



WMS 11.4 Tutorial

EPANET Modeling in WMS

Building a Hydraulic Model Using Shapefiles



Objectives

Open shapefiles containing the geometry and attributes of EPANET links and nodes. Convert the shapefile features into an EPANET network with links and nodes using map data. Export the model and run it within the EPANET interface.

Prerequisite Tutorials

- None

Required Components

- WMS Core
- EPANET

Time

- 35–40 minutes

1	Introduction	2
2	Getting Started	3
2.1	Selecting the EPANET Model and Creating a Water Distribution Coverage .	3
3	Working with Shapefiles	3
3.1	Importing the Shapefiles	3
3.2	Viewing Node Attributes	4
3.3	Viewing Link Attributes	4
3.4	Mapping Shapes to Feature Objects	5
3.5	Creating a 1D Schematic.....	6
4	Importing a Background Map	7
4.1	Importing a Previously Downloaded Background Map	7
5	Editing Model Parameters.....	8
5.1	Editing Project Parameters	8
5.2	Creating the Water Usage Demand Pattern.....	9
5.3	Editing Link Parameters.....	9
5.4	Editing Node Parameters.....	10
6	Saving the Project and Exporting the Model	10
6.1	Saving the Project File.....	10
6.2	Exporting the Model.....	10
7	Reviewing and Running the Model in EPANET	11
7.1	Editing Nodes	12
7.2	Editing Valves	13
7.3	Editing Tanks	13
7.4	Running the Model.....	14
7.5	Saving the Model	14
8	Conclusion	15

1 Introduction

The US Environmental Protection Agency (EPA) developed EPANET, an application to model the hydraulic and water quality behavior of water distribution piping systems. The application is capable of handling models that have varying spatial and temporal water demands. A single, or extended, period analysis can be set up and run for analyzing a water distribution network.

For the hydraulic analysis, EPANET calculates pressures at each node, as well as velocities, flows, and head loss in each link. Minor losses from fittings and major losses due to friction are included in the calculations. For the water quality analysis, EPANET can calculate the water quality at each link and node as well as the relative age of the water in the pipes. Water quality will not be analyzed in the model used in this tutorial.

The model used in this tutorial is from a development near Denver, Colorado. The shapefiles contain several polylines and points representing the links and nodes making up the pipes, valves, junctions, and tank used to build the network.

This tutorial illustrates the method for using shapefiles to create an EPANET water distribution model. The projection will be set first, and the shapefiles will be opened so the attributes can be reviewed. A background map will be imported to aide in visualization.

The shapefiles will then be converted into feature points and feature lines within a water distribution coverage, and then into a 1-D hydraulic schematic. The link, node, and project parameters will be reviewed and updated. The model will then be exported and opened in the EPANET application, where it will be run, and the results will be reviewed.

2 Getting Started

Starting WMS new at the beginning of each tutorial is recommended. This resets the data, display options, and other WMS settings to their defaults. To do this:

1. If necessary, launch WMS.
2. If WMS is already running, select the *File* | **New**  menu item to ensure that the program settings are restored to their default state.
3. A dialog may appear asking to save changes. Click the **Don't Save** button to clear all data.

The Graphics Window of WMS should refresh to show an empty space.

2.1 Selecting the EPANET Model and Creating a Water Distribution Coverage

1. From the *Module* bar, select the **Hydraulic Modeling Module**  option.
2. From the *Model* drop-down, select "EPANET", refer Figure 1.



Figure 1: Model drop-down

3. In the Project Explorer, right-click on the " Drainage" coverage and select the *Type* | **Water Distribution** context menu item.

Notice that the name of the coverage is now " Water Distribution".

3 Working with Shapefiles

Import the shapefiles and review the attributes. Then convert the shapefiles into feature objects so they can be converted into a network of EPANET links and nodes.

3.1 Importing the Shapefiles

1. Click the **Open**  macro to bring up the *Open* dialog.
2. From the *Files of type* drop-down, select "WMS XMDF Project File (*.wms)".
3. Browse to the *EPANET_Shapefile* folder.
4. Select the file "Colorado.wms".
5. Click the **Open** button to import the project and exit the *Open* dialog.

The two shapefiles, *Valley_Network_Link.shp* and *Valley_Network_Node.shp*, should now appear under the " GIS Data" folder in the Project Explorer, and the Main Graphics Window should appear similar to  Figure 2.

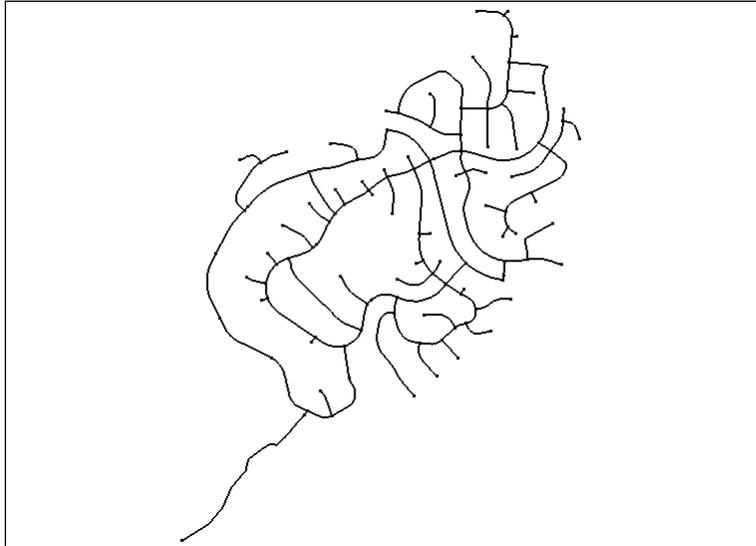


Figure 2: The imported shapefiles

3.2 Viewing Node Attributes

1. Right-click on “ Valley_Network_Node.shp” GIS data shapefile points item and select the **Open Attribute Table** context menu item to bring up the *Attributes* dialog.
2. Review the various attribute fields and note that they cover most of the required node attributes within EPANET.
 - *Node_ID* is the unique node name.
 - *Node_Type* is the type of node as specified within the EPANET model.
 - *Elevation* is the elevation of the node feature.
 - *Node_Tag* is a field which allows group IDs for organization purposes.
 - *Base_Dem* is a field describing the base demand of water usage for each of the nodes.
 - The *Northing* and *Easting* columns are not used by EPANET.
3. Click the **OK** button to exit the *Attributes* dialog.

3.3 Viewing Link Attributes

1. Right-click on the “ Valley_Network_Link.shp” GIS data shapefile arcs item and select the **Open Attribute Table** context menu item to bring up the *Attributes* dialog.
2. Review the various attribute fields and note that they cover most of the required node attributes within EPANET.
 - *Link_ID* is the unique link name.
 - *Pipe_Diam* is the diameter of the pipe link.
 - *Length* is the length of the pipe.

- *Link_Type* is the type of link within EPANET and can be specified as any of the link types such as pipe, valve, and pump
 - *Roughness* is the Hazen-Williams pipe roughness coefficient (C) that is assigned to each pipe to calculate major losses.
 - *ML_Coeff* is the Minor Loss coefficient assigned to the pipe based on the nearby fittings or other minor loss features.
 - *Status* is the status of the link and can be assigned as “Open”, “Closed”, or “None”.
 - *Link_Tag* is a field which allows group IDs for organizational purposes.
3. Click the **OK** button to exit the *Attributes* dialog.

3.4 Mapping Shapes to Feature Objects

1. Select the *Mapping | Shapes* → **Feature Objects** menu item to bring up *Step 1 of 4* of the *GIS to Feature Objects Wizard* dialog.
2. If asked to use all shapes in all visible shapefiles, click the **Yes** button.
3. Accept the defaults and click the **Next >** button to go to the *Step 2 of 4* page of the *GIS to Feature Objects Wizard* dialog.
4. On the *Mapping* row:
 - From the *Node_ID* column drop-down, select the “Node name” item.
 - From the *Node_Type* column drop-down, select the “Node type” item.
 - From the *Elevation* column drop-down, select the “Node elevation or head” item.
 - From the *Node_Tag* column drop-down, select the “Node tag” item.
 - From the *Base_Dem* column drop-down, select the “Node base demand” item.
5. Click the **Next >** button to go to the *Step 3 of 4* page of the *GIS to Feature Objects Wizard* dialog.
6. On the *Mapping* row:
 - From the *Link_ID* column drop-down, select the “Link name” menu item.
 - From the *Pipe_Diam* column drop-down, select the “Pipe diameter” menu item.
 - From the *Length* column drop-down, select the “Pipe length” menu item.
 - From the *Link_Type* column drop-down, select the “Link type” menu item.
 - From the *Roughness* column drop-down, select the “Pipe roughness” menu item.
 - From the *ML_Coeff* column drop-down, select the “Link minor loss coefficient” menu item.
 - From the *Status* column drop-down, select the “Link status” menu item.
 - From the *Link_Tag* column drop-down, select the “Link tag” menu item.

7. Click the **Next >** button to go to the *Step 4 of 4 (Finished)* page of the *GIS to Feature Objects Wizard* dialog.
8. Click the **Finish** button to close the *GIS to Feature Objects Wizard* dialog.

Now that the shapefiles have been mapped, they are no longer needed.

9. In the Project Explorer, under the “GIS Data” folder, use the **Shift** key to select both “Valley_Network_Node.shp” and “Valley_Network_Link.shp”, then right-click on one of them and select the **Delete** context menu item.

The project should appear similar to Figure 3.

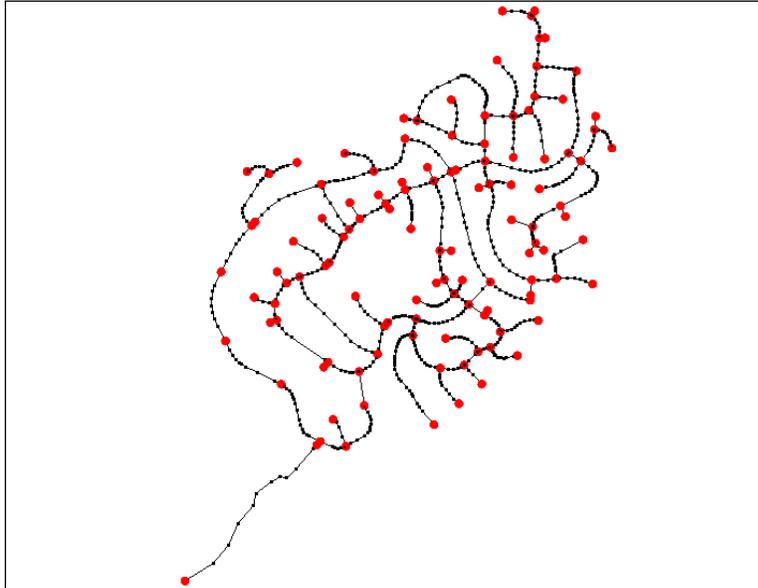


Figure 3: After mapping shapefiles to feature objects

Although they cannot be viewed, all of the attributes from both the polyline and point shapefile are now stored in “Water Distribution”. This coverage will now be used to create a 1D schematic of the links and nodes that make up the network.

3.5 Creating a 1D Schematic

1. From the *Module* bar, switch to the **Map Module**  option.
2. From the *Menu* bar, select the *Water Distribution | Map* → **1D Schematic** menu item.

The project should appear similar to Figure 4.

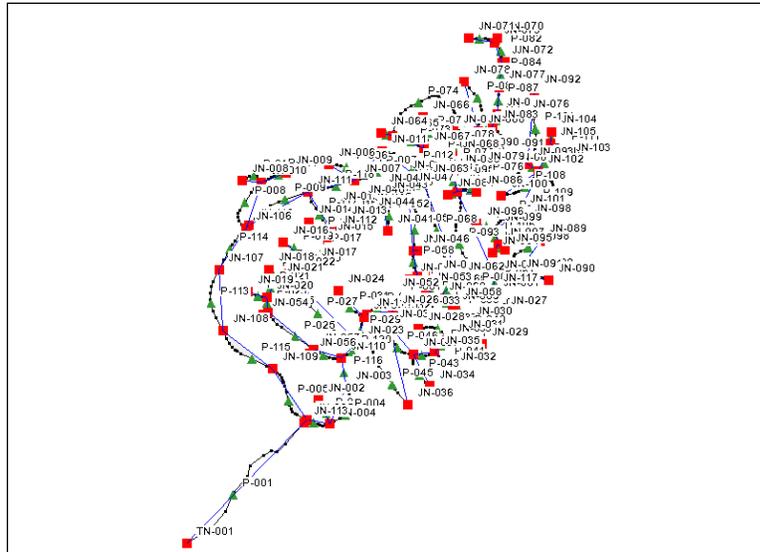


Figure 4: After mapping feature objects to 1D schematic

4 Importing a Background Map

For computers with an internet connection, a background map can be interactively downloaded from the web by using the **Get Data**  tool. For this tutorial, a previously downloaded image file will be used for the background map.

4.1 Importing a Previously Downloaded Background Map

A background map has been previously downloaded and can be imported into the project.

1. Select the *File* | **Open...**  menu item to bring up the *Open* dialog.
2. From the *Files of type* drop-down, select “TIFF Files (*.tif)”.
3. Select the “CO_Imagery.tif” file and click the **Open** button to import the file and exit the *Open* dialog.

The world imagery background map should appear as in  Figure 5. Feel free to review the development and the alignment of the pipe network with the streets and easements.

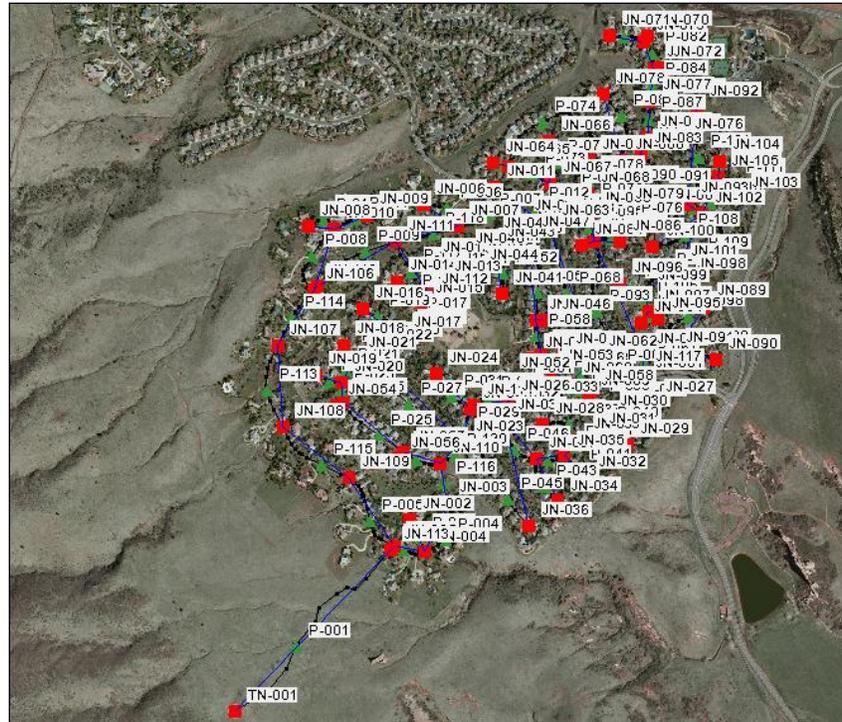


Figure 5: Project with world imagery background map

5 Editing Model Parameters

Now that the link and node networks have been created, it is possible to view the model, link, and node parameters. Parameters will also be entered for the tank node and a demand pattern will be assigned to all junction nodes.

5.1 Editing Project Parameters

First, view the overall project parameters.

1. From the *Module* bar, switch to the **Hydraulic Modeling Module** .
2. From the *Menu* bar, select the **EPANET | Edit Project Parameters** menu item to bring up the *Properties* dialog.

The demands have been entered in units of gallons per minute.

3. On the *Flow Units* row in the *Value* column, select the “GPM” drop-down menu item.
4. On the *Headloss Formula* row, select the “H-W” (Hazen-Williams) drop-down menu item.
5. On the *Total Duration* row, enter “24.0” hours.
6. On the *Hydraulic Time Step* row, enter “1.0” hours.
7. On the *Pattern Time Step* row, enter “1.0” hours.
8. On the *Pattern Start Time* row, enter “0.0” hours.
9. Click the **OK** button to close the *Properties* dialog.

5.2 Creating the Water Usage Demand Pattern

The next step is to create the demand pattern for the water use. The demand pattern is a 24-hour usage pattern and will be assigned to junction nodes at a later step. The demand pattern determines the calculated actual demands at each node in each time step.

1. From the *Menu* bar, select the *EPANET | Define Patterns...* menu item to bring up the *XY Series Editor* dialog.
2. For the *Curve Name*, enter “Residential_Usage”.
3. In a spreadsheet application, browse to the *EPANET_Shapefile* folder and open “Demand_Multipliers.xlsx”.
4. In the spreadsheet, select the contents of cells A1 to A24 and press the **Ctrl-C** key combination to copy them to the clipboard.
5. In WMS, in the *XY Series Editor* dialog, select the cell in the *Multiplier* column on row 1 and press the **Ctrl-V** key combination to paste the spreadsheet contents into that column.

The plot in the *XY Series Editor* dialog should appear similar to Figure 6.

6. Click the **OK** button to close the *XY Series Editor* dialog.

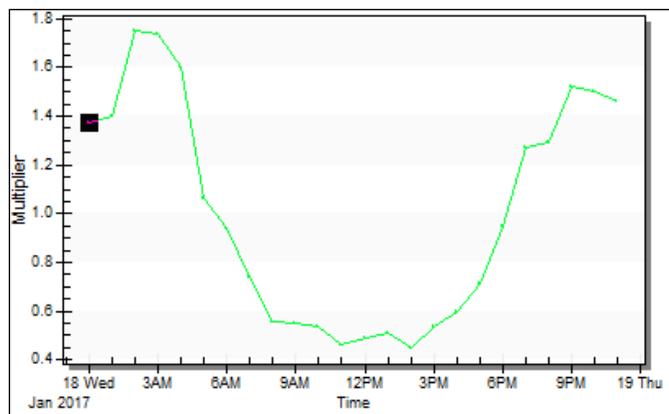


Figure 6: *XY Series Editor* dialog demand pattern plot

5.3 Editing Link Parameters

First, set the parameters for the links.

1. From the *Menu* bar, select the *EPANET | Edit Parameters...* menu item to bring up the *Hydraulic Properties* dialog.
2. From the *Attribute type* drop-down, select “Links”.
3. From the *Sort based on* drop-down, select “Name”.
4. Scroll down to the bottom of the spreadsheet where in the *Link Type* column “Valve” is selected for six entries.
5. In the *Status* column, for each valve entry, select the “None” drop-down menu item.

When a valve link status is “None”, it allows the valve control setting to be applied to the hydraulic model.

6. In the *Valve Type* column, for each valve entry, select the “Pressure Reducing Valve” drop-down menu item.
7. In the *Valve Setting* column, for valves V-001 through V-004, enter “60.0”.
8. In the *Valve Setting* column, for valves V-005 and V-006, enter “75.0”.

5.4 Editing Node Parameters

Next, set the parameters for the nodes.

1. From the *Attribute type* drop-down, select the “Nodes” menu item.
2. From the *Sort based on* drop-down, select the “Name” menu item.
3. In the All row (it has the yellow fields), in the *Demand Pattern* column, select the “Residential_Usage” drop-down menu item.

This assigns the “Residential_Usage” demand pattern to all junction nodes within the network.

4. In the *Name* column, scroll down to the row with the name “TN-001”.
5. In the *Initial Level* column, enter “17.0”.
6. In the *Minimum Level* column, enter “7.0”.
7. In the *Maximum Level* column, enter “20.0”.
8. In the *Diameter* column, enter “120.00”.

The tank is 20 feet in height and 120 feet in diameter.

9. Click the **OK** button to exit the *Hydraulic Properties* window.

6 Saving the Project and Exporting the Model

The next step is to save the WMS project and export the model, by doing the following:

6.1 Saving the Project File

1. Select the *File* | **Save As...** menu item to bring up the *Save As* dialog.
2. From the *Save as type* drop-down, select “WMS XMDF Project File (*.wms)”.
3. As the *File name*, enter “CO_Shape.wms”.
4. Click the **Save** button to save the project and close the *Save As* dialog.
5. If asked if image files should be saved in the project directory, click the **No** button.

6.2 Exporting the Model

1. Select the *EPANET* | **Export EPANET File...** menu item to bring up the *Select an EPANET File* dialog.
2. From the *Save as type* drop-down, select “EPANET file (*.inp)”.
3. As the *File name*, enter “Colorado.inp”.

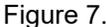
4. Click the **Save** button to export the project and close the *Select an EPANET File* dialog.
5. Close WMS by clicking on the  *Close* button at the top-right corner of the window.

The rest of the tutorial will be within the public domain EPANET application. It may be necessary to download it if it is not already installed.¹

7 Reviewing and Running the Model in EPANET

Now that the model data file has been prepared, it will be reviewed and run within the EPANET application.

1. Open the EPANET application.
2. Select the *File | Open...* menu item to bring up the *Open a Project* dialog.
3. From the *Files of type* drop-down, select “Input file (*.INP)”.
4. Browse to the *EPANET_Shapefile* folder.
5. Select the “Colorado.inp” file and click the **Open** button to import the file and exit the *Open a Project* dialog.

The project should appear similar to  Figure 7.

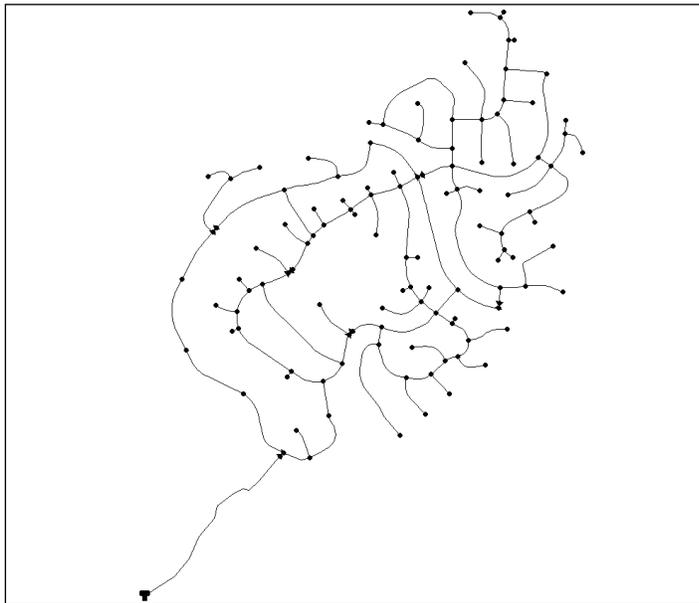


Figure 7: Initial project in EPANET

¹ Download and install the software from <https://www.epa.gov/water-research/epanet>, if needed.

7.1 Editing Nodes

1. Select the *Window* | **1 Browser** menu item to view the *Browser* dialog. Scroll to the right, if necessary, to see it.

The *Browser* dialog should appear similar to Figure 8.

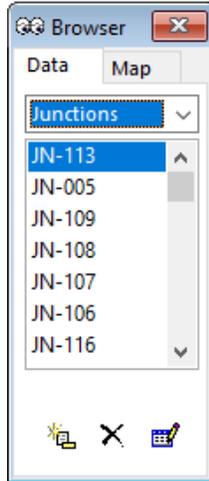


Figure 8: *Browser* dialog

2. In the *Browser* dialog, on the *Map* tab, from the *Nodes* drop-down, select "Elevation".

A color scheme will be applied to the nodes based on their elevations. They are currently all red since the ranges of values are set lower than the elevations found in this development. These value ranges can be changed as needed.

3. Select the *View* | *Legends* | *Modify* | **Node** menu item to bring up the *Legend Editor* dialog.
4. In the *Elevation* fields to the right of the color bar, from top to bottom, enter "6000.0", "6100.0", "6200.0", and "6300.0".

The *Legend Editor* dialog should appear as in Figure 9.

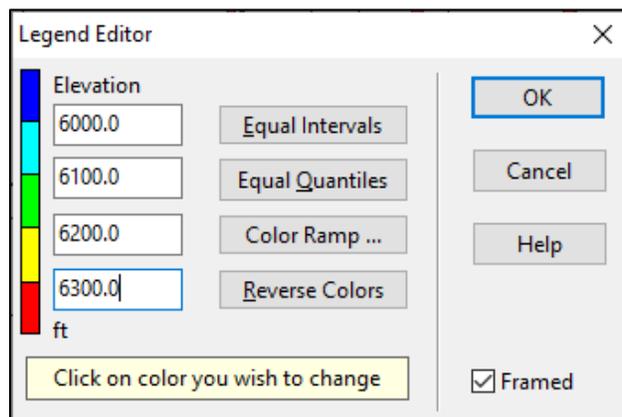


Figure 9: *Legend Editor*

5. Click the **OK** button to close the *Legend Editor* dialog.

The nodes should appear similar to Figure 10. The junctions range from 5986 – 6244 feet in elevation.

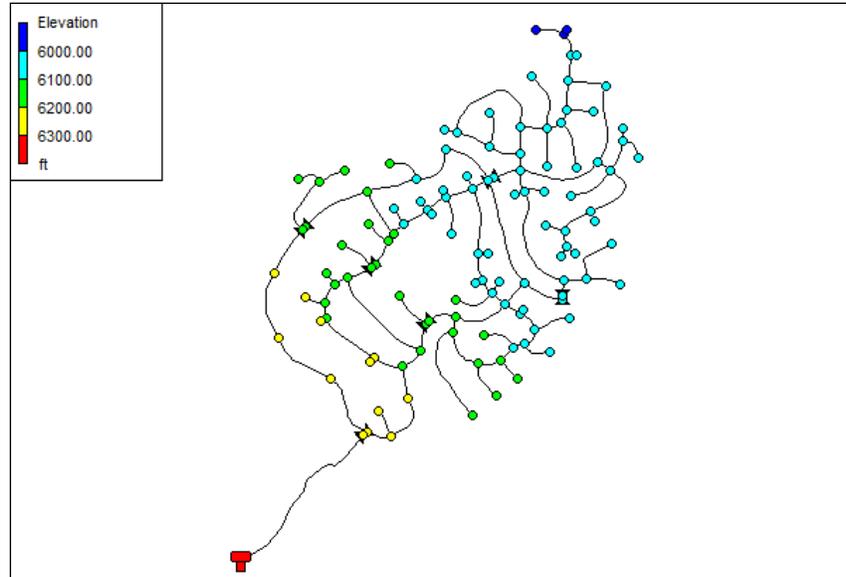


Figure 10: After legend elevations adjusted

7.2 Editing Valves

1. In the *Browser* dialog, on the *Data* tab, from the drop-down, select “Valves”.
2. In the list below the drop-down, double-click on “V-001” to bring up the *Valve V-001* dialog.

Note the *Setting* of “60”. This is the pressure setting (or “psi”) for that valve. The valve is located directly below the tank and sets the initial pressure for water flowing into the development.

3. After reviewing the valve information, click the  **Close** button in the top-right corner to close the *Valve V-001* dialog.
4. Repeat steps 2–3 for each of the other five valves, reviewing their setting as well as their location in the network.

These valves divide the development into three different pressure zones.

7.3 Editing Tanks

1. In the *Browser* dialog, on the *Data* tab, from the drop-down, select “Tanks”.
2. In the list below the drop-down, double-click on “TN-001” to bring up the *Tank TN-001* dialog.
3. After reviewing the tank information, click the  **Close** button in the top-right corner to close the *Tank TN-001* dialog.

This can be done for each of the options in the drop-down on the *Data* tab. Feel free to review any desired information this way.

7.4 Running the Model

1. Once done reviewing information, click the **Run**  macro to execute the model run and bring up the *Run Status* dialog.
2. When the model finishes running, click the **OK** button to close the *Run Status* dialog.
3. In the *Browser* dialog, on the *Map* tab, from the *Nodes* drop-down, select “Pressure”.
4. From the *Links* drop-down, select “Flow”.

The model was run for a 24-hour period and model outputs have been computed at each hour.

5. Use the right arrow  button (directly below the *Time* drop-down) or by clicking the **Forward**  button at the bottom of the *Map* tab to advancing the time steps individually, in order to observe the flows and pressures over time in the 24-hour solution.

More detailed model outputs for individual nodes or links can be extracted using the **Graph**  and **Table**  tools. For information on using these tools, refer to the EPANET documentation.²

7.5 Saving the Model

The last step is to save the network as an EPANET NET file.

1. Select the *File* | **Save** menu item to bring up the *Save Project As* dialog.
2. From the *Save as type* drop-down, select “Network files (*.NET)”.
3. Enter “Colorado.net” as the *File name* and click **Save** to export the network file and close the *Save Project As* dialog.

The file is now ready for use in Aquaveo’s [CityWater](#)³, where further analysis and visualization can be performed on the model.

² See <https://nepis.epa.gov/Adobe/PDF/P1007WWU.pdf>.

³ See https://www.xmswiki.com/wiki/CW:CityWater_Overview.

8 Conclusion

This concludes the “EPANET Modeling in WMS” tutorial. The following key concepts were discussed and demonstrated:

- Setting up an EPANET model with a water distribution coverage.
- Importing shapefiles to define the links and nodes.
- Importing a background map.
- Viewing and editing model and project parameters.
- Exporting the project to EPANET.
- Reviewing and running the model in EPANET.
- Exporting the network model as a NET file.

Feel free to experiment with EPANET as desired.