EPANET FAQ Sheet

- **Where can I find the Mac OSX or other platform versions of EPANET?**

  US EPA only offers a Windows version of EPANET. However, our collaborators at OpenWaterAnalytics do offer a command line version of EPANET. While this version does not include the graphical user interface, it does include all the capabilities in US EPA's release of EPANET 2.2.0. Here is the link for the release: https://github.com/OpenWaterAnalytics/EPANET/releases/tag/v2.2

  If you need help on this release, please go to the "Issues" for the repository at: https://github.com/OpenWaterAnalytics/EPANET/issues

  Also, information and assistance can be found at the Wiki: https://github.com/OpenWaterAnalytics/EPANET/wiki

  and Community Forum: http://community.wateranalytics.org/

- **What do I need to understand to design an appropriate tank for a system?**

  To determine the appropriate size and characteristics of a tank for your system, you first need to understand (1) the size and throughput of your water source, (2) the size of your population that will be served by your water system, and (3) the average and maximum daily demands (water usage) that the population served by your water system will require.

  For (1), you need to know the rate (e.g., liters per second) that your source water well can deliver or provide water. For (2), you need to understand the demand patterns of your population. Finally, for (3), you need to understand the peak water demands that will be required by the people of your water system.

  EPANET cannot directly provide this information. You will first need to research and find an appropriate textbook or research papers and reports to help you design your water system. Once you have designed your water system, you can then use EPANET to examine various operational scenarios to help ensure that you have designed your system appropriately.

- **Why won't EPANET open or show up on screen?**

  If EPANET won't open and gives an error when you try to open it, especially after OS update, try running the non-installing EPANET 2.2. This is the link to the "Non-Installing Software for EPANET 2.2 (ZIP)" from: https://github.com/USEPA/EPANET2.2/releases/download/2.2.0/epanet2.2.zip. After downloading, just extract the contents to your chosen folder location and then double click the executable and EPANET should open.

EPANET 2.2 now works on multi-monitor setups, so that shouldn't be the issue. EPANET might also be slow to show if it is currently running a simulation(s) and other windows were open over it.

- **Can I import GIS, DXF, WaterCAD, or other files into EPANET?**

  EPANET does not have the capability to import DXF, GIS, WaterCAD, or most other commercial water modeling files. EPANET can only use models from its input files (e.g. .inp, .net, or .bak). You can also import an image for the background or map of the model, but it must be a WEM, MAP, or BMP.

  There is an ongoing research project titled, "Re-Engineering SWMM and EPANET User Interface Application Software Architectures". This project, while it still has bugs, provides some connectivity between EPANET and GIS through QGIS. This is an open source software project can be found at https://github.com/USEPA/SWMM-EPANET_User_Interface/tree/master.

- **Can I make an addition or modification to an existing model?**

  EPANET models are always editable. You can easily add, delete, or edit components of the network.

- **Can I set a pattern with a time step other than the default 1 hour and/or a repeat duration other than the default 24 hours?**

  You can change the "Pattern Time Step", within the "Time Options", to something other than the default of 1 hour. This is useful if the pattern needs to change by the minute, by the day, or some other custom interval.

  The pattern will have 24 time periods by default, but more can be added by hitting the **Enter** key when in the last cell of the editor.

  Patterns can even be saved (as a .pat) or loaded for use in other models through the "Pattern Editor" window.

- **Can I do transient analyses in EPANET?**

  EPANET does not support transient state analyses, but there is a 3rd-party open source transient simulation tool using EPANET models called TSNet. Find more about it here: https://tsnet.readthedocs.io/en/latest/content.html

- **Can EPANET simulate fluids other than water?**

  EPANET can be used to model all forms of fluid transport. EPANET will always assume that the fluid is incompressible, part of laminar, transition, and turbulent flow, in a closed pipe (e.g. contaminant injections are modeled as mass/time), and in a full pipe. You just need to adjust the appropriate parameters for the fluid (e.g. viscosity).

- **Are there any security concerns with EPANET?**

  We are not aware of any malware/virus issues associated with downloading EPANET 2.2 from our website. If you are still concerned, you should use the "Non-Installing Software for EPANET 2.2 (ZIP)" from: https://github.com/USEPA/EPANET2.2/releases/download/2.2.0/epanet2.2.zip. After downloading, just extract the contents to your chosen folder location and then double click the executable and EPANET should open.

- **Does EPANET have variable speed pumps (VSP)?**

  EPANET does not have the capability to automatically adjust pump speed (e.g. variable speed pump (VSP)) to achieve a desired tank level, flow, or pressure. You can specify a pump curve to apply a pump utilization procedure to adjust pump speed, but this assumes the user knows what the speed of the pump should be ahead of time.

  Our collaborators at OpenWaterAnalytics are contemplating a future enhancement to EPANET to include a VSP capability. lease see the issue discussion at https://github.com/OpenWaterAnalytics/EPANET/issues/530.

- **Why am I getting the error, "Warning: Pump XYZ closed because cannot deliver head at #.##.## hrs."?**

  This error is issued when a pump is asked to operate outside the range of its pump curve and EPANET will close the pump if that head is greater than the pump's shutoff head.

  Sufficiently increasing the pump curve, particularly at and around the shutoff head, is the most direct way to resolve the warning, but this increase might be unrealistic or impossible to reflect in the physical system.

  The warning could also be resolved by simply increasing the Maximum Trials ("Project" in the taskbar >> "Defaults..." >> "Hydraulics" tab >> increase "Maximum Trials"). This is especially if this warning is accompanied by other warnings.

  The most menial, but realistic and informative way to resolve this warning is to examine the components around the pump to find why the system is demanding the excessive head. Pay particular attention to the Head of Reservoirs and Tanks, flow restricting pipes and valves, and other nearby and stronger pumps (all at the "#.##.## hrs" time).

- **How can I verify the results if they don't seem correct/realistic?**

  EPANET is a calculates the results correctly based on the inputs it's given. If you are questioning the results, instead review the values of the inputs. If the head loss across a particular pipe seems excessively high, then it is still calculated correctly, but is possibly caused by pushing too much water through a too small pipe.

- **Why isn't the water quality changing in some parts of the model?**

  If the water quality isn't changing, the water quality may be in equilibrium in some links and junctions. Alternatively, the water quality not changing as expected at the peripherals of the system may be because EPANET 2.2 doesn't use any trickle flow. This means that dead-end junctions with 0 demand will not get any new (nor start with any) water.

- **Why aren't changes to certain settings saving in EPANET?**

  Some settings in EPANET are saved to the file instead of the program. These settings include more than just the network information, such as the printer margins.

- **How can I import a pipe network created with a CAD or GIS program?**

  EPANET can import a geometric description of a pipe network in a simple text format. This description simply contains the ID labels and map coordinates of the nodes and the ID labels and end nodes of the links. This simplifies the process of using other programs, such as CAD and GIS packages, to digitize network geometric data and then transfer these data to EPANET.

  The format of a partial network text file looks as follows, where the text between brackets (< >) describes what type of information appears in that line of the file:

```
[TITLE]

<optional description of the file>

[JUNCTIONS]

<ID label of each junction>

[PIPES]

<ID label of each pipe followed by the ID labels of its end
junctions>

[COORDINATES]

<Junction ID and its X and Y coordinates>

[VERTICES]

<Pipe ID and the X and Y coordinates of an intermediate vertex point>
```

Note that only junctions and pipes are represented. Other network elements, such as reservoirs and pumps, can either be imported as junctions or pipes and converted later or simply be added in later. The user is responsible for transferring any data generated from a CAD or GIS package into a text file with the format shown above.

In addition to this partial representation, a complete specification of the network can be placed in a file using the format described in Appendix Command Line EPANET. This is the same format EPANET uses when a project is exported to a text file. In this case the file would also contain information on node and link properties, such as elevations, demands, diameters, roughness, etc.

You may also want to try using our research tool that is currently under development. Here is the link: https://github.com/USEPA/SWMM-EPANET_User_Interface/releases/tag/MTP7r0. This executable program installs a python version of EPANET which has the capability to import a map of your piping network and other attributes using a GeoJSON (.geojson) file format. This program is under development and may have issues. If you can import your piping and associated infrastructure elements you will then want to save the file as an EPANET .inp file.

- **How do I model a groundwater pumping well?**

  Represent the well as a reservoir whose head equals the piezometric head of the groundwater aquifer. Then connect your pump from the reservoir to the rest of the network. You can add piping ahead of the pump to represent local losses around the pump.

  If you know the rate at which the well is pumping, then an alternate approach is to replace the well – pump combination with a junction assigned a negative demand equal to the pumping rate. A time pattern can also be assigned to the demand if the pumping rate varies over time.

- **How do I size a pump to meet a specific flow?**

  Set the status of the pump to CLOSED. At the suction (inlet) node of the pump add a demand equal to the required pump flow and place a negative demand of the same magnitude at the discharge node. After analyzing the network, the difference in heads between the two nodes is what the pump needs to deliver.

- **How do I size a pump to meet a specific head?**

  Replace the pump with a Pressure Breaker Valve oriented in the opposite direction. Convert the design head to an equivalent pressure and use this as the setting for the valve. After running the analysis, the flow through the valve becomes the pump's design flow.

- **How can I enforce a specific schedule of source flows into the network from my reservoirs?**

  Replace the reservoirs with junctions that have negative demands equal to the schedule of source flows. (Make sure there is at least one tank or remaining reservoir in the network, otherwise EPANET will issue an error message.)

- **How can I analyze fire flow conditions for a particular junction node?**

  To determine the maximum pressure available at a node when the flow demanded must be increased to suppress a fire, add the fire flow to the node's normal demand, run the analysis, and note the resulting pressure at the node. To determine the maximum flow available at a particular pressure, set the emitter coefficient at the node to a large value (e.g., 100 times the maximum expected flow) and add the required pressure head (2.3 times the pressure in psi) to the node's elevation. After running the analysis, the available fire flow equals the actual demand reported for the node minus any consumer demand that was assigned to it.

- **How do I model a reduced pressure backflow prevention valve?**

  Use a General Purpose Valve with a head loss curve that shows increasing head loss with decreasing flow. Information from the valve manufacturer should provide help in constructing the curve. Place a check valve (i.e., a short length of pipe whose status is set to CV) in series with the valve to restrict the direction of flow.

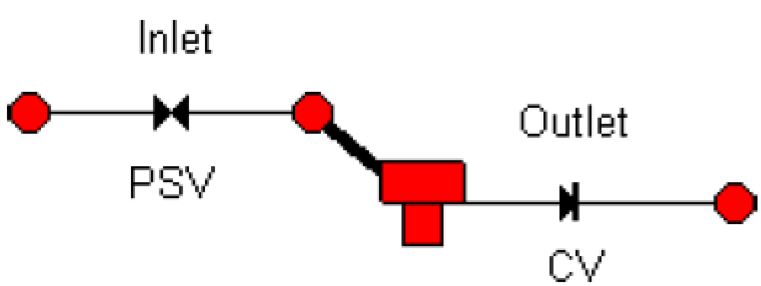
- **How do I model a pressurized pneumatic tank?**

  If the pressure variation in the tank is negligible, use a very short, very wide cylindrical tank whose elevation is set close to the pressure head rating of the tank. Select the tank dimensions so that changes in volume produce only very small changes in water surface elevation.

  If the pressure head developed in the tank ranges between H_1 and H_2, with corresponding volumes V_1 and V_2, then use a cylindrical tank whose cross-sectional area equals $(V_2 - V_1)/(H_2 - H_1)$.

- **How do I model a tank inlet that discharges above the water surface?**

  Use the configuration shown below:



Example of Tank Inlet Discharging above Water Surface.

The tank's inlet consists of a Pressure Sustaining Valve followed by a short length of large diameter pipe. The pressure setting of the PSV should be 0, and the elevation of its end nodes should equal the elevation at which the true pipe connects to the tank. Use a Check Valve on the tank's outlet line to prevent reverse flow through it.

- **How do I determine initial conditions for a water quality analysis?**

  If simulating existing conditions monitored as part of a calibration study, assign measured values to the nodes where measurements were made and interpolate (by eye) to assign values to other locations. It is highly recommended that storage tanks and source locations be included in the set of locations where measurements are made.

  To simulate future conditions, start with arbitrary initial values (except at the tanks) and run the analysis for several repeating demand pattern cycles so that the water quality results begin to repeat in a periodic fashion as well. The number of such cycles can be reduced if good initial estimates are made for the water quality in the tanks. For example, if modeling water age the initial value could be set to the tank's average residence time, which is approximately equal to the fraction of its volume it exchanges each day.

- **How do I estimate values of the bulk and wall reaction coefficients?**

  Bulk reaction coefficients can be estimated by performing a bottle test in the laboratory (see Bulk Reactions in Section 3.4 of EPANET 2.2 User's Manuel). Wall reaction rates cannot be measured directly. They must be backfitted against calibration data collected from field studies (e.g., using trial and error to determine coefficient values that produce simulation results that best match field observations). Plastic pipe and relatively new lined iron pipe are not expected to exert any significant wall demand for disinfectants such as chlorine and chloramines.

- **How can I model a chlorine booster station?**

  Place the booster station at a junction node with zero or positive demand or at a tank. Select the node into the Property Editor and click the ellipsis button in the Source Quality field to launch the Source Quality Editor. In the editor, set Source Type to SETPOINT BOOSTER and set Source Quality to the chlorine concentration that water leaving the node will be boosted to. Alternatively, if the booster station will use flow-paced addition of chlorine then set Source Type to FLOWPACED BOOSTER and Source Quality to the concentration that will be added to the concentration leaving the node. Specify a time pattern ID in the Time Pattern field if you wish to vary the boosting level with time.

- **How would I model trihalomethanes (THM) growth in a network?**

  THM growth can be modeled using first-order saturation kinetics. Select Options – Reactions from the Data Browser. Set the bulk reaction order to 1 and the limiting concentration to the maximum THM level that the water can produce, given a long enough holding time. Set the bulk reaction coefficient to a positive number reflective of the rate of THM production (e.g., 0.7 divided by the THM doubling time). Estimates of the reaction coefficient and the limiting concentration can be obtained from laboratory testing. The reaction coefficient will increase with increasing water temperature. Initial concentrations at all network nodes should at least equal the THM concentration entering the network from its source node.

- **Can I use a text editor to edit network properties while running EPANET?**

Save the network to file as ASCII text (select File >> Export >> Network). With EPANET still running, start up your text editor program. Load the saved network file into the editor. When you are done editing the file, save it to disk. Switch to EPANET and read in the file (select File >> Open). You can keep switching back and forth between the editor program and EPANET, as more changes are needed. Just remember to save the file after modifying it in the editor, and re-open it again after switching to EPANET. If you use a word processor (such as Word) or a spreadsheet as your editor, remember to save the file as plain ASCII text.

- **Can I run multiple EPANET sessions at the same time?**

  Yes. This could prove useful in making side-by-side comparisons of two or more different design or operating scenarios.