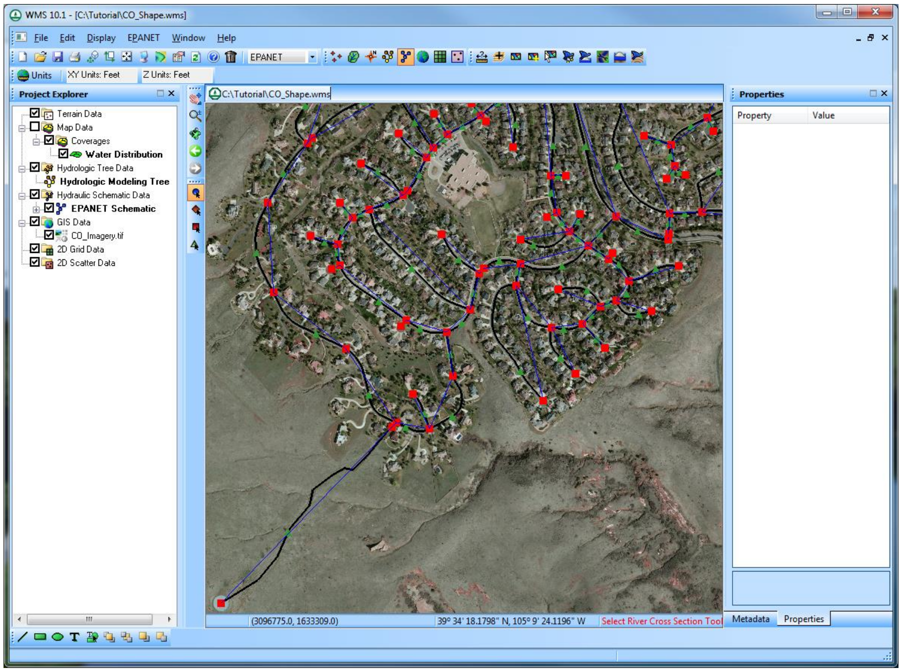
WMS

v. 10.1

WMS 10.1 Tutorial

Water Distribution System Modeling – Working with EPANET

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

- Data
- Map
- Hydrology
- Water Distribution

Time

- 35–40 minutes

AQUAVEO™

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 using two methods.

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, press *Ctrl-N* or select *File* | **New…** to ensure that the program settings are restored to their default state.
3. A dialog may appear asking to save changes. Click **No** to clear all data.

The graphics window of WMS should refresh to show an empty space.

2.1 Setting the Projection

First, specify a coordinate system (projection).

1. Select *Display* | **Display Projection…** to bring up the *Display Projection* dialog.
2. In the *Horizontal* section, select *Global Projection* to bring up the *Select Projection* dialog. If the dialog does not automatically appear, click **Set Projection…** to bring it up.
3. Select “State Plane Coordinate System” from the *Projection* drop-down..
4. Select “Colorado Central (FIPS 502)” from the *Zone* drop-down.
5. Select “NAD83” from the *Datum* drop-down.
6. Select “FEET (U.S. SURVEY)” from the *Planar Units* drop-down.
7. Click **OK** to close the *Select Projection* dialog.
8. In the *Vertical* section, select “Feet (U.S. Survey)” from the *Units* drop-down.
9. Click **OK** to exit the *Display Projection* dialog.

2.2 Selecting the EPANET Model and Creating a Water Distribution Coverage

1. Switch to the **Hydraulic Modeling** module.
2. Select “EPANET” from the Model drop-down (Figure 1).
3. Right-click on “ Drainage” in the Project Explorer and select *Type* | **Water Distribution**.

Notice that the name of the coverage is now “ Water Distribution”.

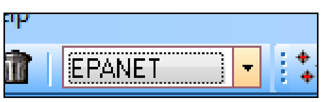


Figure 1 Model drop-down

3 Working with Shapefiles

Now 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 and Viewing the Attributes

1. Click **Open** to bring up the *Open* dialog.
2. Select “Shapefiles (*.shp)” from the *Files of type* drop-down.
3. Browse to the *EPANET_Shapefile\EPANET_Shapefile* folder.
4. While holding down the *Shift* key, select both “Valley_Network_Link.shp” and “Valley_Network_Node.shp”.
5. Click **Open** to import the two shapefiles and exit the *Open* dialog.

The two shapefiles should now appear under “ GIS Data” in the Project Explorer, and the Main Graphics Window should appear similar to Figure 2.

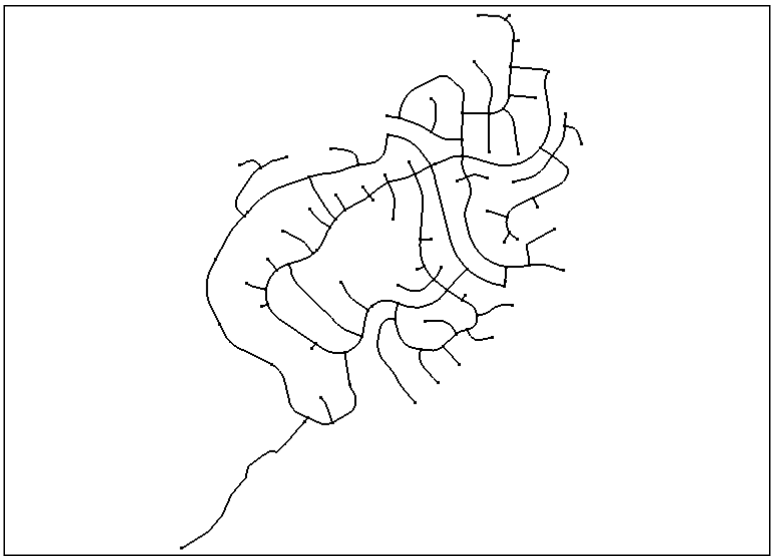


Figure 2 The imported shapefiles

6. Right-click on “ Valley_Network_Node.shp” and select **Open Attribute Table** to bring up the *Attributes* dialog.
7. 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.

8. Click **OK** to exit the *Attribute* dialog.
9. Right-click on “ Valley_Network_Link.shp” and select **Open Attribute Table** to bring up the *Attributes* dialog.

10. 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 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 feature.
    - *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 organization purposes.
11. Click **OK** to exit the *Attribute* dialog.
12. Select *Mapping* | **Shapes → Feature Objects** to bring up the *GIS to Feature Objects Wizard* dialog.
13. Click **Yes** if asked to use all shapes in all visible shapefiles.
14. Select “Water Distribution” from the *Select a coverage for mapping* drop-down.
15. Check the boxes next to *Valley_Network_Link.shp* and *Valley_Network_Node.shp* in the *Map Shapefile?* column in the *Select shapefiles to map* section.
16. Click **Next** to go to the *Step 1 of 3* page of the *GIS to Feature Objects Wizard* dialog.
17. On the *Mapping* row:
    - Select “Node name” from the drop-down in the *Node_ID* column.
    - Select “Node type” from the drop-down in the *Node_Type* column.
    - Select “Node elevation or head” from the drop-down in the *Elevation* column.
    - Select “Node tag” from the drop-down in the *Node_Tag* column.
    - Select “Node base demand” from the drop-down in the *Base_Dem* column.
18. Click **Next** to go to the *Step 2 of 3* page of the *GIS to Feature Objects Wizard* dialog.
19. On the *Mapping* row:
    - Select “Link name” from the drop-down in the *Link_ID* column.
    - Select “Pipe diameter” from the drop-down in the *Pipe_Diam* column.
    - Select “Pipe length” from the drop-down in the *Length* column.

- Select "Link type" from the drop-down in the *Link_Type* column.
- Select "Pipe roughness" from the drop-down in the *Roughness* column.
- Select "Link minor loss coefficient" from the drop-down in the *ML_Coeff* column.
- Select "Link status" from the drop-down in the *Status* column.
- Select "Link tag" from the drop-down in the *Link_Tag* column.

20. Click **Next** to go to the *Step 3 of 3* page of the *GIS to Feature Objects Wizard* dialog.
21. Click **Finish** to close the *GIS to Feature Objects Wizard* dialog.

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

22. 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 **Delete**.

The project should appear similar to Figure 3.

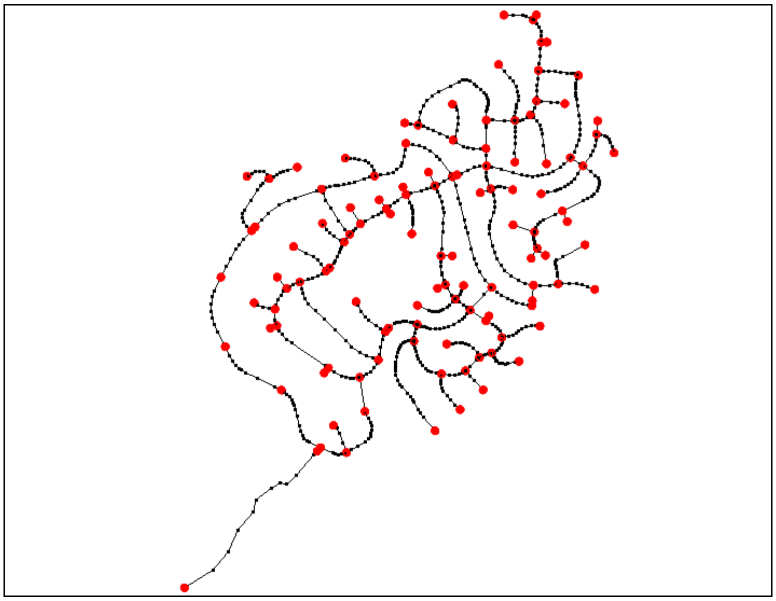


Figure 3 After mapping shapefiles to feature objects

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.

23. Switch to the **Map** module.
24. Select *Water Distribution* | **Map** → **1D Schematic**.

The project should appear similar to Figure 4.

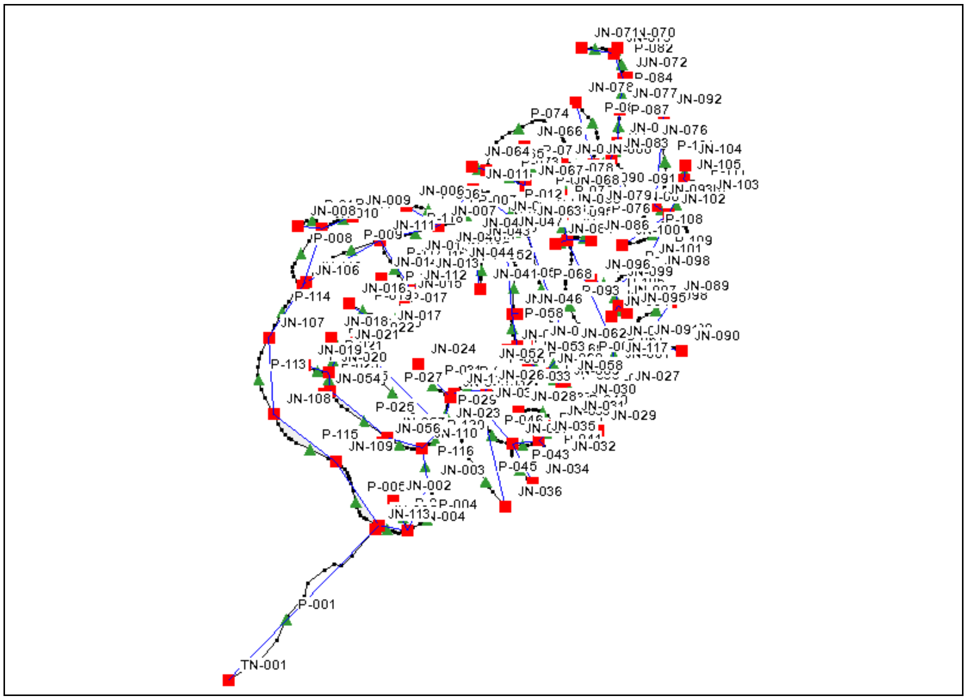


Figure 4 After mapping feature objects to 1D schematic

4 Background Map

4.1 Downloading 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. If an internet connection is not available, skip to the next section.

1. Click **Get Data** and click and drag a box around all feature arcs and feature nodes shown in the display to bring up the *Data Service Options* dialog.
2. Select *Web Services*, then scroll to the right and select *World Imagery*
3. Click **OK** to close the *Data Service Options* dialog and bring up the *Save Web Services Data File(s)* dialog.
4. Select “Web Services Files (*.web)” from the *Save as type* drop-down.
5. Enter “CO_Imagery.web” as the *File name*.
6. Click **Save** to close the *Save Web Services Data File(s)* dialog and bring up the *Image Pixel Size* dialog.
7. Click **Yes** when asked to create the file.
8. Enter “3.0” in the *Enter the image pixel size (ft)* field and click **OK** to close the *Image Pixel Size* dialog.

An export progress dialog will appear. When the GeoTIFF is done exporting, the dialog will close automatically.

9. Click No if asked to generate image pyramids.

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.

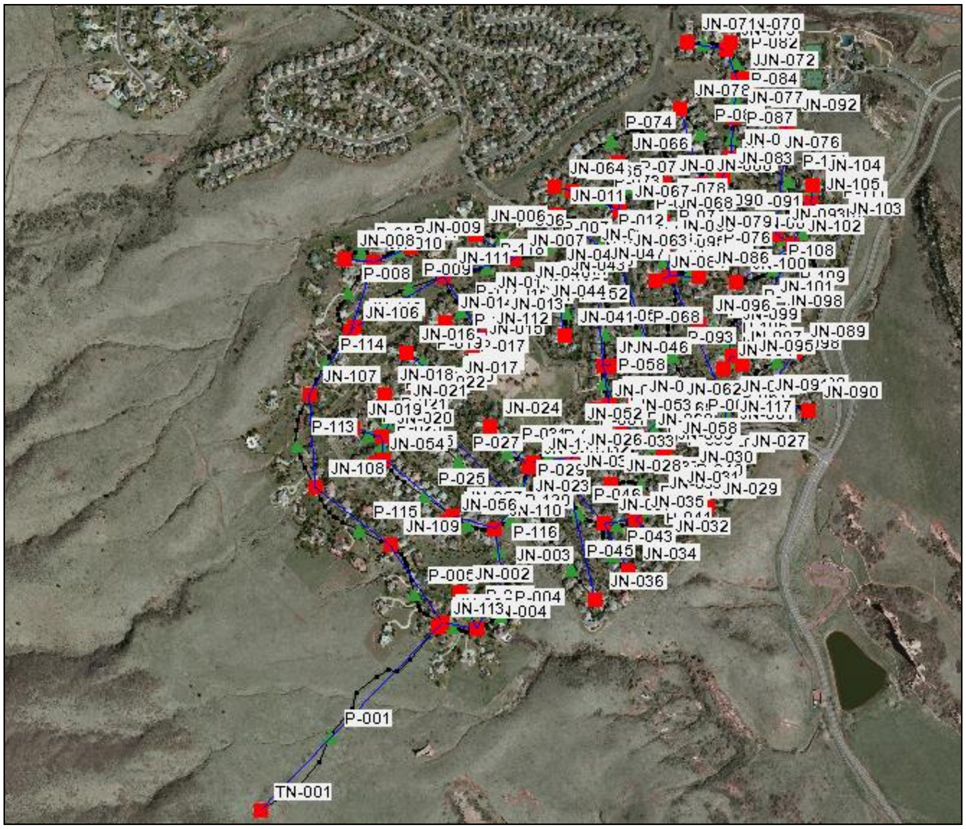


Figure 5 Project with world imagery background map

4.2 Importing a Previously Downloaded Background Map

A background map has been previously downloaded and can be imported into the project. If the previous section was completed successfully, skip this section.

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Select “TIFF Files (*.tif)” from the *Files of type* drop-down.
3. Select “CO_Imagery.tif” and click **Open** to import the file and exit the *Open* dialog.
4. If asked to generate image pyramids, click **No**.

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.

5 Viewing Model Parameters

Now that the link and node network has been created, 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. Switch to the **Hydraulic Modeling** module.
2. Select *EPANET* | **Edit Project Parameters** 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 “GPM” from the drop-down.
4. Select “H-W” (Hazen-Williams) from the drop-down on the *Headloss Formula* row.
5. Enter “24.0” on the *Total Duration* row.
6. Enter “1.0” on the *Hydraulic Time Step* row.
7. Enter “1.0” on the *Pattern Time Step* row.
8. Enter “0.0” on the *Pattern Start Time* row.
9. Click **OK** to close the *Properties* dialog.

5.2 Creating the Water Usage Demand Pattern

Next, create the water use demand pattern. 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. Select *EPANET* | **Define Patterns…** to bring up the *XY Series Editor* dialog.
2. Enter “Residential_Usage” as the *Curve Name*.
3. Outside of WMS, browse to the *EPANET_Shapefile\EPANET_Shapefile* folder.
4. Open “Demand_Multipliers.xlsx” into a spreadsheet application.
5. Select the contents of cells A1 to A24 in the spreadsheet and press *Ctrl-C* to copy them to the clipboard.
6. In WMS in the *XY Series Editor* dialog, select the cell in the *Multiplier* column on row *1* and press *Ctrl-V* to paste the spreadsheet contents into that column.

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

7. Click **OK** to close the *XY Series Editor* dialog.

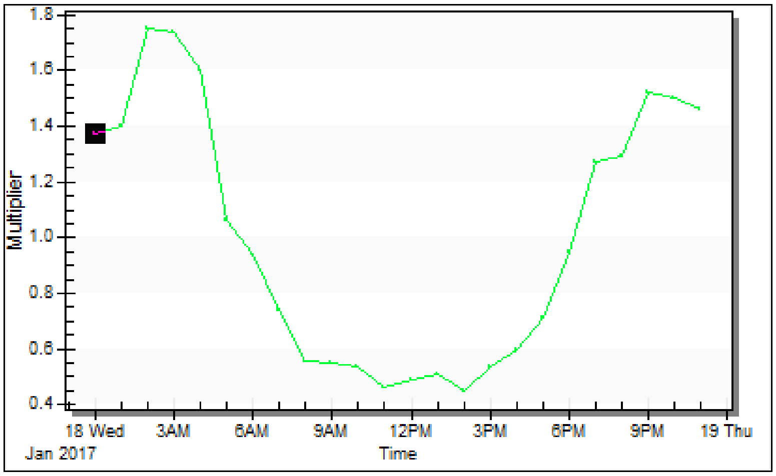


Figure 6 XY Series Editor dialog demand pattern plot

5.3 Editing Link Parameters

First, set the parameters for the links.

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

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

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

5.4 Editing Node Parameters

Next, set the parameters for the nodes.

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

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

4. Scroll down to the row with “TN-001” in the *Name* column.

5. Enter “17.0” in the *Initial Level* column.
6. Enter “7.0” in the *Minimum Level* column.
7. Enter “20.0” in the *Maximum Level* column.
8. Enter “120.00” in the *Diameter* column.

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

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

6 Saving the Project and Exporting the Model

Now save the WMS project and export the model.

6.1 Saving the Project File

1. Select *File* | **Save As…** to bring up the *Save As* dialog.
2. Select “WMS XMDF Project File (*.wms)” from the *Save as type* drop-down.
3. Enter “CO_Shape.wms” as the *File name*.
4. Click **Save** to save the project and close the *Save As* dialog.
5. Click **No** when asked if image files should be saved in the project directory.

6.2 Exporting the Model

1. Select *EPANET* | **Export EPANET File…** to bring up the *Select an EPANET File* dialog.
2. Select “EPANET file (*.inp)” from the *Save as type* drop-down.
3. Enter “Colorado.inp” as the *File name*.
4. Click **Save** to export the project and close the *Select an EPANET File* dialog.
5. Close WMS by clicking on the  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

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 *File* | **Open…** to bring up the *Open a Project* dialog.

[1] It is located in the *C:\Program Files (x86)\EPANET* folder by default. Download and install the software from https://www.epa.gov/water-research/epanet if needed.

3. Select "Input file (*.INP)" from the *Files of type* drop-down.
4. Browse to the *EPANET_Shapefile\EPANET_Shapefile* folder.
5. Select "Colorado.inp" and click **Open** to import the file and exit the *Open a Project* dialog.

The project should appear similar to Figure 7.

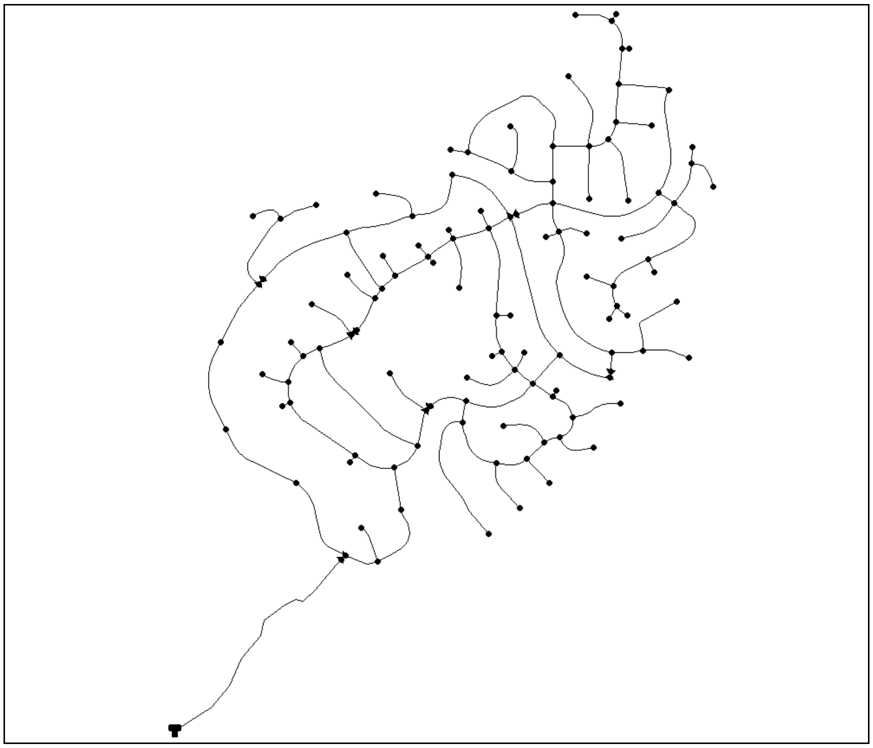


Figure 7 Initial project in EPANET

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

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

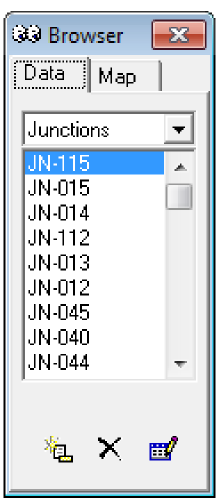


Figure 8 Browser dialog

7. On the *Map* tab, select “Elevation” from the *Nodes* drop-down.

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.

8. Select *View* | *Legends…* | *Modify* | **Node** to bring up the *Legend Editor* dialog.
9. From top to bottom in the *Elevation* fields to the right of the color bar, enter “6000.0”, “6100.0”, “6200.0”, and “6300.0”.

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

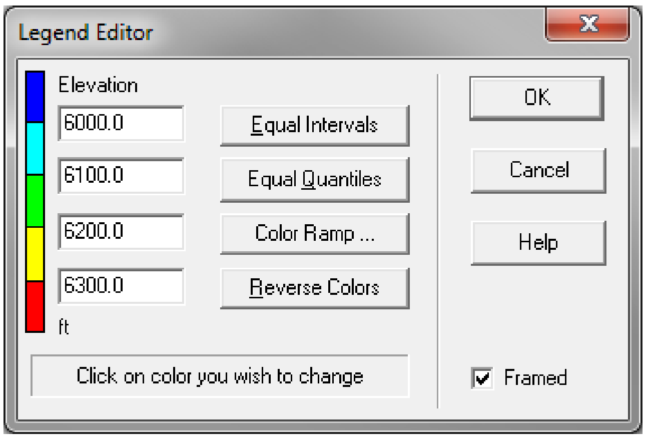


Figure 9 Legend Editor

10. Click **OK** to close the *Legend Editor* dialog.

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

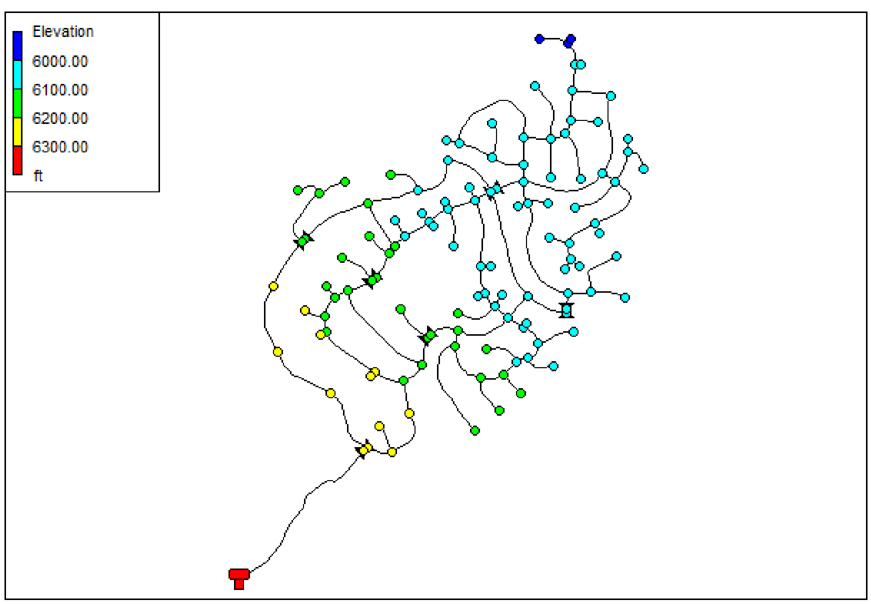


Figure 10 After legend elevations adjusted

11. On the *Data* tab in the *Browser* dialog, select “Valves” from the drop-down.
12. 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.

13. When done reviewing the valve information, click the in the top right corner to close the *Valve V-001* dialog.
14. Repeat steps 12–13 for each of the other five valves.

Review their setting as well as their location in the network. These valves divide the development into three different pressure zones.

15. On the *Data* tab in the *Browser* dialog, select “Tanks” from the drop-down.
16. In the list below the drop-down, double-click on “TN-001” to bring up the *Tank TN-001* dialog.
17. When done reviewing the tank information, click the 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.

18. Once done reviewing the information for the various parts of the network, click **Run** to execute the model run and bring up the *Run Status* dialog.
19. When the model finishes running, click **OK** to close the *Run Status* dialog.
20. On the *Map* tab in the *Browser* dialog.
21. Select “Pressure” from the *Nodes* drop-down.
22. Select “Flow” from the *Links* drop-down.

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

23. Observe the flows and pressures over time in the 24 hour solution by advancing the time steps individually using the right arrow (directly below the *Time* drop-down) or by clicking **Forward** at the bottom of the *Map* tab.

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]

7.1 Saving the Model

Finally, save the network as an EPANET NET file.

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

---

[2] See https://nepis.epa.gov/Adobe/PDF/P1007WWU.pdf.

8 Conclusion

This concludes the “Water Distribution System Modeling – Working with EPANET” 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 in EPANET as desired.