Chapter 6 Tutorial Examples – WaterGEMS

The following tutorials give step-by-step instructions on how to solve example problems using the WaterGEMS CONNECT Edition Update 3 computer program developed by Bentley Systems.

Tutorial 1 – Three Pumps in Parallel

Problem Statement

A pump station is designed to supply water to a small linen factory. The factory, at an elevation of 58.0 meters, draws from a circular, constant-area tank at a base elevation of 90.0 meters with a minimum water elevation of 99.0 meters, an initial water elevation of 105.5 meters, a maximum water elevation of 106.5 meters, and a diameter of 10.0 meters.

Three main parallel pumps draw water from a source with a water surface elevation of 58.0 meters. Two pumps are set aside for everyday usage, and the third is set aside for emergencies. Each pump has a set of controls that ensure it will run only when the water level in the tank reaches a certain level. Use the Hazen-Williams equation to determine friction losses in the system. The network layout is given in Figure 6-1; the pump and pipe data are given in the tables below.

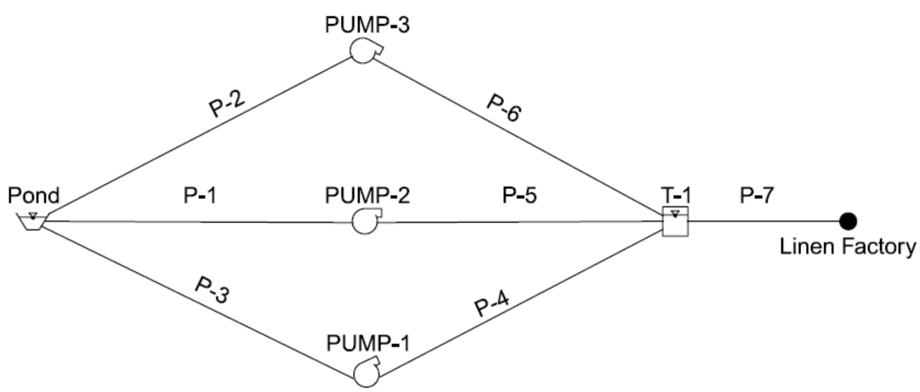


Figure 6-1: Schematic of Example Problem

a) Can the pumping station support the factory's 20 l/s demand for a 24-hour period?

b) If there were a fire at the factory that required an additional 108 l/s of water for hours 0 through 6, would the system with the pump controls given in the problem statement be adequate? Supply the Extended Period Simulation report describing the system at each time step.

c) How might the system be operated so that the fire flow requirement in part (b) is met?

Pipe Information for Tutorial Problem

Pipe	Length (m)	Diameter (mm)	Material	Roughness
P-1	6	150	Cast Iron	90
P-2	6	150	Cast Iron	90
P-3	6	150	Cast Iron	90
P-4	71	150	Cast Iron	90
P-5	72	150	Cast Iron	90
P-6	73	150	Cast Iron	90
P-7	18	200	Cast Iron	90

Pump Information for Tutorial Problem

Pump	Elevation (m)	Pump Curve		Controls
		Flow (l/s)	Head (m)	
PUMP-1	57.0	0 32 63	78.0 58.5 0	On when T-1 is below 105.5 meters Off when T-1 is above 106.0 meters
PUMP-2	57.0	0 32 63	78.0 58.5 0	On when T-1 is below 105.2 meters Off when T-1 is above 106.0 meters
PUMP-3	57.0	0 32 63	67.0 50.3 0	On when T-1 is below 99.25 meters Off when T-1 is above 103.00 meters

PART (a): *Can the pumping station support the factory's 20 l/s demand for a 24-hour period?*

Solution

Setting up the Project

When you start WaterGEMS, you should be prompted with the **Welcome to WaterGEMS** dialog. From this dialog, you can access Quick Start Lessons, create new projects, and open existing projects. Select **Create New Hydraulic Model**. If you are asked to **Enable Change Tracking?**, select **No**.

If the **Welcome to WaterGEMS** dialog does not appear, WaterGEMS is set to **Hide Welcome Page on startup**. To start a new project, select **New** from the **File** tab. You can change from **Hide Welcome Page** mode to **Show Welcome Page** mode in the **Global Options** dialog, which is accessible by selecting **Options** from the **Tools** tab.

As always when starting a new project, the file should be saved frequently to avoid losing data or simulations. To save a new project, select **Save As** under the **File** tab. Enter your project title **Tutorial 1** and at any time you can save your project by clicking the **Save** button .

A more descriptive project title and other general information can be entered into the **Info** window found under the **File** tab.

Before starting, you should setup the general default settings for the project. You can find the default settings in **Options** under the **Tools** tab. In the **Drawing** tab select **Schematic** from the **Drawing Mode** field. This option will allow you to define the pipe lengths and node locations without having to worry about scale and spatial placement on the x-y plane.

To define the default units go to the **Units** tab found under the **Options** tab. Select **SI** (System International) from the list box in the **Results Default** field if it is not already selected. You can define any of the default label units by clicking the unit field and selecting the desired unit from the list. For example, to change the **Angle** units from radians to degrees, click on **Radians** in the unit field, then select **Degrees** by locating it on the dropdown list of available units.

Laying Out the System

Begin with the pipeline running horizontally through the center of the system. Because you selected **Schematic** in the **Drawing Mode** field, you do not have to layout the system exactly as shown in the problem statement. You can roughly sketch the schematic by following the instructions here. You will likely have to rename many of the elements to match the names shown in Figure 6-1. The steps below will describe how to do this.

Click the **Layout** tab to select different components for your system. Select **Reservoir**. To place the reservoir, simply click the left mouse button in the layout area.

Select **Pipe** and place the crosshair of the cursor directly over the reservoir in your layout area. Left-click then move away from the reservoir to place your first pipe. Right-click to open a menu to select **Pump**, then place your first pump. Right-click again to select **Tank** and place the tank. Right click again to select **Junction** to place the final junction (Linen Factory). When you placed the final junction, right-click to select **Done**.

Continue by entering the remaining two pumps and four pipes in the same way as described previously. To connect a pipe to an object already placed in the layout area, click the object with the cursor crosshair while in the pipe layout mode. The object should turn red when it is selected.

To exit the pipe layout mode, click the **Select** button on the Layout tab.

Except for the scale, your schematic should look roughly like Figure 6-1 in the problem statement. If a pump is reversed (flowing in the wrong direction), double-click on the pump to open the **Properties** window; you can set the pumping direction in the **Downstream Pipe** line.

To rename pipes to match Figure 6-1, double-click a pipe that is be renamed to open its **Properties** window. You can rename a pipe by typing the new name in the **Label** field.

Entering the Data

Double-click the reservoir node to open its dialog editor. Change the name to "Pond" in the **Label** field. Enter 58 m in the **Elevation** field. Close the dialog editor.

Double-click the tank. Enter the given diameter for the circular section and the appropriate elevations from the problem statement. Disregard the inactive volume field. Be sure that **Elevation** is selected in the **Operating Range Type** field. Close the dialog editor.

Double-click the bottom pump. Change the name to "PUMP-1" in the **Label** field. Enter the appropriate elevation from the pump data table in the problem statement into the **Elevation** field. Click the **Pump Definition** field and select **Edit Pump Definitions** to open the Pump Definitions dialog. Add a new pump definition and label it "Pumps 1 and 2". In the **Head** tab select **Standard (3 Point)**. Enter the pump curve data given for PUMP-1. If you need to change the units, right-click on the Flow or Head table headings and open the **Units and Formatting** dialog to change the units. Click **Close** to close **Pump Definition** dialog. Now select "Pumps 1 and 2" in the **Pump Definitions** field. Close the dialog editor.

Repeat the above process for the other pumps. When entering the data for PUMP-3, you will have to create a new pump definition titled "Pump 3" for the **Pump Definitions** field.

Enter the pump controls given in the problem statement. Click **Controls** on the **Components** tab.

Select the **Conditions** tab to enter the five Tank conditions as described from the problem statement information. Enter each condition as **New** and **Simple**. The **Condition Type** is **Element**; select the Tank from the layout screen by clicking the selection icon ; select **Hydraulic Grade** as the **Storage Attribute**; the **Operator** and **Hydraulic Grade** is entered based on the problem statement information.

Select the **Actions** tab to enter whether the pump is on or off. The default setting is generally with the pumps on. Enter the six actions (each pump either on or off) as **New** and **Simple**. For example, to turn off PUMP-1, the **Element** is entered by clicking the selection icon and selecting PUMP-1 from the layout screen; the **Pump Attribute** would be Pump Status; the **Operator** would be the default "="; then select Off for the **Pump Status**.

Select the **Controls** tab to enter the six controls. The controls are all Simple and entered as If/Then statements. For example, click **New** then click the **Evaluate as Simple Control** box. Each condition and action are available to select. To select a condition, click the arrow down icon next to the IF Condition field, then select **Find Condition** to open a window to select {"Tank" level > 106.00 m}. To select an action, click the arrow down icon next in the THEN Action field, then select **Find Action** to open a window to select {"PUMP-1" pump status = off}. After you enter all six control statements, close the **Controls** dialog.

Double-click the junction node. Change the name to "Linen Factory". Enter 58 m in the **Elevation** field. Click the **Demand Collection** field to enter a demand of 20 l/s as a **Fixed** demand pattern as after clicking the ellipse button. Close both dialog editors.

For the pipes, you can edit the data as you have been by clicking each element individually, and then entering in the appropriate data. However, this method can be time consuming, especially as the number of pipe elements increase. It is often easier to edit the data in a tabular format.

Click the **Flex Tables** button found on the **Home** tab. Select **Pipe Table** from the available tables.

The fields highlighted in the **Pipe Table** are output fields. The fields in white are input fields and can be edited as you would edit data in a spreadsheet.

Warning: If you did not rename each pipe, the pipes may not be listed in the table in numerical order, nor have the same labels as seen in Figure 6-1. You may want to re-label each pipe to match what is in Figure 6-1; look at the Start Node and Stop Node columns on the flex table to help properly label each pipe to match Figure 6-1. After you re-label each pipe, you may want to sort the pipe labels in ascending order. To do this, move the cursor to the top of the table and place it on the **Label** column. Right-click, select **Sort,** and then select **Sort Ascending.** The pipes should then be listed in numerical order.

Enter the correct pipe lengths into the **Length (User Defined)** column found on the Pipe Table. Also enter the pipe diameters and Hazen-Williams C value. Close the Pipe Table.

Note: You can customize which columns appear in the **Pipe Table** by clicking the **Edit** button in the toolbar at the top of the table. Table columns can be added or removed as desired. You can also move columns in the table using the double arrow icons and .

Running the Model

To run the model, first click the **Compute** button on the main toolbar. Arrows should appear on your layout screen indicating the flow direction in each pipe. If you click on any of the objects, you will see the results in the dialog. You could look at the results for all similar objects by opening the **Flex Tables** button. For example, if you want to look at flows in all the pipes, select the **Pipe Table**. To examine the flow through the system over a 24-hour period, select the **Options** under the **Analysis** or **Home** tab. Double-click the **Base Calculation Options,** then in the **Time Analysis Type** field select **EPS**. Set the start time to 12:00:00 AM and the duration to 24 hours. Adjust the **Hydraulic Time Step** to 0.01 hours. **Close** the **Calculation Options** window and then click the **Compute** button.

There are a couple of ways to determine whether your model meets the target demand:

Scroll through the **Calculation Summary** and check to see if there are any disconnected node warnings. When the level in Tank T-1 drops to the minimum tank elevation of 99 m (tank level of 9 m), the tank closes off, preventing any more water from leaving. This closure will cause the linen factory to be disconnected from the rest of the system (that is, it will not get the required 20 l/s).

-OR-

Close the Calculation Summary window and select the Linen Factory junction. To create a graph of the pressure at this node, click the Graphs button on the **Home** tab. Create a **Line-Series Graph** from the **New** button in the Graphs dialog. Select the **Pressure** box in the **Graph Series Options** window then close the options window. You should see the calculated pressure at the Linen Factory and notice that it never reached zero (no water pressure).

Answer

As you will see for this problem, all the pressures at the linen factory hover around 465 kPa, and no disconnected nodes are detected. If you make a graph of how full the tank is (Percent Full), you will see that the tank is never less than 86% full. Therefore, the pumping station can support the factory’s 20 l/s demand for a 24-hour period.

PART (b): *If there were a fire at the factory that required an additional flow of 108 l/s for hours 0 through 6, would the system with the pump controls given in the problem statement be adequate?*

Solution

Add another demand to the Linen Factory node. To do this, double-click the Linen Factory junction and enter into the **Demand Collection** field a second fixed demand of 108 l/s in the row below the 20 l/s demand after clicking the **Ellipse** button. Close the dialog editor.

Select the **Options** under the **Home** or **Analysis** tab, and then double-click **Base Calculation Options.** You only need to run this model for six hours, so change 24 to 6 in the **Duration (hours)** field. **Close** the **Calculation Options** window and click **Compute** to run the model.

As you scroll through the results, you will see warning messages (yellow or red indicators instead of green) indicating a disconnected node at the linen factory after 3 hours. Close the **User Notifications** window. Select the tank and create a **Line-Series Graph** of the water level in the tank by selecting **Level (Calculated)** in the **Graph Series Options** window**.** The graph indicates that the water level in the tank reaches the minimum level of 9 meters at about 3 hours and cannot supply water to the linen factory. **Note:** When the tank is empty, water flowing to the linen factory with drop to zero (disconnected node) but the pumps are still on giving the appearance that water level in the tank goes up – this is an artifact – the water level in the tank does not truly go up.

Answer

If there were a fire at the factory, the existing system would NOT be able to supply water after about 3 hours.

PART (c): *How might the system be operated so that the fire flow requirement in Part (b) is met?*

Solution

PUMP-3 could be manually switched on at the beginning of the fire to supply the flow necessary to fight the fire at the linen factory. To do this delete the pump controls for PUMP-3. Then PUMP-3 will be always on during the model simulation.

Open the Controls window by clicking **Controls** on the **Components** tab. Delete the two controls for Pump 3 by highlighting them one at a time and clicking **Delete** . **Close** the Controls window.

Click **Compute** to run the model. As you scroll through the **User Notifications** window, there should not be any warning messages indicating a disconnected node.

Select the tank and create a **Line-Series Graph** of the water level in the tank by selecting **Level (Calculated)** in the **Graph Series Options** window**.** The graph indicates that the water level in the tank continues to decrease but can supply water to the linen factory for the entire 6-hour period.

Answer

If pump 3 is set to turn on at the start of a fire alarm in the linen factory, water can be supplied for at least 6 hours.

Tutorial 2 – Water Quality

Problem Statement

A local water company is concerned with the water quality in its distribution network. The company wishes to determine the age and chlorine concentration of the water as it exits the system at different junctions. The water surface at the reservoir is 70 meters.

Chlorine is injected into the system at the source of flow, R-1, at a concentration of 1.0 mg/l. It has been determined through a series of bottle tests that the average bulk reaction rate of the chlorine in the system (including all pipes and tanks) is approximately –0.5/day.

The network model may be entered in WaterGEMS CONNECT Edition Update 3 using the layout in Figure 6-2 and the data that follows.

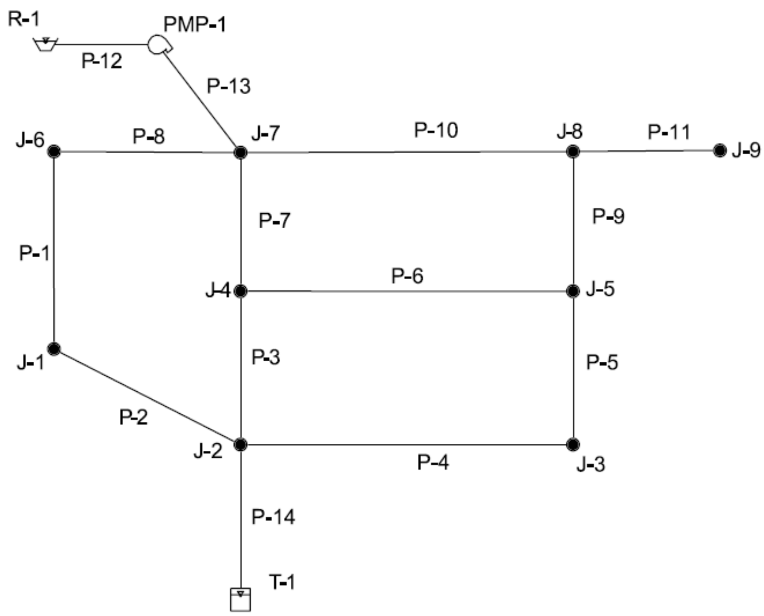


Figure 6-2: Schematic for Water Quality Tutorial

The tank is circular with a diameter of 15.0 meters. The minimum elevation is 99.0 meters. The maximum elevation is 104.0 meters, and the initial elevation is 103.4 meters. The base elevation is 98.0 meters, and the inactive volume is 10.0 m^3. The elevation of the pump is 69.0 meters and is initially on.

Pump Information for Water Quality Tutorial

Flow (l/min)	Head (m)	Controls
0	40	Off if node T-1 above 103.5 m On if node T-1 below 99.5 m
3,000	35	
6,000	24	

Continuous Demand Pattern Data for Water Quality Tutorial

Time from Start (hr)	Multiplier
Start	0.80
1	0.60
2	0.50
3	0.50
4	0.55
5	0.60
6	0.80
7	1.10
8	1.50
9	1.40
10	1.30
11	1.40
12	1.40

Time from Start (hr)	Multiplier
13	1.30
14	1.40
15	1.50
16	1.60
17	1.80
18	1.80
19	1.40
20	1.20
21	1.00
22	0.90
23	0.80
24	0.80

Junction Data for Water Quality Tutorial

Junction	Elevation (m)	Demand (l/min)
J-1	73	151
J-2	67	227
J-3	85	229
J-4	61	212
J-5	82	208
J-6	56	219
J-7	67	215
J-8	73	219
J-9	55	215

Pipe Data for Water Quality Tutorial

Pipe	Length (m)	Diameter (mm)	Roughness
P-1	300	200	130
P-2	305	200	130
P-3	225	200	130
P-4	301	200	130
P-5	225	200	130
P-6	301	200	130
P-7	225	200	130
P-8	301	200	130
P-9	200	200	130

P-10	301	200	130
P-11	300	200	130
P-12	1	250	130
P-13	3,000	300	130
P-14	300	300	130

a) Perform an age analysis on the system using a duration of seven days and a time step of one hour. Determine the youngest and oldest water in the distribution system and the storage tank. Explain why water age varies.

b) Perform a constituent analysis using the same duration and time step as in part (a). Determine the range of concentrations in the system and the storage tank. Explain the behavior of the system with regard to chlorine.

c) Are the simulation results consistent with the known behavior of chlorine?

d) Why is it necessary to run the model for such a long period of time? Do you feel seven days is too long or too short a time period to test the model? Why?

<u>Solution</u>

Use the same steps as in Tutorial 1 to setup the project, layout the system, and enter the data for the reservoir, pump, tank, all pipes and all junctions. Be sure to set units to **System International** and the drawing mode to **Schematic.** You will likely have to rename many of the elements after you draw the general layout to make sure data is correctly entered for each element. It is easier to rename the junctions before renaming the pipes to match Figure 6-2. If you need to change the units, right-click over the line heading and open the **Units and Formatting** dialog to change the units.

The pump definition and control dialogs are found on the **Components** tab. Refer to Tutorial 1 to review how to enter data into these windows.

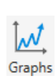
The demand pattern data can be entered by selecting **Patterns** on the **Components** tab. Right-click **Hydraulic** to select **New**. The Start Time is **12:00:00 AM**, the Starting Multiplier is **0.80**, and the Pattern Format is **Continuous**. Enter the data from the problem statement table under the **Hourly** tab.

The demand pattern can be assigned to a selected junction by clicking a junction and entering **Hydraulic Pattern – 1** into the **Demand Collection** field, or the pattern can be assigned to all junctions as a Global Edit. In this case, assign the demand pattern to all junctions by selecting **Demand Control Center** on the **Components** tab. Click **Yes** to continue. On the **Junctions** tab, right-click the **Pattern (Demand)** table heading to select **Global Edit**. Select **Hydraulic Pattern – 1** in the **Value:** field. You can also easily input the **Demand (Base)** values for each junction in this window. Click **Close** and close the Demand Control Center dialog.

Base Scenario

Run the model for a 24-hour period by selecting the **Options** under the **Analysis** or **Home** tab. Double-click the **Base Calculation Options** then in the **Time Analysis Type** select **EPS.** The **Duration (hours)** is 24 hours and the **Hydraulic Time Step (hours)** is 1.0 hour. Close the Base Calculation Options dialog.

Click on the **Scenarios** icon. Notice that the Compute button is in the Scenario window toolbar. Click the **Compute** button. WaterGEMS calculates the system parameters for a 24-hour simulation period. Details of the calculation can be viewed on the **Calculation Summary** window. **Close** the **Calculation Summary** and **Scenarios** windows.

Click on the tank then the Graphs button 

on the **Home** tab. Create a **Line-Series Graph** from the **New** button in the Graphs dialog. Select the **Percent Full** box under "Results" under the **Fields** field in the **Graph Series Options** window. Close the Graph Series Options window. Size the graph window to fit on the layout screen such that you can see most of the layout and the toolbar in the Graph window that contains the **Play** button

. Play the 24-hour simulation by clicking the **Play** button in the Graph window. Pipe flows are indicated by the arrows. Note that that there is no flow in pipes P-12 and P-13 when the pump is not operating. The flow direction reverses in pipes P-1, P-2, P-3, P-4, P-5, P-7, P-8, P-9, and P-14 over the 24-hour period. The volume of the water in tank T-1 is indicated by the **Percent Full (%)** on the y-axis of the graph.

Age Analysis

Click **Options** to open the Calculation Options window. Duplicate by highlighting the Base Calculations Options line then clicking the **Duplicate** button and a copy of **Base Calculation Options** will be created. With the copy highlighted, click **Rename** and change the copy's name to "Age Analysis Calculation Options". Double-click the age calculation options you just created and select **Age** in the **Calculation Type** field. The **Duration** is 168 hours (7 days), and the **Hydraulic Time Step** is 1.00 hr.

The analysis of the age of water within the network may be performed by defining and running an age analysis scenario. From the **Analysis** or **Home** tab, select **Scenarios**.

Create a new **Base Scenario** by clicking the **New** button. Enter "Age Analysis" as the name of the scenario.

Right-click the Age Analysis scenario, and select **Make Current**. The red check should now be on the Age Analysis scenario. Double-click on the Age Analysis scenario and select the Age Analysis Calculation Options in the **Steady State/EPS Solver Calculation Options** field.

From the Scenarios dialog, click the **Compute** button.

Close the Calculation Summary window and Scenarios dialog to view the layout screen.

Results: The oldest water in the network will be found in tank T-1. Click on the tank then the **Graphs** button in the **Home** tab. Create a **Line-Series Graph** from the **New** button in the Graphs dialog. Select the **Age Analysis** box in the **Scenarios** field and **Age (Calculated)** found under "Results (Water Quality)" in the **Fields** field in the **Graph Series Options** window. You should also unclick any other selected lines in the **Fields** field. Close the options window. The resulting graph is shown in Figure 6-3.

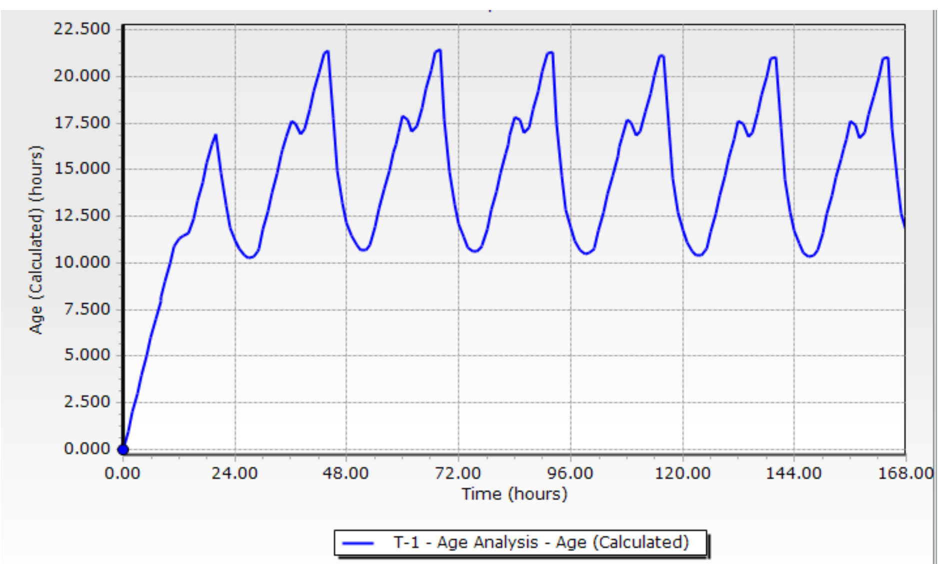


Figure 6-3: Age of water in Tank T-1

Note that the water distribution network reaches dynamic equilibrium after two days of the simulation. After 48 hrs, the maximum age at T-1 is approximately 21.5 hours, and the minimum age is approximately 10.5 hours.

To view the variation of water age in the network, click on the tank, J-2, J-3, J-7, and J-9 while holding the shift on the keyboard to select each object. Then click the **Graphs** button on the **Home** tab. Create a **Line-Series Graph** from the **New** button in the Graphs dialog. Select the **Age Analysis** box in the **Scenarios** field and **Calculated Age**, both under Tank and Junction in the **Fields** field in the **Graph Series Options** window. You should also unclick any other selected lines in the **Fields** field. Close the options window to look at the graph. Notice that the water age at the junctions is much less (2 to 4 hours) than the water in the tank while the pump is on and feeding fresh water into the system. Then the water age at the junctions greatly increases when the system is fed by the tank water after the pump turns off.

Water Quality Analysis

In order to analyze the behavior of chlorine in the network, the properties of chlorine must be defined in the engineering library.

From the **Components** tab, select **Engineering Libraries**. Expand Constituent Libraries, then right-click on ConstituentLibrary.xlm and select **Save As**. Type "Chlorine" then click **Save**. Select **No** if ask to unregister ConstituentLibrary.xlm. Rename "Constituent" under the new Chlorine.xlm line by right-clicking and renaming it "Chlorine".

Click **Chlorine** under the Chlorine.xlm library to open the properties window. Enter a **Diffusivity** value of 1.122e-010 m^2/s. Enter the **Bulk Reaction Order** as 1 and the **Bulk Reaction Rate** as –0.5 $(mg/l)^{1-n}$/day. Because n =1, the units of the rate constant are day^{-1}. **Close** the **Engineering Libraries**.

Click **Options** from the **Analysis** or **Home** tab to open the Calculation Options window. Duplicate by highlighting the Age Analysis Calculations Options line then clicking the **Duplicate** button and a copy of **Age Analysis Calculation Options** will be created. With the copy highlighted, click **Rename** and change the copy's name to "Chlorine Analysis Calculation Options". Double-click the chlorine calculation options you just created and select **Constituent** in the **Calculation Type** field. The **Duration** is 168 hours (7 days), and the **Hydraulic Time Step** is 1.00 hr. **Close** the Calculation Options dialog.

The analysis of the chlorine in the water within the network may be performed by defining and running a constituent analysis scenario. From the **Analysis** or **Home** tab, select **Scenarios**.

Create a new **Base Scenario** by clicking the **New** button. Enter "Chlorine Analysis" as the name of the scenario.

Right-click the Chlorine Analysis scenario, and select **Make Current**. The red check should now be on the Chlorine Analysis scenario. Double-click on the Chlorine Analysis scenario and select the Chlorine Analysis Calculation Options in the **Steady State/EPS Solver Calculation Options** field. **Close** the Scenarios window.

Click **Alternatives** found under the **Home** or **Analysis** tab. Expand on **Constituent**; then double-click on the **Base Constituent** alternative to open the Constituents dialog. On the **Constituent System Data** tab click the ellipse button . Click the **Synchronization Options** button and select **Import from Library**. Check the box for **Chlorine** from the **Constituent Libraries** list then click **Select**. The Chlorine properties should be added to the dialog. Close the **Constituents** dialog. Select **Chlorine** in the **Constituent** field on the **Constituents: Base Constituents Alternative** window. **Close** the **Constituent Alternative** window.

Double-click the reservoir to define the loading of chlorine. Select **True** in the **Is Constituent Source?** field. The **Constituent Source Type** is **Concentration** and the baseline concentration is 1.0 mg/L. The constituent source pattern is fixed.

The bulk reaction rate in the pipes can be adjusted using the **Tables** tool. Click the **Flex Table** button, then select **Pipe Table**. Add columns for the **Bulk Reaction Rate (Local)** and **Specify Local Bulk Reaction Rate?** to the table by clicking the **Edit** button in the toolbar at the top of the table**.** You can also move columns in the table using the double arrow icons and .

You can input bulk reaction rates for individual pipelines by placing a check mark in the **Specify Local Bulk Reaction Rate?** column for each pipe you want to adjust, and then entering a reaction value for the pipe in the

Bulk Reaction Rate (Local) column. However, in this case the global bulk reaction rate of –0.5 $(mg/l)^{1-n}$/day is acceptable for all system pipes, so the default values do not need to be adjusted. **Close** the **Pipe Table**.

Double-click the tank. Set the initial chlorine concentration to 0.000 mg/l, select **True** in the **Specify Local Bulk Rate?** field, then enter the bulk reaction rate of –0.5 /day. **Close** the tank T-1 dialog.

From the **Analysis** or **Home** tab, select **Scenarios**. Make sure **Chlorine Analysis** is the current scenario then run the scenario by clicking the **Compute** button .

Open the EPS results **Time Browser** from the **Analysis** tab by clicking the Times icon. To create a contour map of the chlorine concentration, click the **Contours** button found on the **Analysis** tab. After clicking the **New** button, the contour **Field** should be **Concentration (Calculated)**; select **All Elements**; set the **Minimum** to 0.0, **Maximum** to 1.0, **Increment** to 0.025, and **Index** to 0.1 mg/L. Click the **Apply** button to add the contours to the system layout. **Close** the Contour definition dialogs.

Click the **Play** button on the **Time Browser** toolbar. The chlorine concentrations for each time step can be viewed through time as shown in Figure 6-4.

Save your simulation as "Tutorial 2" since this network will be used in the following tutorials.

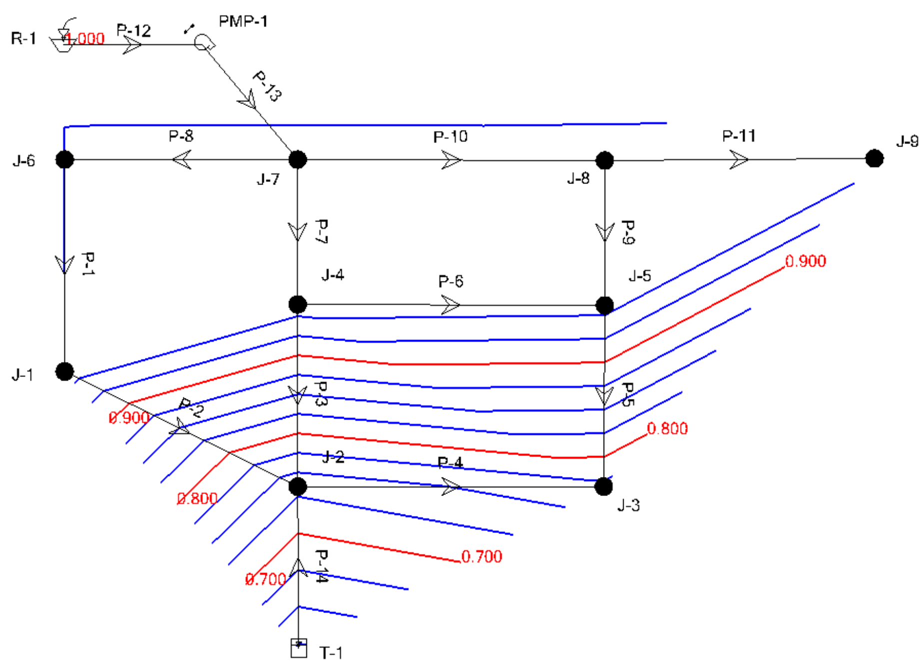


Figure 6-4: Contours of Chlorine Concentration at 42 Hours

Results: To view the variation of the chlorine concentration in the network, click on the tank, J-2, J-3, J-7, and J-9 while holding the shift on the keyboard to select each object. Then click the **Graphs** button in the **Home** tab. Create a **Line-Series Graph** from the **New** button in the Graphs dialog. Select the **Chlorine Analysis** box in the **Scenarios** field, and **Concentration (Calculated)** found under "Results (Water Quality)" for both the Tank and Junction in the **Fields** field in the **Graph Series Options** window. You should also unclick any other selected lines in the **Fields** field. Close the options window to look at the graph which should like Figure 6-5. The lowest chlorine concentration is found in tank T-1. Junctions J-2 and J-3 each have similar chlorine concentration values. In addition, the water distribution network reaches dynamic equilibrium during the second day of the simulation.

During the last 24 hours of the simulation (hours 144 through 168) when dynamic equilibrium has been achieved, the maximum chlorine concentration at tank T-1 is 0.822 mg/l, and the minimum concentration is 0.657 mg/l.

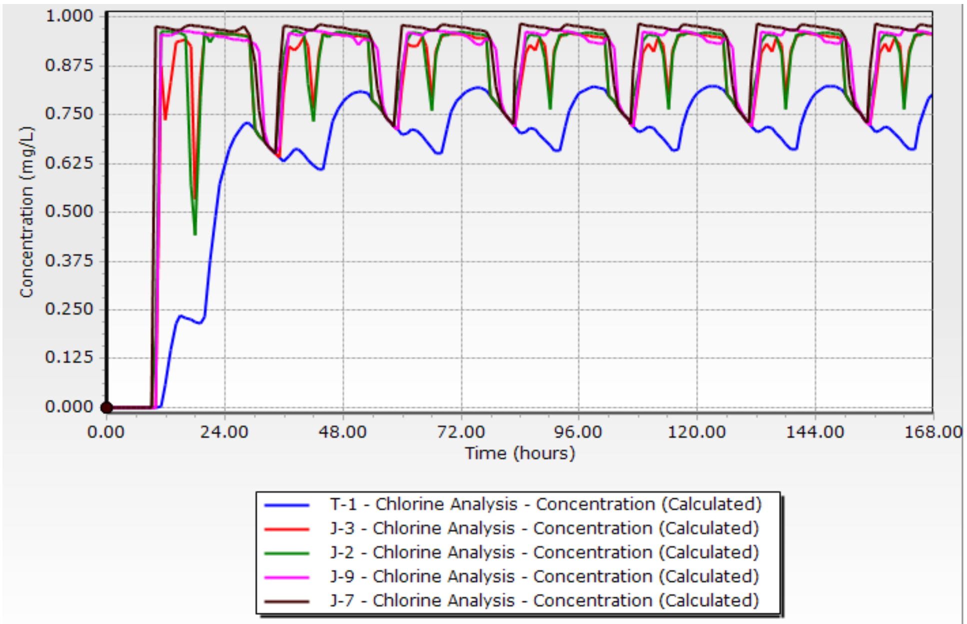


Figure 6-12: Chlorine Concentration in Tank T-1 and selection junctions

To compare water age against chlorine concentration at a selected junction, open the graph that plotted the **Calculated Age** for the tank and junctions. The graphs of age versus time and chlorine concentration should now be open on the desktop. Move the graphs so that both are visible and the axes are aligned. Comparison of the two graphs suggests an inverse correlation between age and chlorine concentration.

<u>Answers</u>

The oldest water is found in the storage tank. It is far from the source, and incoming water is mixed with the tank's contents. In the distribution system, the oldest water is found at J-2. The newest water is found at J-7 when pump PMP-1 is running.

The lowest chlorine concentration is in Tank T-1 when it is nearly empty. In the distribution system, the lowest chlorine concentration is found at J-2 when T-1 is emptying. The highest concentration is at J-7 when pump PMP-1 is running.

These results are consistent with the fact that chlorine concentration declines over time. For example, during the third day of operation, the minimum chlorine concentration at Junction J-2 is coincident with the maximum age, and the maximum chlorine concentration is coincident with the minimum age.

Inspection of the graph of chlorine concentration in tank T-1 suggests that the system stabilizes into a daily pattern on the second day. However, if the initial tank level or the demands are changed, stabilization may take longer. It appears that the seven-day simulation period is adequate for this network.

Tutorial 3 – Pumping Costs

Problem Statement

Calculate the daily electrical costs for the network in Tutorial 2 using the following data:

Energy price	$0.10/kWh
Motor efficiency	90%
Pump efficiency	50% at 2,000 L/min 60% at 2,500 L/min 55% at 3,000 L/min

Solution

Open Tutorial 2 after it is completed.

The first step is to add the pump and motor efficiency data to PMP-1. Open the **Pump Definitions** found under the **Components** tab. You can also double-click **PMP-1** in the layout view to open the pump characteristics window**.** In the **Pump Definition** field, select **Edit Pump Definitions**. The pump definition dialog appears.

On the **Efficiency** tab, select the **Multiple Efficiency Points** option for the pump efficiency. In the **Efficiency Points** table, add the efficiency data from the problem statement.

In the **Motor** section, enter 90% for the motor efficiency.

Close the **Pump Definition** tool, and the pump dialog if it is open.

The **Energy Cost** tool is used to calculate energy costs. From the **Analysis**s tab, click the **Energy Costs** button on the toolbar and select **Scenario Energy Cost**.

Click the Energy Pricing button to open the **Energy Pricing** dialog.

Create a **New** label "Energy Pricing-1" to enter the electricity cost. Select **Constant** from the **Tariff Type** drawdown list. For the **Energy Price** enter $0.10/kWh. Close the **Energy Pricing** dialog to return to the **Energy Costs** window.

Set the scenario to **Chlorine Analysis**, select "Energy Pricing-1" in the **Energy Pricing** field located on the "Pumps" tab, and then click the **Compute** button **.**

Answer

On the left panel of the Energy Cost window, highlight the **Chlorine Analysis** line. On the right panel, select the **Summary** tab. For a detailed summary of the computed cost with time along with the detailed pump performance values, you can select **PMP-1** in the left panel then select the **Results** tab on the right panel. For the seven-day simulation, the following data were calculated:

Pump energy used	4,028 kWh
Volume pumped	21,210 m^3
Pump cost	$402.80
Daily cost	$58.60

Tutorial 4 – Pipe Sizing using Darwin Designer

Problem Statement

Prepare a minimal cost estimate for pipe materials and installation portion of the project in Tutorial 2. The system pipes should be sized using a demand multiplier of 3.4 (peak flow factor) with a calculated pressure for each junction between 170 and 550 kPa. In addition, the system should supply to an industry located at junction 9 an additional 1,500 l/min with a minimum pressure of 275 kPa. Use the following cost data:

Pipe Material and Cost		
Material	**Diameter (mm)**	**Cost ($/m)**
Ductile Iron	75	40.32
Ductile Iron	150	56.64
Ductile Iron	200	79.36
Ductile Iron	250	114.72
Ductile Iron	300	156.16
Ductile Iron	350	201.92

Do not consider the cost of the reservoir, tank, pump, or pipes P-12 and P-13.

Solution

Open Tutorial 2 after it is completed.

Click **Options** from the **Home** or **Analysis** tab to open the Calculation Options window. Create a new **Calculation Option** by clicking the **New** button and enter "Designer Calculation Options" as the name. Double-click the calculation option you just created. The **Time Analysis Type** should be set to **Steady State**.

From the **Home or Analysis** tab, select **Scenarios**. Create a new **Base Scenario** by clicking the **New** button. Enter "Designer Analysis" as the name of the scenario.

Right-click the Designer Analysis scenario, and select **Make Current**. The red check should now be on the Designer Analysis scenario. Double-click on the Designer Analysis scenario and select the Designer Calculation Options in the **Steady State/EPS Solver Calculation Options** field.

From the Scenarios dialog, click the **Compute** button.

Click the **Darwin** button  found on the **Analysis** tab and select **Darwin Designer** to create a scenario to determine the minimum design cost**.** Click the **New** button and select **New Designer Study**. Check to make sure the Representable Scenario is **Designer Analysis**.

Enter design criteria on the **Design Events** tab in the dialog by clicking the **New** button. The top window is used to enter general design criteria; the bottom window is for entering specific design criteria for individual or groups of elements (i.e. pipe, junction, tank, etc.). It is typical that pipe networks are designed for high flow conditions. Scroll across to set the **Demand Multiplier** to 3.4. Criteria are set to maintain a working network to avoid low flow or pressure conditions. Set the **Minimum Pressure (Default)** to 170 kPa, and the **Maximum Pressure (Default)** to 550 kPa. The default values for flow velocity criteria (0 to 2.44 m/s) are acceptable and do not need to be adjusted.

Enter the design criteria for the industry at junction J-9 using the bottom window. In the **Demand Adjustment** tab, use the **Select from Drawing** button  to select junction J-9 from the drawing. Click on J-9 and then click the green check-mark **Done** button  in the **Select** dialog. Enter 1,500 l/min into the **Additional Demand** cell on the **Demand Adjustments** tab. Select the **Pressure Constraints** tab and again select junction J-9 from the drawing. Place a check-mark in the box in the **Override Defaults?** cell, and then enter 275 kPa and 550 kPa for the minimum and maximum pressures, respectively.

The next step is to define the pipes to be sized by Darwin Designer. Pipes with similar properties can be grouped together and will be designed the same, or the software can analyze each pipe separately. In this case, each pipe will be analyzed separately. From the problem statement, all pipes will be considered except P-12 and P-13, which are the pipes at the reservoir and pump. On the **Design Groups** tab, click the **Select Elements for Design Group** button to select all the pipes. A table with all of the pipes should be displayed. If not, highlight "<All Available>" in the **Selection Set** window, and then click **OK**. Delete pipes P-12 and P-13 from the table to remove them from the analysis.

The pipe material, properties, and costs to be used in this design scenario are entered in the **Cost/Properties** tab. Open the new pipe table by highlighting **New Pipe** in the window, then selecting **Design Option Groups** under the **New** button. Rename "New Pipe-1" to "New Ductile Iron Pipe". Enter the pipe type, diameter, and cost per linear meter from the table in the problem statement. The Hazen-Williams C is 130 for ductile iron pipe. If it is necessary to change the units for pipe costs, right-click the column heading **Unit Cost**, and then select **Units and Formatting**. In the **Unit** field, select **$/m**, and then click **OK**.

The objective of this scenario is to size the pipes to deliver the required flow while maintaining reasonable pressures throughout the network at the minimum design cost. On the **Design Type** tab, select the **Minimize Cost** criteria in the **Objective Type** date field.

After the design criteria is entered, start the simulation by highlighting **New Design Study-1** in the left panel of the window then click **New** to select **New Optimized Design Run**. Several different types of design runs may be performed by selecting different design events or design groups to be compared. Since there is only one design run in this demonstration, comparisons of different potential design solutions will not be possible. In many cases, different solutions need to be compared to determine the optimum solution for the design situation. The left panel of the Darwin Designer window helps to organize different solutions.

On the **Design Events** tab, select the design criteria to be evaluated. **New Design Event-1** should be checked as Active.

On the **Design Groups** tab, select the ductile iron properties and cost that were entered. The data is entered in the **Cost/properties** column. Since all designed pipe will come from the same ductile iron table, enter the data using global edit. Right-click the **Cost/properties** field and select **Global Edit**. Select "New Ductile Iron Pipe" in the **Value** window. Click **OK** and all of the **Cost/properties** fields should fill in.

The **Options** tab allows Darwin Designer parameters to be adjusted. In this case the default values are acceptable and will be used.

To start the run, click the **Compute** button in the **Darwin Designer** toolbar. When the run is completed, **Close** the **Designing…** window. The top three solutions will be listed under the "New Optimized Design Run" in the left panel of the **Darwin Designer** window.

Answer

A summary table of the three solutions is shown by clicking the **Solutions** folder and beneath this are tables that have the determined pipe diameters for each solution. In this example, the minimal pipe cost is $209,949. A summary of each solution cost and the design pipe diameters for Solution 1 are shown in the tables below. Pipe diameters range from 75 to 250 mm and the cost for each pipe is determined based on the entered pipe length. Open **Solution 1** and select the **Simulated Results** tab. The calculated pressure at the industry (junction J-9) is 284 kPa, which is within the required range.

It also should be noted that there are a number of 75 mm (3-inch) pipes in this solution, and the pipe connecting the network to the tank (P-14) is only a 200 mm (8-inch diameter) pipe. If this solution was evaluated for fire flow conditions, it is likely that these pipes would not deliver the required fire flow. Further simulations should be conducted on this solution to ensure that this design can deliver the required flow during a fire event.

Darwin Designer Solutions	
Solution	**Total Cost ($)**
Solution 1	209,949
Solution 2	209,984
Solution 3	210,777

Pipe Diameters and Costs for Solution 1		
Pipe	**Diameter (mm)**	**Cost ($)**
P-1	75	12,096
P-2	75	12,298
P-3	250	25,812
P-4	75	12,136
P-5	75	9,072
P-6	200	23,887
P-7	75	9,072
P-8	75	12,136
P-9	150	11,328
P-10	200	23,887
P-11	250	34,416
P-14	200	23,808
Total Cost:		**209,949**

Tutorial 5 – Model Calibration using Darwin Calibrator

WaterGEMS has the ability to use measured field data to calibrate the model. In many cases, data that are entered into the model are an approximation or guess. When the model results do not match field data, parameters in the model need to be adjusted. Also, the Darwin Calibrator with field data can be used to locate potential differences between the real network and the model which could be caused by problems in the system (blockages, closed valves, etc.).

Problem Statement

Adjust the Hazen-Williams C factor (roughness factor) for pipes P-1, P-2, P-3, P-7 and P-8 from the pipe network used in Tutorial 2. The tank level was at 3.93 meters during field measurements, the pump was off, and a hydrant with a measured flow of 3,400 l/min at junction J-7 was opened to increase the head loss in the pipe network. The measured field data is shown in the table below.

Field Data with Hydrant open	
Junction	**Pressure (kPa)**
J-1	263.5
J-2	327.0
J-6	406.5
J-7	296.0

Solution

Open Tutorial 2 after it is completed.

Click **Options** from the **Home** or **Analysis** tab to open the Calculation Options window. Create a new **Calculation Option** by clicking the **New** button and enter "Calibrator Calculation Options" as the name. Double-click the calculation option you just created. The **Time Analysis Type** should be set to **Steady State**.

From the **Home or Analysis** tab, select **Scenarios**. Create a new **Base Scenario** by clicking the **New** button. Enter "Calibrator Analysis" as the name of the scenario.

Right-click the Calibrator Analysis scenario, and select **Make Current**. The red check should now be on the Calibrator Analysis scenario. Double-click on the Calibrator Analysis scenario and select the Calibrator Calculation Options in the **Steady State/EPS Solver Calculation Options** field.

From the Scenarios dialog, click the **Compute** button.

Click the **Darwin** button found on the **Analysis** tab and select **Darwin Calibrator** to create a scenario to conduct a calibration study to determine the pipe roughness factors**.** Click the **New** button and select **New Calibration Study**. Check to make sure the Representable Scenario is **Calibrator Analysis**.

The top window is used to enter general criteria; the bottom window is for entering specific criteria for individual or groups of elements (i.e. pipe, junction, tank, etc.). Enter the field data in the **Field Data Snapshots** tab. Click the **New** button to on the top panel enter new data.

Enter the measured pressures on the **Observed Target** tab of the lower panel. Select the junctions from the drawing by clicking the select button then clicking junctions J-1, J-2, J-6, and J-7 from the drawing. Click the green check-mark **Done** button in the **Select** dialog. Select Pressure (kPa) for each junction in the **Attribute** field. Enter the measured pressure for each junction in the **Value** field.

Enter the tank level in the **Boundary Overrides** tab. Select the tank from the drawing by clicking the select button then clicking the tank. Click the green check-mark **Done** button in the **Select** dialog. Select **Tank Level (m)** in the **Attribute** field, and enter 3.93 meters in the **Value** field.

To make sure the pump is off, in the **Boundary Overrides** tab select the pump by clicking the select button , then clicking the pump. Click the green check-mark **Done** button in the **Select** dialog. Select **Pump Status** in the **Attribute** field, then select **Off** in the **Value** field.

Enter the additional demand at junction 7 in the **Demand Adjustments** tab. Select this junction from the drawing by clicking the select button , then clicking J-7. Click the green check-mark **Done** button in the **Select** dialog. Enter 3,400 l/min in the **Value** field.

Select the pipes where the roughness values are to be determined in the **Roughness Groups** tab. Pipes can be evaluated separately, or pipes with similar roughness characteristics can be grouped together. In this case, Darwin Calibrator will determine the pipe roughness values for P-1, P-2, and P-8 separately, and pipes P-3 and P-7 will be grouped together. Click the **New** button, then the ellipse button in the **Elements IDs** field. Click the select button , and then select pipe P-1 from the drawing. Click the green check-mark **Done** button in the **Select** dialog. Click **OK** to enter this pipe into the table as **New Roughness Group - 1**. Repeat these steps to add pipes P-2 and P-8 to the table as separate roughness groups. When completed, there should be three defined roughness groups in the table on the **Roughness Groups** tab.

Add the grouped pipes by clicking the **New** button, then the ellipse button in the **Elements IDs** field. Click the select button , then select pipes P-3 and P-7 from the drawing. Click the green check-mark **Done** button in the **Select** dialog. Click **OK** to enter these pipes into the table as a group. There should now be four roughness groups in the table on the **Roughness Groups** tab.

Darwin calibration method settings are found on the **Calibration Criteria** tab. In this case, the default settings are acceptable and will be used for the calibration analysis.

To use this data to determine the pipe roughness values, create a new run by clicking the **New** button on the left panel of the Darwin Calibrator window and then select **New Optimized Run**.

The pipe roughness values are assumed to have a range between 5 and 140 for each pipe. This is wide range and any chokes (blockages, partially closed valves, etc.) in a pipe could greatly reduce the pipe roughness value. It is not expected that a roughness value above 140 would be observed for ductile iron pipe. On the **Roughness** tab in the **Operation** field, **Set** should be selected; the indicated minimum (5) and maximum (140) roughness values should be entered in the **Minimum Value** and **Maximum Value** fields. Enter the increment to be used to analyze roughness values by entering 5 into the **Increment** field.

For this simulation, the top 5 solutions will be displayed. Click the **Options** tab and enter 5 into the **Solutions to Keep** field.

To start the run, click the **Compute** button in the Darwin Calibrator toolbar. When the run is completed, close the **Calibration…** window. The top five solutions will be listed under the "New Optimized Run" in the left panel of the Darwin Calibrator window.

Answer

The fitness values of the five solutions are shown by clicking the **Solutions** folder. In this example, the fitness values ranged from 0.288 to 0.343 where a lower fitness value indicates a "better" solution. A summary of each solution with the determined roughness values are shown in the table below. To view the determined roughness values for the "best" solution, click the **Solution 1** summary and highlight **Roughness** in the **Adjustments Results** window under the **Solutions** tab. To view the observed and simulated Hydraulic Grade Line (HGL), click the **Simulated Results** tab.

Darwin Calibrator Solutions				
	Roughness Value			
Solution	**P-1**	**P-2**	**P-8**	**P-3 and P-7**
Solution 1	20	50	30	135
Solution 2	20	50	45	140
Solution 3	25	65	55	130
Solution 4	20	50	85	140
Solution 5	20	50	130	140

These results indicate that the grouped pipes P-3 and P-7 have roughness values that are about what would be expected for the installed ductile iron pipe. However, the results for pipe P-1, P-2, and P-8 show that the roughness value is much lower. This could indicate a partially closed valve, the pipe is blocked, or that the pipe diameter may be smaller than expected. In this case, pipe P-1 should be investigated to determine the cause of this low roughness value. If there is a problem and that problem was fixed, new field measurements should be taken.

These roughness values can be entered into the model and further simulations can be conducted for different conditions. With enough field data for different hydraulic conditions, a model that closely simulates the actual system can be created. Keep in mind that many times the person doing the modeling must decide what values to put into the model. The software can only calculate values based on what is entered. The person doing the modeling must judge how accurate the model is and whether the model can be used to make decisions.