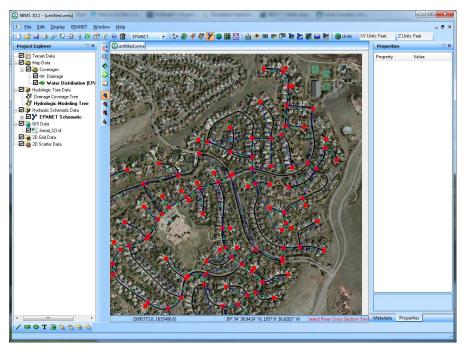


# WMS 10.1 Tutorial

# Water Distribution System Modeling – EPANET Hydraulic Model

Import an existing water distribution model and modify link and node parameters within WMS



# Objectives

View an existing water distribution model within the EPANET interface. Import the model into WMS and make changes to the link/node properties. Export and run the model to visualize changes in the water distribution system.

## Prerequisite Tutorials

• None

## **Required Components**

- Data
- Map
- Hydrology
- Water Distribution

#### Time

• 25–30 minutes





1	Introduction2		
2	Getting Started2		
3	3 Visualizing the Existing Conditions Model		
	3.1 O	Opening and Reviewing the Model Within EPANET3	
	3.2 R	tunning the Model and Reviewing the Results5	
	3.3 E	Exporting the Pipe Network6	
4	Importing the Pipe Network into WMS6		
	4.1 S	etting up the Projection6	
		mporting the Pipe Network6	
5			
		Oownloading a Background Image	
	5.2 Ir	mporting a Background Image Without an Internet Connection8	
6	Makin	g Modifications to the Network9	
	6.1 C	Creating the New Link and Node9	
		Modifying the New Link and Node11	
7	Export	ting and Running the Modified Network in EPANET11	
		Exporting the Modified Network	
		Opening the Modified Network in EPANET and Re-running the Model12	
8	Conclu		

#### 1 Introduction

The US Environmental Protection Agency (EPA) developed an application called EPANET to model the hydraulic and water quality behavior of water distribution piping systems. It is capable of handling models with varying water demands both spatially and temporally. A single or extended period analysis can be set up and run for a water distribution network.

For the hydraulic analysis, EPANET will calculate pressures at each node and velocities, flows, and headloss in each link. Minor losses from fittings and major losses due to friction are included in the calculations. For the water quality analysis, EPANET calculates 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 for this tutorial.

This tutorial first opens an existing model into the external EPANET application, runs it to observe the existing conditions solution, and exports the network as an EPANET input file. It then opens the input file into WMS where the coordinate projection is set and a base map is downloaded.

Link and node properties will be changed, editing options will be explored and the model will be exported so it can be imported back into the external EPANET application. In EPANET, the model will then be run and reviewed.

The model used is from a development near Denver, Colorado. It has several links and nodes that make up the pipes, valves, junctions, and tank used to build the network. After some of the parameters in the links and nodes are modified, the model will be run again to observe the changes in the results.

# 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.

## 3 Visualizing the Existing Conditions Model

This part of the tutorial will be completed within the public domain EPANET application. It may be necessary to download it if it is not already installed.<sup>1</sup>

### 3.1 Opening and Reviewing the Model Within EPANET

To begin, the existing conditions EPANET model will be opened and viewed.

- 1. Launch the EPANET application.
- 2. Select *File* | **Open...** to bring up the *Open a Project* dialog.
- 3. Select "Network files (\*.NET)" from the *Files of type* drop-down.
- 4. Browse to the *EPANET\_Edits*\*EPANET\_Edits*\ folder and select "Water\_Dist\_CO.NET".
- 5. Click **Open** to import the network file and exit the *Open a Project* dialog.

The network should appear similar to Figure 1. Notice the water tank at the lower left of the network. The pump that supplies water to the tank has not been included in this model since the tank has sufficient water to supply the development for a 24 hour period.

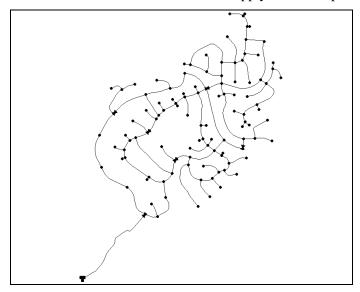


Figure 1 Pipe network

-

<sup>&</sup>lt;sup>1</sup> 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.

Throughout the development, nodes have been created at pipe junctions and different intervals. Water demand has been assigned to the nodes based on the water usage of the surrounding homes.

6. Select *Window* | **1 Browser** to view the *Browser* dialog.

The *Browser* dialog should appear similar to Figure 2.

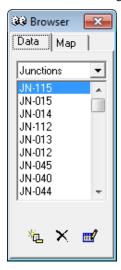


Figure 2 EPANET Browser dialog Data tab

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. Currently they are all red since the ranges of values are set lower than the elevations found in this development. These values can be edited.

- 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 3.

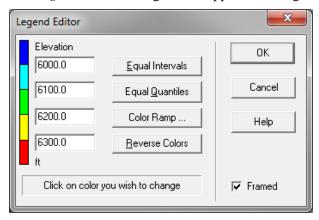


Figure 3 EPANET Legend Editor dialog

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

The nodes should appear similar to Figure 4. The junctions range from 5986 - 6244 feet in elevation.

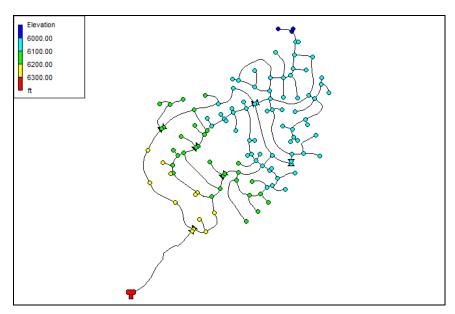


Figure 4 Nodes showing elevations

- 11. In the *Browser* dialog, select the *Data* tab, and select "Valves" from the drop down.
- 12. Double-click on "V-001" to bring up the *Valve V-001* dialog.

The "60" on the *Setting* line indicated the pressure setting (psi) for the valve. This valve is located directly downstream from the tank and sets the initial pressure for water flowing into the development.

- 13. When done reviewing the property values, click the to close the *Valve V-001* dialog.
- 14. Repeat steps 12–13 to review the property values for the remaining valves.

These valves divide the development into 3 different pressure zones.

## 3.2 Running the Model and Reviewing the Results

Now run the model and view the results.

- 1. Click **Run 5** button to bring up the *Run Status* dialog.
- 2. When the model has finished running, click **OK** to close the *Run Status* dialog.
- 3. In the *Browser* dialog, select the *Map* tab. If the *Browser* dialog is not visible on the screen select *Window* | **1 Browser** to bring it up.
- 4. Select "Pressure" from the *Nodes* drop-down.
- 5. 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.

6. 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* dropdown) 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.<sup>2</sup> Feel free to explore these options if desired.

#### 3.3 Exporting the Pipe Network

Now export the network from EPANET to an INP file.

- 1. Select File | Export | Network... to bring up the Save Network As dialog.
- 2. Select "Input files (\*.INP)" from the Save as type dialog.
- 3. Enter "Water\_Dist\_CO.inp" as the *File name*.
- 4. Click **Save** to export the INP file and close the *Save Network As* dialog.
- 5. Click at the top right corner of the EPANET application window to close the application and return to WMS.
- 6. Click **No** if asked to save changes in the current project.

# 4 Importing the Pipe Network into WMS

Next, specify a coordinate projection system in WMS and import the INP file into WMS.

#### 4.1 Setting up the Projection

In WMS, do the following to set up the projection:

- 1. Select *Display* | **Display Projection...** to bring up the *Display Projection* dialog.
- 2. In the *Horizontal* section, select *Global Projection* and click **Set Projection...** to bring up the *Select Projection* dialog (the dialog may appear automatically).
- 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 exit 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.

## 4.2 Importing the Pipe Network

1. Switch to the **Hydraulic Modeling** \*module.

2. Select "EPANET" from the Model drop-down on the toolbar (Figure 5).

<sup>&</sup>lt;sup>2</sup> See https://nepis.epa.gov/Adobe/PDF/P1007WWU.pdf.

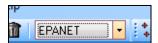


Figure 5 Model drop-down

- 3. Select *EPANET* | **Import EPANET File...** to bring up the *Select an EPANET File* dialog.
- 4. Select "EPANET file (\*.inp)" from the *Files of type* drop-down.
- 5. Browse to the *EPANET\_Edits\EPANET\_Edits\* folder and select "Water Dist CO.inp".
- 6. Click **Open** to import the file and close the *Select an EPANET File* dialog.

After importing the network, a new coverage will be displayed called "Water Distribution [EPANET File]". Since this model was created with real world coordinates, this coverage contains the georeferenced geometry of the network. A 1D schematic was created in the Project Explorer called "PEPANET Schematic". The schematic (Figure 6) will contain all of the links and nodes as well as their respective parameters.

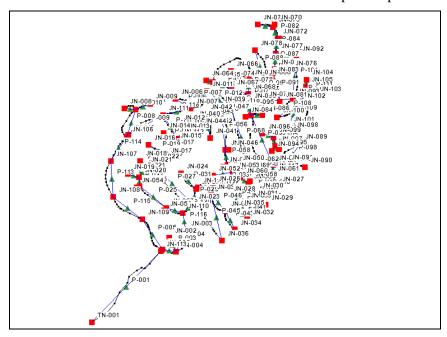


Figure 6 Imported EPANET schematic

# 5 Background Image

The following section requires an internet connection. If no internet connection is available, skip to section 5.2 to use the previously-downloaded background image provided with the tutorial files.

## 5.1 Downloading a Background Image

Projects with georeferenced models can have a background map interactively downloaded from the web by using the **Get Data** tool.

- 1. Using the **Get Data** tool, click and drag a selection box around the entire pipe network to bring up the *Data Service Options* dialog.
- 2. Select Web Services and scroll to the right to find 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 "Aerial\_CO.web" as the *File name* and click **Save** to close the *Save Web Services Data File(s)* dialog.
- 6. Click **Yes** when asked to "create the following file(s)" to bring up the *Image Pixel Size* dialog.
- 7. Enter "3.0" in the Enter the image pixel size (ft) field.
- 8. When the download completes, click **No** if asked to build image pyramids.

The world imagery will now be shown (Figure 7). Feel free to **Zoom** in to review the development and the alignment of the pipe network with the streets and easements.



Figure 7 Background image after import

#### 5.2 Importing a Background Image Without an Internet Connection

This section should only be completed if no internet connection is available. If section 5.1 was completed successfully, please skip to section 6.

A background map was previously downloaded and can be imported into the project by doing the following:

- 1. Click **Open** if to bring up the *Open* dialog.
- 2. Select "All Files (\*.\*)" from the *Files of type* drop-down.

- 3. Select "Aerial\_CO.tif" and click **Open** to import the background image and exit the *Open* dialog.
- 4. Click **No** if asked to generate image pyramids.

The world imagery will now be shown (same as in Figure 7). Feel free to **Zoom** in to review the development and the alignment of the pipe network with the streets and easements.

# 6 Making Modifications to the Network

#### 6.1 Creating the New Link and Node

Now modify the EPANET model by adding a new junction and a new pipe to the network for a new community clubhouse and by changing the Hazen Williams pipe roughness coefficient.

- 1. Switch to the **Map**  $\clubsuit$  module.
- 2. Click **Display Options I** to bring up the *Display Options* dialog.
- 3. Select "River Data" from the list on the left.
- 4. On the 1-D Hydraulic River tab, turn off River Hydraulic Schematic.

This turns off the display of the links and nodes in the EPANET Schematic.

- 5. Click **OK** to exit the *Display Options* dialog.
- 6. **Zoom** in to the community clubhouse area highlighted in Figure 8.

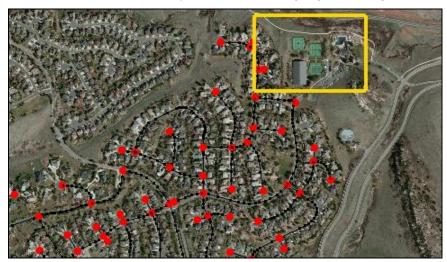


Figure 8 Zoom in to the area around the clubhouse

7. Using the **Create Feature Arc** tool, create a new arc from the corner node visible in the street below the clubhouse to the intersection of the road and the parking lot across the street from the clubhouse (Figure 9).

Ensure that the new arc is connected to the existing network at the node located at the starting point in Figure 9. This is accomplished when starting the arc by clicking close

enough to the existing node that the new arc snaps to it. If it is not connected, delete it and recreate it.

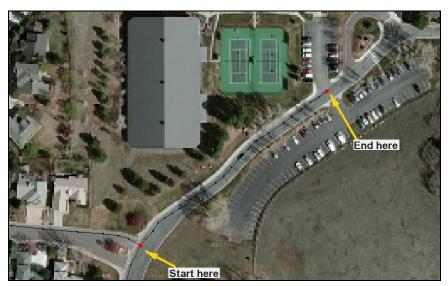


Figure 9 Location of the new pipe to the clubhouse

- 8. Click **Display Options I** to bring up the *Display Options* dialog.
- 9. Select "River Data" from the list on the left.
- 10. On the 1-D Hydraulic River tab, turn on River Hydraulic Schematic.

This turns on the display of the links and nodes in the EPANET Schematic.

- 11. Click **OK** to exit the *Display Options* dialog.
- 12. Select *Water Distribution* |  $Map \rightarrow 1D$  Schematic.

This adds the new link and node to the existing network (Figure 10).



Figure 10 New link and node

#### 6.2 Modifying the New Link and Node

Now assign the appropriate parameters to the new link and node.

- 1. Switch to the **Hydraulic Modeling** \* module.
- 2. Select *EPANET* | **Edit Parameters...** to bring up the *Hydraulic Properties* dialog.
- 3. Select "Link" from the *Attribute type* drop-down.
- 4. Select "All" from the *Show* drop-down.
- 5. Select "Name" from the *Sort based on* drop-down.

The sort tool allows the data to be sorted based on any of the parameter columns. This makes it easier to navigate through the list of link parameters, especially if there are a large number of them.

- 6. On the Link 30 row, enter:
  - "P-121" in the *Name* column (replacing "Link 30").
  - "12.0" in the *Diameter/Height* column.
  - "Residential" in the *Tag* column.
- 7. On the *All* row (the one highlighted in yellow), enter "130.0" in the *Roughness* column.

This changes the Hazen Williams roughness coefficient for all of the pipe links in the project.

- 8. Select "Nodes" from the *Attribute type* drop-down.
- 9. Select "Name" from the Sort based on drop-down.
- 10. On the Node 30 row:
  - Enter "JN-119" in the *Name* column (replacing "Node 30").
  - Enter "5987.54" in the *Elevation/Head* column.
  - Enter "3.1" in the *Base Demand* column.
  - Select "Residential Usage" from the drop-down in the *Demand Pattern* column.
  - Enter "Residential" in the *Tag* column.
- 11. Make sure all of the Nodes have "Residential\_Usage" selected from the drop-down in the *Demand Pattern* column.
- 12. Click **OK** to exit the *Hydraulic Properties* dialog.

# 7 Exporting and Running the Modified Network in EPANET

Now that all of the modifications have been made to the pipe link network, it needs to be exported to a format readable by the external EPANET application. The project should then be saved.

#### 7.1 Exporting the Modified Network

Now that the network has been modified in WMS, export it to run in the external EPANET application.

- 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 "Water\_Dist\_CO\_Modified.inp" as the *File name* and click **Save** to export the INP file and close the *Select an EPANET File* dialog.

The pipe link network has now been exported with all the changes made in this tutorial. Before switching to the external EPANET application, save the WMS project.

- 4. Select *File* | **Save** to bring up the *Save As* dialog.
- 5. Select "WMS XMDF Project File (\*.wms)" from the *Save as type* drop-down.
- 6. Enter "Colorado\_Network.wms" as the *File name* and click **Save** to save the project under the new name and close the *Save As* dialog.
- 7. Click **No** if asked to save the image files in the project directory.
- 8. Close WMS by clicking the in the top right corner of the WMS window.

#### 7.2 Opening the Modified Network in EPANET and Re-running the Model

- 1. Open the external EPANET application.
- 2. Click **Open** if to bring up the *Open a Project* dialog.
- 3. Select "Input file (\*.INP)" from the *Files of type* drop-down.
- 4. Select "Water\_Dist\_CO\_Modified.inp" and click **Open** to import the input file and exit the *Open a Project* dialog.

The project should appear similar to Figure 11.

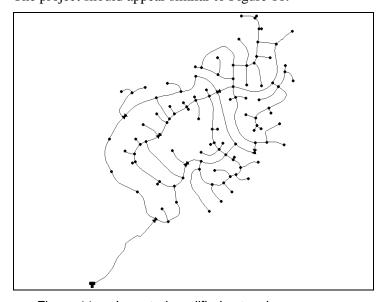


Figure 11 Imported modified network

5. Using the **Select Object** tool, double-click on the new link to bring up the *Pipe P-121* dialog.

This dialog allows the various property values of the link to be reviewed.

- 6. Click the in the top right corner to close the *Pipe P-121* dialog.
- 7. Repeat steps 5–6 as desired for any other links and nodes.

Notice that the Roughness values have changed for all of the links.

- 8. Click **Run** § to bring up the *Run Status* dialog.
- 9. When the run completes, click **OK** to close the *Run Status* dialog.
- 10. In the *Browser* dialog, on the *Map* tab, select "Pressure" from the *Nodes* dropdown and "Flow" from the *Links* drop-down.

This allows visualization of the computed pressures and flows.

- 11. Observe the flows and pressures over time in the 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.
- 12. Open the original model ("Water\_Dist\_CO.inp") in another instance of EPANET for a side-by-side comparison.

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.<sup>3</sup>

## 8 Conclusion

This concludes the "Water Distribution System Modeling – EPANET Hydraulic Model" tutorial. The following key concepts were discussed and demonstrated:

- Opening an existing model in the EPANET interface and running it to observe the existing conditions solution
- Exporting the network as an EPANET input file (\*.inp)
- Opening the existing model input file in WMS
- Setting the coordinate projection
- Downloading a basemap
- Making changes to link and node properties
- Exploring editing options
- Exporting the modified model and opening it within the EPANET interface
- Running and viewing the modified EPANET model

<sup>&</sup>lt;sup>3</sup> See https://nepis.epa.gov/Adobe/PDF/P1007WWU.pdf.