parallel pipes need to use **Design Cost Table 1**, for that option group contains data for a pipe size of 0. Parallel pipes have the prefix **P**.
   b. All groups containing new, single pipes need to use **Design Cost Table 2**, for that option group does not use a 0 pipe size.



7. Click the **Rehabilitation Groups** tab.
   a. Make sure all the groups are checked as **Is Active**.
   b. Set all the groups to use your rehabilitation option group. (Right-click the heading of the check box column and globally edit them.)

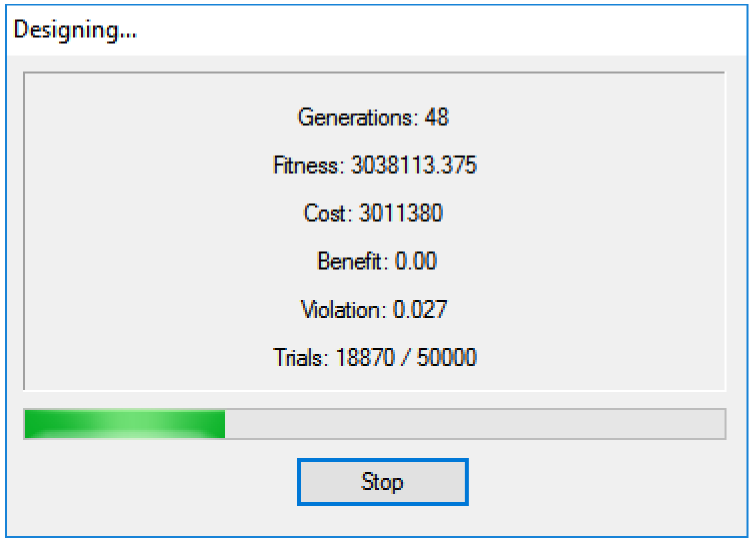


8. Click the **Options** tab to set the GA parameters for the optimization.
   – Under Stopping Criteria, set Maximum Trials to **100000**.
   – Under Top Solutions, set Solutions to Keep to **5**.

Calculate and Verify the Optimal Solution

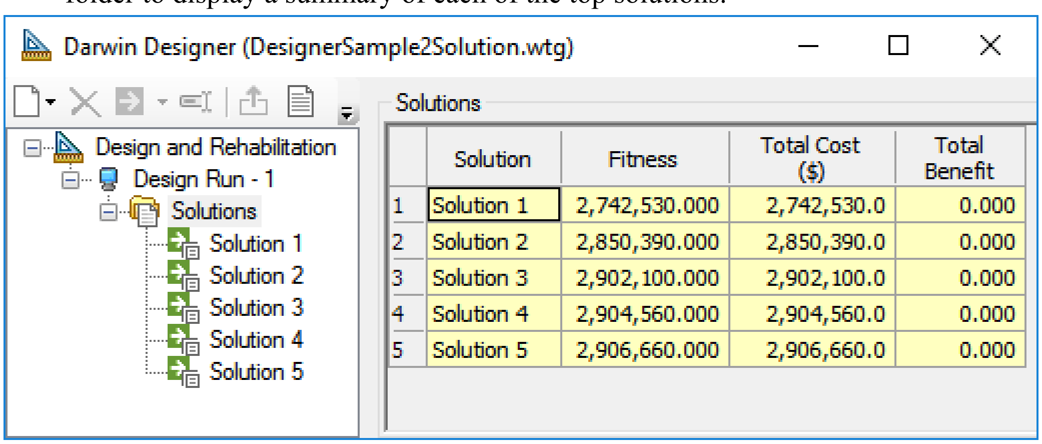
After you calculate your solutions, it is important that you look at them and verify they do what you need.

1. Click **Compute**. A dialog box opens that displays the progress and certain statistics of the calculation.



2. After the calculation is complete, click **Close**. (If the calculation did not complete successfully, you would check the Messages tab.)

   Under the Solutions folder you see five solutions numbered 1 through 5 These are the five top solutions Darwin Designer has calculated. Highlight the Solutions folder to display a summary of each of the top solutions.



   Solutions are stored in order of optimization fitness, with Solution 1 providing a better calculated solution than Solution 2, which has a better calculated solution that Solution 3, etc.

3. Export the solutions to your model, so you can review tank levels.

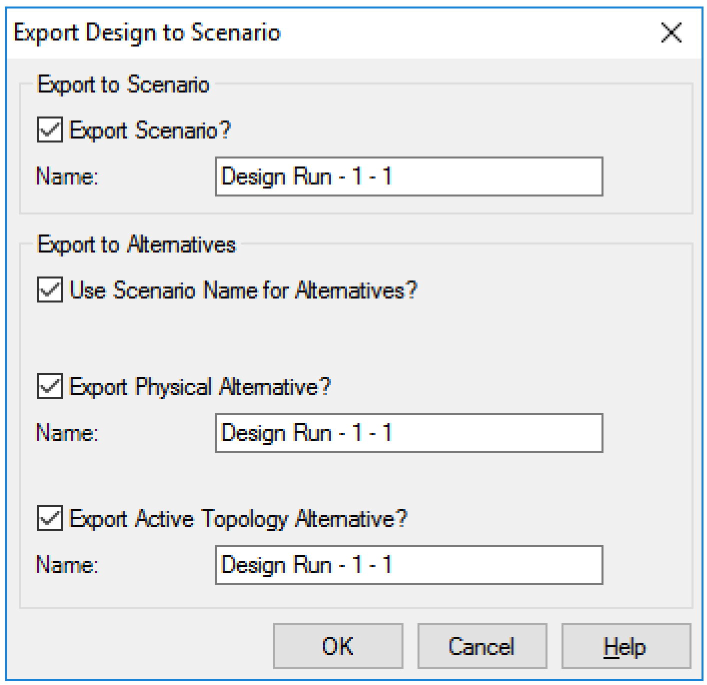
Note that the optimization calculations consider your pressure requirements (that pressure be greater than 45 psi) but not your tank levels.

a. Highlight **Solution 1**

b. Click **Export to Scenario**.  The Export to Scenario dialog box opens.

c. Select the **Use Scenario Name for Alternatives** check box. The default name is the design run name plus an incremental number starting at 1.

d. Check the **Export Physical Alternative?** and **Export Active Topology Alternative?** checkboxes.



e. Click **OK**. This exports Solution 1.

f. Select **Solution 2** from the solutions list.

g. Export Solution 2.

h. Export the remaining solutions as it was done in steps ‘b’ through ‘d’ above.

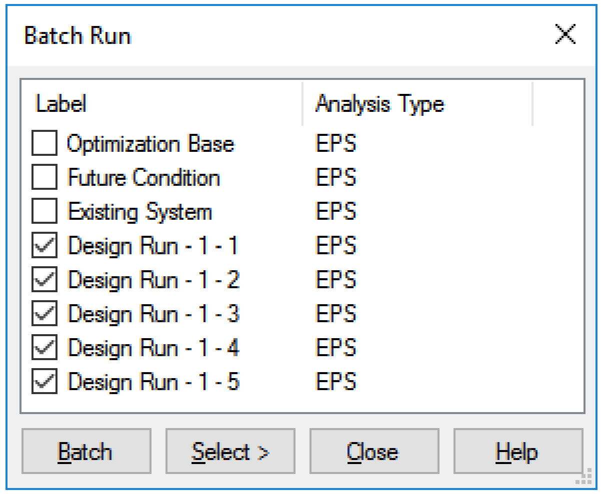
4. **Close** Darwin Designer so you can review the solutions you exported.

5. Click **Analysis > Scenarios** to open the Scenarios manager. .

6. Compute the scenarios you exported in a batch run. This lets you graph those results and look at what is happening with your tank levels.

   a. Click the black down arrow next to the compute button at the top of the scenario manager and choose **Batch Run**.

   b. Select the Scenarios you want to run. (Design Run - 1 - 1, Design Run - 1 - 2, Design Run - 1 - 3, Design Run - 1 - 4, and Design Run - 1 - 5).



   c. Click **Batch**, click **Yes** in the prompt, and close the message boxes that appear before and after the calculations.

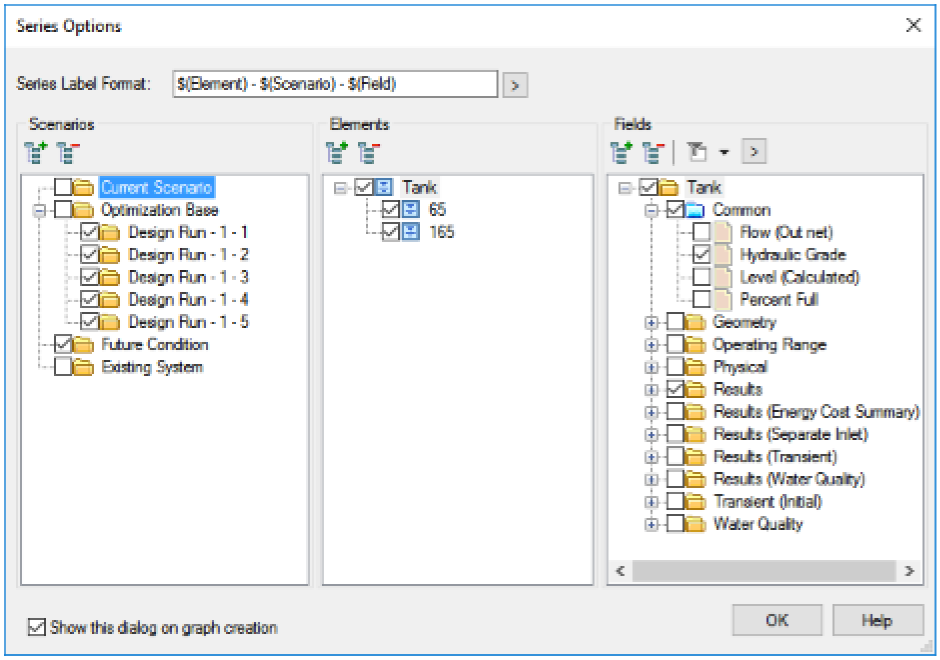
   d. After the batch run finishes, close the Scenarios manager and the User Notifications dialogs.

7. You will use graphing to inspect your tank levels. Click **View > Graphs**

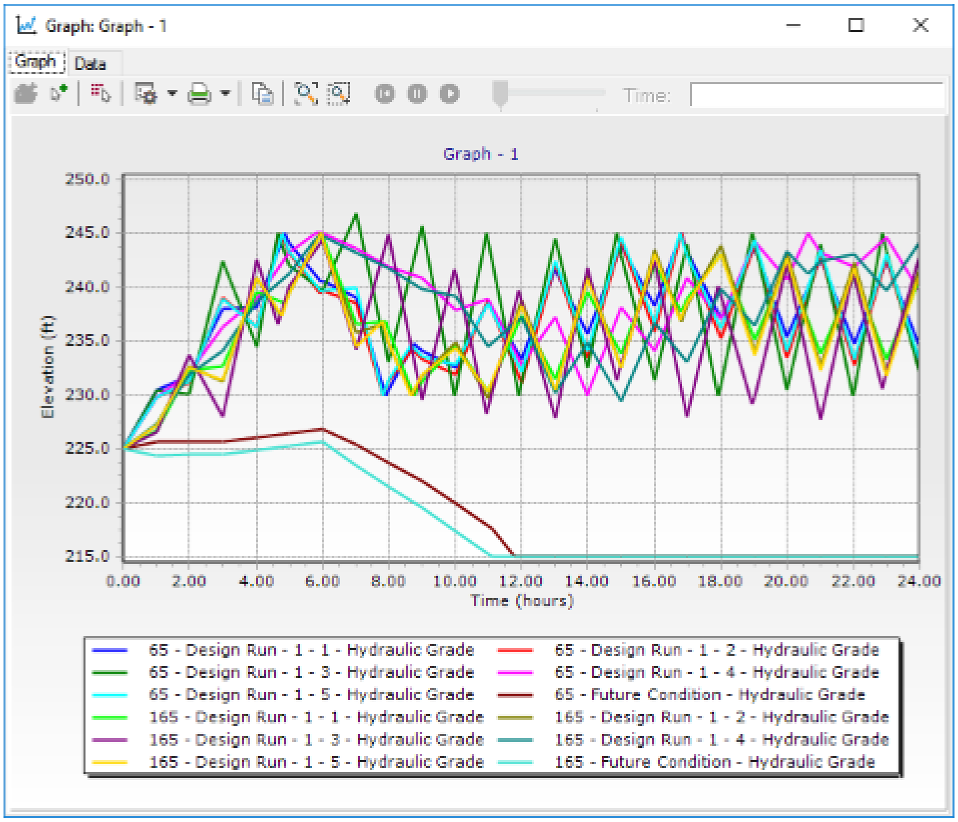
   e. Click the **New** button and select **Line Series Graph.** A Select toolbar appears to allow you to select the elements you want to graph from the drawing view. Click on both tanks, then select **Done**.

   f. In the Scenarios list of the Series Options dialog, check the boxes next to the **Design Run - 1 - 1, Design Run - 1 - 2, Design Run - 1 - 3, Design Run - 1 - 4, Design Run - 1 - 5,** and **Future Condition** scenarios (uncheck Optimized Base if it is checked).

g. In the Fields list make sure **Hydraulic Grade or Level (Calculated)** is selected.



h. Click **OK**



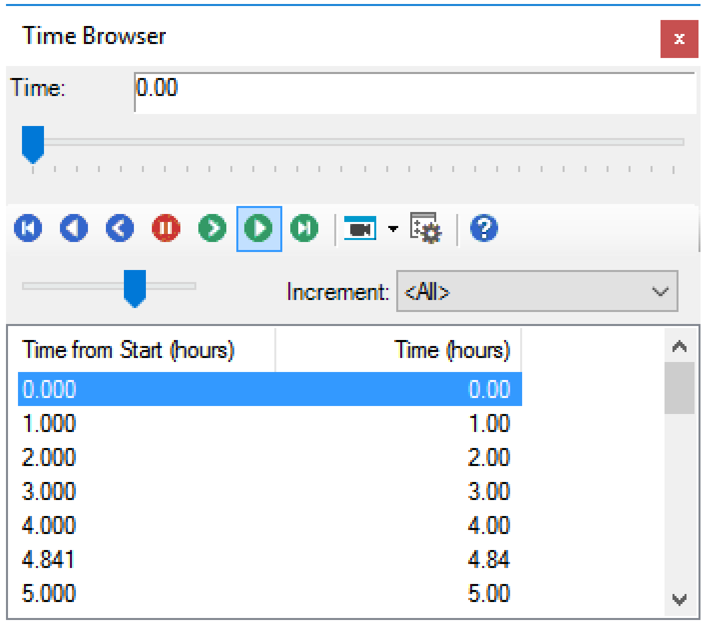
i. Review the graph. Notice that each of the design runs are able to keep the tanks full.

j. While all of the design runs do keep the tanks full, Solution 1 is the best optimal solution that meets your pressure and tank fill requirements while minimizing costs.

8. Close the Graph window.

9. In the Scenario drop-down list, choose **Design Run - 1-1**, which represents Solution 1 that Darwin Designer calculated. From looking at the results in the graph, you know this solution keeps your tanks full.

10. Inspect your tank pressure by animating the scenario over 24 hours. Click **Analysis > Times**. Click **Play**.



Note the color coding for pressure:

– <= 45 psi is red

– <= 70 psi is blue

– <= 100 psi is magenta

– <= 130 psi is green

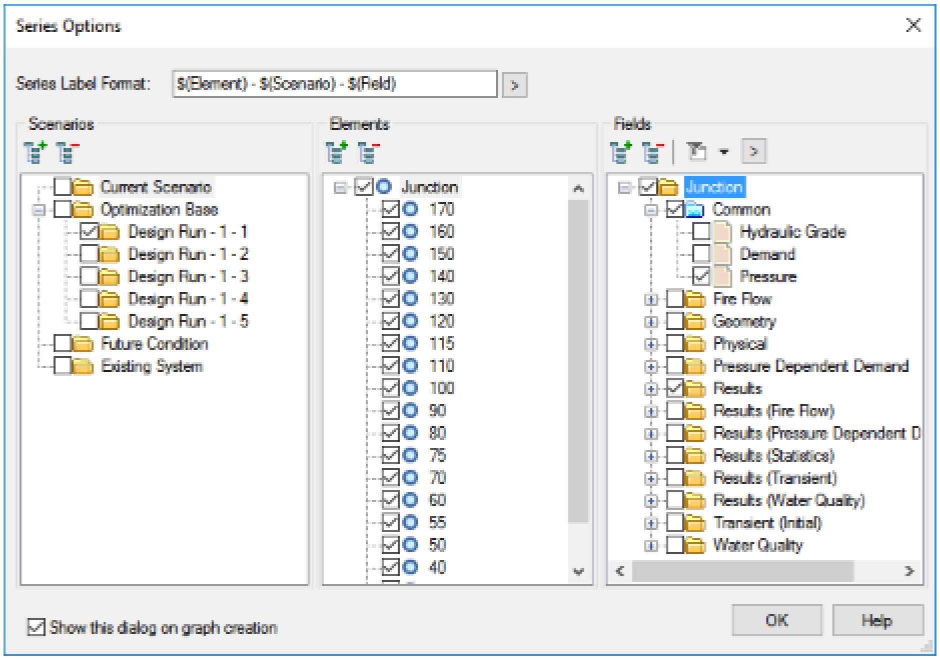
11. Make sure none of the junctions is red during the animation.

12. Inspect a graph of junction pressures.

    a. Click **Home > Select by Element > Junction** to select all of the junctions.

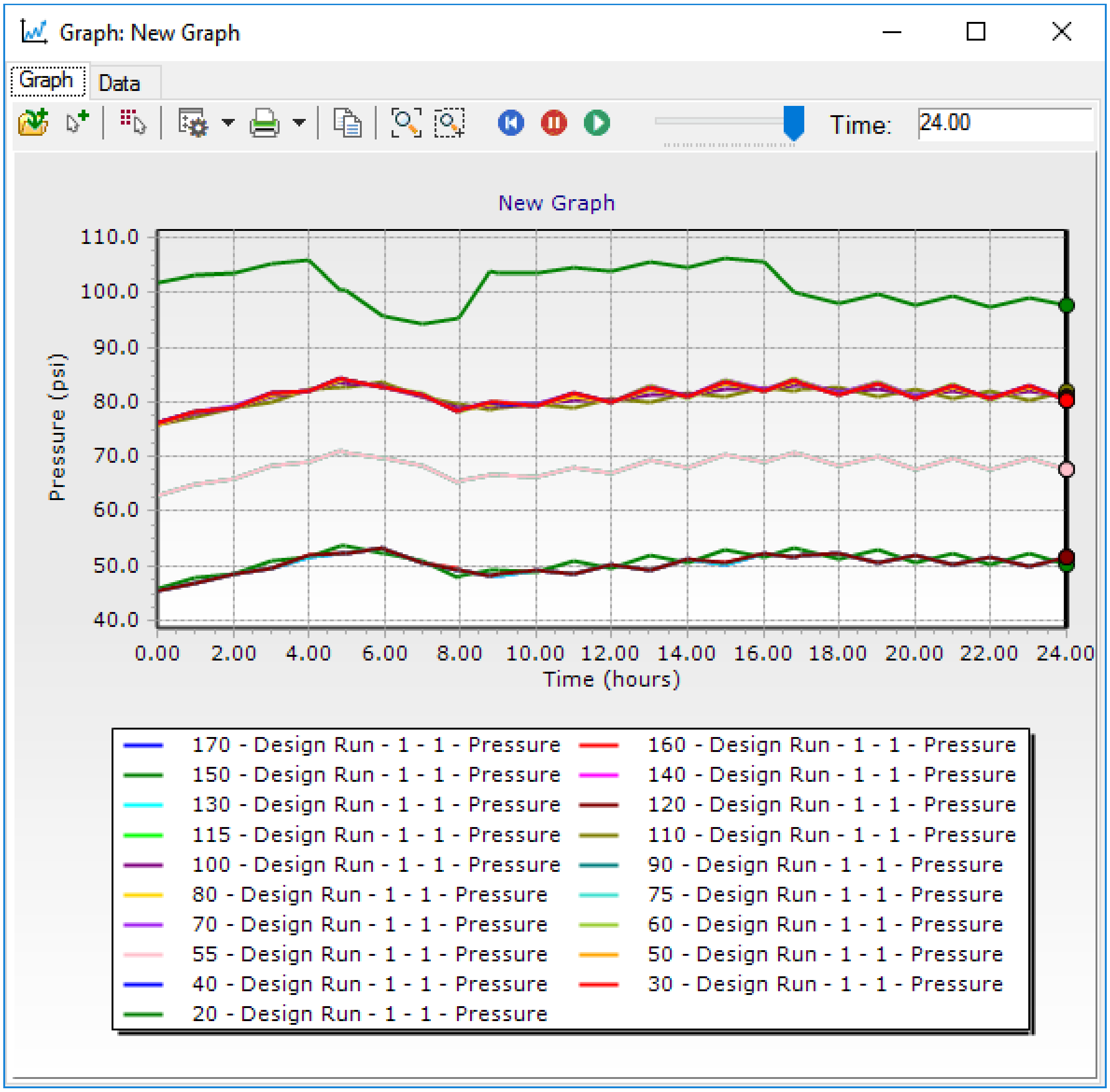
    b. Right-click one of the junctions and select **Graph**. Click **Yes** to the prompt asking if you want to graph all of the selected elements.

c. In the Series Options dialog, uncheck the Hydraulic Grade Field and check the **Pressure** box.



d. Click **OK.**

The Graph dialog box opens and displays pressures for the junctions you selected. Note that none of the junctions fall below 45 psi.



Darwin Designer computed Solution 1 to be the most optimal solution, meaning the least costly, and it also kept the tanks full. You also verified that Solution 1 was able to maintain pressures above 45 psi.

Scenario Energy Costs

Scenario Energy costs calculates energy usage and cost based on an extended period simulation (EPS). It also determines a number of intermediated values such as efficiency, power, and peak energy use.

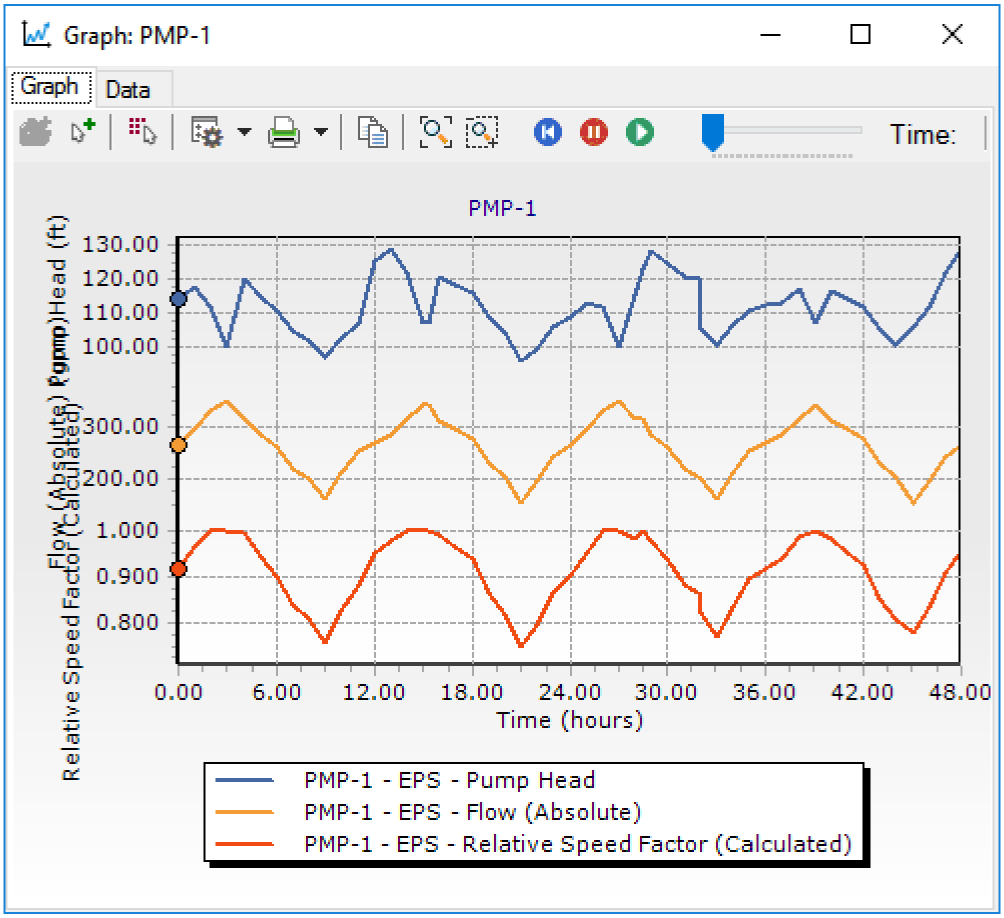
Steps for running an energy cost calculation

1. Run EPS simulation.
2. Open energy cost manager and set up energy pricing.
3. Select scenario and run energy cost calculation.
4. Review Results.

Step 1: Run EPS Model

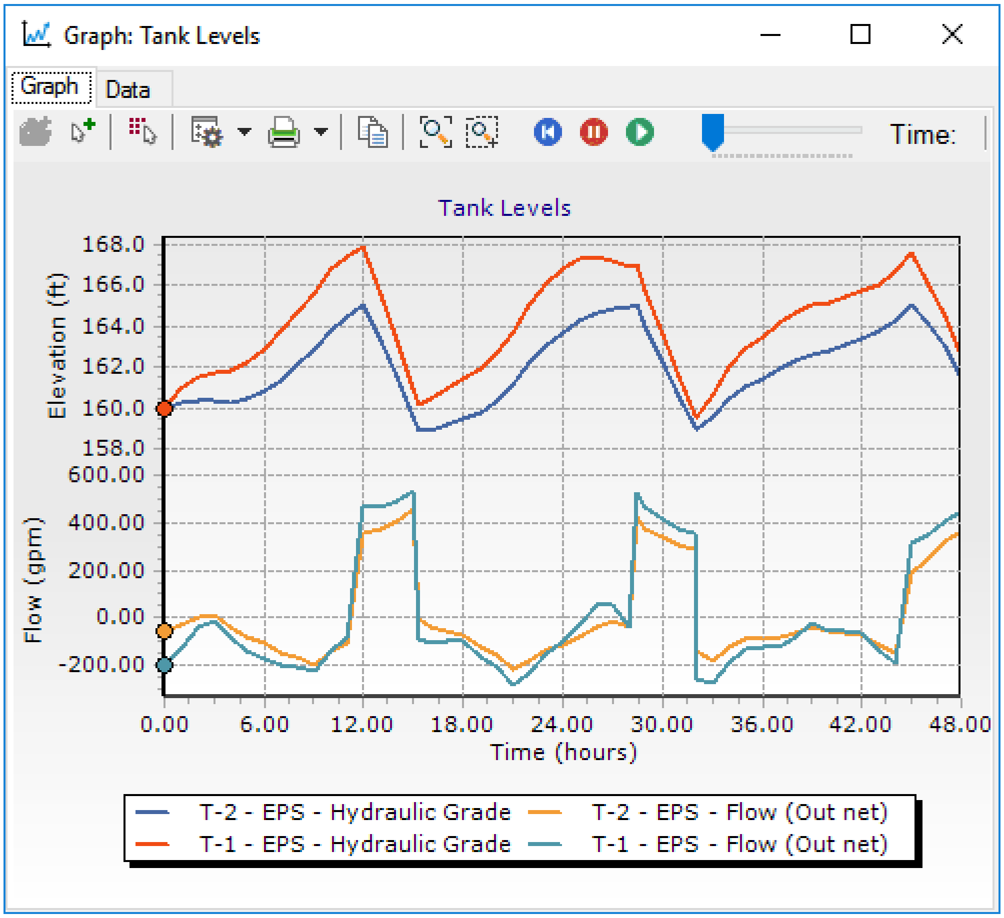
1. Open the **EngCostLessonStart.wtg** file in the Lessons directory.
2. Click **OK** to the message dialog that comes up about CONNECTED Projects.
3. Compute the **EPS** Scenario in the model  .

4. Choose **View** > **Graphs** and double-click on **PMP-1** .



Notice that the pump reaches 100% full speed several times.

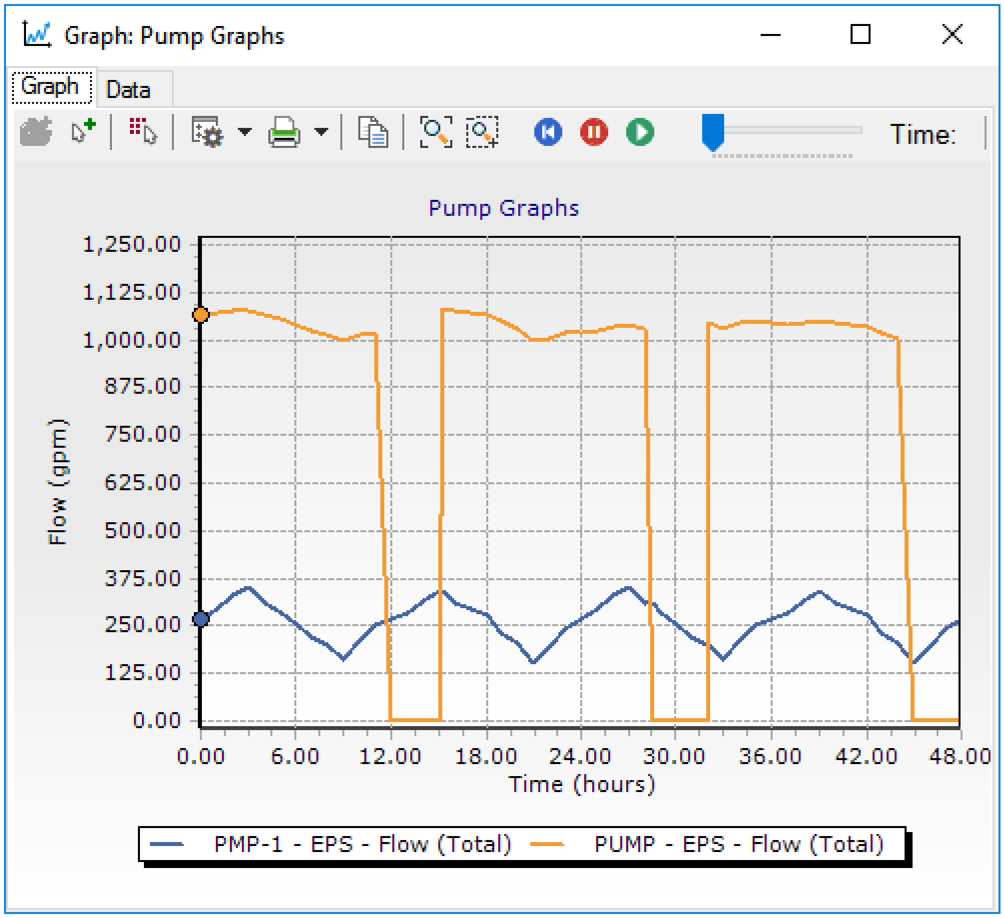
5. Close the graph. In the Graphs manager, double-click the **Tank Levels** graph
.



The tanks fill gradually during this run and empty slightly quicker when the main PUMP cycles off.

6. Close the graph.

7. In the Graphs manager, double-click the **Pump Graphs**.



You can see the relative flow of the main pump and the booster bump.

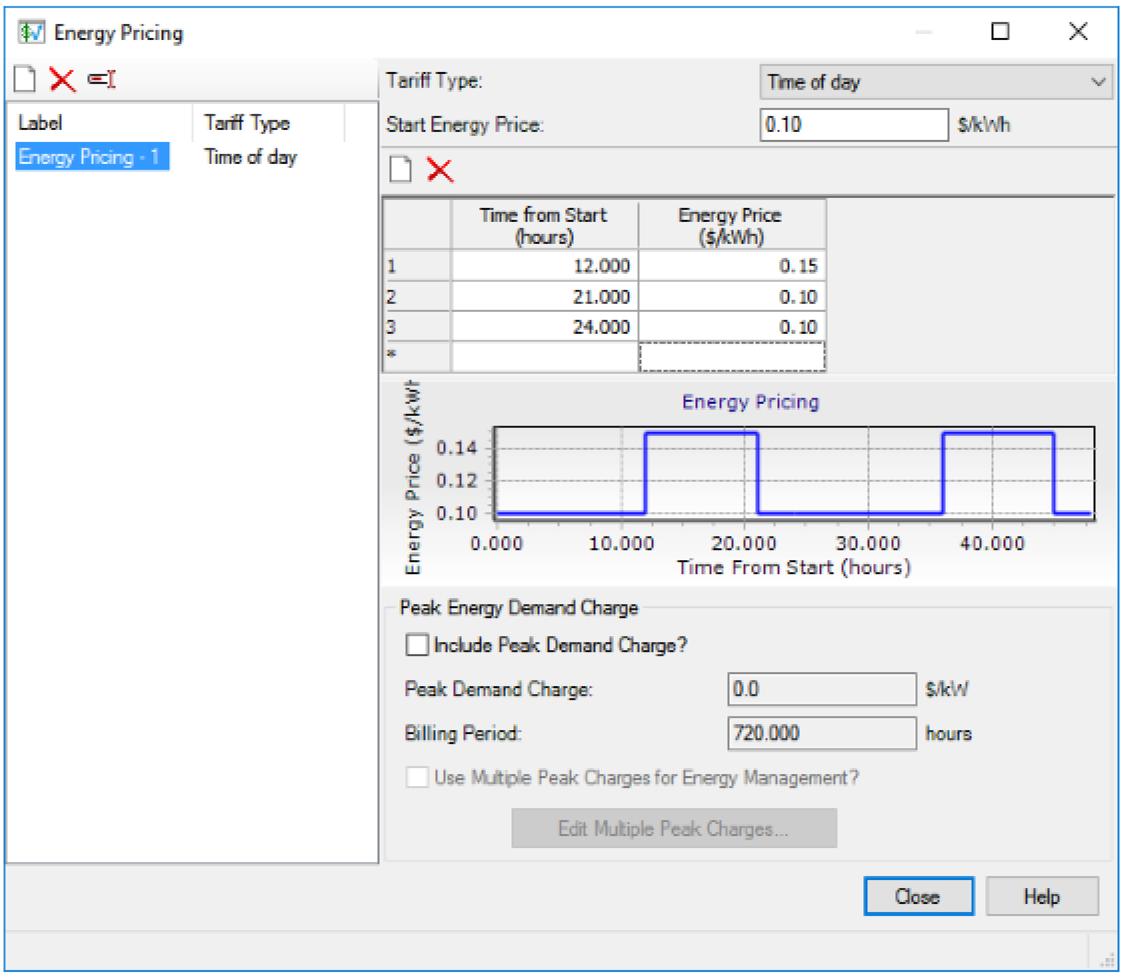
8. Close the graph and the Graphs manager.
9. Save the file as **MyLesson9.wtg**

Step 2: Setting up energy pricing

1. Choose **Analysis > Energy Cost > Scenario Energy Cost**.
2. Click **Energy Pricing** .

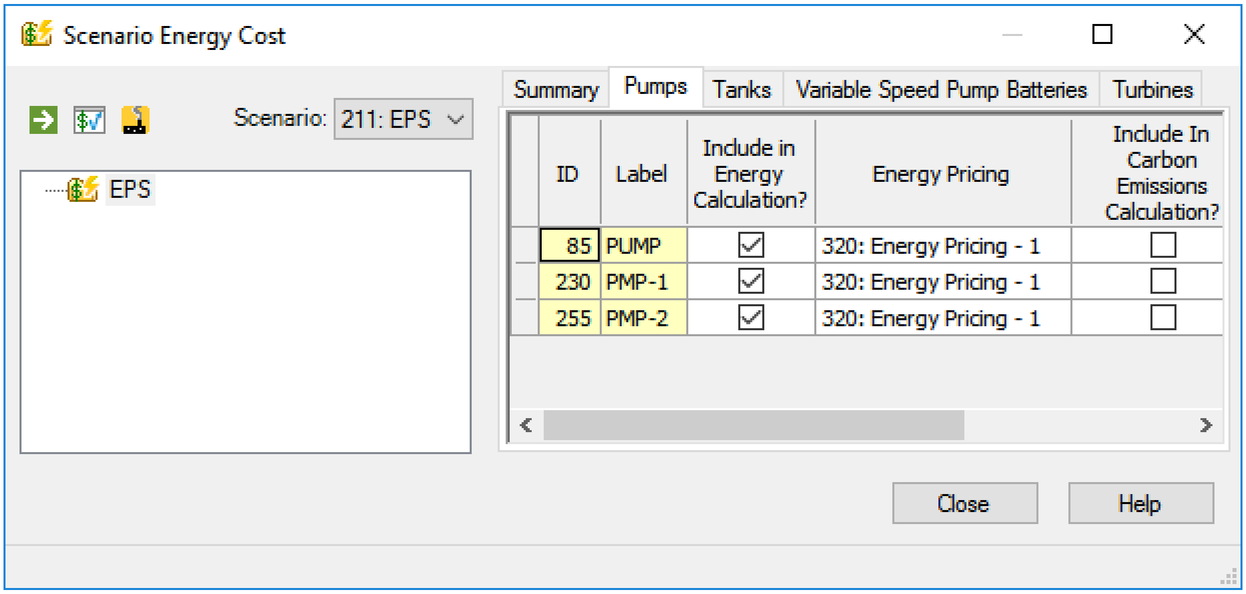
3. Set the start energy price to **0.10 $/kWh**. Use the Add icon on the right side of the dialog to add entries to the energy price table and enter the energy price data shown in the table below.

Time From Start	Energy Price ($/kWh)
12	0.15
21	0.10
24	0.10



4. Click **Close**.
5. In the Scenario Energy Cost manager, select **EPS** from the **Scenario** menu.

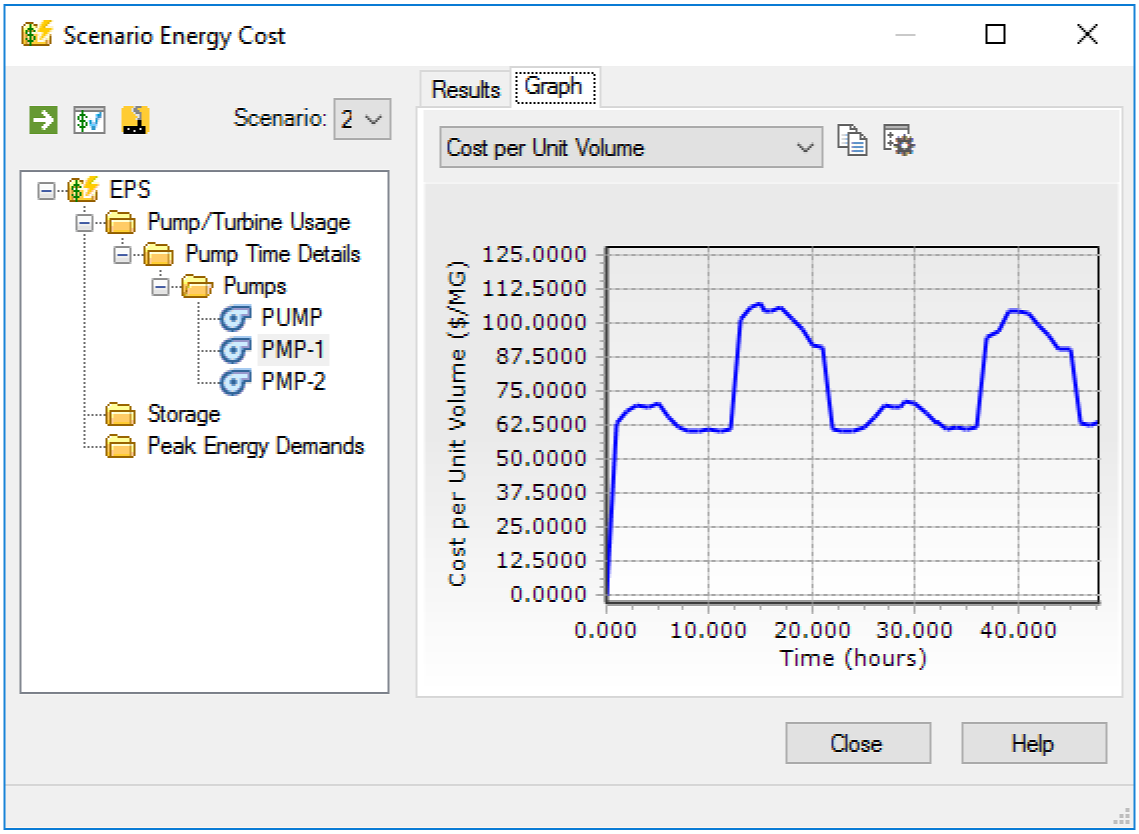
6. In the **Pumps** tab, check the **Include in Energy Calculation?** boxes for each of the pumps. Select **Energy Pricing -1** in the Energy Pricing column for all pumps.
7. Click the **Tanks** tab. Make sure the **Include in Energy Calculation?** boxes are checked for both tanks.



Step 3: Run the energy cost analysis

1. In the Scenario Energy Cost manager, click Compute  .
2. Highlight the **Pump/Turbine Usage** entry in the list. You can see that the efficiency of the constant speed **PUMP** is higher than that of the variable speed P**MP-1** and PMP-2 was not called during this run.

3. Highlight **PMP-1** and click the **Graph** tab. Change the attribute being graphed to **Cost per Unit Volume** and see how the cost changes as a result of pump status and time of day energy charges.

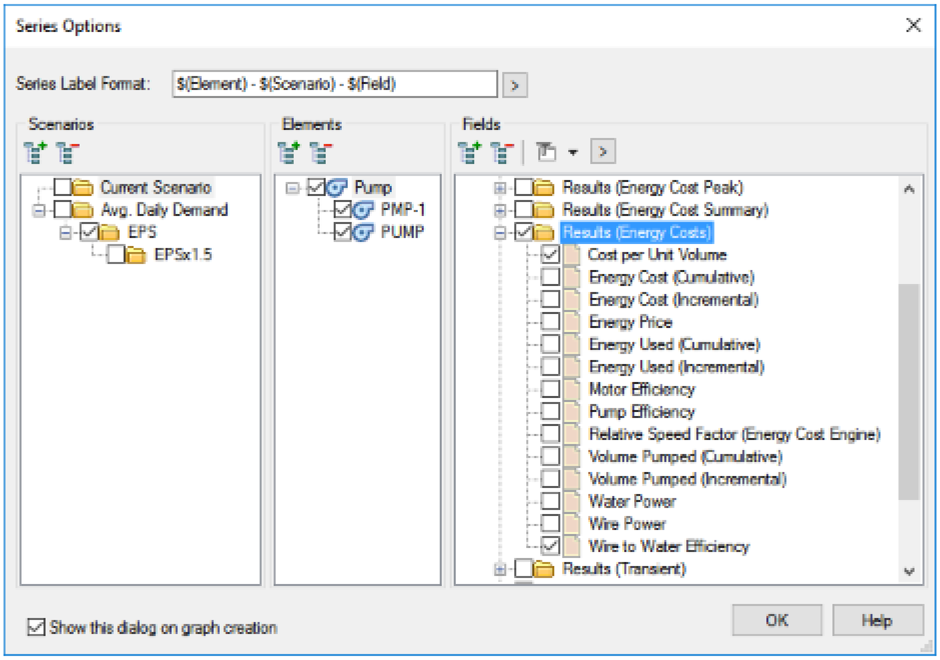


4. Close the Energy Cost manager.

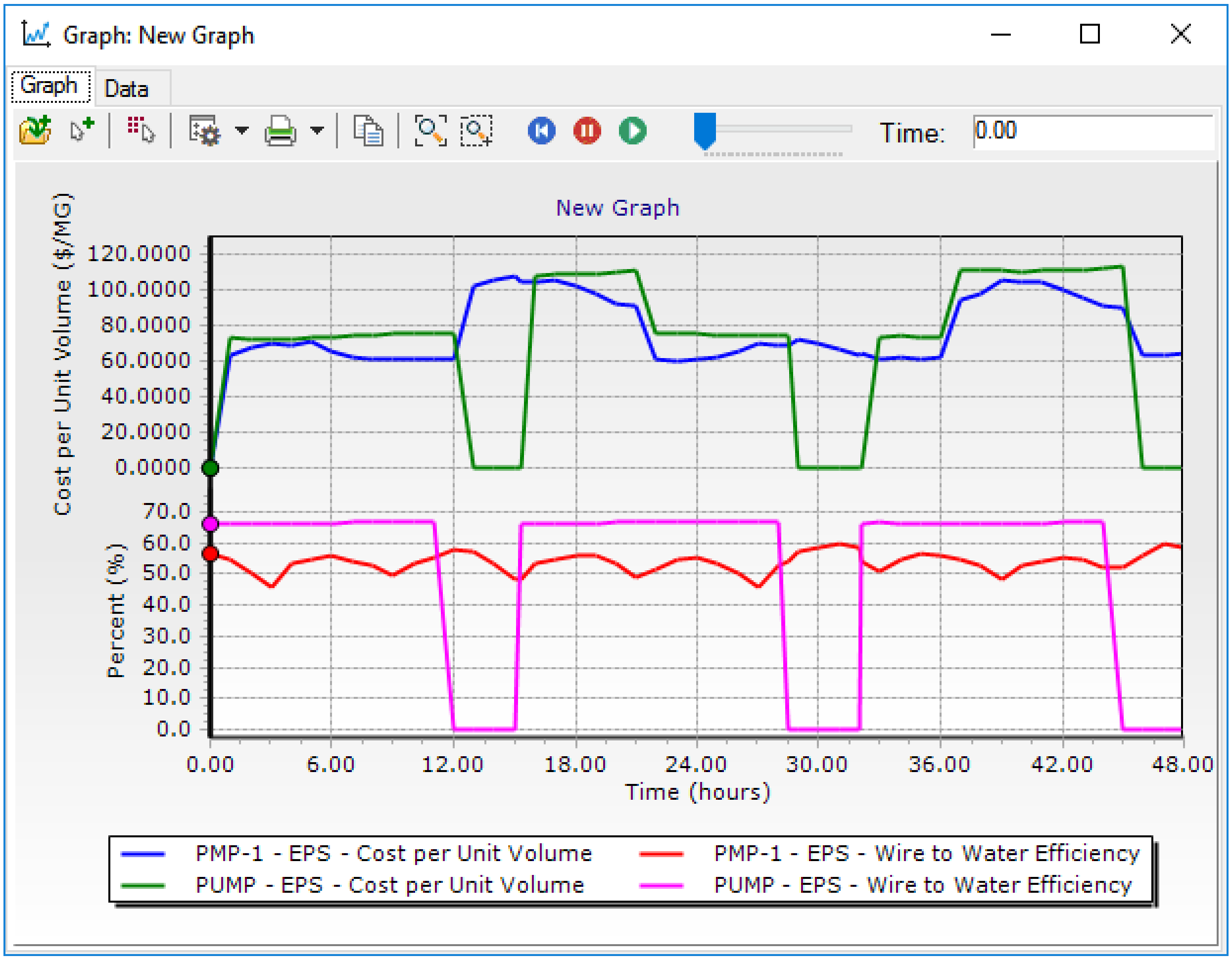
Step 4: Making graphical comparisons between pumps

1. In the drawing, select **PMP-1** and then, while holding down the **<Ctrl>** key, click on the **PUMP** element. Right-click and select **Graph** to open the Series Option manager.
2. Un-check the **Flow (Total)** checkbox and expand the **Results (Energy Costs)** category (click the + button)

3. Check the **Wire to Water Efficiency** and **Cost per Unit Volume** boxes.



4. Click **OK** to open the graph.



The efficiency of the constant speed pump is higher than the variable speed pump whenever it is on. The cost per volume pumped is comparable since the PUMP usually pumps against a higher head. In order to view the head attribute, click on Series Options and check the **Pump Head** box under the Results folder.

5. PUMP pumped into a pressure zone that required a higher pump head.

6. Click the **Add to Graph Manager** button to save the graph, enter a name and click **OK**, and then close the graph window.

Pressure Dependent Demands

Pressure dependent demands (PDD) are used to simulate situations where a change in pressure affects the quantity of water used.

To use PDD

1. Set up a model.
2. Create a PDD function.
3. Create a scenario that assigns a PDD function to an alternative.
4. Run the scenario.

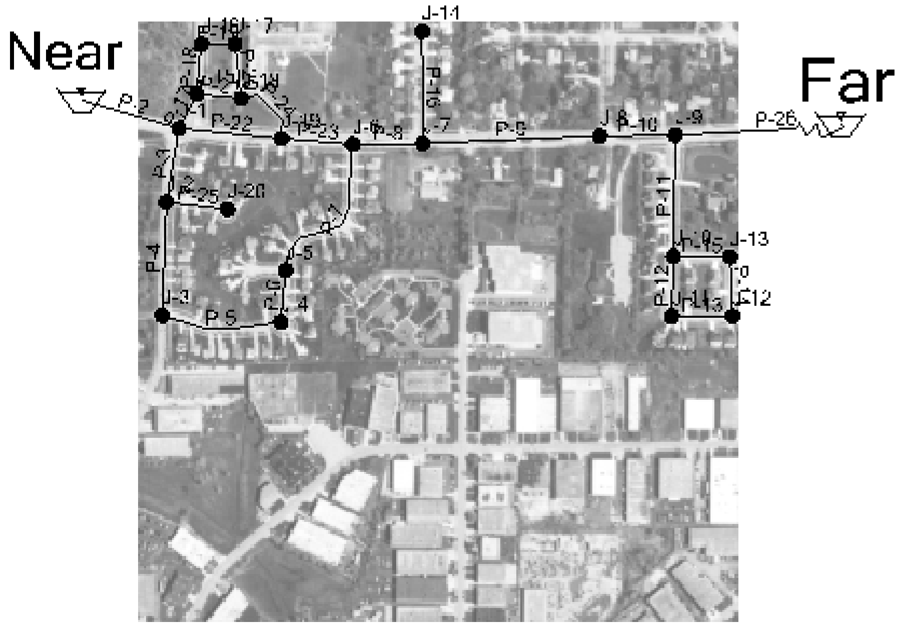
This lesson uses the example of a neighborhood that receives water from two sources, reservoirs that are near and far and both have a hydraulic grade of 150 ft. In this lesson, you will simulate the system without considering PDD and all elements operating. Then the analysis will be run with PDD. In order to simulate a situation where pressure significantly drops, the Near source is taken out of service and the behavior with and without consideration of PDD is made.

The starter file consists of a model with two non-PDD scenarios, SteadyNoPDD and EPSNoPDD. The demands have been loaded and the diurnal demand function has been created.

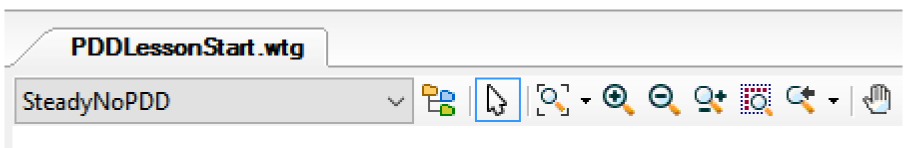
Step 1: Run the initial NoPDD Model

1. Open the **PDDLessonStart.wtg** file in the Lessons directory.
2. Click **OK** to the message dialog that comes up about CONNECTED Projects.
3. Go to **File > Save As** and name the file **MyPDDLesson.**

4. The Near source is on the left and the Far source is on the right.

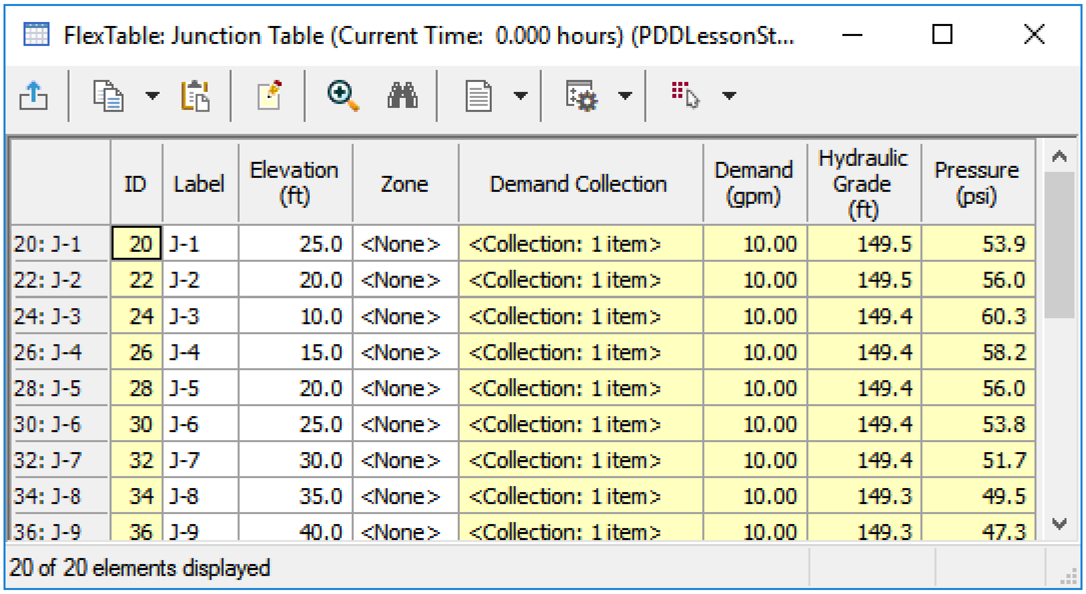


5. Verify the current scenario is **SteadyNoPDD**.



6. Compute the model  and make sure results are green, then close the Calculation Summary.

7. Click **Report** > **Junction**.

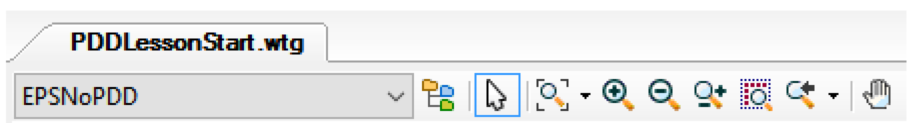


	ID	Label	Elevation (ft)	Zone	Demand Collection	Demand (gpm)	Hydraulic Grade (ft)	Pressure (psi)
20: J-1	20	J-1	25.0	<None>	<Collection: 1 item>	10.00	149.5	53.9
22: J-2	22	J-2	20.0	<None>	<Collection: 1 item>	10.00	149.5	56.0
24: J-3	24	J-3	10.0	<None>	<Collection: 1 item>	10.00	149.4	60.3
26: J-4	26	J-4	15.0	<None>	<Collection: 1 item>	10.00	149.4	58.2
28: J-5	28	J-5	20.0	<None>	<Collection: 1 item>	10.00	149.4	56.0
30: J-6	30	J-6	25.0	<None>	<Collection: 1 item>	10.00	149.4	53.8
32: J-7	32	J-7	30.0	<None>	<Collection: 1 item>	10.00	149.4	51.7
34: J-8	34	J-8	35.0	<None>	<Collection: 1 item>	10.00	149.3	49.5
36: J-9	36	J-9	40.0	<None>	<Collection: 1 item>	10.00	149.3	47.3

20 of 20 elements displayed

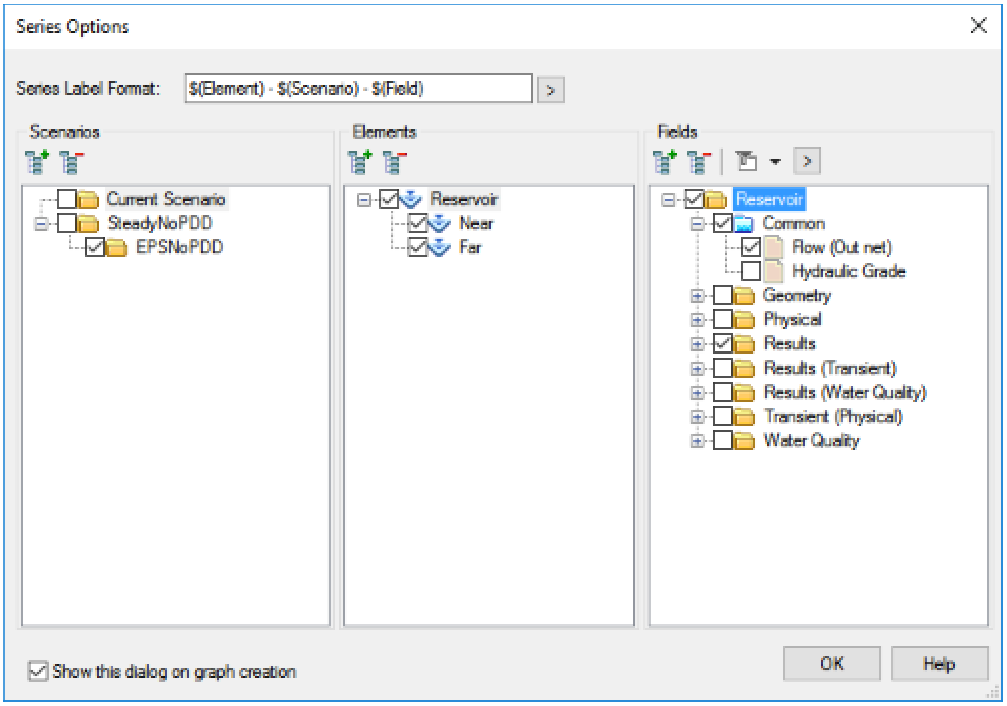
Note that the pressures range from 43 to 60 psi.

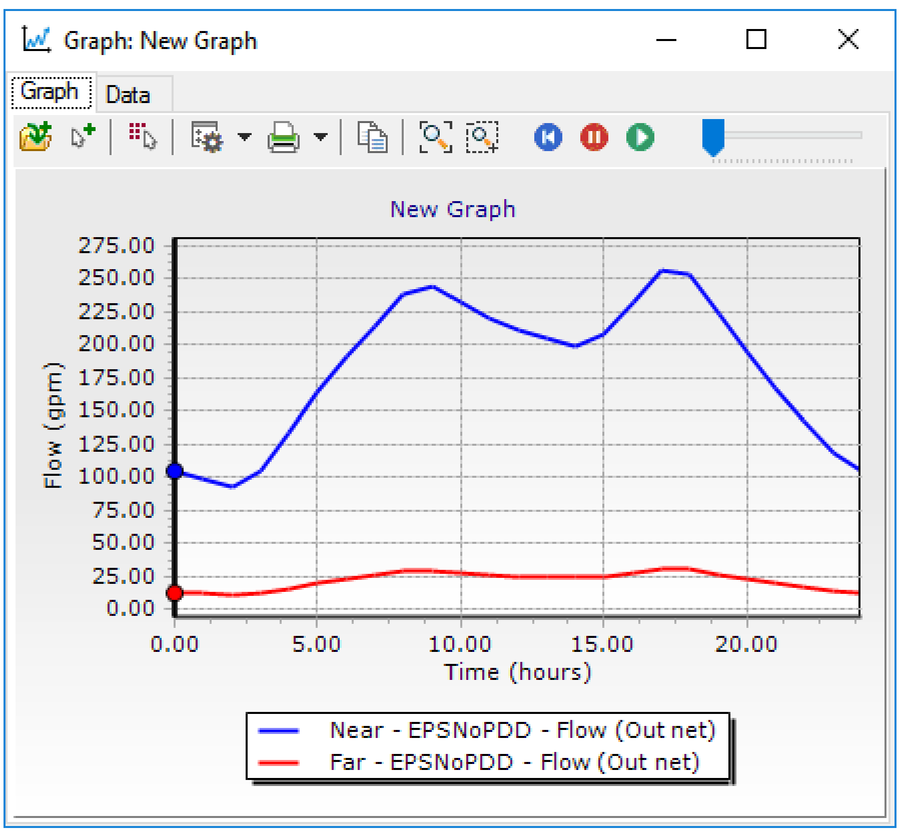
8. Close the FlexTable.
9. Make **EPSNoPDD** the current scenario.



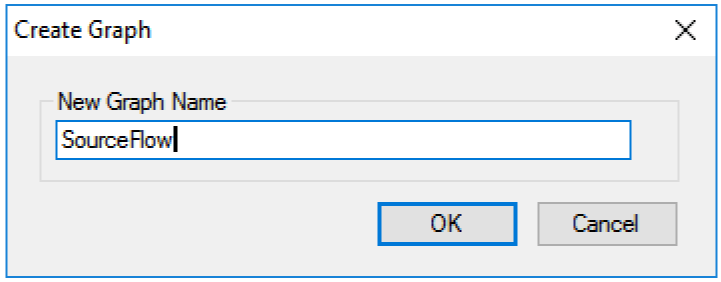
10. Compute the scenario  , make sure results are green, then close the Calculation Summary.
11. In the drawing, hold the **<Ctrl>** key and click the **Near** reservoir, then the **Far** reservoir, and then right-click and select **Graph**.

12. Uncheck **Hydraulic Grade,** then check **Flow (Out net)** and then click **OK** to view the graph.

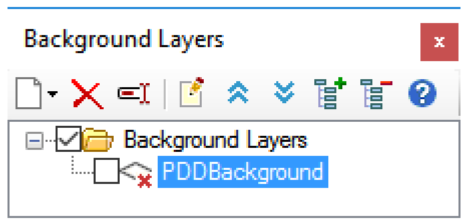




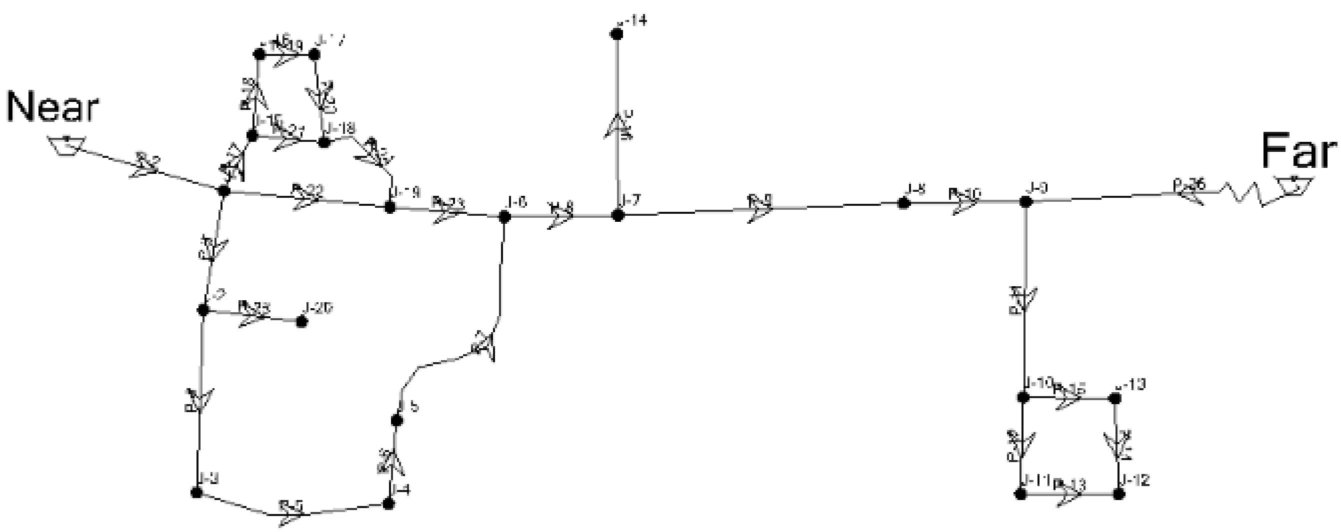
13. Click **Add to Graph Manager**  to save the graph and name it **SourceFlow**.



14. Click **OK** and then close the graph.
15. If you want to turn off the background layers of the drawing choose View > Backgrounds and turn off PDD Background.

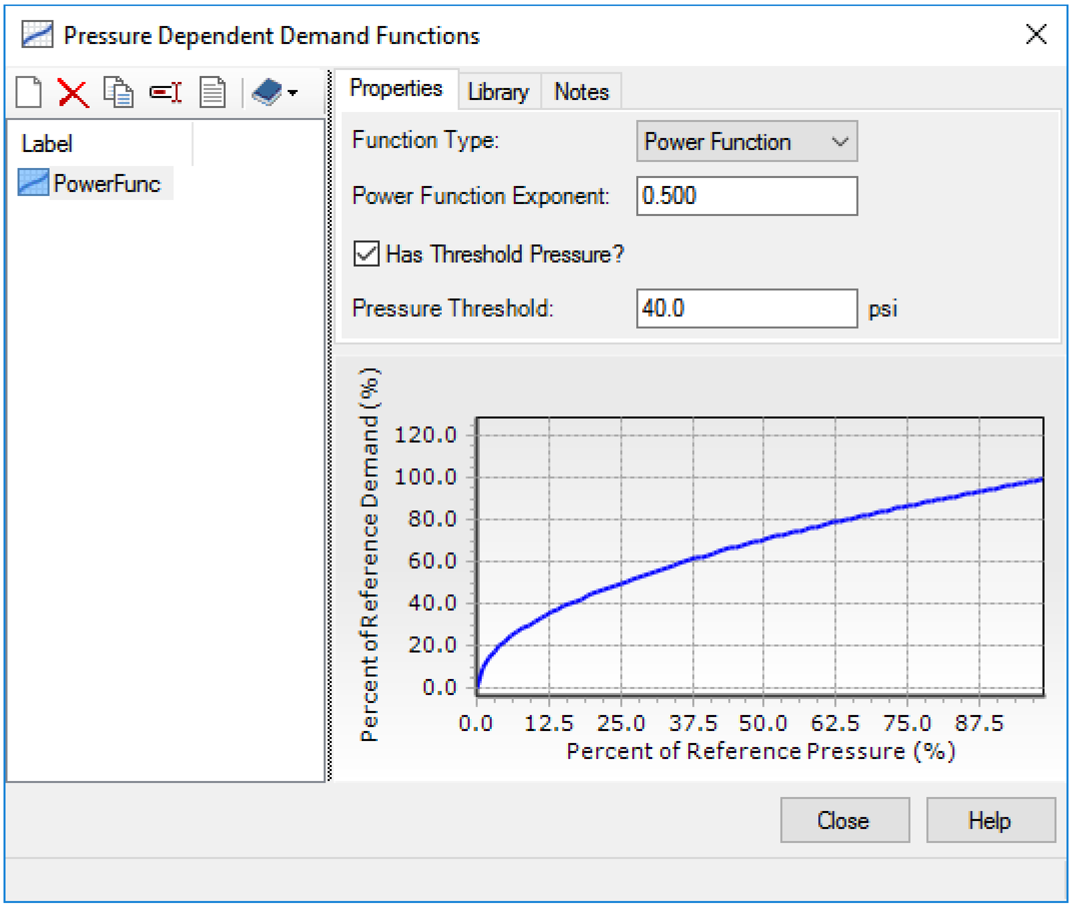


and the drawing will look like the following:



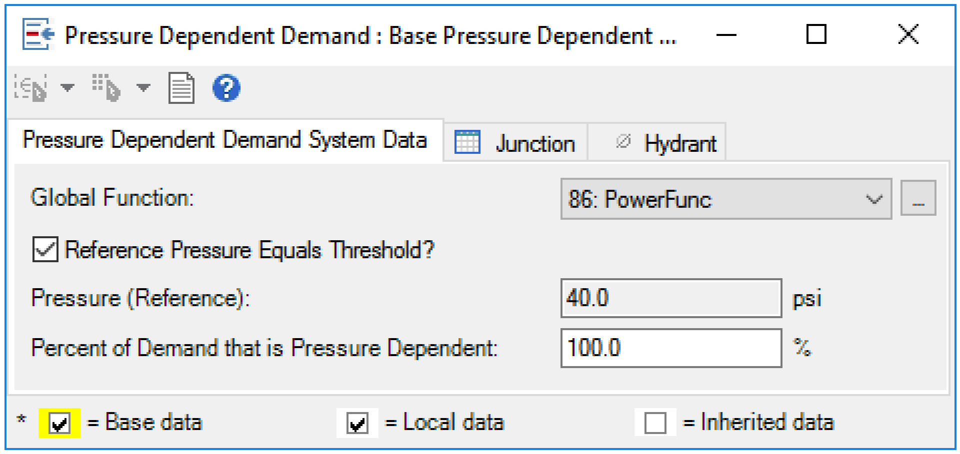
Step 2: Setting up PDD function

1. Choose **Components** > **More** > **Pressure Dependent Demand Functions.** Click **New** and then rename to **PowerFunc.**
2. **Has Threshold Pressure?** should be checked and type in **40 psi** for the pressure threshold.



3. Close the PDD Function manager.
4. Choose **Analysis** > **Alternatives** and click the **Pressure Dependent Demand** Alternative and double-click the **Base Pressure Dependent Demand** Alternative to edit it..

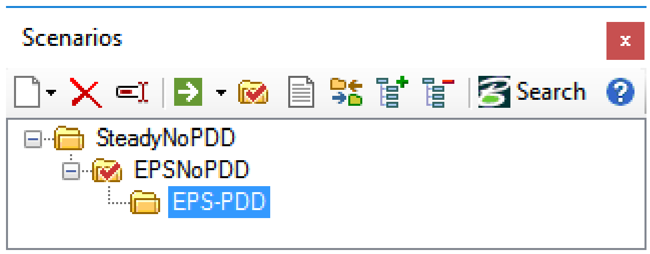
5. Select **PowerFunc** from the Global Function menu



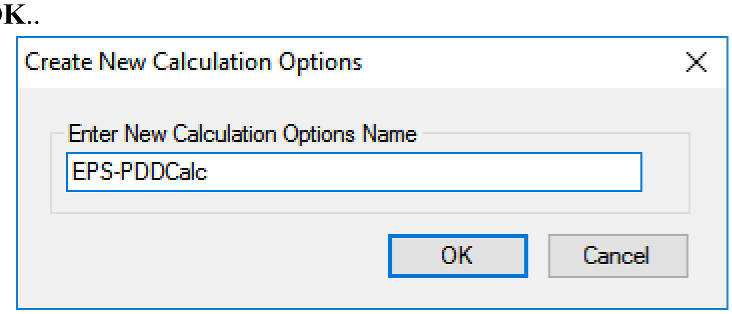
6. Click Close.

Step 3: Run the model with PDD

1. Choose **Analysis** > **Scenarios** and create a child scenario of **EPSNoPDD**.
2. Right-click on **EPSNoPDD** > **New** > **Child Scenario** and rename it **EPS-PDD**

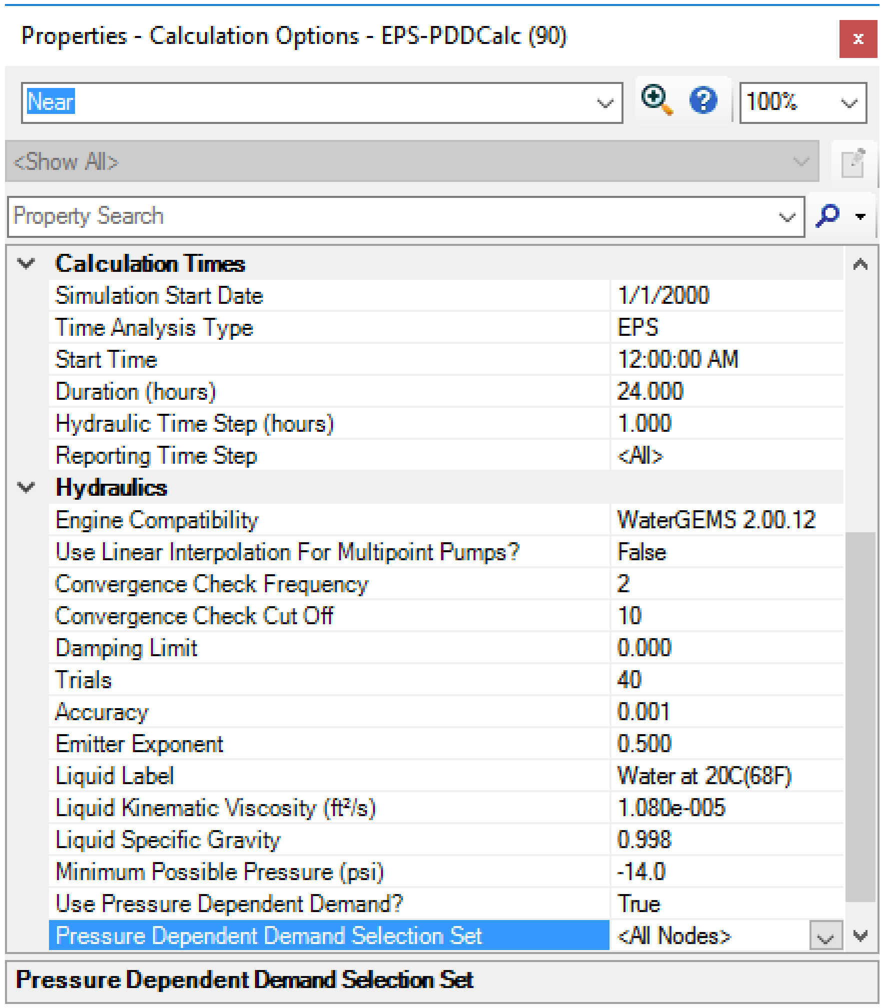


3. Double-click on the **EPS-PDD** scenario to open the Scenarios Property Editor. Under Calculations Options, click the **Steady State/EPS Solver Calculation Options** menu and select **New**. Rename the new option **EPS-PDDCalc** and then click **OK**..



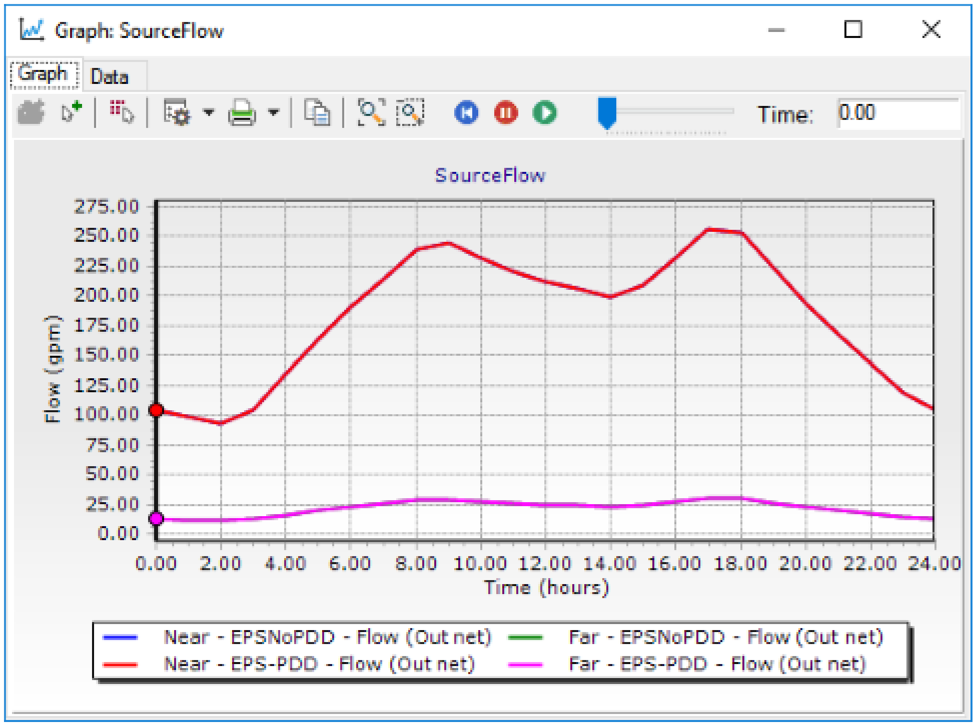
4. Make **EPS-PDD** the current scenario. 

5. Choose **Analysis > Options** and double-click on **EPS-PDDCalc** to open the Property Editor.
6. Set **Time Analysis Type** to **EPS**
   Set **Use Pressure Dependent Demand?** to **True.**
   Set **Pressure Dependent Demand Selection** to **<All Nodes>**



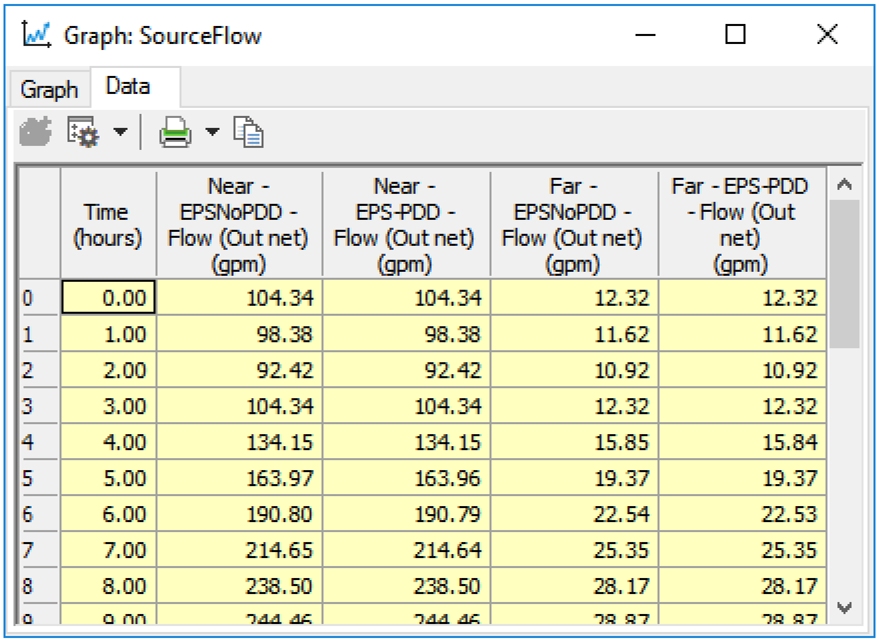
7. In the Scenarios manager, make sure the **EPS-PDD** scenario is current, then click **Compute**.
8. Review the calculation summary and then close it.
9. Review the results by plotting a graph of flow vs. time. Choose **View > Graphs** and double-click on **SourceFlow** graph.

10. Click **Series Options**  and check both **EPSNoPDD** and **EPS-PDD** Scenarios and then **OK**.



There are four lines on the graph but only two are visible. This is because the lines for both scenarios are identical.

11. Click the Data tab to see that the pressure did not drop below the reference pressure during the run.

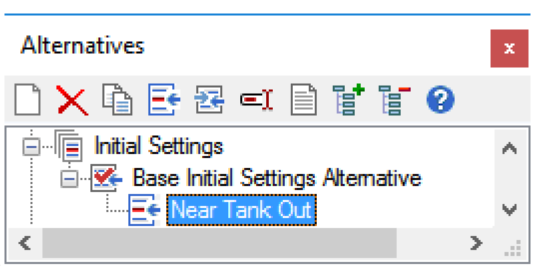


	Time (hours)	Near - EPSNoPDD - Flow (Out net) (gpm)	Near - EPS-PDD - Flow (Out net) (gpm)	Far - EPSNoPDD - Flow (Out net) (gpm)	Far - EPS-PDD - Flow (Out net) (gpm)
0	0.00	104.34	104.34	12.32	12.32
1	1.00	98.38	98.38	11.62	11.62
2	2.00	92.42	92.42	10.92	10.92
3	3.00	104.34	104.34	12.32	12.32
4	4.00	134.15	134.15	15.85	15.84
5	5.00	163.97	163.96	19.37	19.37
6	6.00	190.80	190.79	22.54	22.53
7	7.00	214.65	214.64	25.35	25.35
8	8.00	238.50	238.50	28.17	28.17
9	9.00	244.46	244.46	28.87	28.87

Step 4: Running non-PDD models with outage

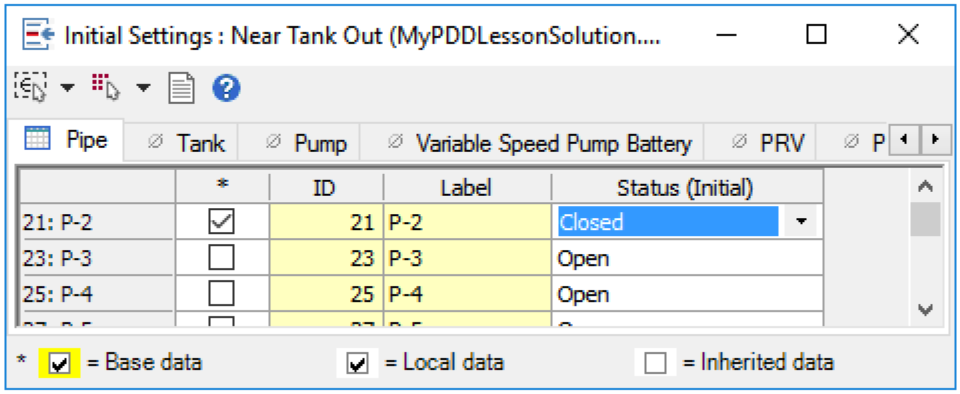
In order to examine the effect of a drop in pressure, create a scenario where the pressures will drop. In this example, Near tank will be taken out of service. Create a new scenario where pipe P-2 is closed.

1. Click **Analysis** > **Alternatives** . Expand the Initial Settings Alternative node and right-click the **Base Initial Settings Alternative.** Select **New > Child Alternative**.
2. Rename to **Near Tank Out**

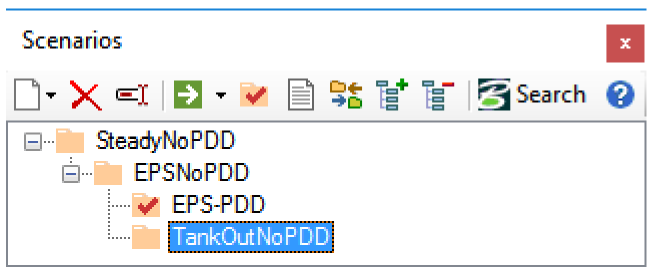


.

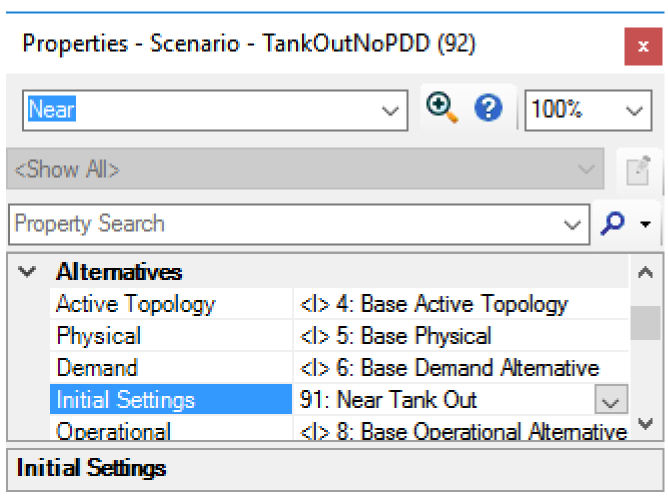
3. Double-click on **Near Tank Out** and change the status of **P-2** to **closed**. When the status has been changed to Closed a check shows in the first column to show that it is different from its parent.



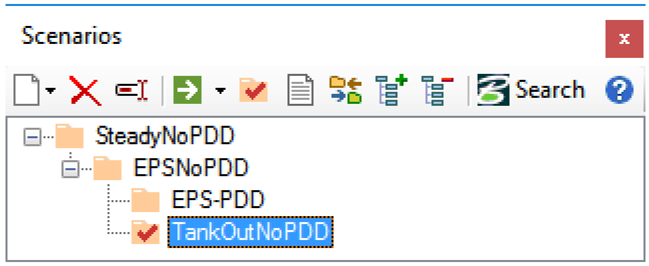
4. Close the alternative editor.
5. In the Scenarios Manager create a **new child scenario** off of **EPSNoPDD** called **TankOutNoPDD**.



6. Double-click the new scenario to open the scenario Property Editor. Change the **Initial Alternative** to **Near Tank Out** and then close the Property Editor.

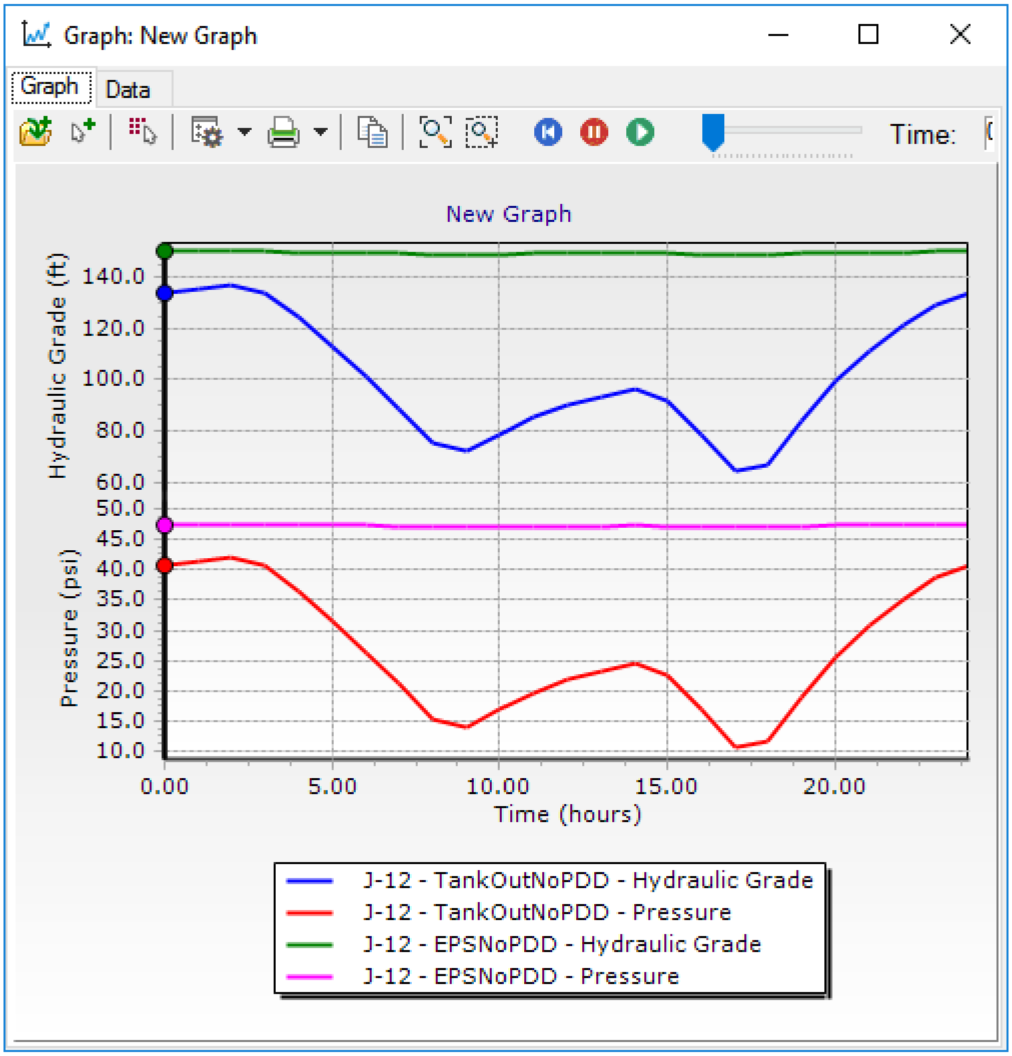


7. Make the **TankOutNoPDD** the current scenario and then click **Compute**.



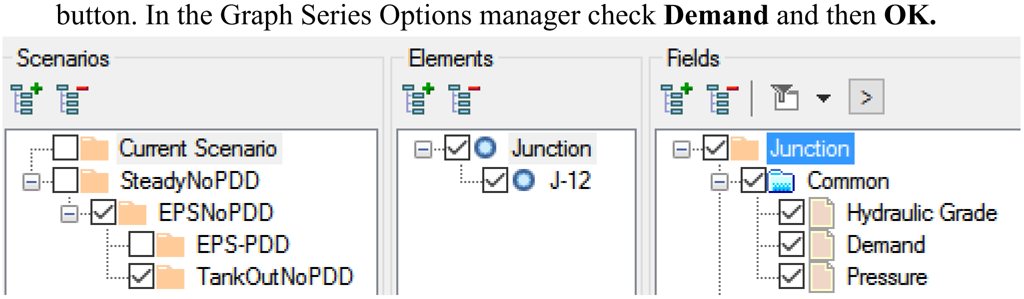
8. Review the calculation summary and then close it.
9. Right-click on **J-12** and select **Graph**.

10. In Series Options check the boxes for the **EPSNoPDD** and **TankOutNoPDD** scenarios. Check the box next to the **Pressure** field (Hydraulic Grade is checked by default; leave it checked) and click **OK**.

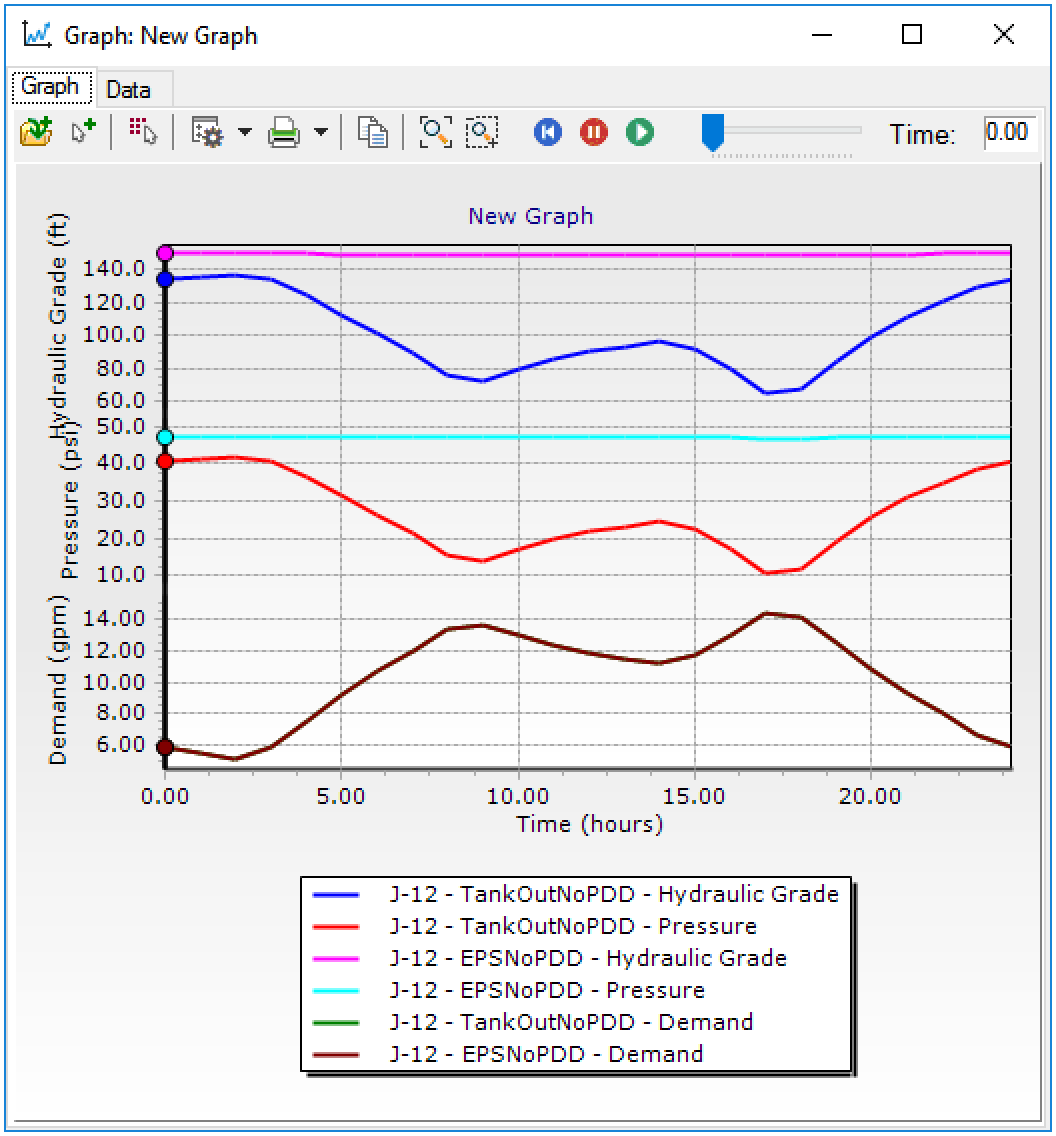


When the **Near** Tank is out of service there is a significant drop in pressure.

11. Examine the effect of the drop in pressure on Demand. Click the **Series Options** button. In the Graph Series Options manager check **Demand** and then **OK.**



12. The demand did not change with pressure because it is not a PDD run, demand is independent of pressure, so there is a single line for Demand. Notice that when flow increases due to the time of day, there is not a corresponding drop in flow because of pressure drop.

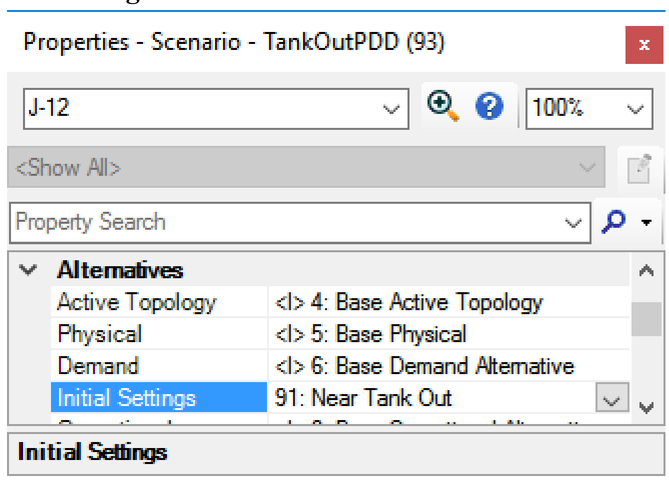


13. Click the **Add to Graph Manager** button, rename the graph as **Pressure Demand J-12** and click **OK**.

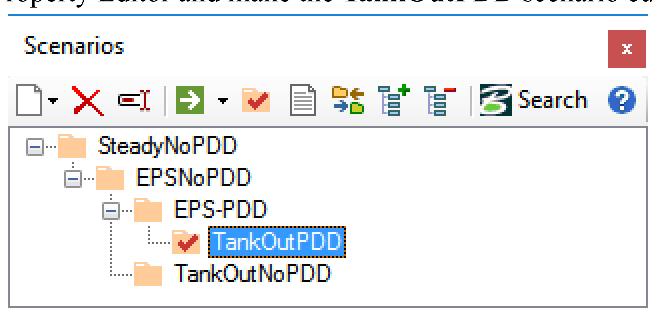
14. Close the graph.

Step 5: Run PDD model with outage

1. Click **Analysis > Scenarios.**
2. Right-click **EPSPDD** and select **New > Child Scenario**. Rename the new scenario **TankOutPDD.**
3. Double-click on **TankOutPDD** to open the scenario Property Editor.
4. Set the **Initial Settings** alternative to **Near Tank Out**

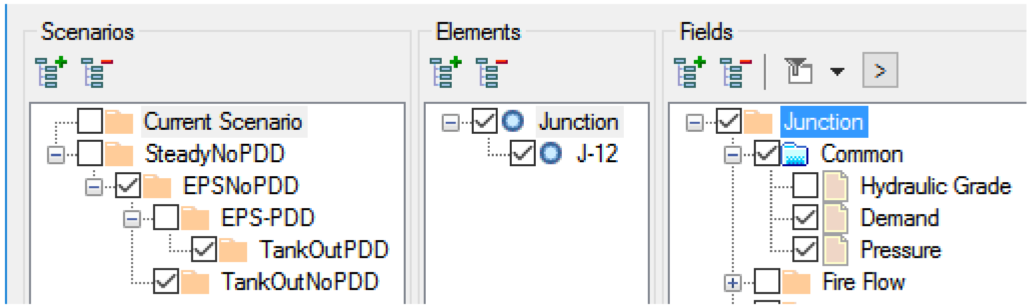


5. Close the Property Editor and make the **TankOutPDD** scenario current.

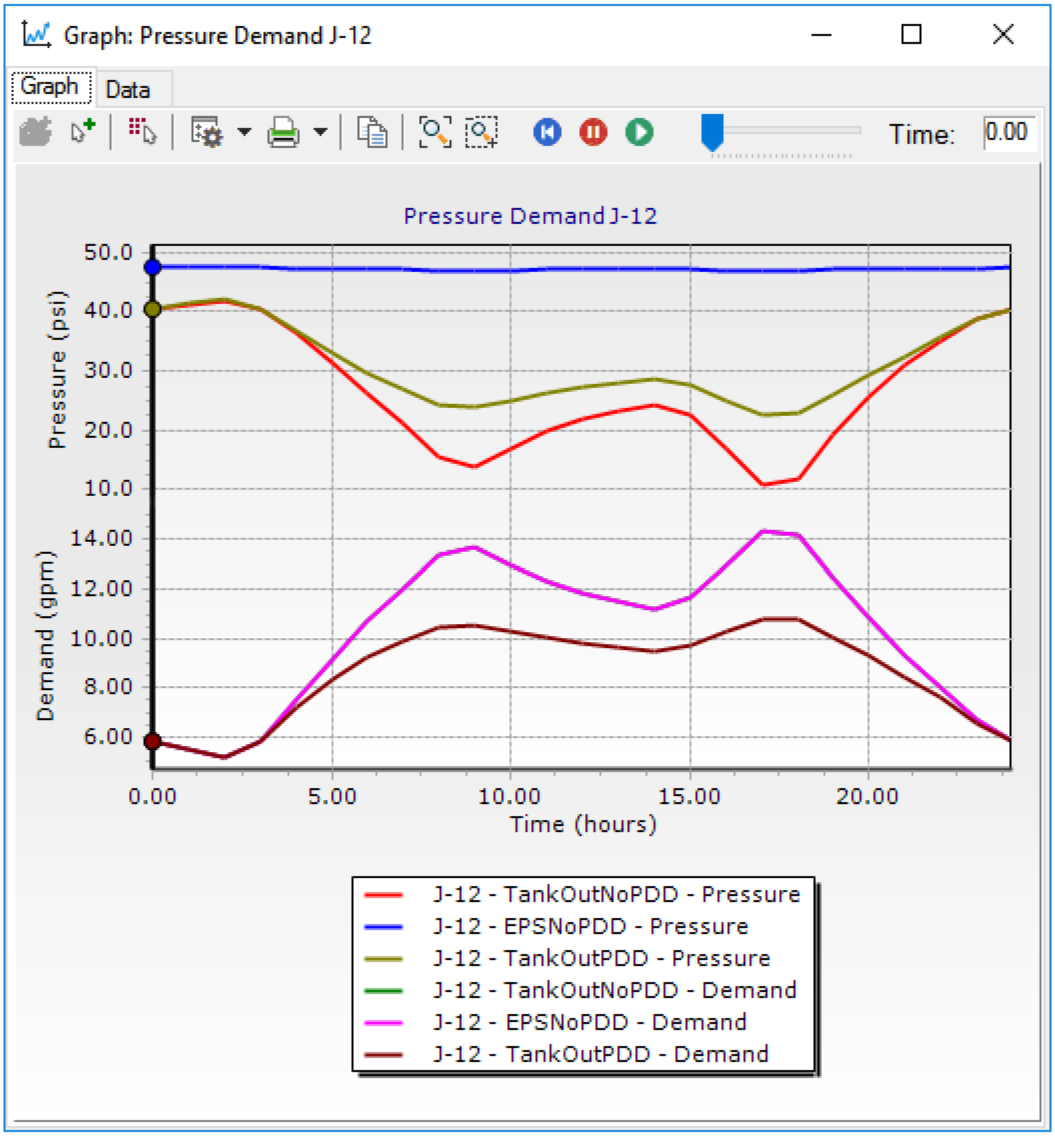


6. **Compute** the scenario, review the calculation summary, and close it.
7. Click **View > Graphs** and open the **Pressure Demand J-12** graph.

8. Click the **Series Options** button and check **TankOutPDD** in the list of Scenarios, uncheck **Hydraulic Grade** in the list of Fields, and then click **OK**.



9. When PDD is used, the demand decreases when the pressure drops, so the overall pressure drop is not as great as when the pressure dependency of demands is ignored.

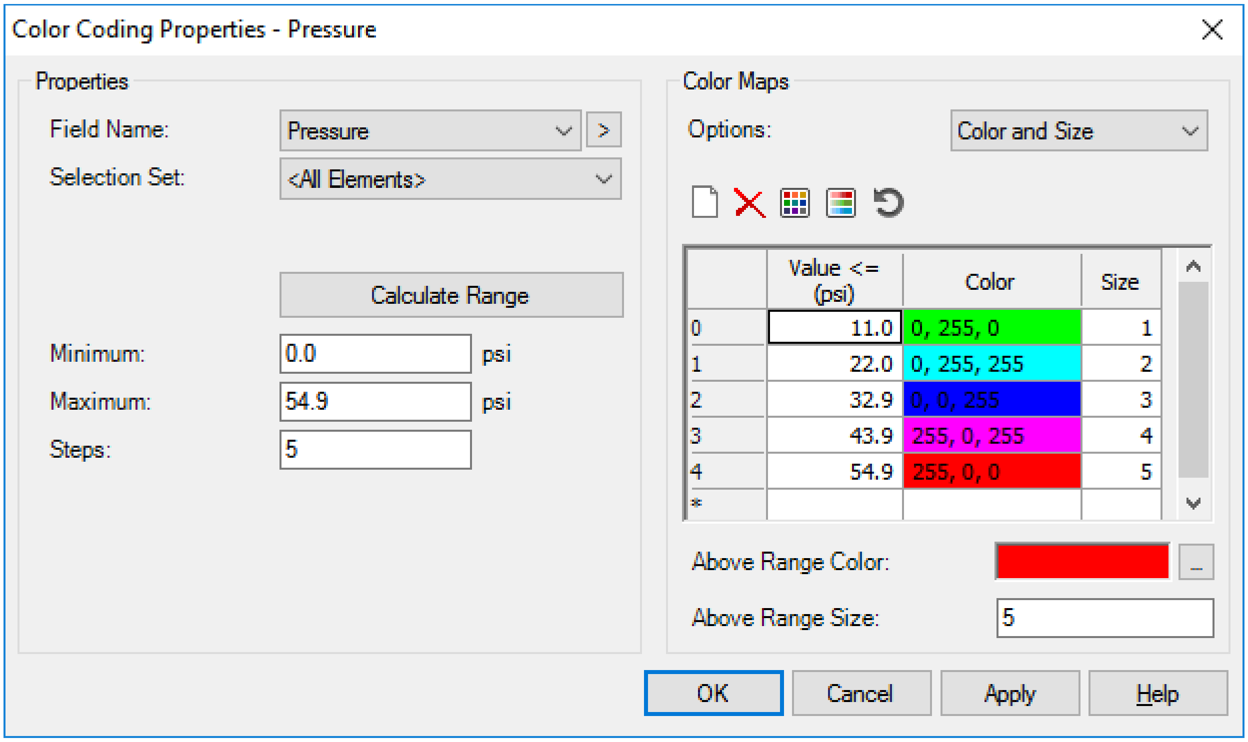


10. Close the graph.

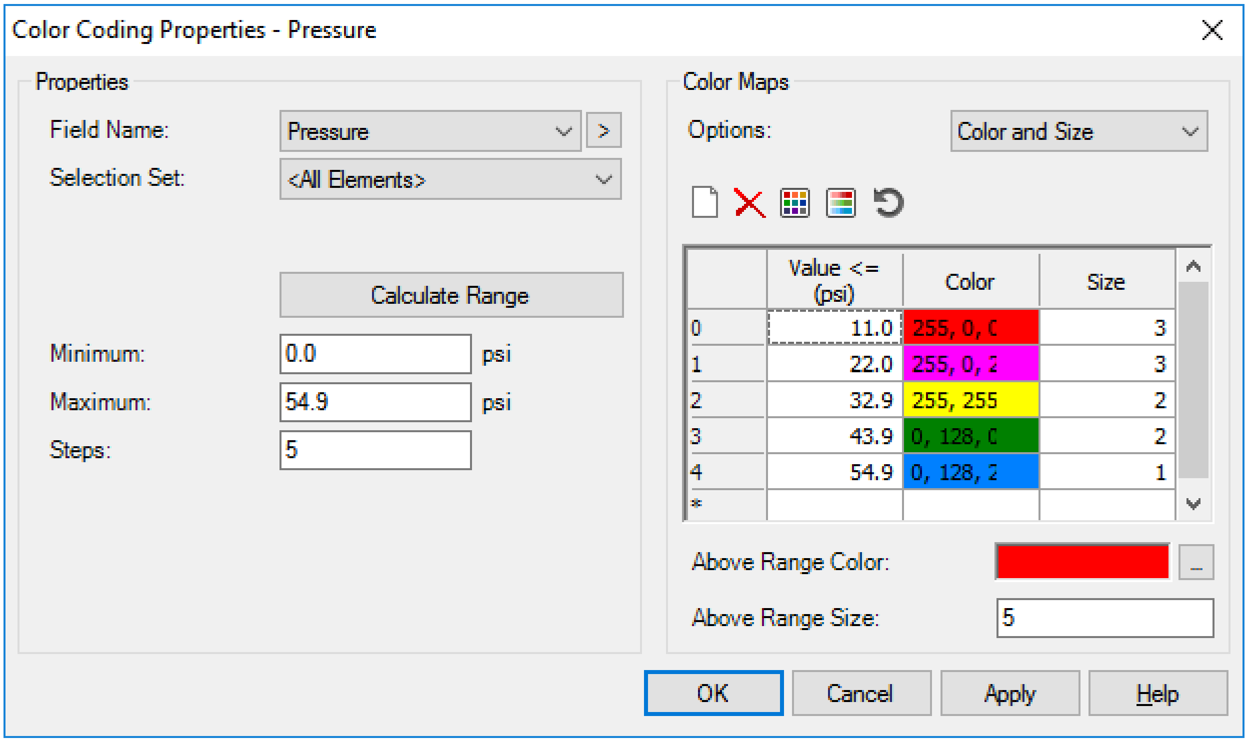
Step 6: Animating Results

1. Click **Analysis** > **Scenarios** and make the **TankOutNoPDD** scenario current.
2. Choose **View** > **Symbology** and expand the Junction entry
3. Right-click on **Junction** and then select **New** > **Color Coding**.

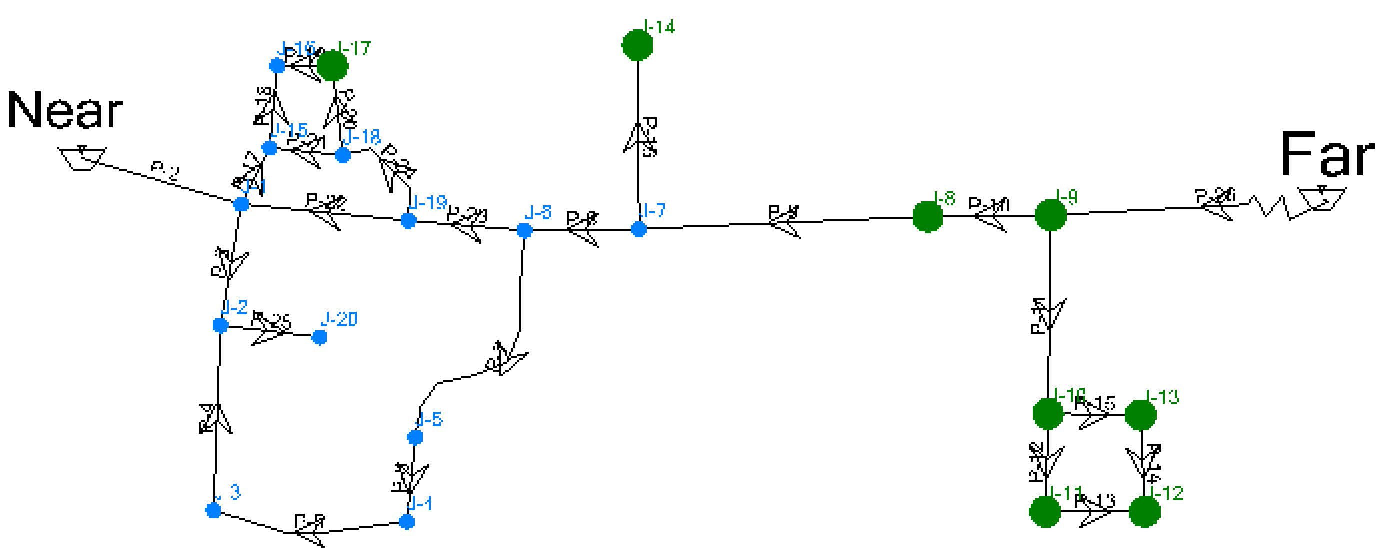
4. Select **Pressure** from the Field Name menu and **Color and Size** from the Options menu.
5. Click **Calculate Range** , select **Full Range** from the submenu, this gives you the pressure ranges for this scenario.
6. Click the **Initialize** button



7. Manually modify the ranges, color and size fields to look like the following example. The colors, in order of appearance are: Red, Magenta, Yellow, Green, and Royal Blue. Change the sizes to 3, 3, 2, 2, and 1 respectively.

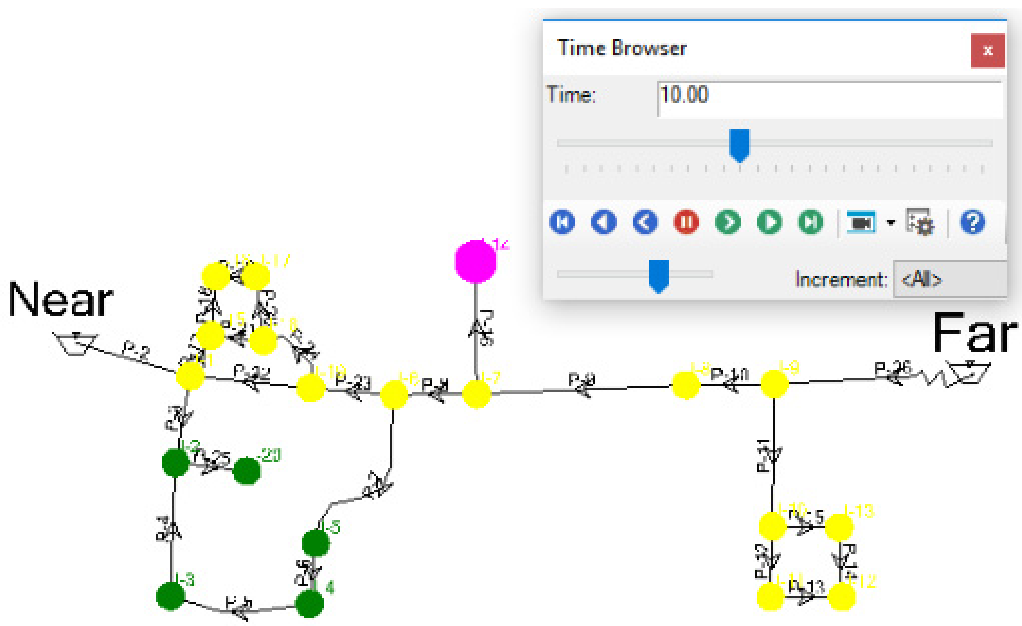


8. Click **OK**.



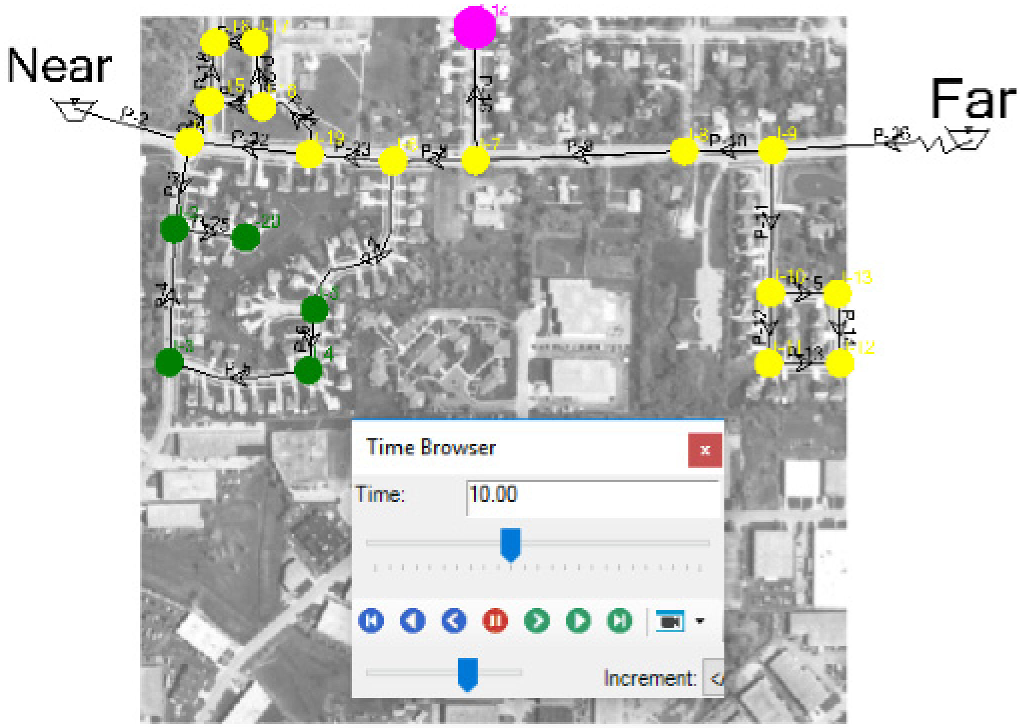
9. Click **Analysis** > **Times** and click **Play** . Observe how the colors and pressures change over the course of a day. Then click Pause .
10. Switch to the **TankOutPDD** scenario.

11. **Compute**, and then close the calculation summary.
12. Click **Play** and observe how the pressures in this run do not drop as low.



13. Pause the animation and click **View** > **Backgrounds** and check the PDDBackground box.

.



14. Close the open dialogs.

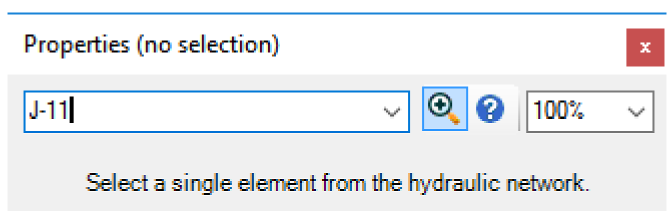
Criticality and Segmentation

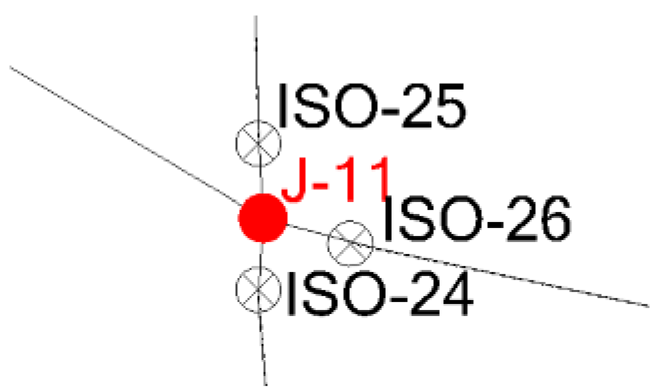
A criticality analysis can be performed on a water model to identify the most critical or important hydraulic elements within the model. Once the options have been set in a Criticality Studies level of the Segmentation and Criticality manager, you must decide which scenario is to be used for the analysis and set the rules for use of valving in the options tab.

This lesson assumes that you have already constructed a model that has isolating valves and that these valves reference pipes and pressure dependent demand functions that have been set up.

Step 1: Check the Isolation Valves

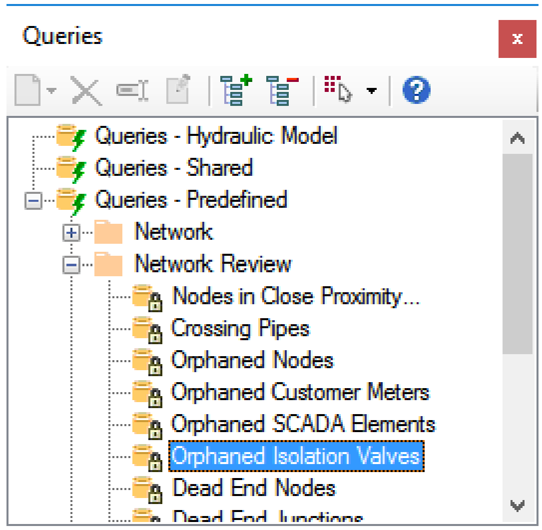
1. Open **CritStart.wtg** from the Lessons folder
2. Click **OK** to the message dialog that comes up about CONNECTED Projects.
3. Go to **File > Save As** and name the file **MyCritSolution**.
4. Use **Pan**  to look at the placement of isolation valves (or hold the middle mouse button to pan)
5. Click **Home > Find Element**  (The top section of the Property Editor contains the Find Element tool). Type **J-11** in the field and then click **Find**.





6. Check for valves not assigned to pipes.

   a. Click **View > Queries**. Under Queries - Predefined, expand the **Network Review** folder and double-click **Orphaned Isolation Valves**.

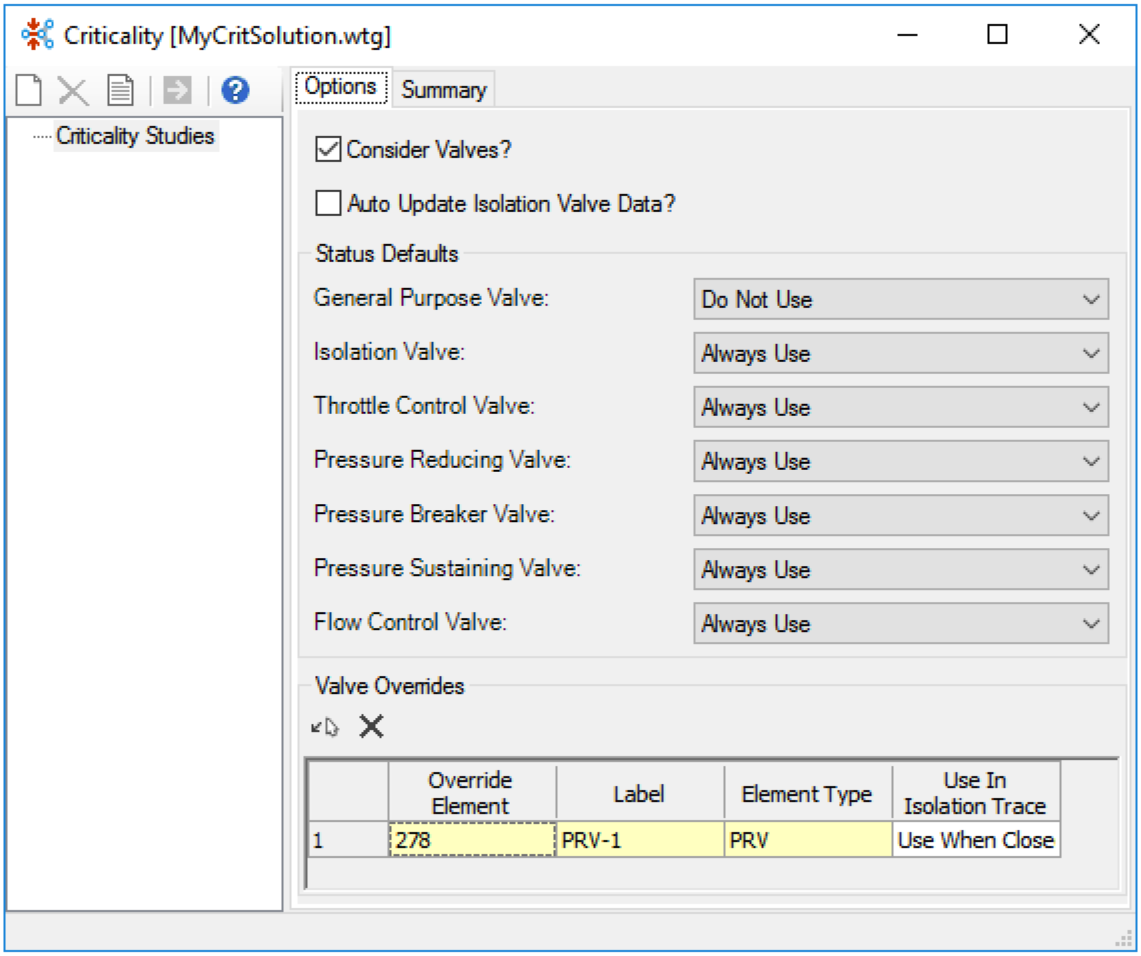


   b. All valves are assigned, however if the query turned up orphaned valves then you could delete the isolation valve, leave it orphaned, or select the valve and choose the Referenced Pipe option in the valve Property Editor and select the pipe where the valve is located.

7. Close the queries manager and the Property Editor.

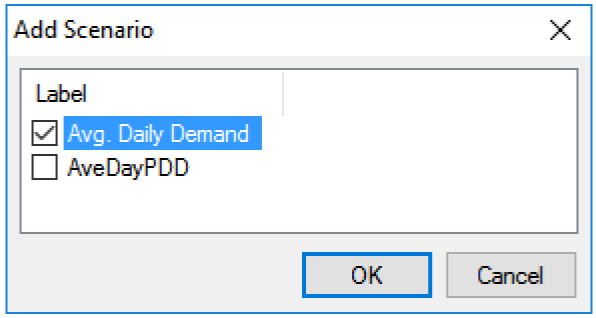
Step 2: Start the Criticality Manager and set up segmentation

1. Choose **Analysis** > **Criticality**

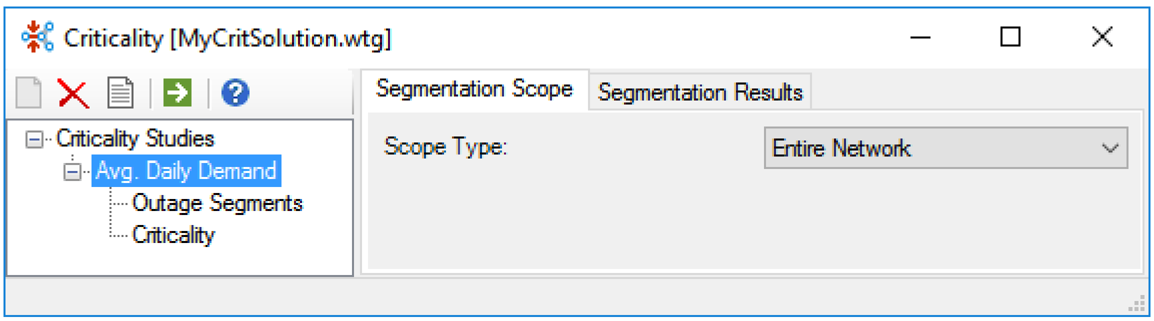


2. In the **Options** tab, verify that **Consider Valves**? is checked and that **Always Use** is selected in the **Isolation Valve** field.

3. Click **New.** In the Add Scenario dialog, check **Avg. Daily Demand** and click **OK**.



4. Select **Entire Network** from the **Scope Type** menu.



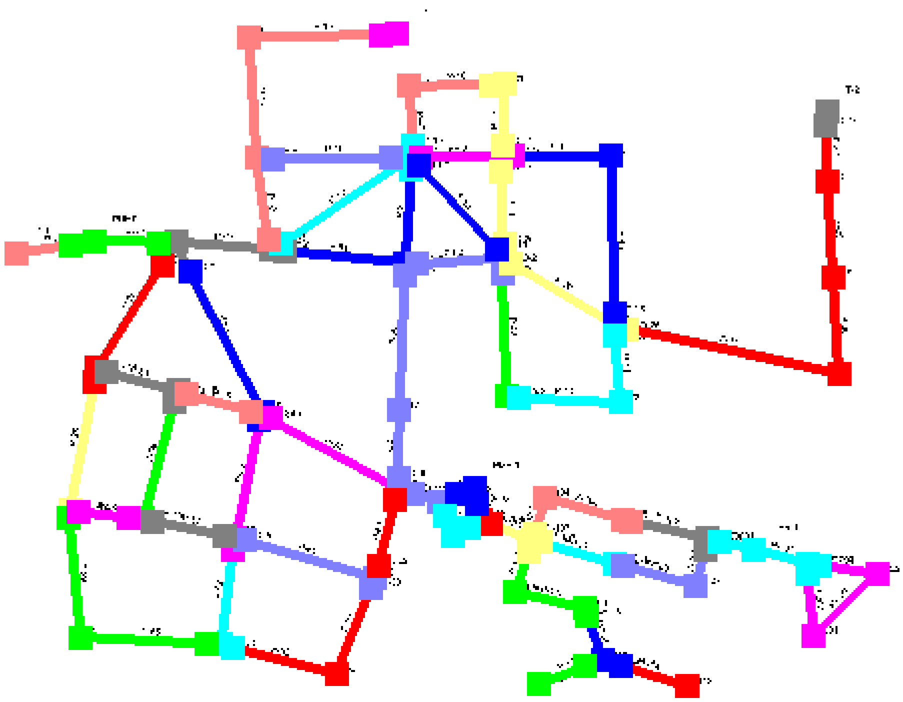
5. Click **Compute**  to perform the segmentation analysis, and click **Yes** at the prompt.

   Label - List of segments that were identified in the analysis. If **Consider Valves?** was not checked, there is one pipe per segment and the label of the pipe is listed next to the segment name. In this case, **Consider Valves?** was checked so the segments consist of a variety of pipes and nodes.

   General statistics are given for each segment.

   Affected Elements - The elements that make up or bound the segment.

6. Click Highlight Segments to view the color coded segments in the drawing.



The results of segmentation can be advantageous. You can identify which segments require successful operation of a large number of valves in order to achieve a shutdown.

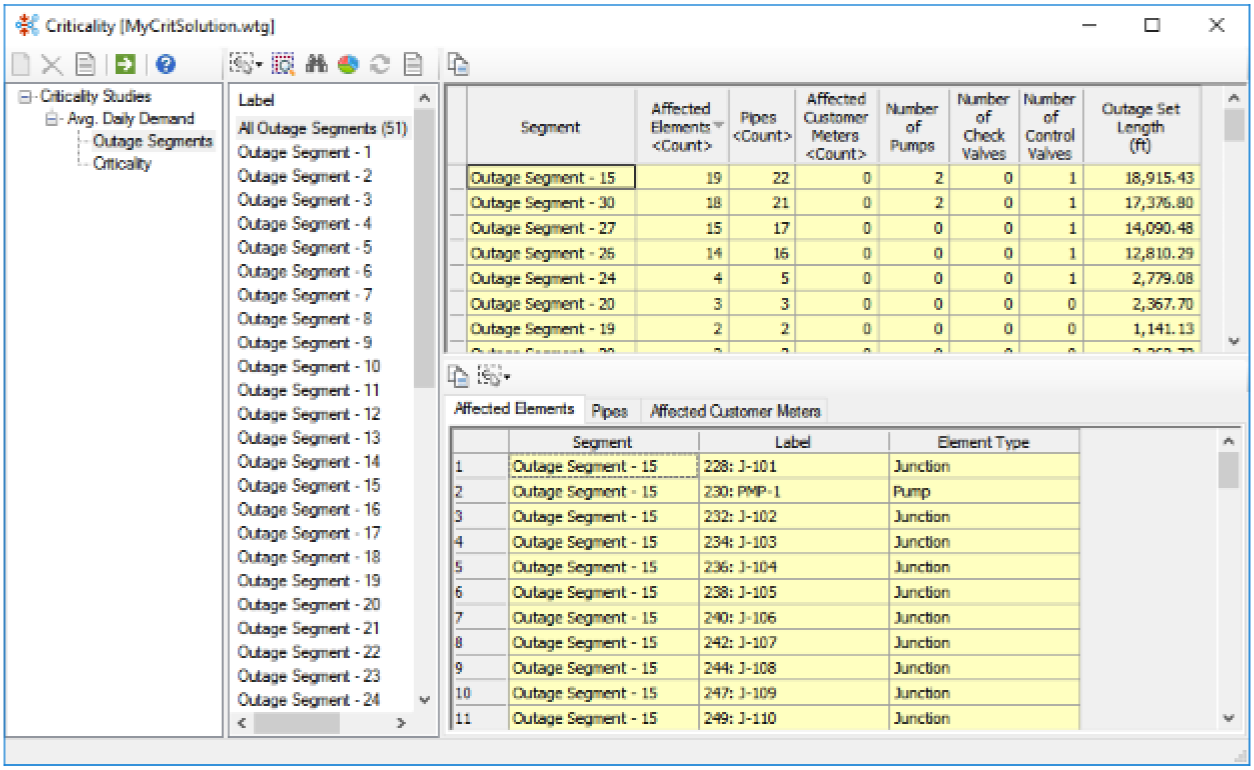
7. Right-click on the **Isolation Nodes <Count>** column and select **Sort > Sort Descending**.

Segment	Affected Elements <Count>	Isolation Nodes <Count>	Pipes <Count>	Affected Customer Meters <Count>	Segment Length (ft)	Fluid Volume of Segment (ft³)	Segment Color
Segment - 7	2	6	7	0	1,216.33	250.70	28,128,128
Segment - 13	1	6	6	0	1,457.84	325.26	0,255,255
Segment - 5	1	5	5	0	901.04	276.16	28,128,255
Segment - 2	1	4	4	0	1,814.44	514.51	0,0,255
Segment - 6	1	4	4	0	640.69	125.80	255,0,255
Segment - 11	1	4	4	0	1,110.33	374.50	0,0,255

The segments at the top of the list usually prove to be the most difficult to isolate and may require investigation to make them less susceptible to issues that arise due to an inoperative valve.

Step 3: Perform outage analysis to identify if isolating a segment causes other segments to be isolated

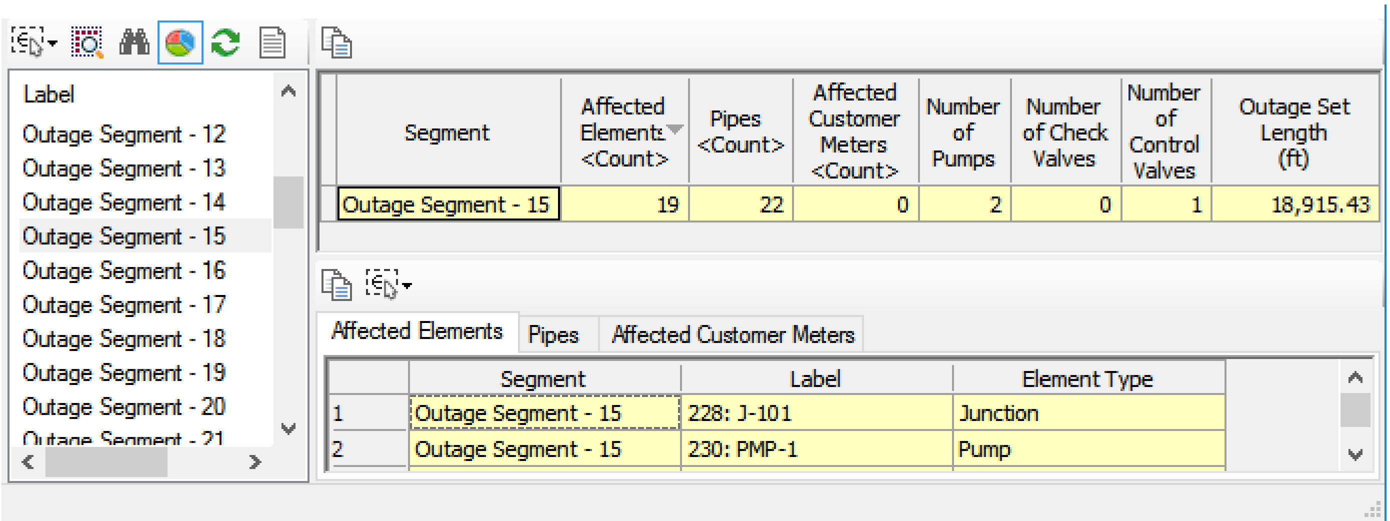
1. Click on **Outage Segments** and then **Compute**  .. Click **Yes** at the prompt.



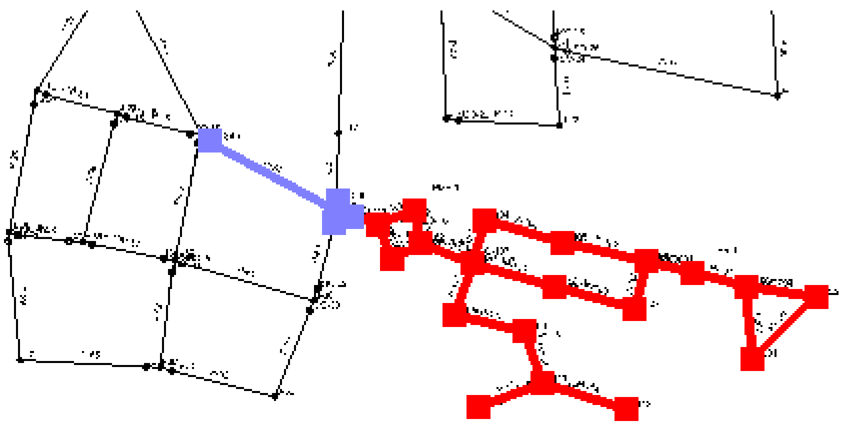
2. Right-click on **Outage Set Length > Sort > Sort Descending** to find out which segments have outages that will cause significant downstream outages.

Segment	Affected Elements <Count>	Pipes <Count>	Affected Customer Meters <Count>	Number of Pumps	Number of Check Valves	Number of Control Valves	Outage Set Length (ft)
Outage Segment - 15	19	22	0	2	0	1	18,915.43
Outage Segment - 30	18	21	0	2	0	1	17,376.80
Outage Segment - 27	15	17	0	0	0	1	14,090.48
Outage Segment - 26	14	16	0	0	0	1	12,810.29
Outage Segment - 24	4	5	0	0	0	1	2,779.08

3. Select the Outage Segment with the **highest Outage Set Length** from the **Label** column. Click Highlight Segments  to view the color coded segments in the drawing.



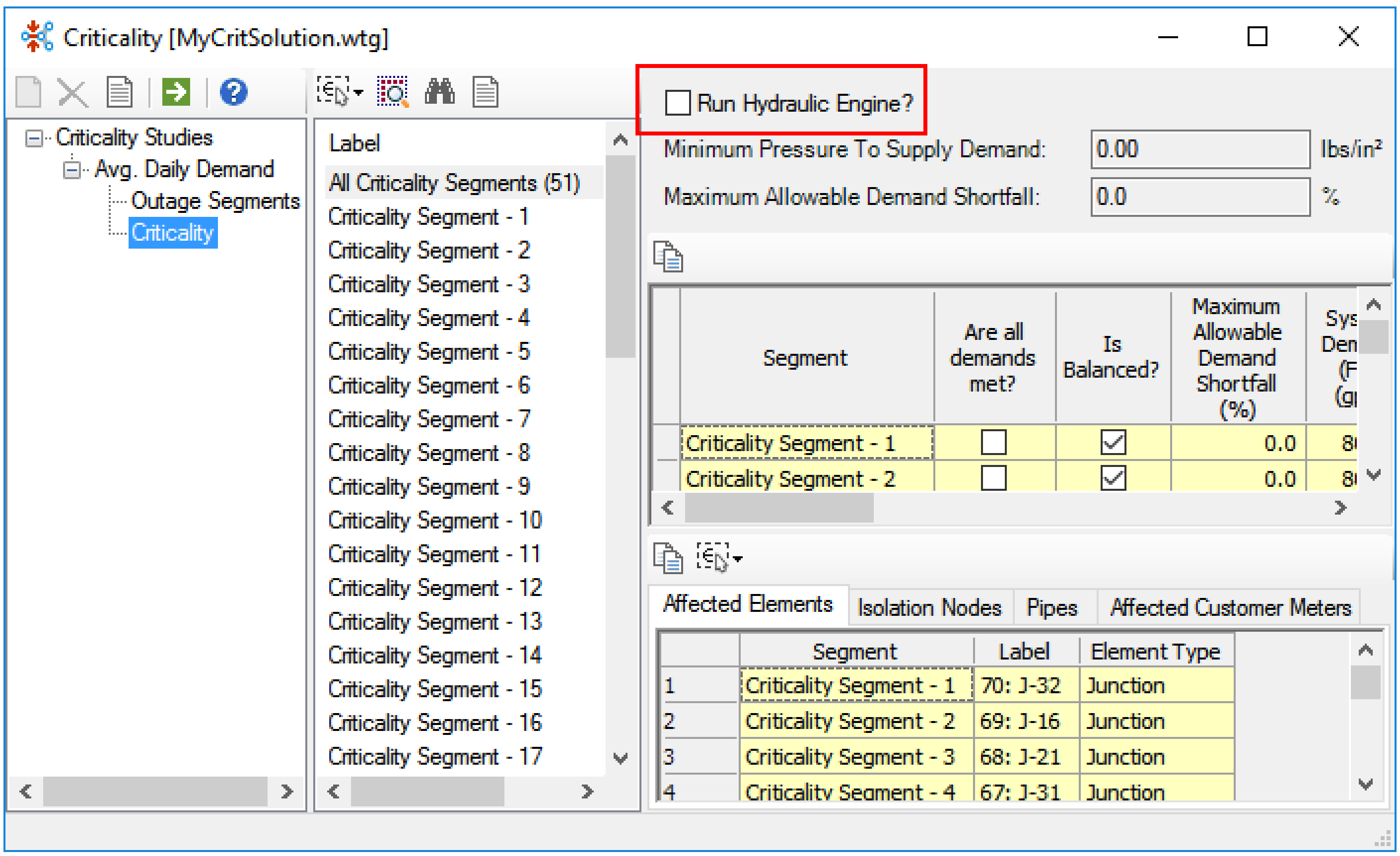
4. View the drawing to see that the pipe with the highest Outage Set Length is in blue and the downstream outage segments that will be out of service are in red.



Step 4: Run criticality analysis

The most important function of criticality analysis is the ability of the system to meet demands given a segment outage. A form of this analysis is the case where the shortfalls are determined solely based on connectivity. If the node is connected back to the source, it is assumed the demands are met. This type of run does not involve the hydraulic engine and is calculated more quickly.

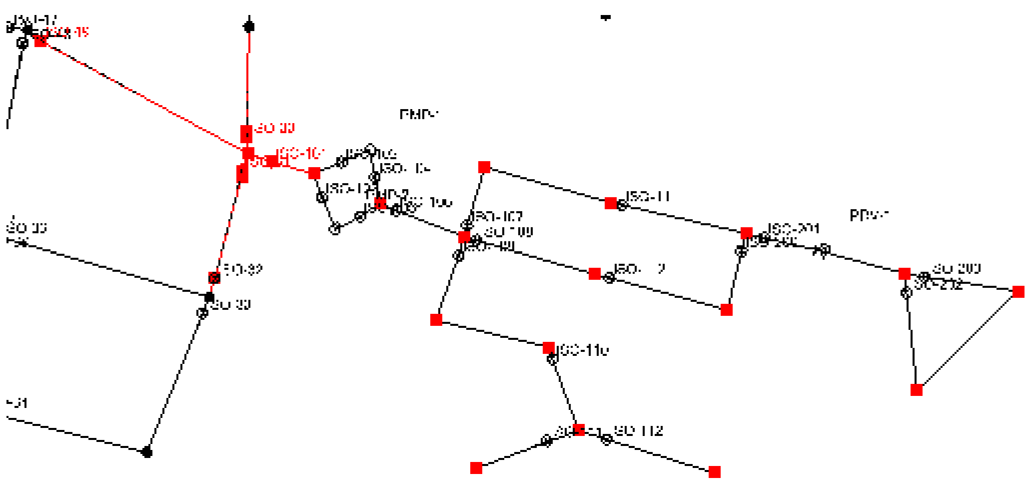
1. Select **Criticality** and make sure **Run Hydraulic Engine?** is unchecked. Then click **Compute** .



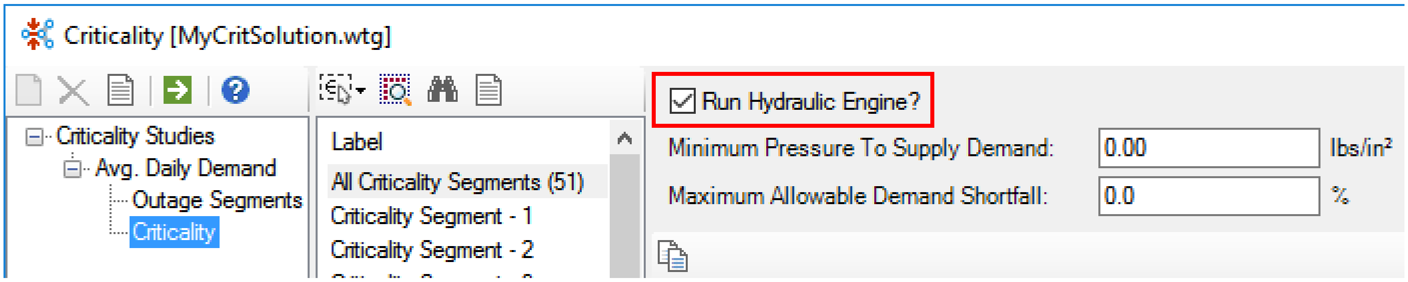
2. Right-click on the **System Demand Shortfall %** column and then **Sort > Sort Descending**.

	Segment	Are all demands met?	Is Balanced?	Maximum Allowable Demand Shortfall (%)	System Demand (Full) (gpm)	System Demand (Met) (gpm)	System Demand Shortfall (%)	Node with Largest Percent Demand Shortfall
	Criticality Segment - 15	☐	☑	0.0	804.00	528.00	34.3	56: J-13
	Criticality Segment - 27	☐	☑	0.0	804.00	543.00	32.5	232: J-102
	Criticality Segment - 30	☐	☑	0.0	804.00	543.00	32.5	232: J-102
	Criticality Segment - 42	☐	☑	0.0	804.00	543.00	32.5	232: J-102
	Criticality Segment - 26	☐	☑	0.0	804.00	547.00	32.0	234: J-103
	Criticality Segment - 7	☐	☑	0.0	804.00	704.00	12.4	64: J-1
	Criticality Segment - 10	☐	☑	0.0	804.00	704.00	12.4	77: J-4
	Criticality Segment - 20	☐	☑	0.0	804.00	712.00	11.4	247: J-109
	Criticality Segment - 36	☐	☑	0.0	804.00	724.00	10.0	79: J-2
	Criticality Segment - 13	☐	☑	0.0	804.00	749.00	6.8	58: J-3
	Criticality Segment - 25	☐	☑	0.0	804.00	752.00	6.5	236: J-104
	Criticality Segment - 19	☐	☑	0.0	804.00	764.00	5.0	251: J-111
	Criticality Segment - 24	☐	☑	0.0	804.00	769.00	4.4	240: J-106
	Criticality Segment - 14	☐	☑	0.0	804.00	769.00	4.4	57: J-30
	Criticality Segment - 21	☐	☑	0.0	804.00	778.00	3.2	244: J-108

3. Select the segment with the highest **System Demand Shortfall** from the **Label** list and then click **Zoom to Segments** .



4. Now run a criticality analysis that uses the hydraulic network engine to determine the impact of segment outages. Check the **Run Hydraulic Engine** box and click Compute  .



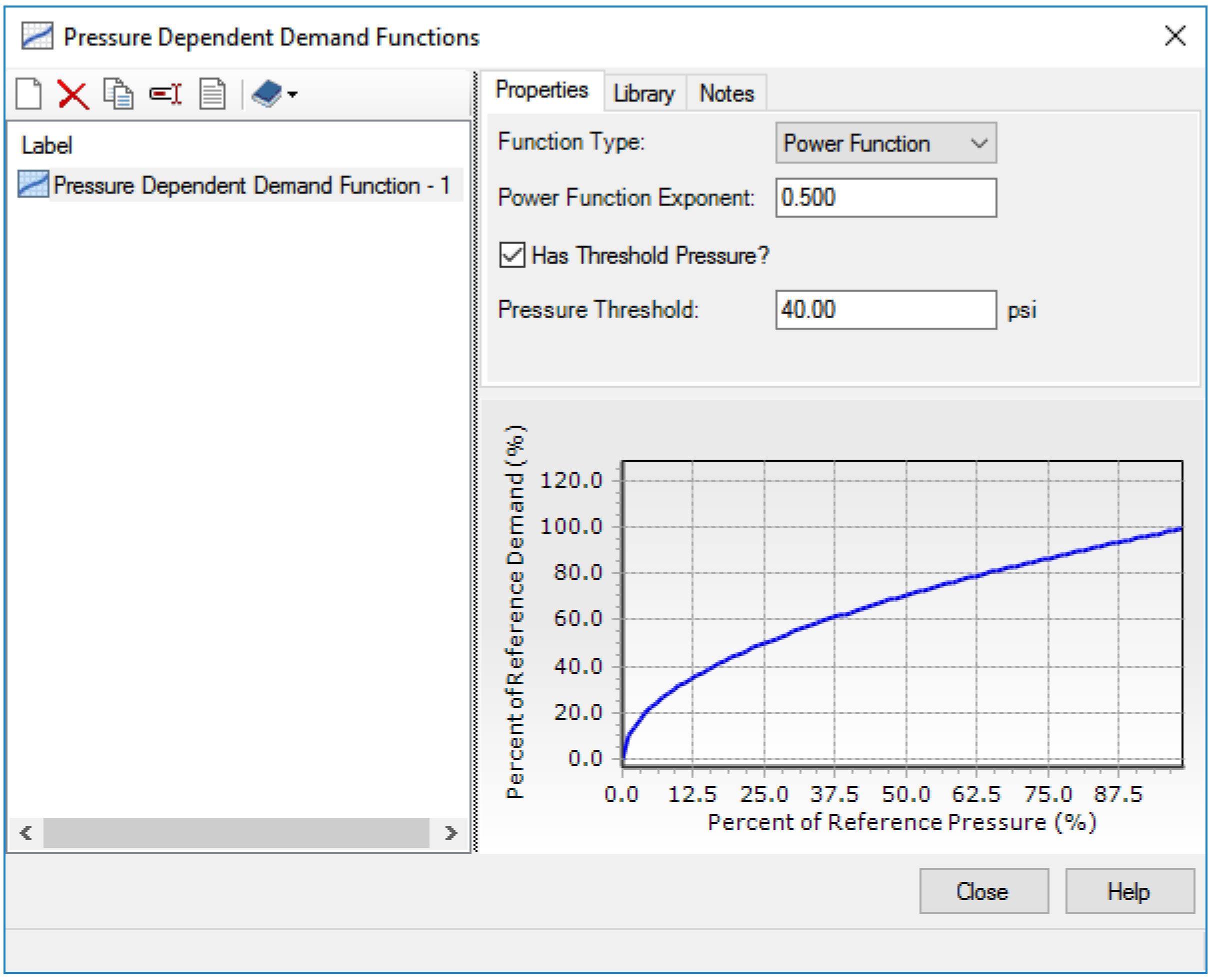
The **System Demand Shortfall %** are the same as the run without hydraulic calculations. This is because the flows are delivered to all nodes that are connected regardless of the pressure.

Step 5: Run criticality analysis hydraulic with PDD

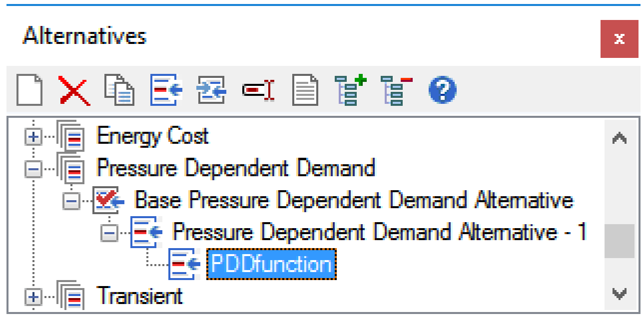
While other types of runs can indicate which segment outages cause the most demand to be isolated from the system, they are not the way to determine the impact on nodes that remain connected to the source but receive much less flow due to the outage.

In order to make these calculations, the demand in the system must be modeled using pressure dependent demands (PDD).

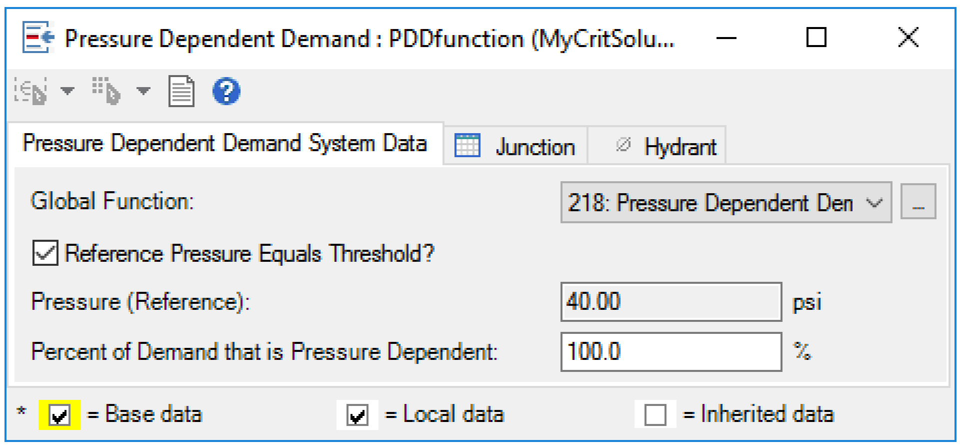
1. Close the criticality manager and click **Components** > **More** > **Pressure Dependent Demand Functions.**
2. Set the Pressure Threshold to **40 psi** and then close the PDD Function manager.



3. Choose **Analysis > Alternatives** , expand the **Pressure Dependent Demand Alternative** and select **PDDfunction**.



4. Double-click to open **PDDfunction** to verify which PDD function is being used, that the reference pressure (the pressure at which all demand is met) is equal to the threshold pressure, and that 100% of the demand is pressure dependent.

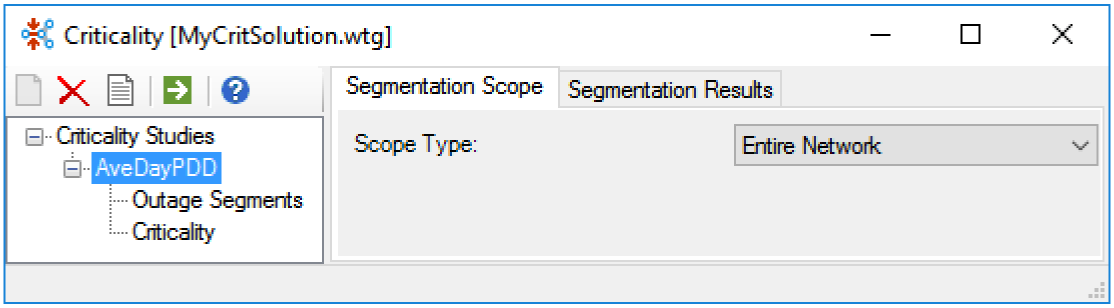


5. Close the alternative editor and the Alternatives manager.

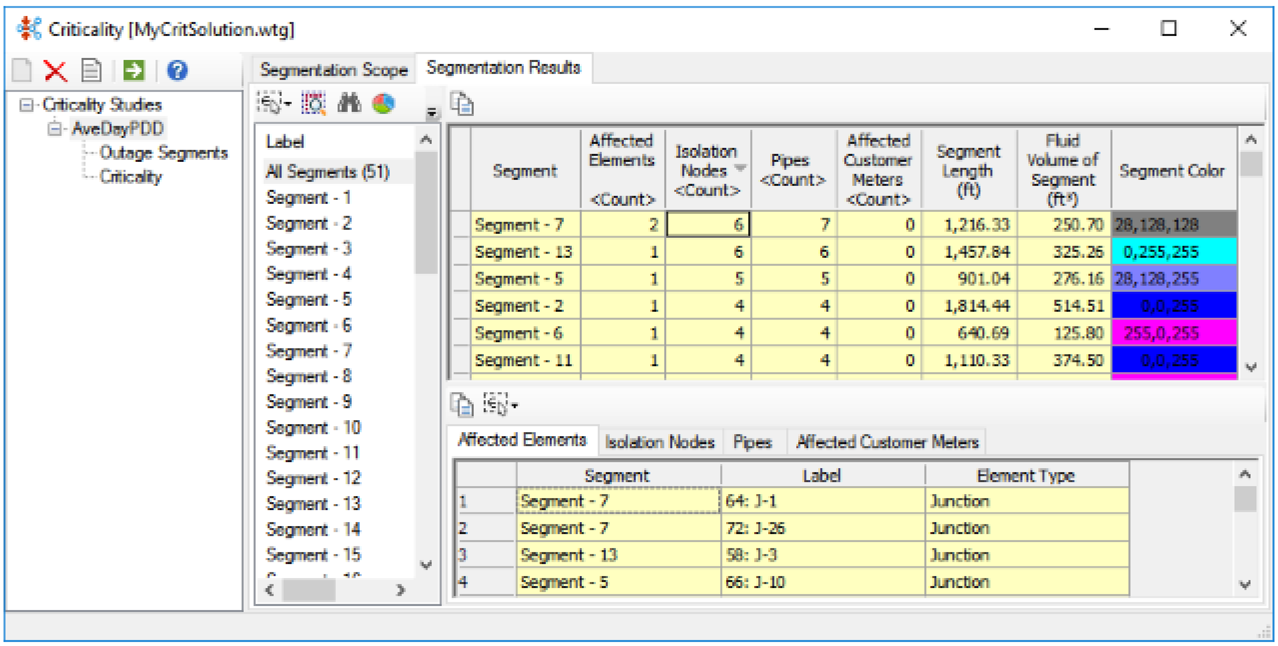
6. Click **Analysis > Criticality.** Highlight **Criticality Studies** and click the **New** button. Check the box for **AveDayPDD**.

7. Click **OK**.

8. From the **Segmentation Scope** tab, Select **Entire Network** from the **Scope Type** menu..



9. Select **AveDayPDD** and click **Compute**  .. Click **Yes** in the prompt that appears.



The segmentation results are the same as the first scenario because the same valving is used.

10. Select **Criticality** below AveDayPDD and check **Run Hydraulic Engine**? and click Compute .

11. Right click on the **System Demand Shortfall** column and select **Sort > Sort Descending**.

Segment	Are all demands met?	Is Balanced?	Maximum Allowable Demand Shortfall (%)	System Demand (Full) (gpm)	System Demand (Met) (gpm)	System Demand Shortfall (%)	Node with Largest Percent Demand Shortfall
Criticality Segment - 15	☐	☑	0.0	804.00	524.74	34.7	56: J-13
Criticality Segment - 42	☐	☑	0.0	804.00	539.83	32.9	232: J-102
Criticality Segment - 27	☐	☑	0.0	804.00	539.83	32.9	232: J-102
Criticality Segment - 30	☐	☑	0.0	804.00	539.83	32.9	232: J-102
Criticality Segment - 26	☐	☑	0.0	804.00	543.81	32.4	234: J-103
Criticality Segment - 7	☐	☑	0.0	804.00	674.29	16.1	64: J-1
Criticality Segment - 10	☐	☑	0.0	804.00	684.71	14.8	77: J-4
Criticality Segment - 36	☐	☑	0.0	804.00	704.95	12.3	79: J-2
Criticality Segment - 20	☐	☑	0.0	804.00	708.46	11.9	247: J-109
Criticality Segment - 13	☐	☑	0.0	804.00	731.78	9.0	58: J-3
Criticality Segment - 25	☐	☑	0.0	804.00	748.40	6.9	236: J-104
Criticality Segment - 14	☐	☑	0.0	804.00	749.76	6.7	57: J-30
Criticality Segment - 19	☐	☑	0.0	804.00	757.88	5.7	251: J-111
Criticality Segment - 38	☐	☑	0.0	804.00	758.59	5.6	75: J-17

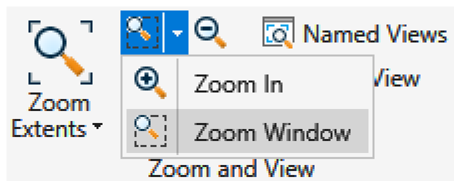
Notice that the shortfalls have increased over the previous runs because the runs that incorporate PDD account for the impact on nodes that receive water but at a lower pressure than under normal circumstances.

12. Close the Criticality manager.

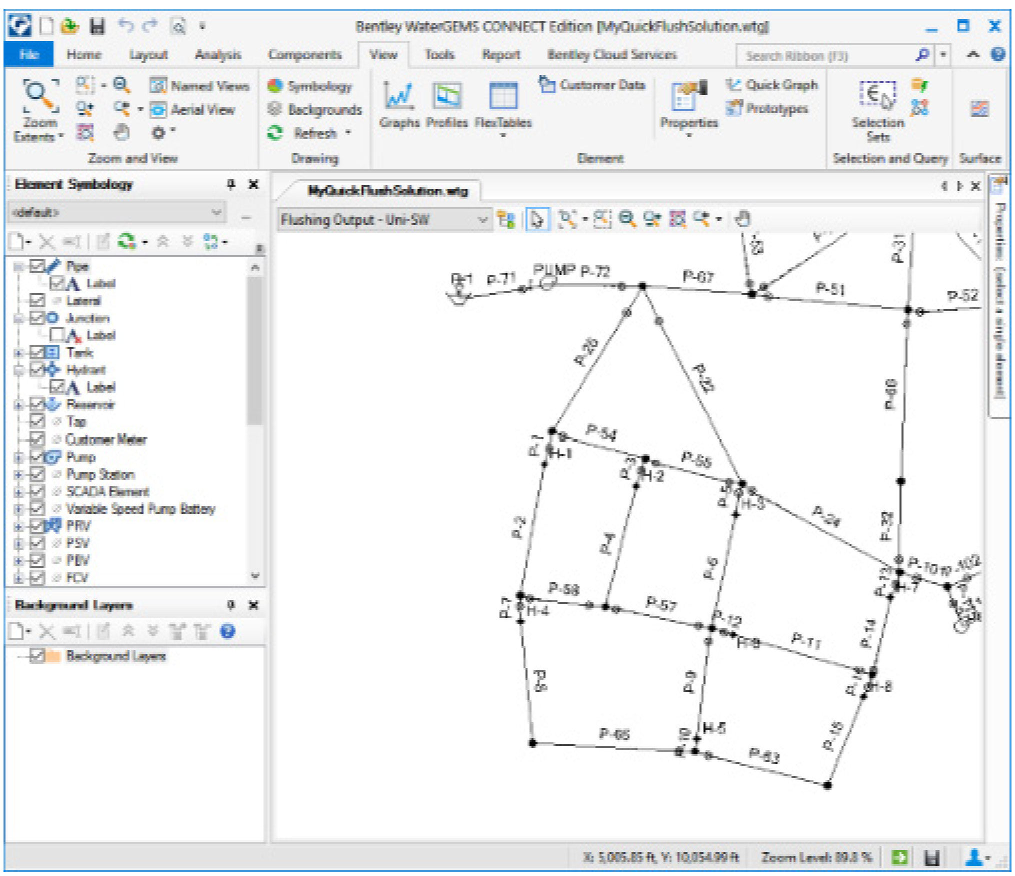
Flushing

In this Quick Start lesson, you will set up and run a series of conventional flushes (no valve operation) and one unidirectional flush.

1. Open the model called **QuickStartFlush.wtg** in the Lessons folder.
2. Click **OK** to the message dialog that comes up about CONNECTED Projects.
3. Click **File** > **Save As** and name the file **MyQuickFlushSolution**.
4. Zoom to the south west portion of the model **View** > **Zoom** > **Zoom Window**

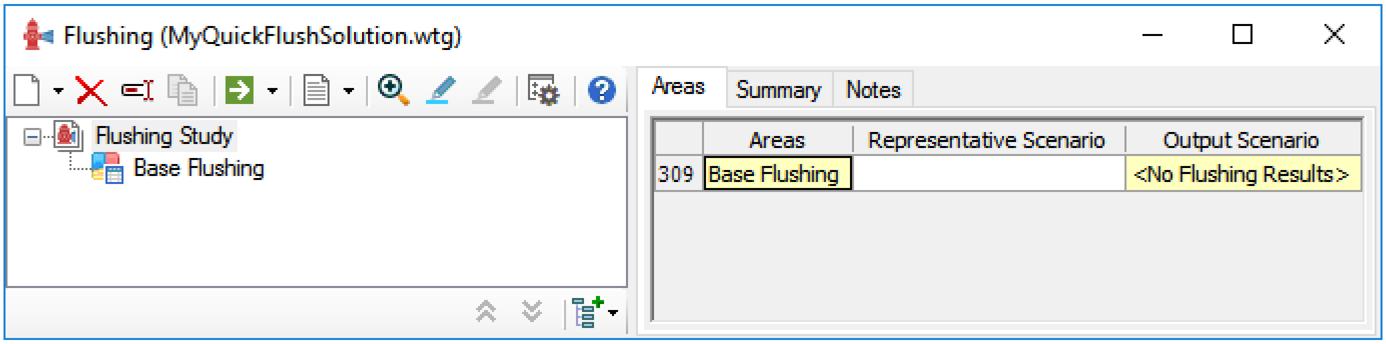


so that it looks like below:

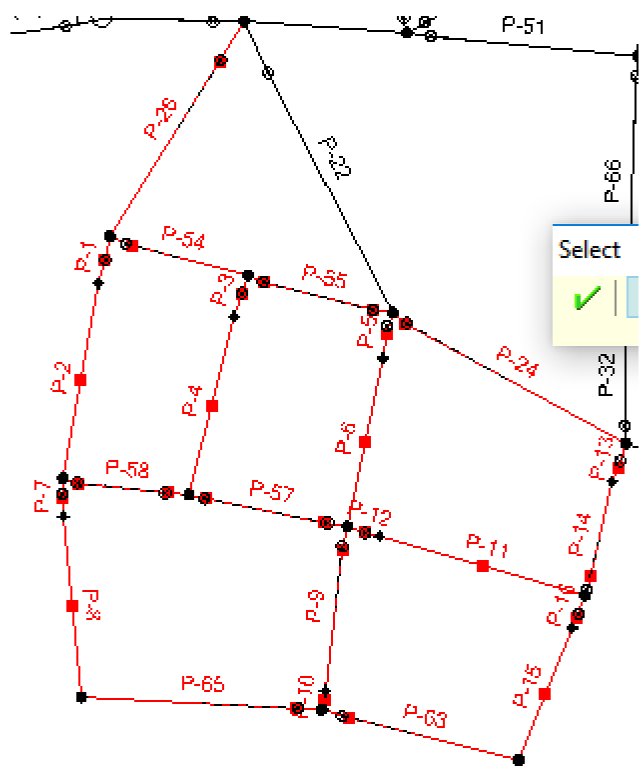


Step 1: Conventional Flushing

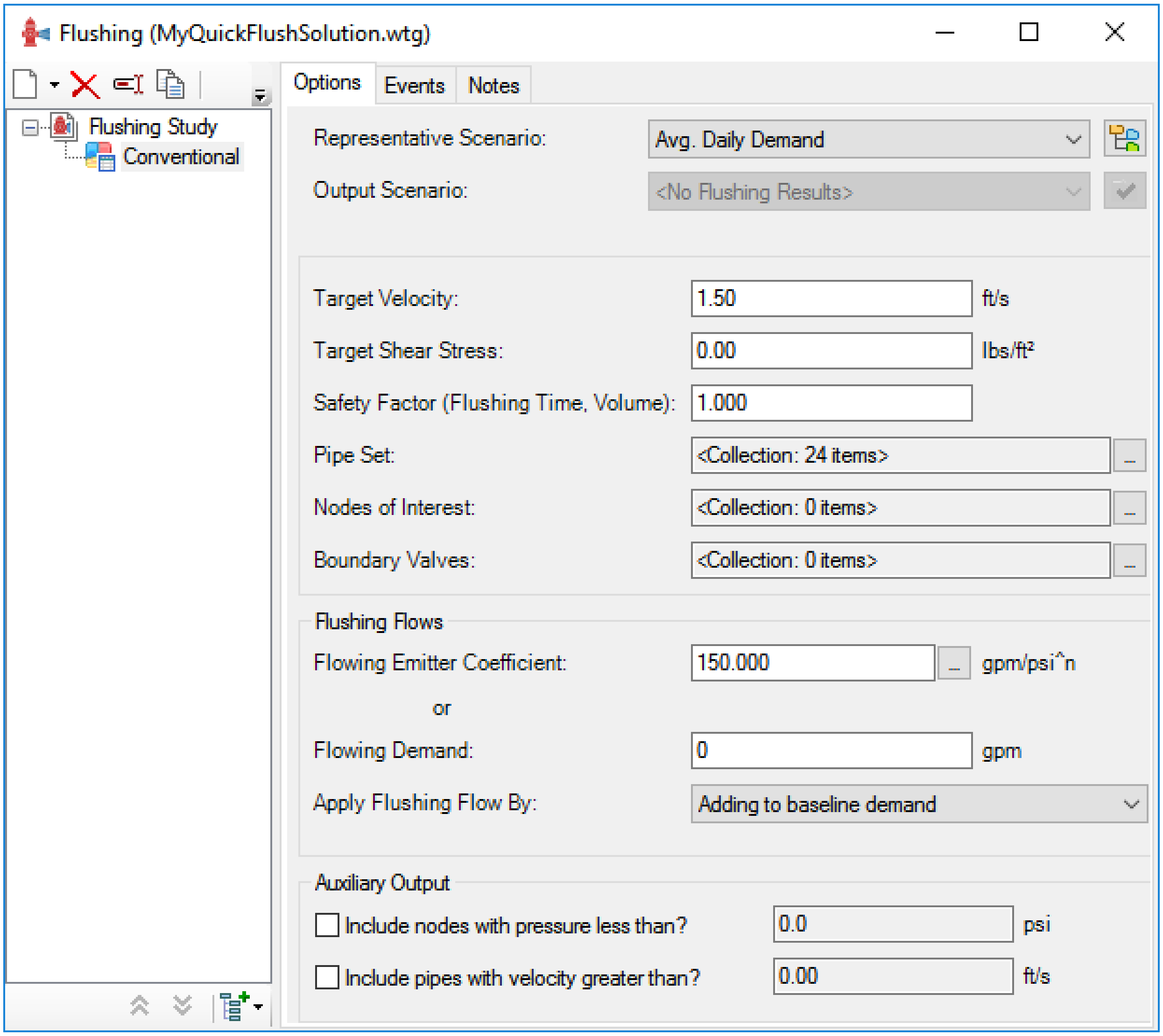
1. Pick **Analysis** > **Flushing.** This opens the flushing manager dialog.



2. Pick **Avg. Daily Demand** as the Representative Scenario in the right pane.
3. In the left pane, highlight **Base Flushing**, pick the Rename button (third from left on top) and change the name to **Conventional**.
4. In the right pane, create a pipe set for which you will calculate flushing properties by picking the ellipsis button next to **Pipe Set,** and **Select From Drawing**. Select the pipes shown below. (It may be necessary to zoom in to some of the shorter pipes to select them.)

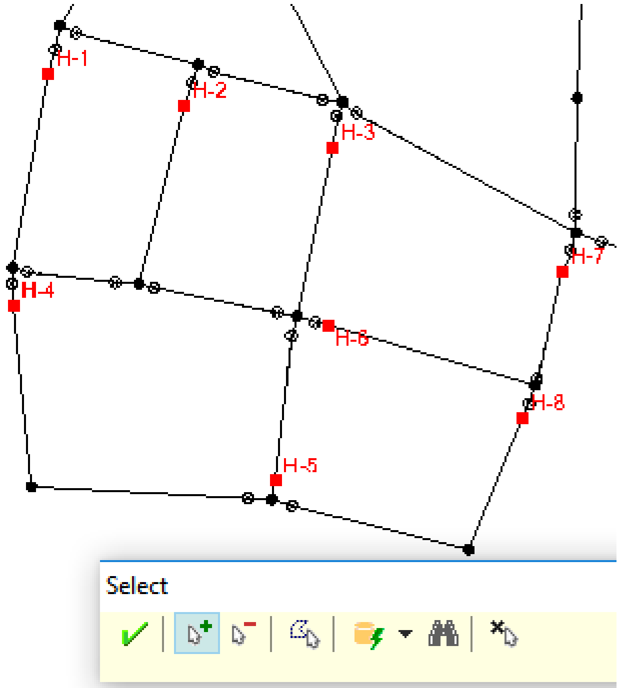


5. Pick the green check mark to complete the selection.
6. Click **OK**.
7. Set the **Flow Emitter Coefficient** to **150**. The dialog should look like this:

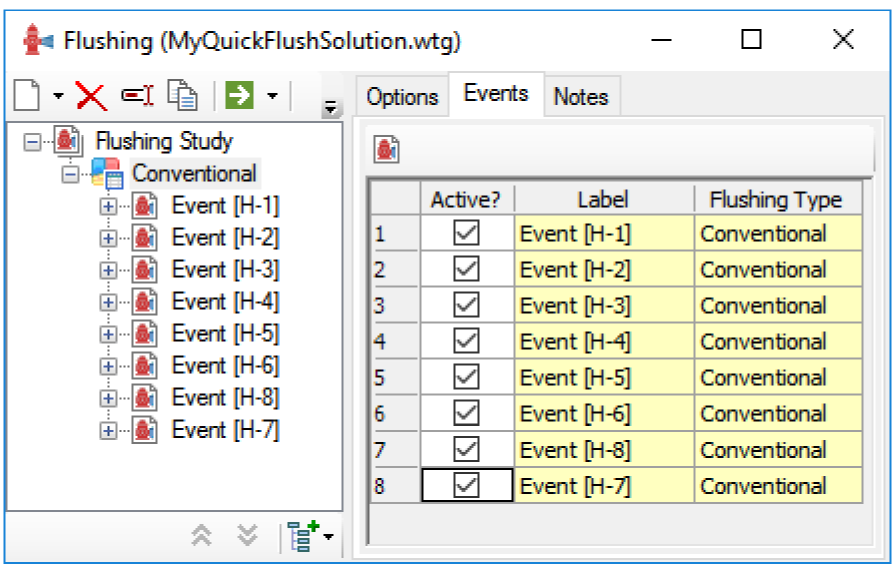


8. Click the **Events** tab on the right pane.
9. Right-click on **Conventional** in the left pane. Select **Add > New Conventional Events (Batch)**

10. Select hydrants **H-1 through H-8** and click the green check mark.

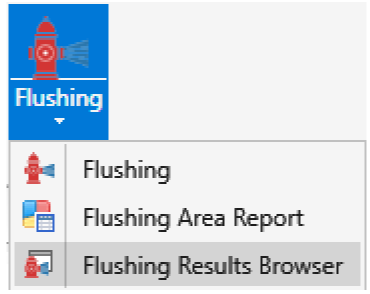


The Flushing Manager will look like this:

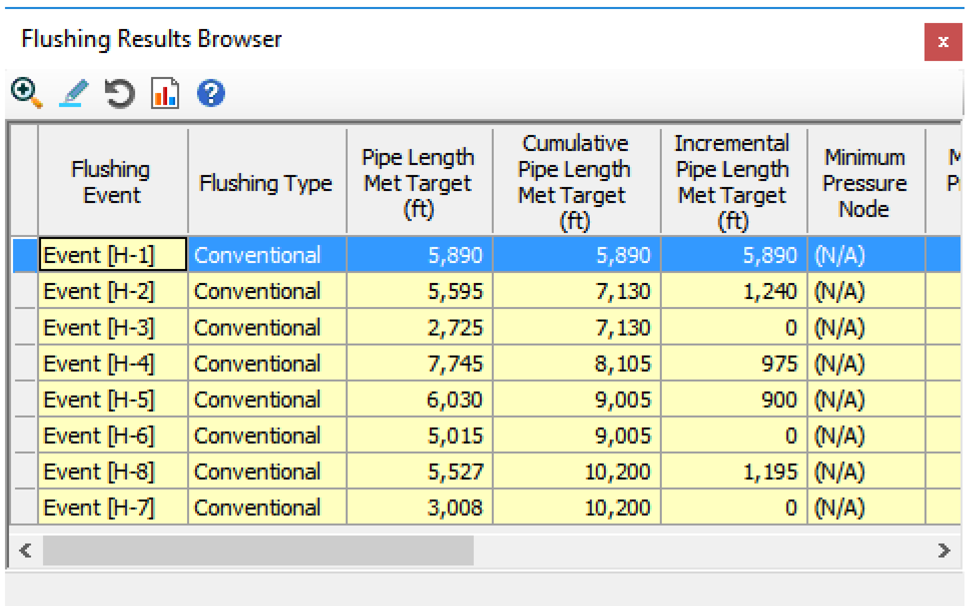


11. Run the 8 events in sequence by clicking the **Compute** button. Close the Calculation Summary.

12. To check the results, open the Flushing Results Browser by clicking **Analysis**, then click the down arrow below **Flushing** and select **Flushing Results Browser**.



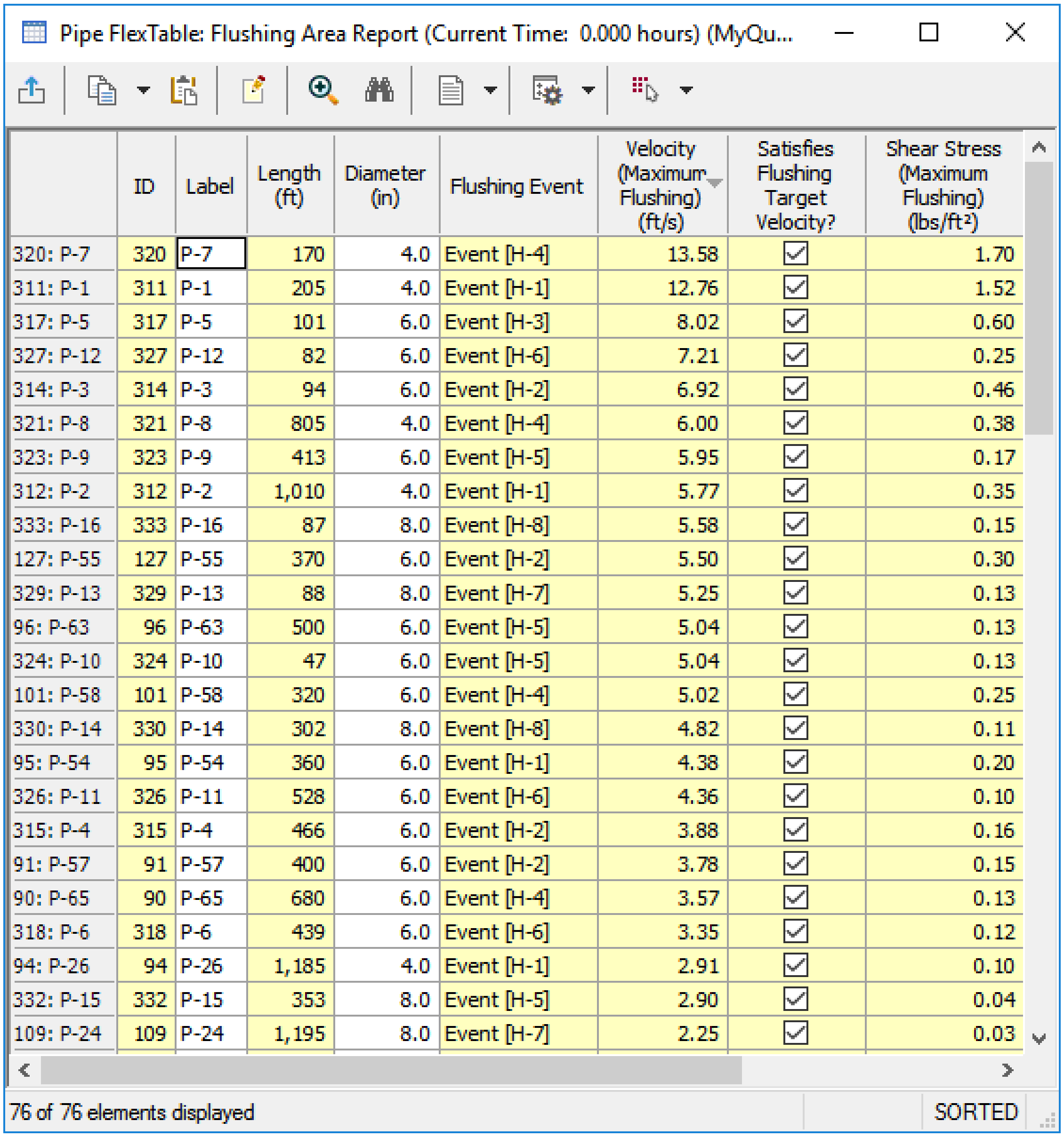
The Flushing Results Browser shows the effect of each event.


Flushing Results Browser

Flushing Event	Flushing Type	Pipe Length Met Target (ft)	Cumulative Pipe Length Met Target (ft)	Incremental Pipe Length Met Target (ft)	Minimum Pressure Node	M P
Event [H-1]	Conventional	5,890	5,890	5,890	(N/A)	
Event [H-2]	Conventional	5,595	7,130	1,240	(N/A)	
Event [H-3]	Conventional	2,725	7,130	0	(N/A)	
Event [H-4]	Conventional	7,745	8,105	975	(N/A)	
Event [H-5]	Conventional	6,030	9,005	900	(N/A)	
Event [H-6]	Conventional	5,015	9,005	0	(N/A)	
Event [H-8]	Conventional	5,527	10,200	1,195	(N/A)	
Event [H-7]	Conventional	3,008	10,200	0	(N/A)	

13. Close this Browser and open the Flushing Area Report by clicking **Analysis**, then click the down arrow below **Flushing** and select **Flushing Area Report.**

14. Right click on the **Velocity Maximum Flushing** column and **Sort > Descending**. This table shows the Velocity and Shear Stress for the pipes in the Pipe Set.



	ID	Label	Length (ft)	Diameter (in)	Flushing Event	Velocity (Maximum Flushing) (ft/s)	Satisfies Flushing Target Velocity?	Shear Stress (Maximum Flushing) (lbs/ft²)
320: P-7	320	P-7	170	4.0	Event [H-4]	13.58	☑	1.70
311: P-1	311	P-1	205	4.0	Event [H-1]	12.76	☑	1.52
317: P-5	317	P-5	101	6.0	Event [H-3]	8.02	☑	0.60
327: P-12	327	P-12	82	6.0	Event [H-6]	7.21	☑	0.25
314: P-3	314	P-3	94	6.0	Event [H-2]	6.92	☑	0.46
321: P-8	321	P-8	805	4.0	Event [H-4]	6.00	☑	0.38
323: P-9	323	P-9	413	6.0	Event [H-5]	5.95	☑	0.17
312: P-2	312	P-2	1,010	4.0	Event [H-1]	5.77	☑	0.35
333: P-16	333	P-16	87	8.0	Event [H-8]	5.58	☑	0.15
127: P-55	127	P-55	370	6.0	Event [H-2]	5.50	☑	0.30
329: P-13	329	P-13	88	8.0	Event [H-7]	5.25	☑	0.13
96: P-63	96	P-63	500	6.0	Event [H-5]	5.04	☑	0.13
324: P-10	324	P-10	47	6.0	Event [H-5]	5.04	☑	0.13
101: P-58	101	P-58	320	6.0	Event [H-4]	5.02	☑	0.25
330: P-14	330	P-14	302	8.0	Event [H-8]	4.82	☑	0.11
95: P-54	95	P-54	360	6.0	Event [H-1]	4.38	☑	0.20
326: P-11	326	P-11	528	6.0	Event [H-6]	4.36	☑	0.10
315: P-4	315	P-4	466	6.0	Event [H-2]	3.88	☑	0.16
91: P-57	91	P-57	400	6.0	Event [H-2]	3.78	☑	0.15
90: P-65	90	P-65	680	6.0	Event [H-4]	3.57	☑	0.13
318: P-6	318	P-6	439	6.0	Event [H-6]	3.35	☑	0.12
94: P-26	94	P-26	1,185	4.0	Event [H-1]	2.91	☑	0.10
332: P-15	332	P-15	353	8.0	Event [H-5]	2.90	☑	0.04
109: P-24	109	P-24	1,195	8.0	Event [H-7]	2.25	☑	0.03

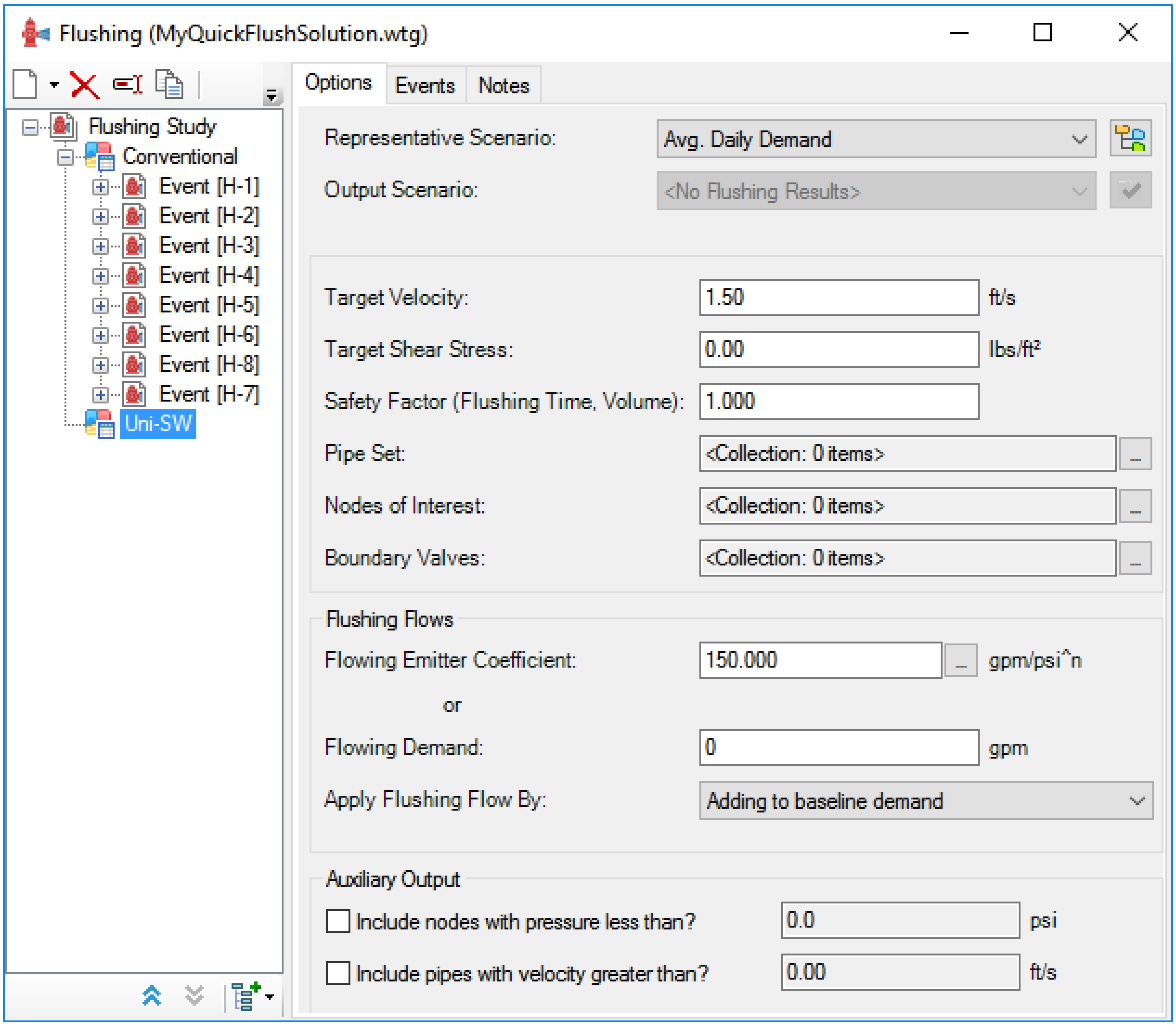
All 24 pipes in this conventional flushing study exceeded the Target Velocity of 1.5 ft/s.

15. Close the Flushing Area Report.

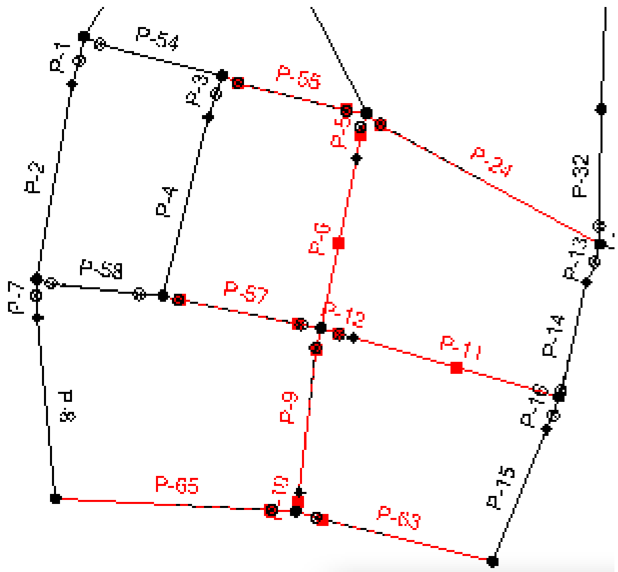
Step 2: Unidirectional Flushing Event

Next you will set up a unidirectional flushing event to increase the velocity in a run of pipes along the southwest edge of the system.

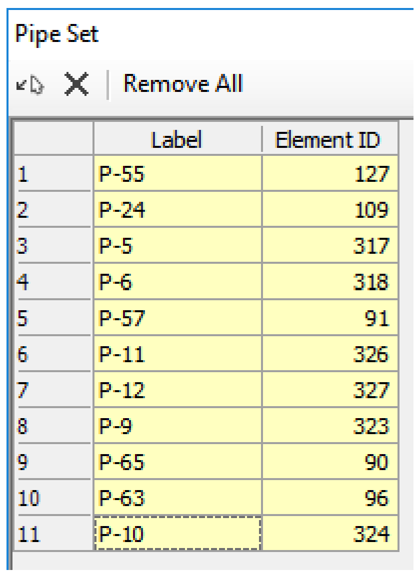
1. Highlight **Flushing study** in the left pane, right-click and Select **Add > New Area**
2. Rename the new area **Uni-SW**.



3. Pick **Avg. Daily Demand** as the Representative Scenario, set the **Emitter coefficient to 150** and create the Pipe Set as shown below:



4. Click the green check mark and view the Pipe set.


Pipe Set

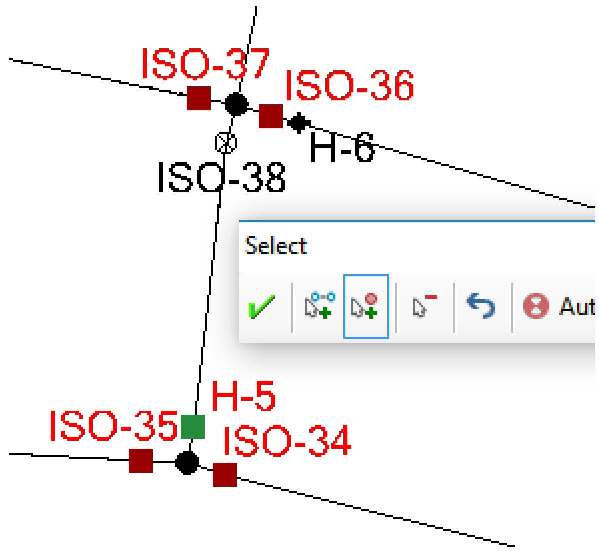
Remove All

	Label	Element ID
1	P-55	127
2	P-24	109
3	P-5	317
4	P-6	318
5	P-57	91
6	P-11	326
7	P-12	327
8	P-9	323
9	P-65	90
10	P-63	96
11	P-10	324

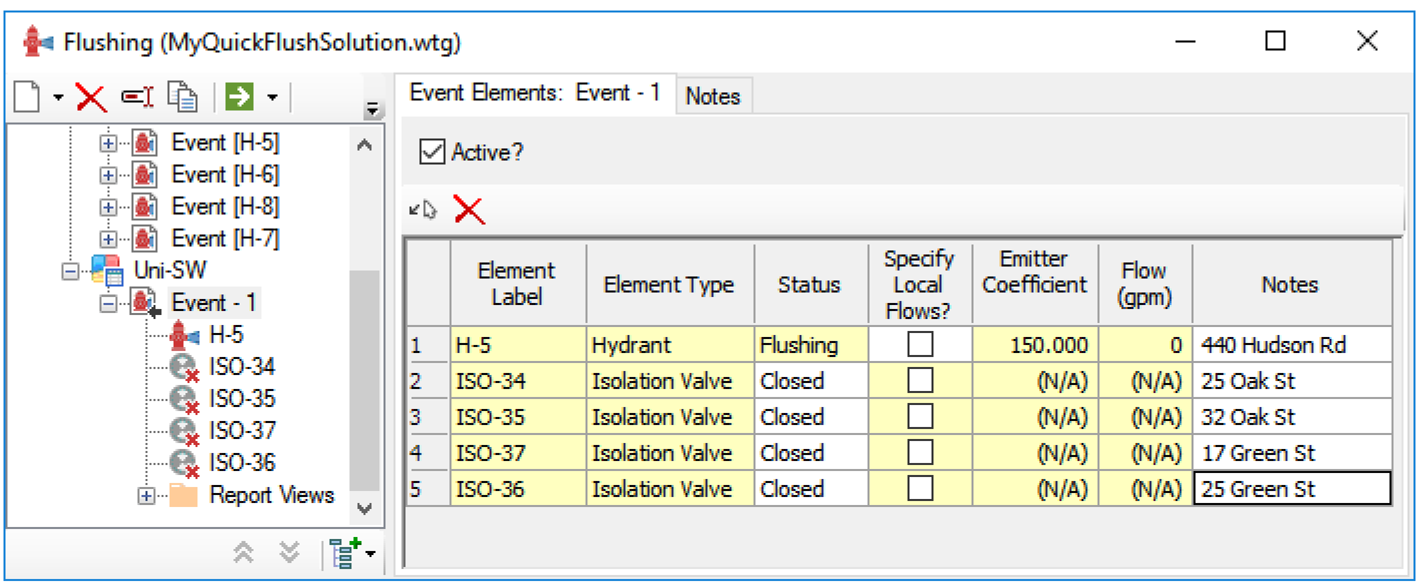
5. Click **OK**.

6. Right-click on **Uni-SW** in the left pane. Select **Add** > **New Unidirectional Event.**

7. In the **Select** dialog that comes up, click the **Add Operational Elements button** . Then pick the valves **ISO-34, ISO-35, ISO-36 and ISO-37 to close** and hydrant **H-5 to flow** as shown below:

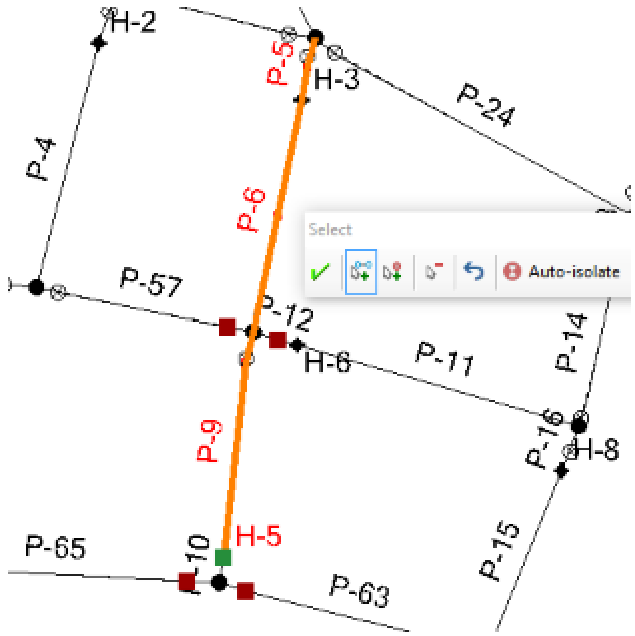


8. After picking the elements and clicking the green check mark, review the list of elements to be operated. Feel free to add some descriptive notes to the elements to be operated.

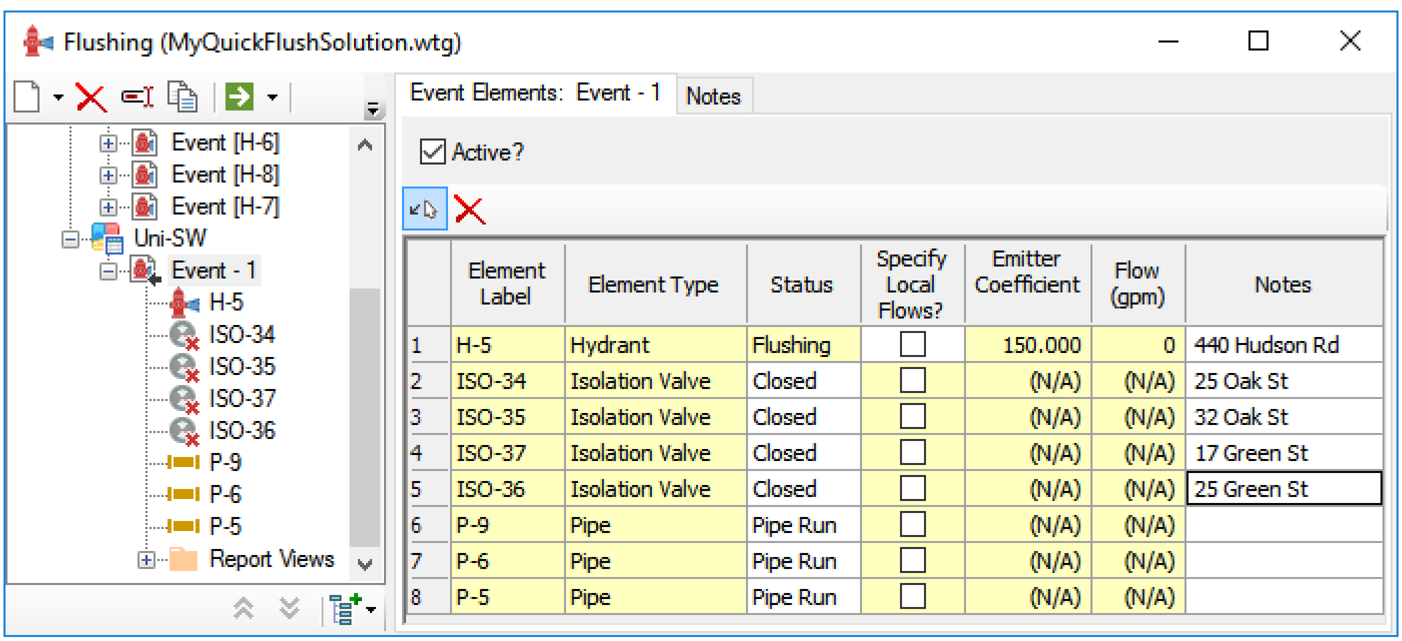


9. Identify the pipes to be part of the Pipe Run by picking the **Select from Drawing** button on the right pane.

10. In the **Select** dialog that comes up, click the **Add Pipe Run Elements button**  . Then pick the pipes **P-5, P-6 and P-9** as shown below: Once again, it may be desirable to use the mouse wheel to zoom in to the shorter pipes.

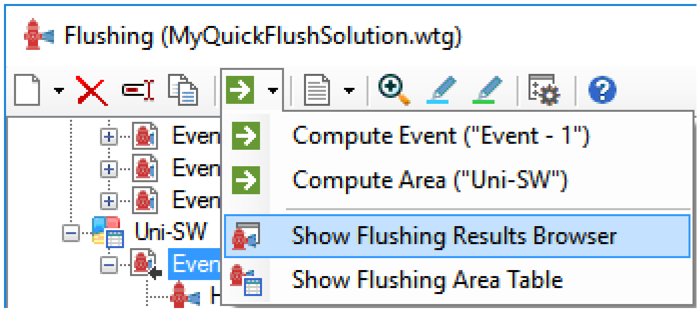


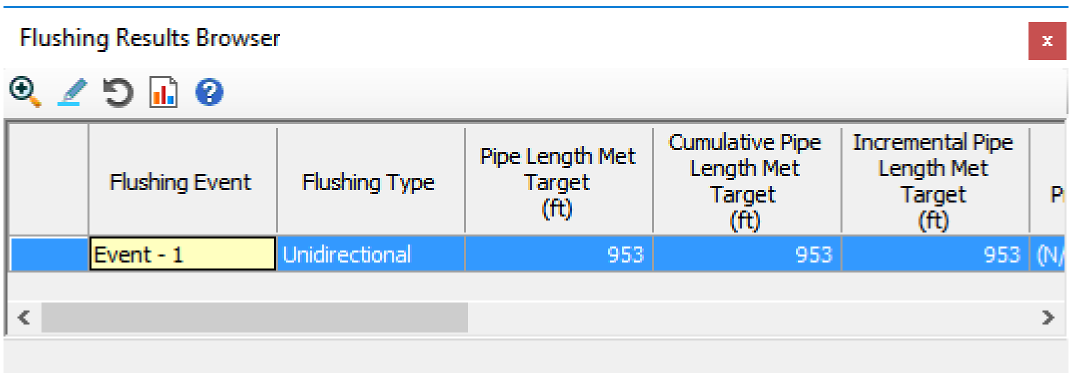
11. With all of the elements identified, select the **Compute** button. Click **Yes** to the **Compute Single Flushing Event** prompt.



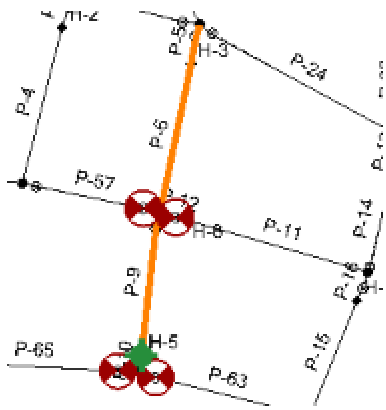
	Element Label	Element Type	Status	Specify Local Flows?	Emitter Coefficient	Flow (gpm)	Notes
1	H-5	Hydrant	Flushing	☐	150.000	0	440 Hudson Rd
2	ISO-34	Isolation Valve	Closed	☐	(N/A)	(N/A)	25 Oak St
3	ISO-35	Isolation Valve	Closed	☐	(N/A)	(N/A)	32 Oak St
4	ISO-37	Isolation Valve	Closed	☐	(N/A)	(N/A)	17 Green St
5	ISO-36	Isolation Valve	Closed	☐	(N/A)	(N/A)	25 Green St
6	P-9	Pipe	Pipe Run	☐	(N/A)	(N/A)	
7	P-6	Pipe	Pipe Run	☐	(N/A)	(N/A)	
8	P-5	Pipe	Pipe Run	☐	(N/A)	(N/A)	

12. Once the run is complete, open the **Flushing Results Browser** by clicking the down arrow to the right of **Compute** and selecting **Show Flushing Results Browser**. and view the results



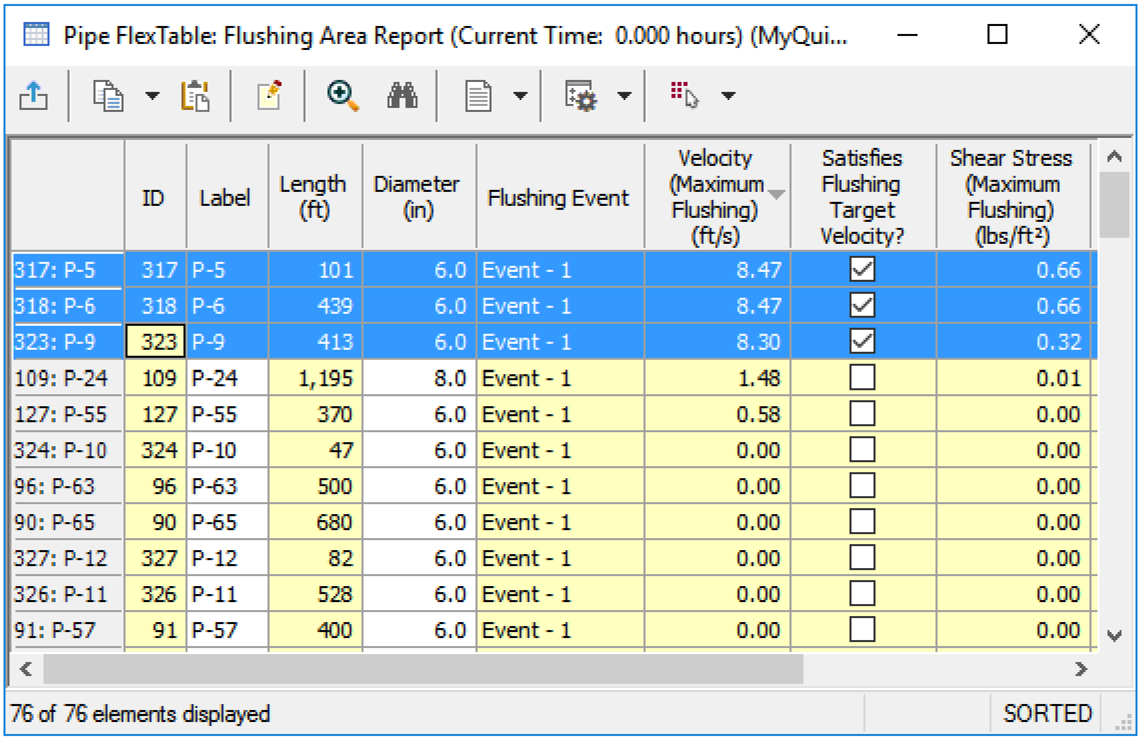


13. Make sure the whole row for the event is highlighted (as shown above), then click the highlight button (second from left) and view the event in the drawing.



14. Close the Flushing Results Browser

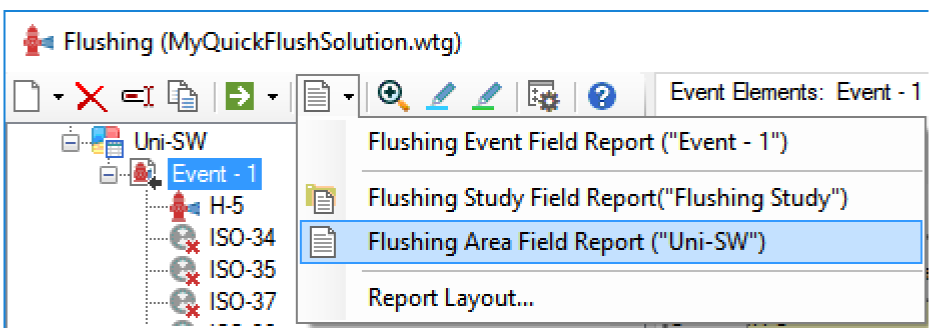
15. Open the **Flushing Area Table** by clicking the down arrow to the right of **Compute** and selecting **Show Flushing Area Table** and view the results



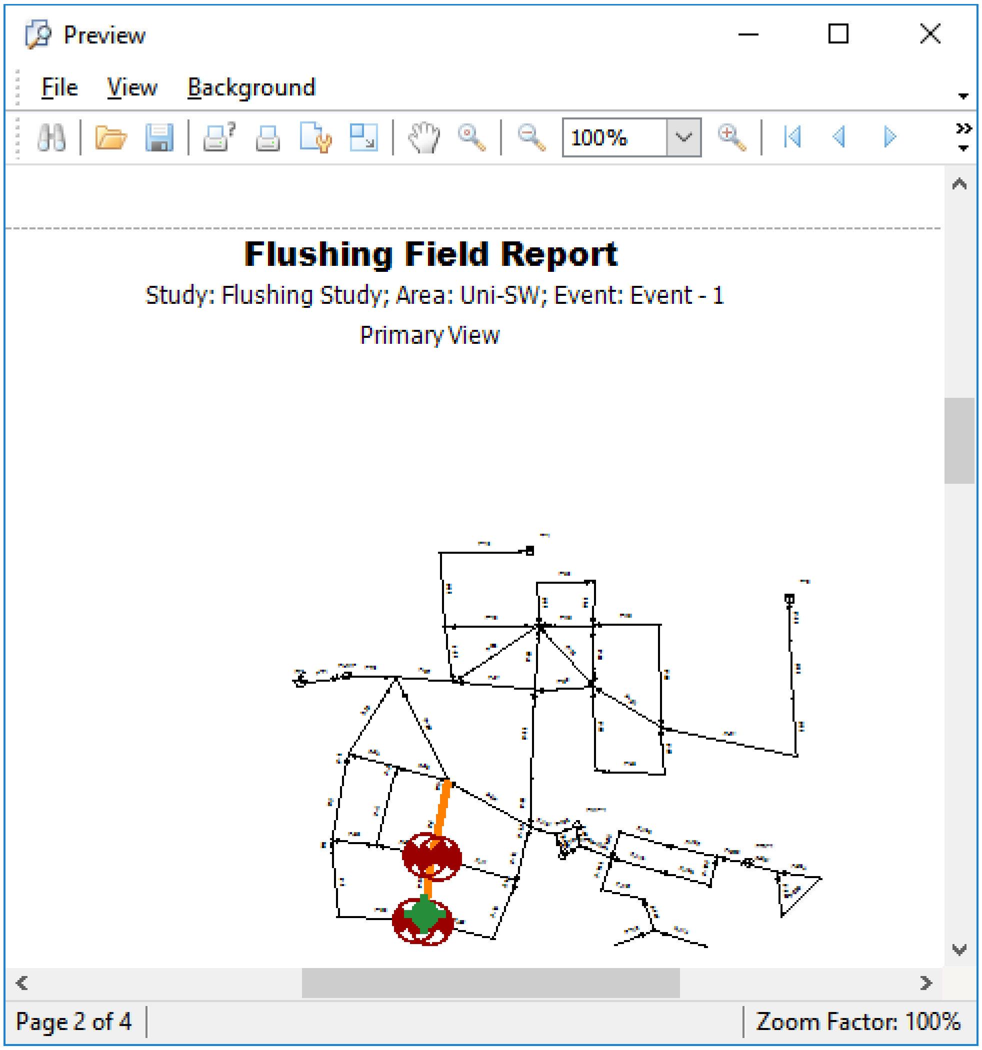
	ID	Label	Length (ft)	Diameter (in)	Flushing Event	Velocity (Maximum Flushing) (ft/s)	Satisfies Flushing Target Velocity?	Shear Stress (Maximum Flushing) (lbs/ft²)
317: P-5	317	P-5	101	6.0	Event - 1	8.47	☑	0.66
318: P-6	318	P-6	439	6.0	Event - 1	8.47	☑	0.66
323: P-9	323	P-9	413	6.0	Event - 1	8.30	☑	0.32
109: P-24	109	P-24	1,195	8.0	Event - 1	1.48	☐	0.01
127: P-55	127	P-55	370	6.0	Event - 1	0.58	☐	0.00
324: P-10	324	P-10	47	6.0	Event - 1	0.00	☐	0.00
96: P-63	96	P-63	500	6.0	Event - 1	0.00	☐	0.00
90: P-65	90	P-65	680	6.0	Event - 1	0.00	☐	0.00
327: P-12	327	P-12	82	6.0	Event - 1	0.00	☐	0.00
326: P-11	326	P-11	528	6.0	Event - 1	0.00	☐	0.00
91: P-57	91	P-57	400	6.0	Event - 1	0.00	☐	0.00

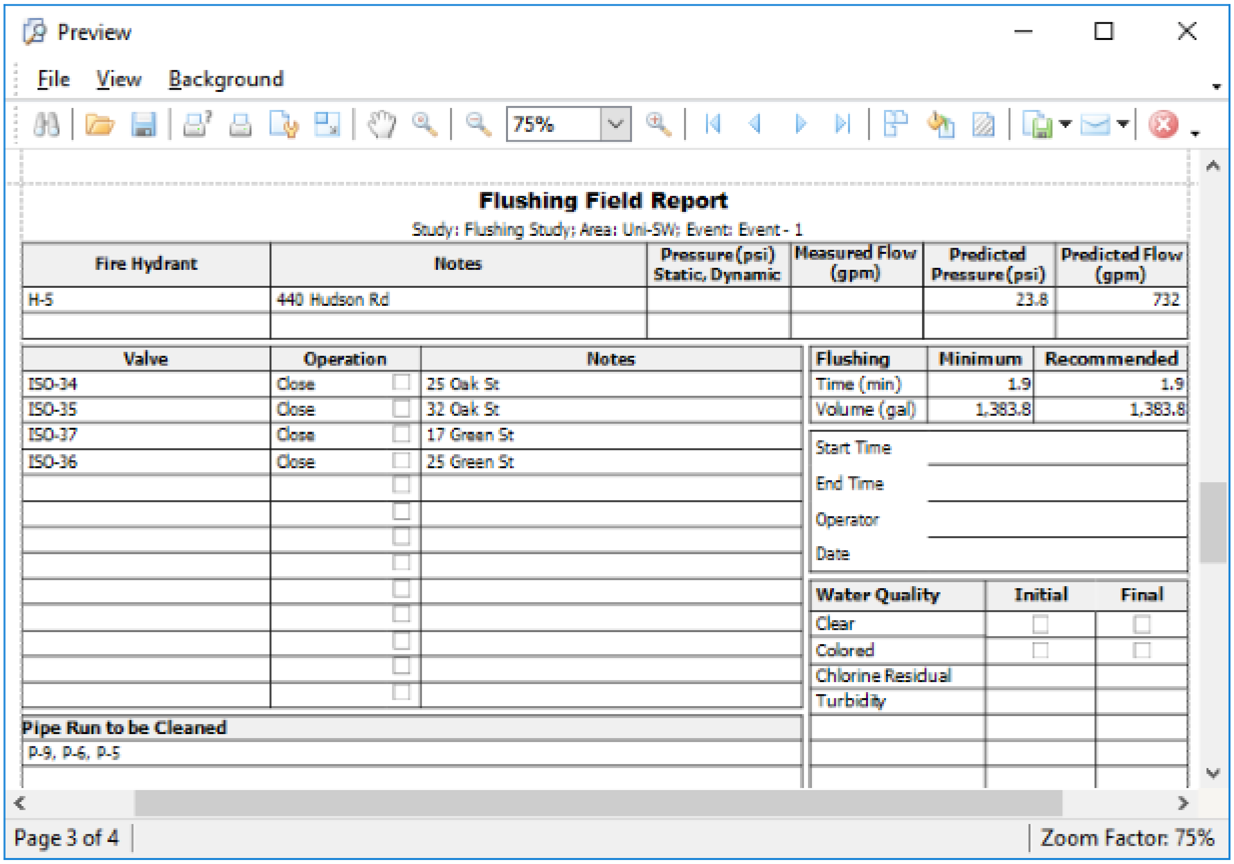
Notice that the velocity in Pipes P-5, P-6 and P-9 exceeds 8 ft/s.

16. Click the down arrow next to **Field Report** and select **Flushing Area Field Report ("Uni-SW")**.



A report is generated including a map of the area and a tabular description of the event indicating which elements to operate.





You performed a conventional flushing analysis (no valve operation required) and confirmed that all pipes in the area of interest met a minimum velocity of 1.5 ft/s. A second, Uni-directional flushing analysis, was performed to evaluate the increase in velocity for a run of pipes (P-5, P-6, P-9) in the same area. Four valves were closed and hydrant H-5 was flushed. Velocity for those pipes increased to over 8 ft/s.

17. Close the report to get back to WaterGEMS.

Importing SCADA Data

In this lesson, you will import historical SCADA data that has been stored in an Excel file and compare the historical values from that file with the current model scenario. To accomplish this, you will

- Identify SCADA Data Source
- Import SCADA data
- Create SCADA Elements
- Link SCADA data and modeling elements
- Compare SCADA data with model results

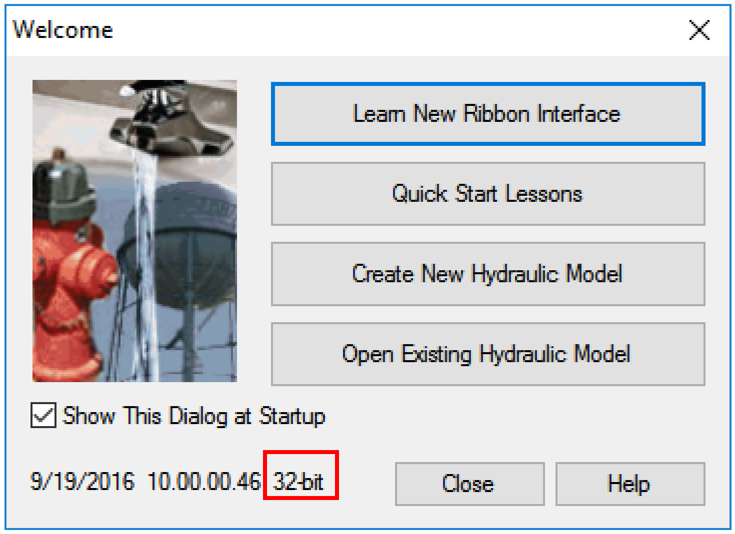
The SCADA data is stored in the Excel file **SCADA1.xls** in the Lessons folder.

1. Open the **SCADA1.xls** file to view its format, which is called *multiple rows per record*. Each record contains a time stamp (24 hour format), pump flow and tank hydraulic grade. In case you don't want to open Excel file, the first few rows of the file look like this.:

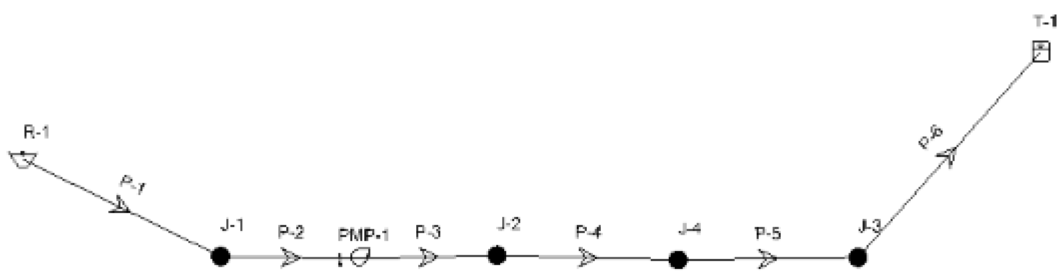
Time (hours)	PMP-1 - Flow (Total) (L/s)	T-1 - Hydraulic Grade (m)
9/12/2016 0:00	21	108
9/12/2016 1:00	20	108.62
9/12/2016 1:39	0	109
9/12/2016 2:00	0	108.82
9/12/2016 3:00	0	108.26
9/12/2016 4:00	0	107.64
9/12/2016 5:00	0	106.94
9/12/2016 6:00	0	106.16
9/12/2016 7:00	0	105.27
9/12/2016 7:15	22	105
9/12/2016 8:00	22	105.13
9/12/2016 9:00	22	105.17
9/12/2016 10:00	22	105.17
9/12/2016 11:00	22	105.26
9/12/2016 12:00	22	105.43
9/12/2016 13:00	22	105.63
9/12/2016 14:00	22	105.79
9/12/2016 15:00	22	105.92
9/12/2016 16:00	21	106.07

2. This Excel file is a 32-bit file. Therefore, it is best to use a 32-bit version of WaterGEMS. If you are running a 64-bit version, close WaterGEMS and start **C:\Program Files (x86)\Bentley\WaterGEMS\WaterGEMS.exe**

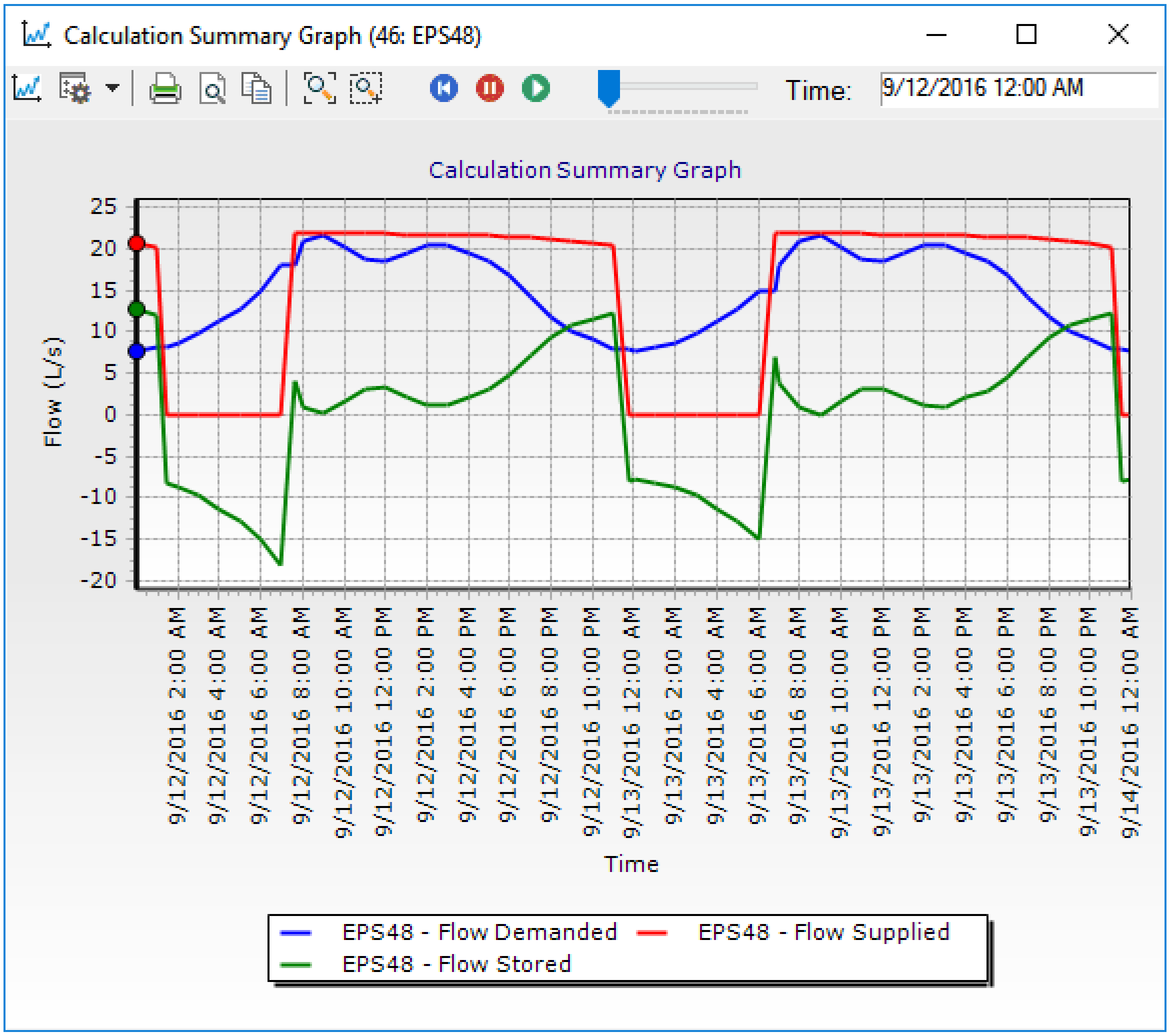
3. The welcome dialog should indicate 32-bit.



4. Click **Open Existing Hydraulic Model** and navigate to **SCADADataStarter.wtg** in the Lessons folder.
5. Click **OK** to the message dialog that comes up about CONNECTED Projects
6. After you have opened the file, click **File > Save As**. Name the file **MySCADADataSolution**.
7. This is a simple model with one pump and one tank as shown below.



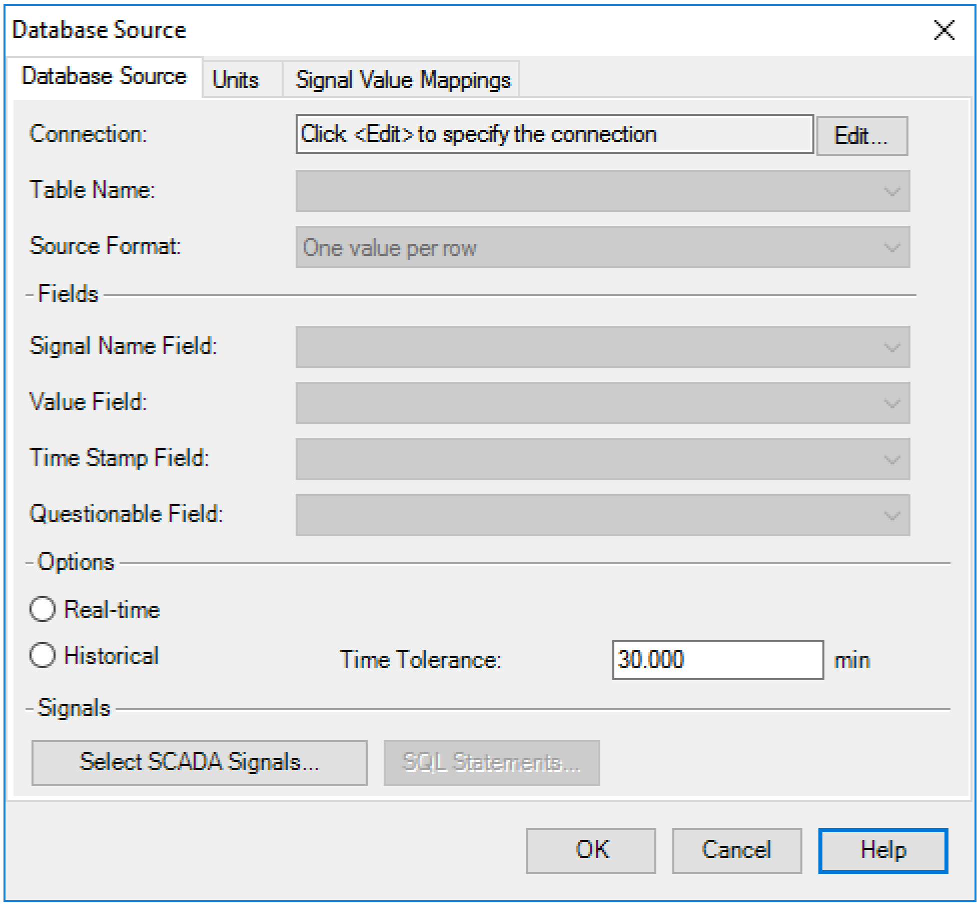
8. Pick Scenario **EPS48** and **Compute**. View the calculation summary graph by clicking on the Graph button at the top of the Calculation Summary dialog (third button on top of calculation summary). It should look like this.



9. Review the data then close the Calculation Summary graph, and Calculation Summary.

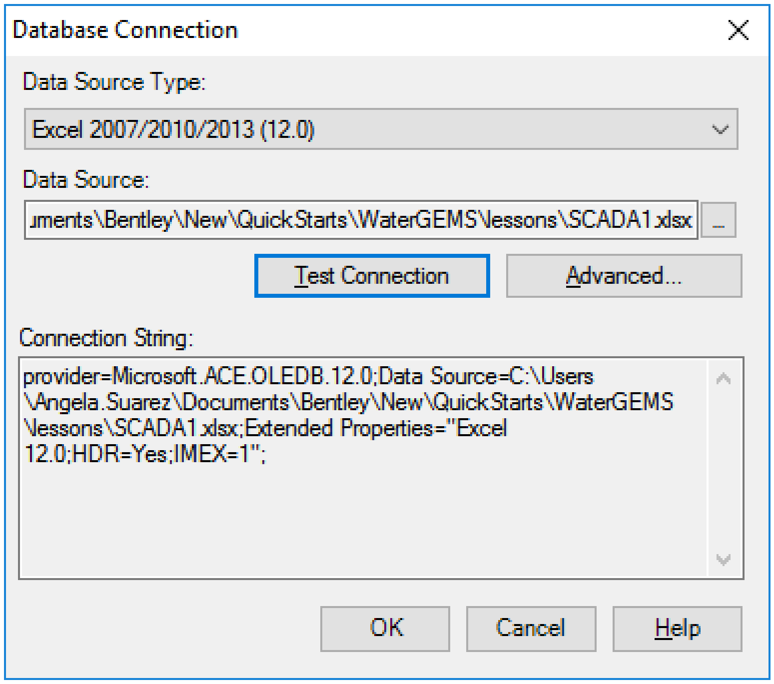
Identify SCADA Data Source

1. Open SCADA Signal Manager using **Analysis** > **SCADAConnect Simulator** > **SCADA Signals**.
2. Click **New** > **Database Source**. The following dialog opens

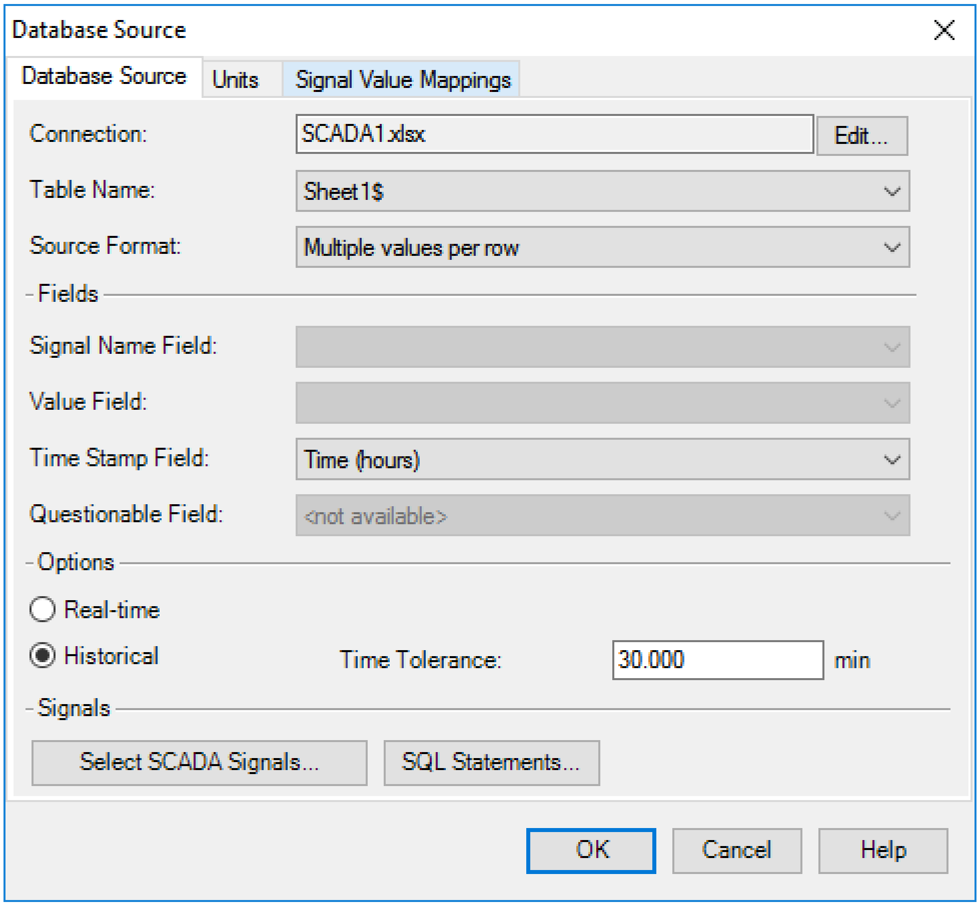


3. Click the **Edit**... button in front of **Connection**
4. In the Database Connection Dialog that comes up, set **Data Source Type** to **Excel2007/2010/2013 (12.0)**

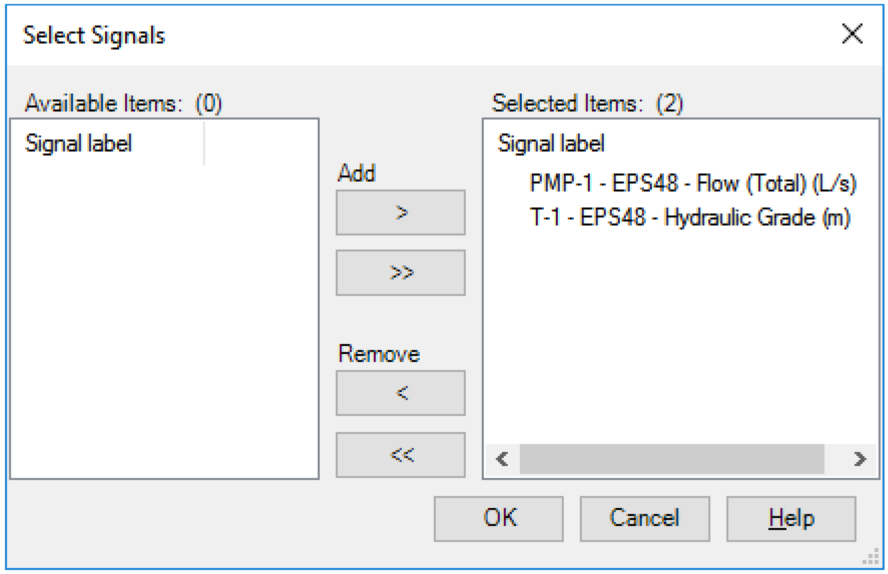
5. Set **Data Source** to **SCADA1.xlsx**. in the Lessons folder.



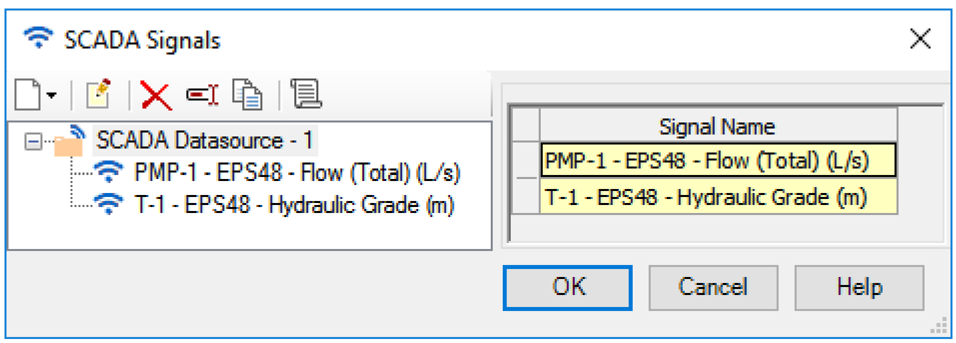
6. Pick **Test Connection.** It should respond **Connection Succeeded**.
7. Click **OK**, then close the Database Connection edit dialog.
8. Back in the Database Source manager, set the following:
   - Table name: **Sheet1$**
   - Source format: **Multiple values per row**
   - Time Stamp Field: **Time (hours)**
   - Options: **Hstorical**
   - Time Tolerance: **30 min**.


Database Source
Database Source
Units
Signal Value Mappings
Connection:
SCADA1.xlsx
Edit...
Table Name:
Sheet1$
Source Format:
Multiple values per row
Fields
Signal Name Field:
Value Field:
Time Stamp Field:
Time (hours)
Questionable Field:
<not available>
Options
Real-time
Historical
Time Tolerance:
30.000
min
Signals
Select SCADA Signals...
SQL Statements...
OK
Cancel
Help

9. Click **Select SCADA Signals** to identify which SCADA signals you want to use. In this case it is both values. Add them by clicking on each of them.

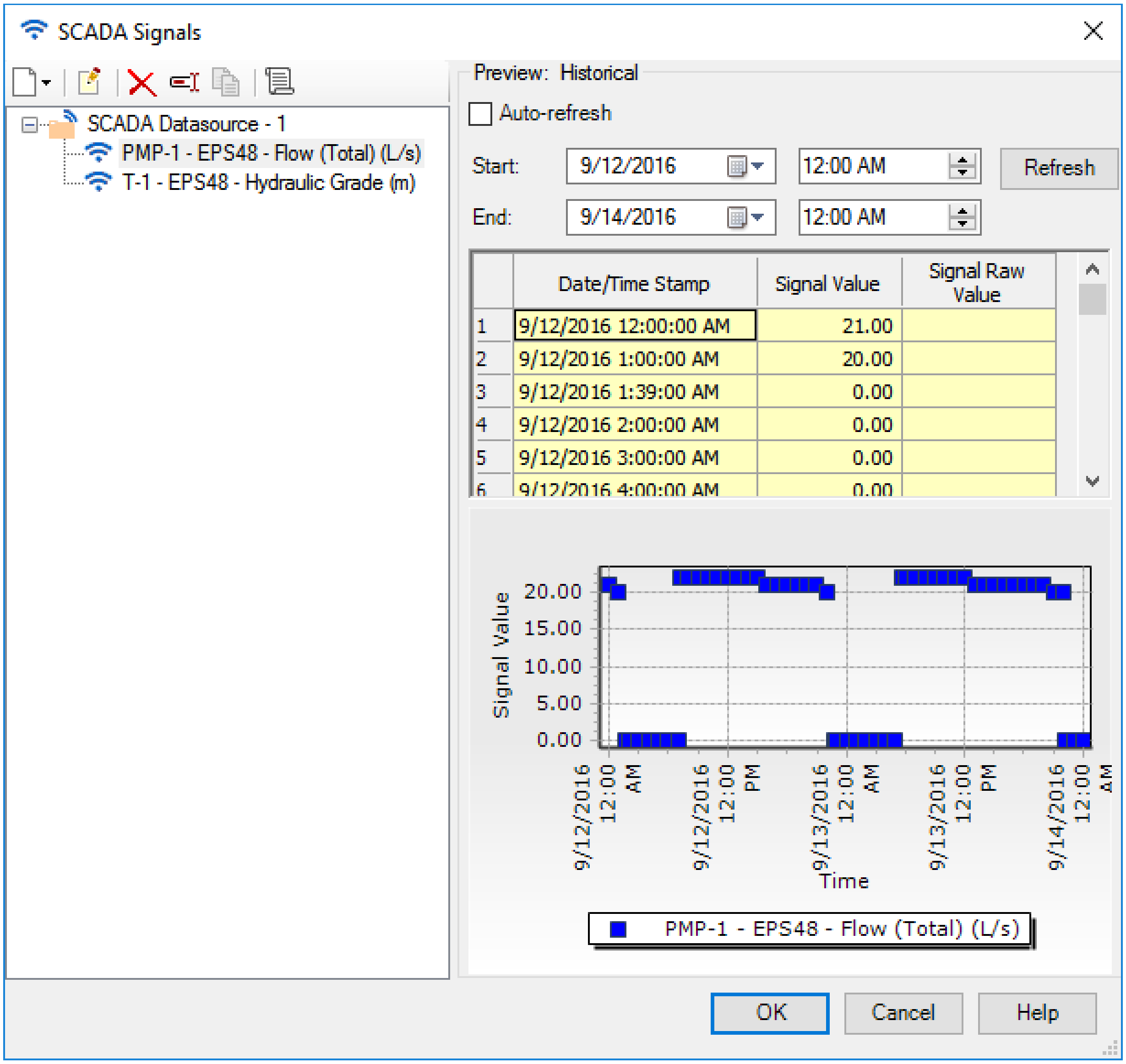


10. Click **OK** to leave signal selection.
11. Click **OK** to leave Database Source setup. This brings you back to SCADA Signals manager.

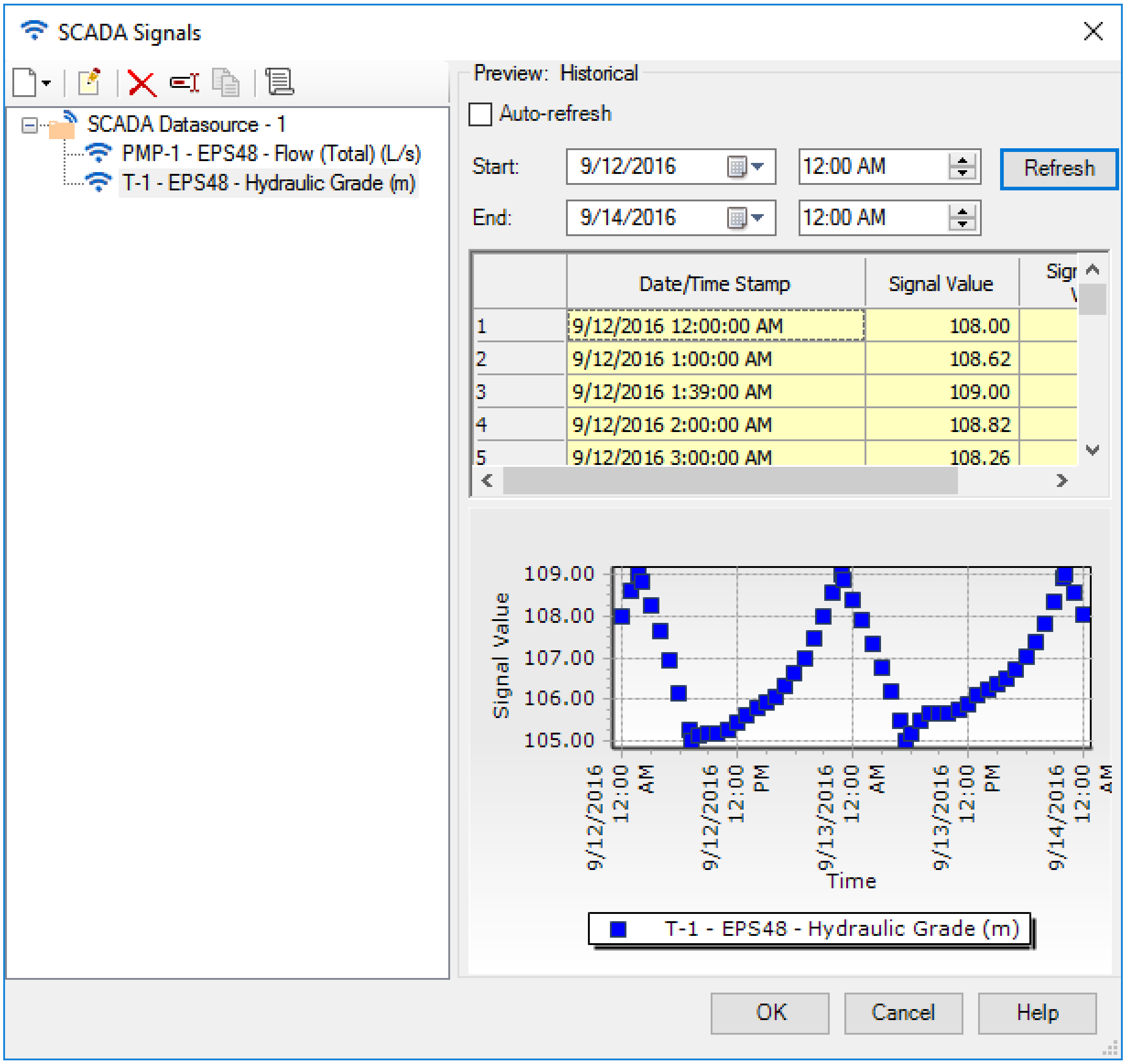


View SCADA Data in WaterGEMS

1. Click **PMP-1** and set **Start** to **9/12//2016 12:00 am** and **End** to **9/14/2016 12:00 am** and select **Refresh**



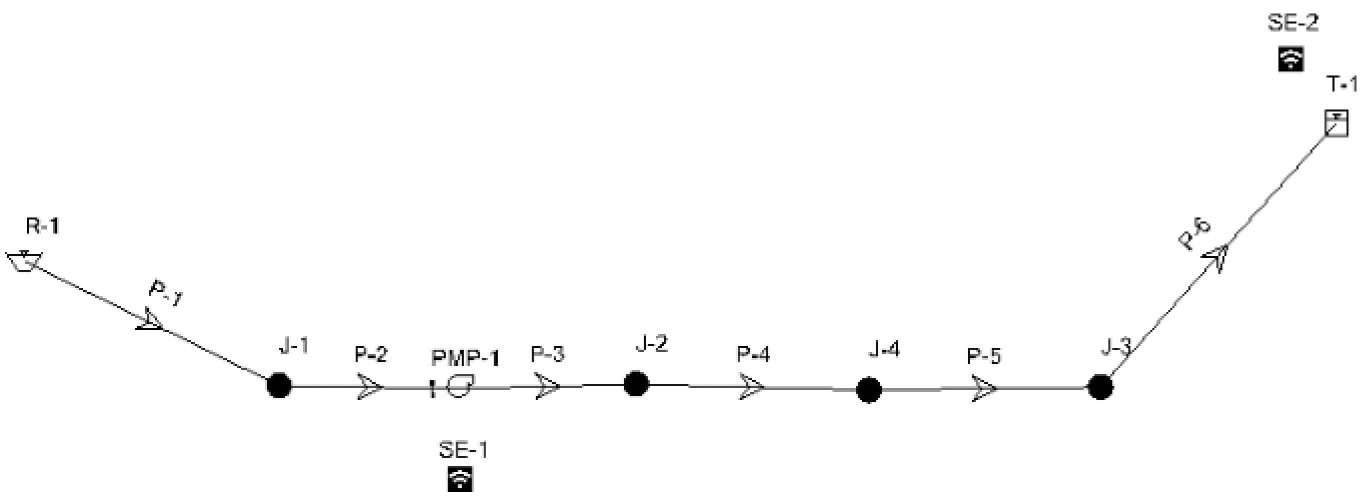
2. Pick **T-1** and repeat the above steps.



3. You have now imported the data. You must now associate that data with a model element using a SCADA signal element. Click **OK**.

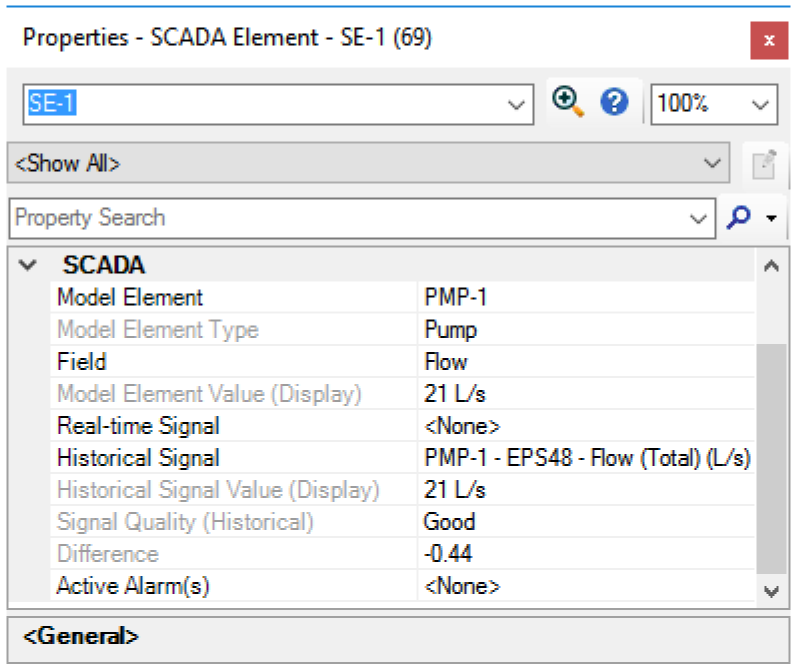
Creating and Mapping SCADA Signal Elements.

1. Place two SCADA elements using **Layout > SCADA Element** near the tank and pump as shown below.

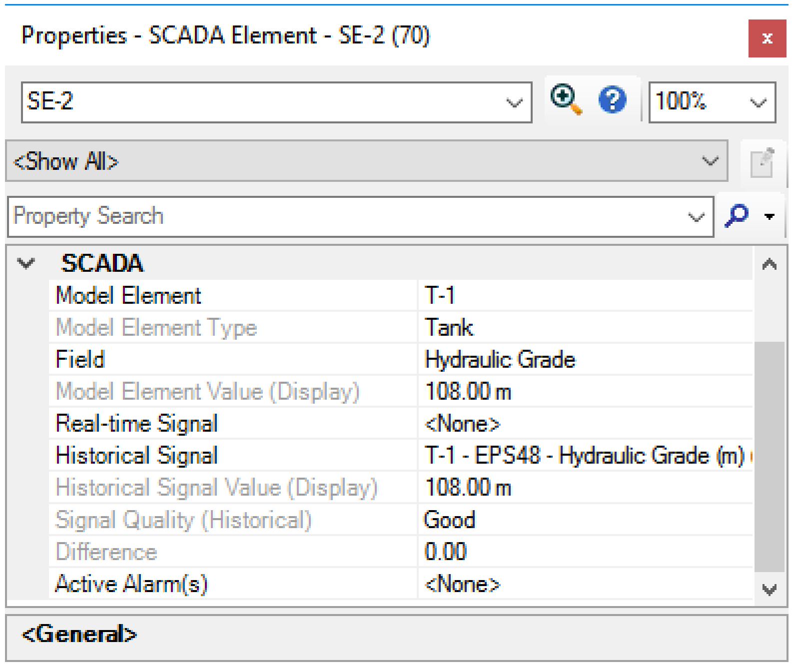


2. Make sure the labels in your model match the labels in this lesson.

3. Associate the signals to the pump using the **SE-1** Property Editor as shown below, by double clicking on **SE-1**"
   - Model Element: **PMP-1**
   - Field: **Flow**
   - Historical Signal: **PMP-1 - EPS48 - Flow (Total) (L/s)**
   - Active Alarm: **None**



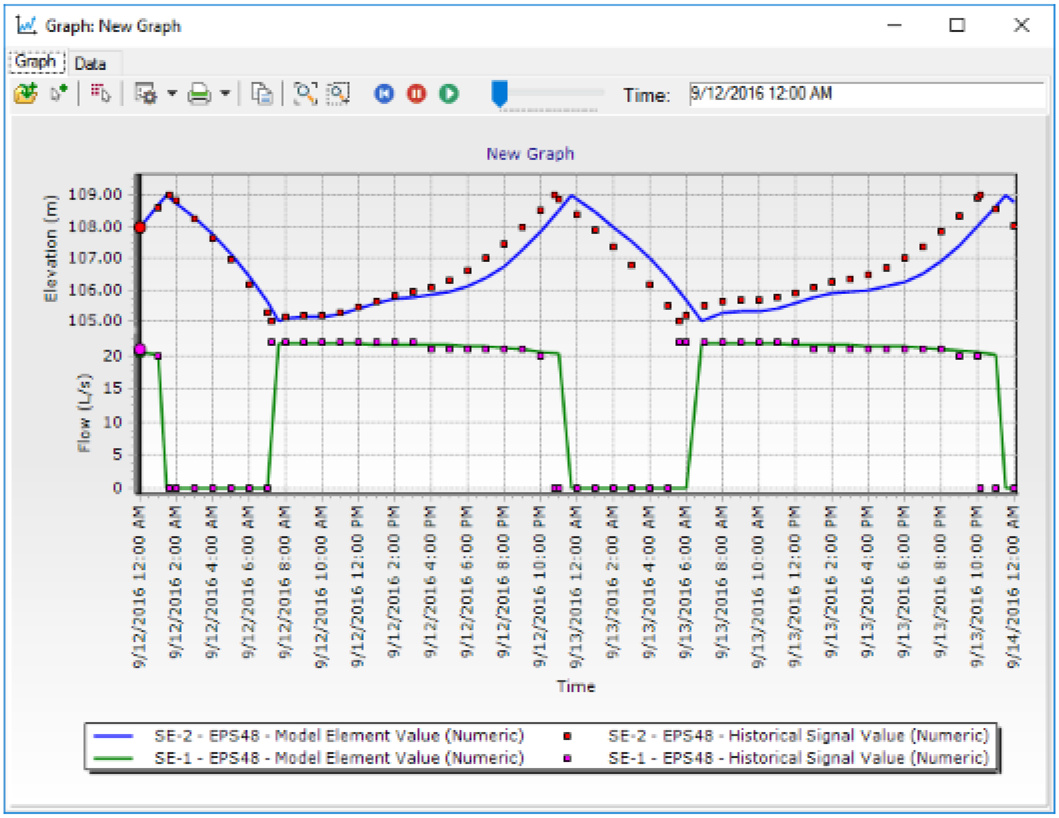
4. Associate the signals to the tank using the **SE-2** Property Editor as shown below:
   – Model Element: **T-1**
   – Field: **Hydraulic Grade**
   – Historical Signal: **T-1 - EPS48 - Hydraulic Grade (m)**
   – Active Alarm: **None**



5. Note the Historical signal values.
6. Close the Property Editor

Compare Model and SCADA Data

1. Compare calculated model results with SCADA data by graphing each SCADA element. Select both signal elements, **SE-1** and **SE-2,** right click and Select **Graph**.
2. Leave the default fields and click **OK**.

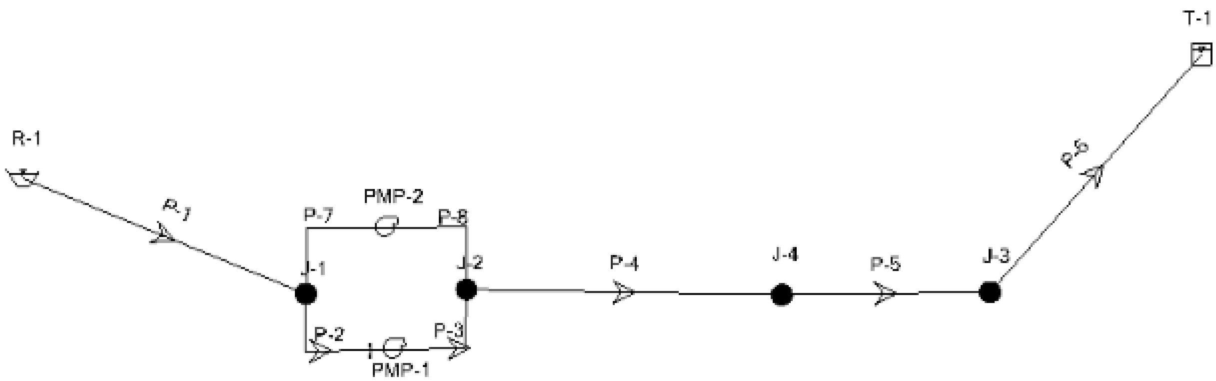


3. Note that the SCADA values (points) compare well with the calculated model results (lines).

SCADAConnect Simulator

In this lesson, you will simulate a fire event and the response in a water system using the simple user interface in SCADAConnect Simulator. You will make three runs: no fire, fire and fire response.

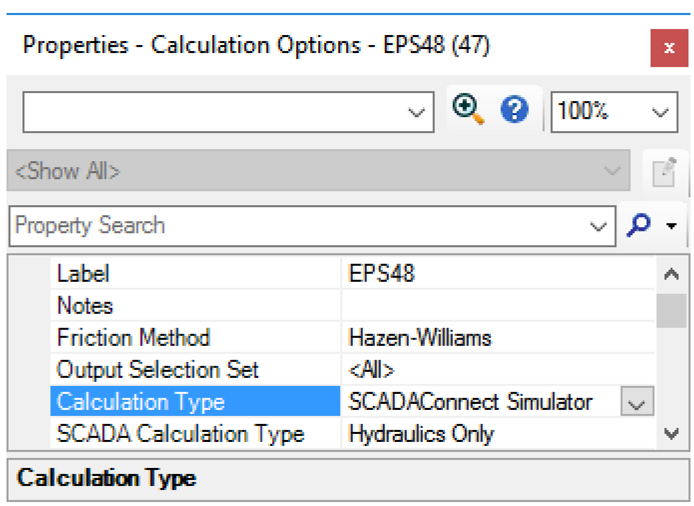
1. Start WaterGEMS and open the file **SCADASimStart.wtg** in the Lessons folder. It should look like the model below.



2. Click **OK** to the message dialog that comes up about CONNECTED Projects.
3. Click **File > Save As**. Name the file **MySCADAConnectRun**

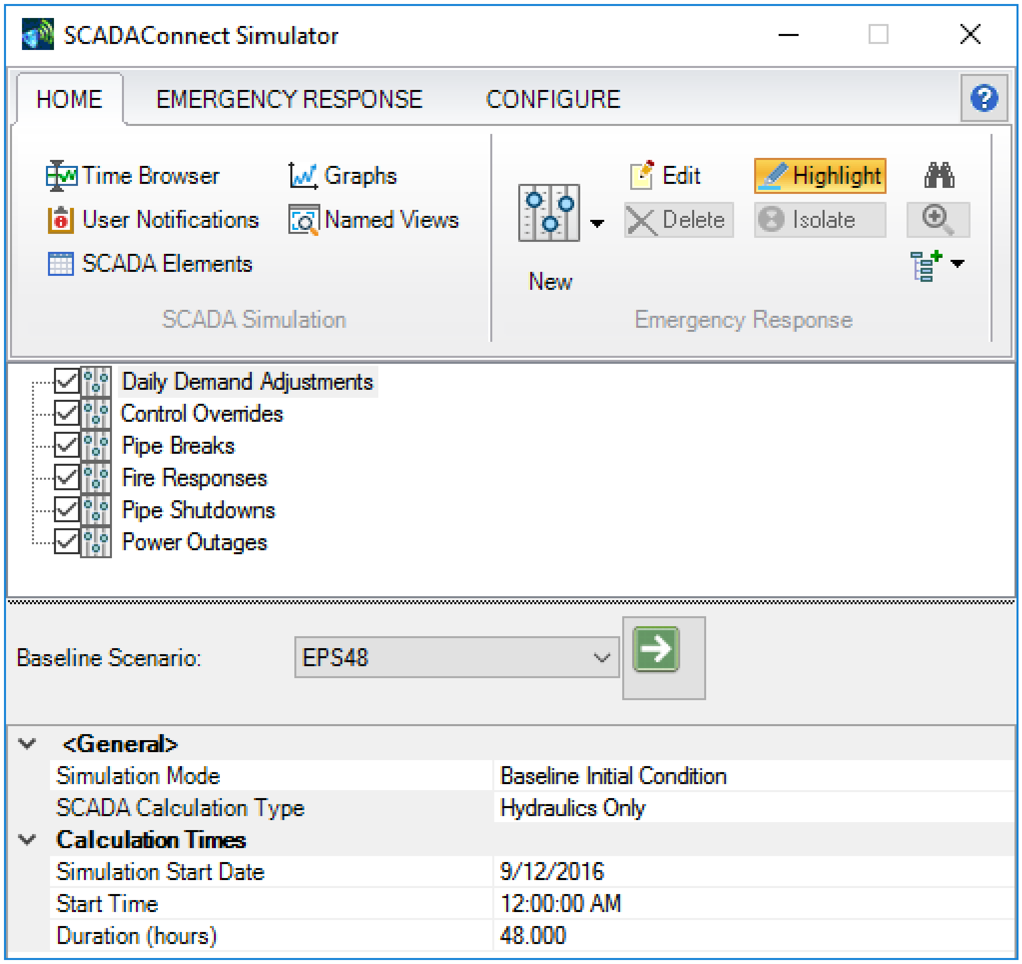
PMP-1 controls are based on the level in tank T-1. PMP-2 is a standby pump.

4. Change the existing **EPS48** scenario into a SCADAConnect simulator scenario by selecting **Analysis > Options > EPS48**. Change the **Calculation Type** to **SCADAConnect Simulator,** then close the Property Editor, and Calculation Options manager.



Be sure to occasionally save your file

5. Open SCADAConnect Simulator using **Analysis** > **SCADAConnect Simulator** > **SCADAConnect Simulator.** Review the manager that appears.

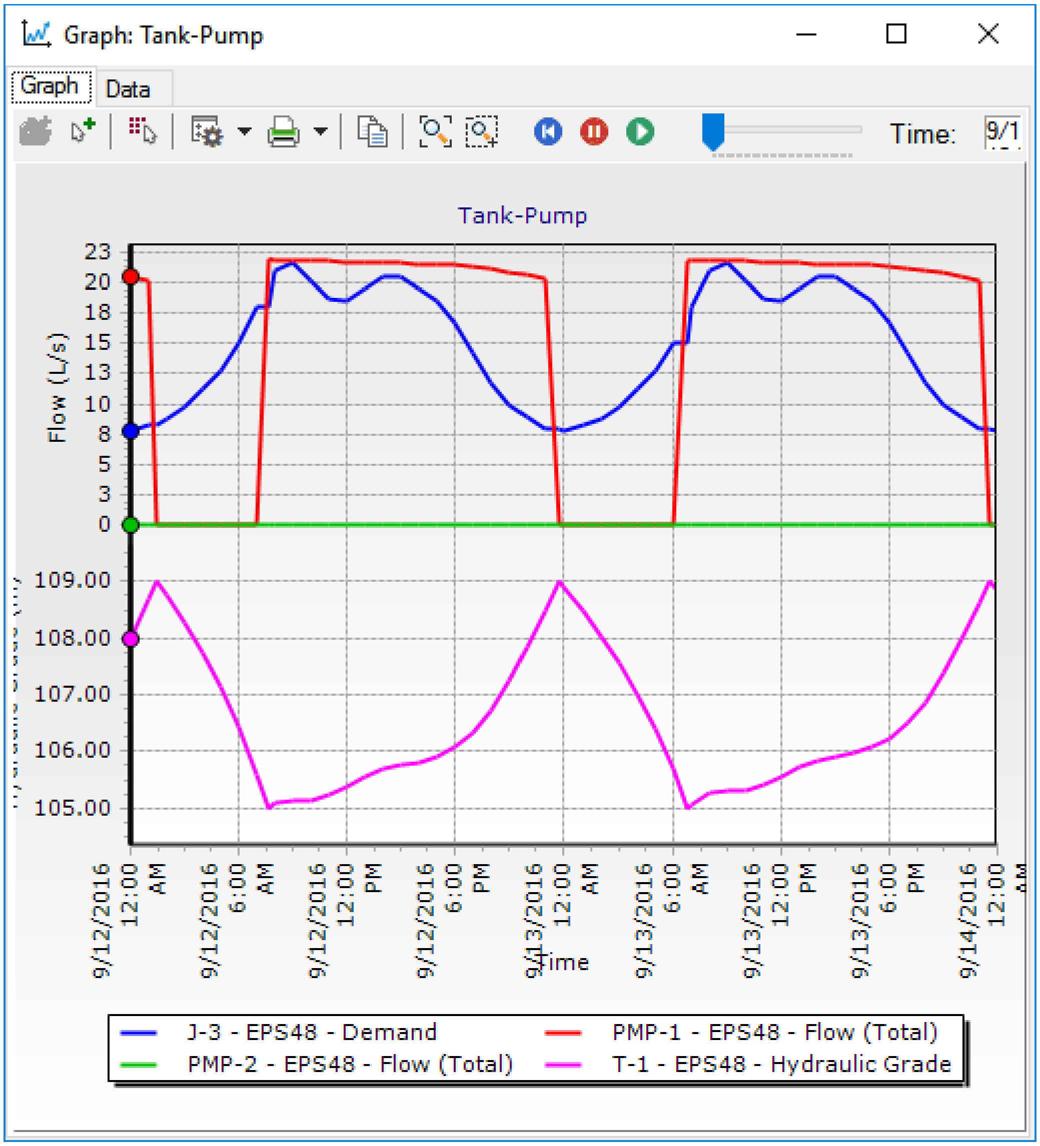


6. Run a normal day by selecting the **Compute** button on the **EPS48** Baseline Scenario.

7. The graph of the Calculation Summary (third button on top of Calculation Summary) should look like this.

8. Close the graph and the Calculation Summary.

9. Open the graph **View > Graphs > Tank-Pump** which should look like this where the red and green lines are pump flows, the blue line is system demand and the magenta graph is tank hydraulic grade line.

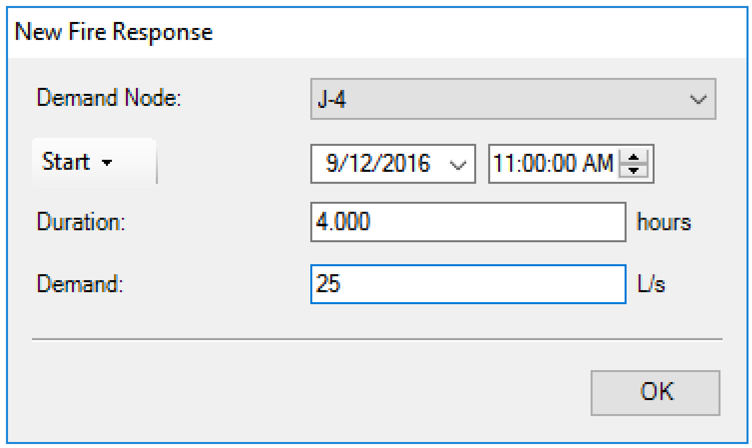


10. Close the graph and the Graphs Manager.

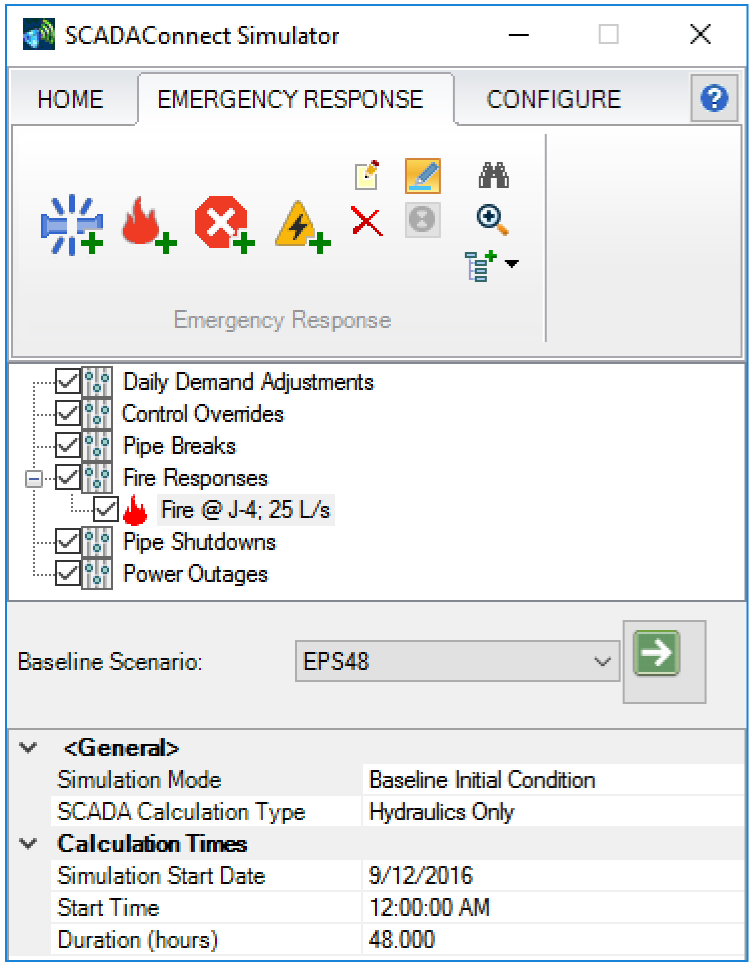
Simulating a Fire Response

Next you will simulate a fire flow at **J-4** of **25 L/s** from **11 am to 3 pm** on **9/12/2016**.

1. Back in the **SCADAConnect Simulator**, click the **Emergency Response** tab, then the **Fire Response** icon. 
2. Select junction **J-4** from the drawing and complete the dialog as shown below.
3. Click **OK**.

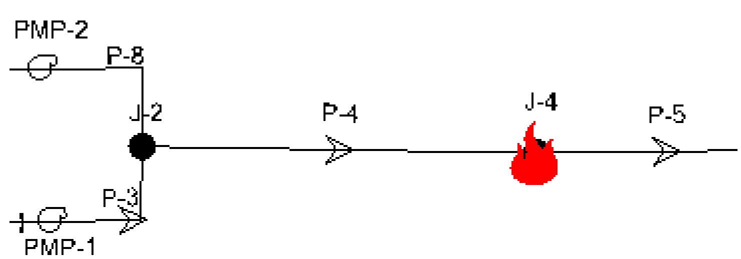


4. The SCADAConnect Simulator dialog will show the fire flow.



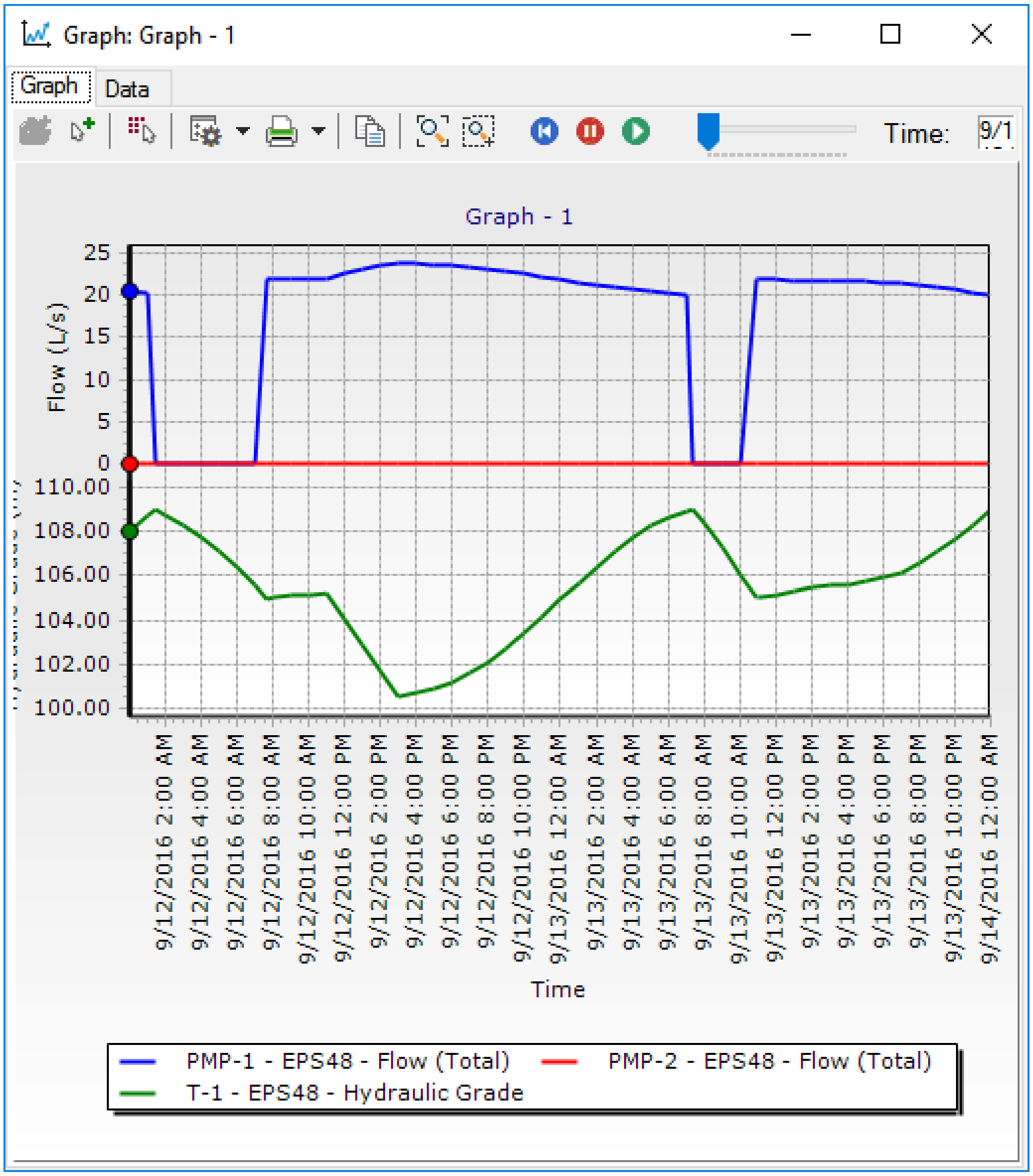
5. Click **Compute**. Close the Calculation Summary.

6. Notice that a flame has been placed on J-4 (in the drawing) to indicate the location of the fire simulation.



7. After the run, select **View > Graphs** and create a **New** graph. Select **PMP-1, PMP-2** and **T-1** from the drawing. Click the green check mark to complete the selection. Leave the default graph fields (Flow for Pumps and HGL for Tanks) and click **OK.**

8. The graph will look like this.

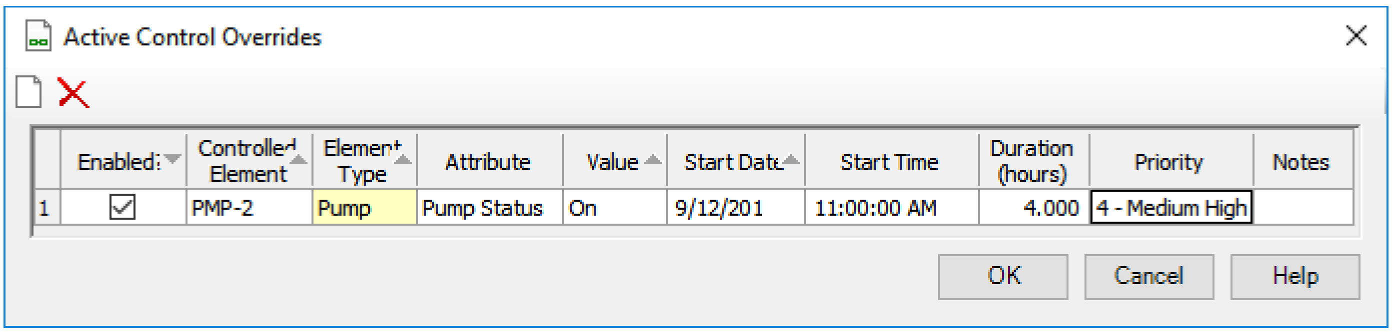


Notice that the HGL in the tank will drop very low, near the tank bottom at 100 m.

9. Close the graph and Graphs Manager.This graph will be automatically saved as **Graph -1.**

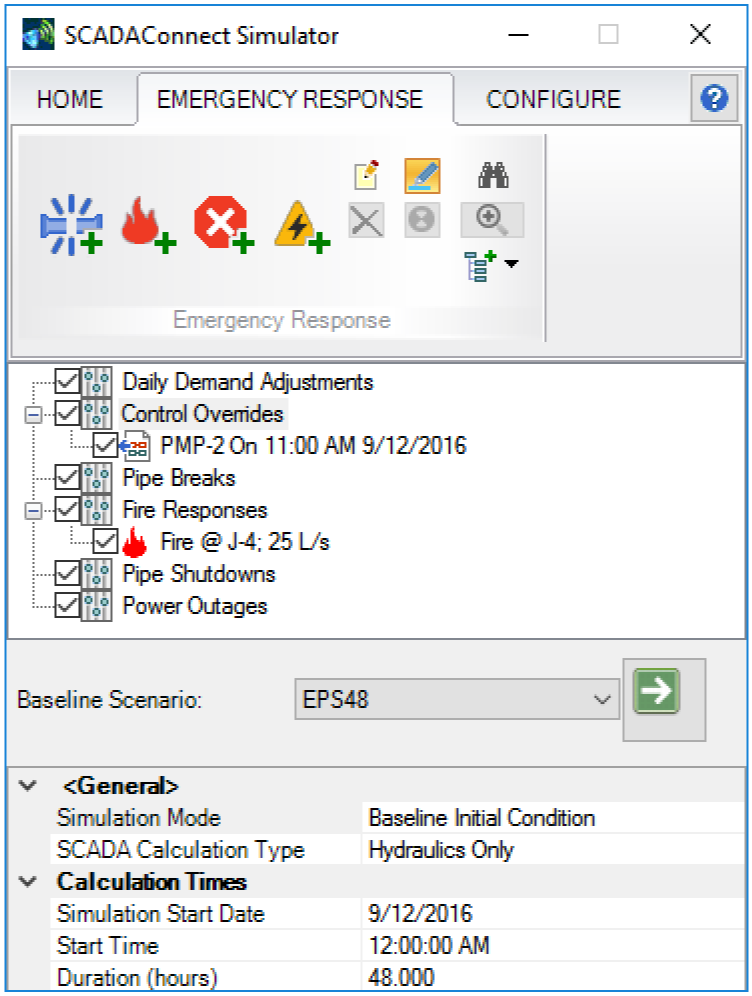
You would like to prevent the tank level from dropping this low. To do this, you will turn on pump PMP-2 at ll:00 am and leave it on during the fire. To do this, you will need to override the pump control for PMP-2.

10. Back in the **SCADAConnect Simulator**, Double click on **Control Overrides** and click **New** under Active Control Overrides. Set the following
    - Controlled Element: **PMP-2**
    - Attribute: **Pump Status**
    - Value: **On**
    - Start Date: **9/12/2016**
    - Start Time: **11:00:00 AM**
    - Duration: **4 hours**
    - Priority: 4 - Medium High.



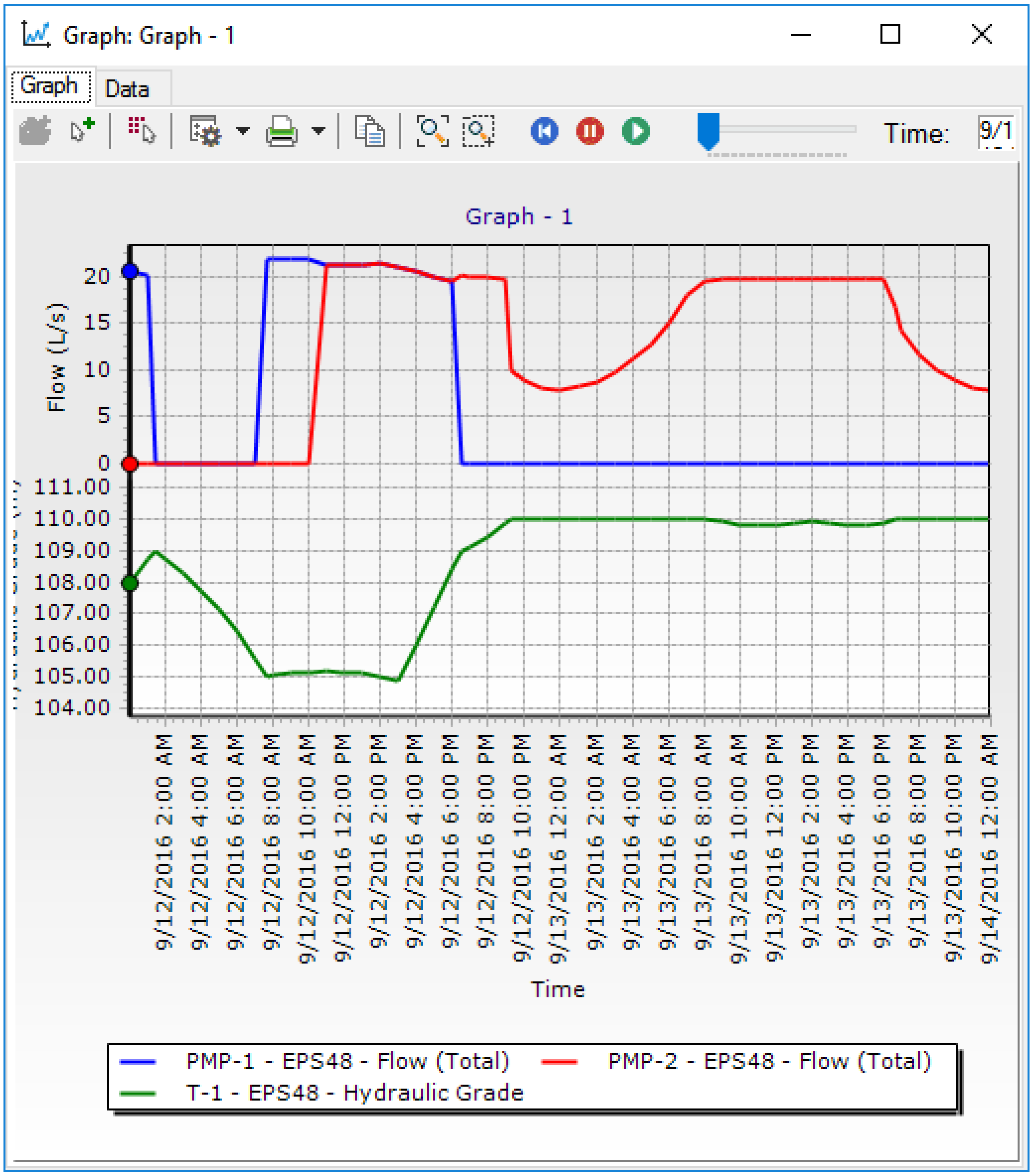
11. Click **OK**.

12. The SCADAConnect Simulator should look like this:



13. Click **Compute**. Review any user notifications, then close User Notifications and Calculation Summary.

14. Look again at the previously created graph. **View > Graphs > Graph -1.**



With PMP-2 running, the water level in T-1 stays in a reasonable range as shown in the graph.