Simulation Options

The **Simulation Options** dialog is used to adjust some facet of a model prior to running a model. Simulation options are usually altered to be associated with a specific scenario.

For example, suppose the user would like to specify a global bulk chlorine decay of -0.5 for one simulation and have another simulation with a decay of -0.7. This can be accomplished through the Simulation Options, where one option can be tied to one scenario while another option is specified for a different scenario.

To create a simulation option, the Simulation Options dialog can be accessed from the Operation tab of the Model Explorer or from the Simulation Options' Browse button in the Run Manager.

General Options

ID and Description

The Simulation Options ID and Description area shows the name and description of the defined options. The red check on the ID folder indicated that the options are currently active.
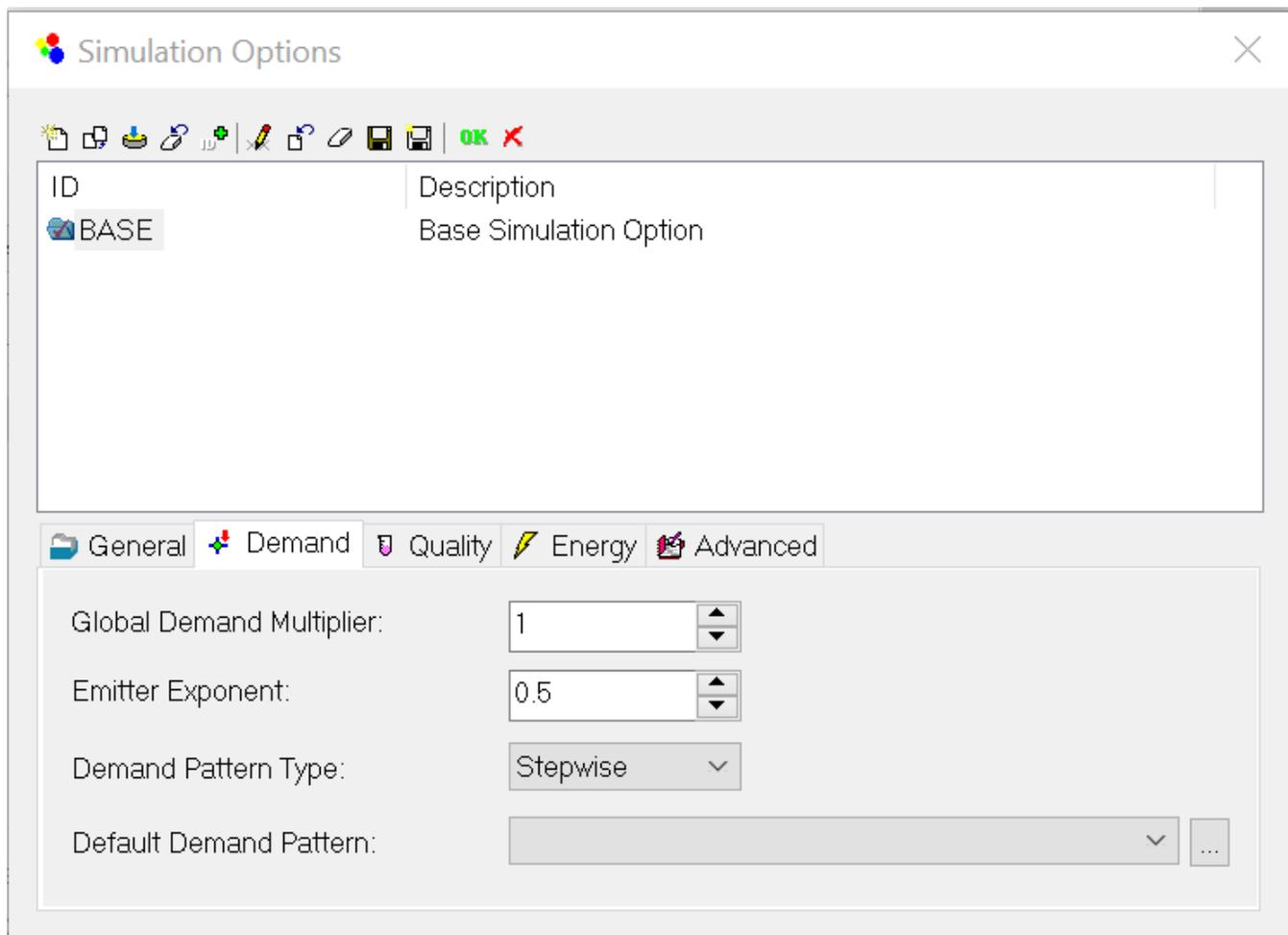

General Tab

- **Flow Unit** - Units of flow measurement (gpm, cfs, mdg, imgd, afd, lps, lpm, mld, cmh, and cmd). The selection of flow units will determine if this model is to use US (Imperial) units of SI units. If required, it is possible to mix US and SI.
- **Headloss Equation** - Determines which headloss equation (Hazen-Williams, Darcy-Weizbach, Chezy-Manning) will be used.
- **Trials** - Sets the maximum number of trials used to solve the nonlinear set of equations that govern network hydraulics at a given point in time. The suggested value is 40. A more complex system may require more trials to reach the required accuracy set below. If system to be modeled consistently gives Maximum Trials Exceeded errors, It may be useful to adjust the maximum trials. Please note: If more trials are required, it will increase the simulation time. If the accuracy can not be attained in 100 to 200 iterations, the model may not be solvable. Please check the simulation report for warning and error messages.
- **Accuracy** - Convergence criterion used to signal that a solution has been found to the nonlinear equations that govern network hydraulics. Trials end when the sum of all flow changes divided by the sum of all link flows is less than this number. Suggested value is 0.001. Please note: A smaller value will provide a more accurate solution but it will usually require more trials and increase run time. A larger number will give a less accurate solution and will be solved with fewer trials.
- **Specific Gravity** - The SG of the liquid being modeled (potable water is 1.0).
- **Unbalanced** - Action to take if a hydraulic solution is not found within the maximum number of trials. Choices are **STOP** to stop the simulation at this point or **CONTINUE** to use the number of trials entered into the Extended Run drop down box. The key to this feature is that the software does not allow link status changes to occur which, in effect, may force the model to converge. Please Note: Using the Continue option may provide inaccurate results that may or may not affect subsequent timestep simulations.
- **Extended Run** - Used in conjunction with the Unbalanced option. By entering a number the user is specifying the number of additional trials to converge the model.
- **Enable Rule Control** - When the box is ON (checked), the current *active* logical control set will be enabled for the hydraulic simulation. When OFF (unchecked), the logical control set specified during a simulation run will be disabled. The user is allowed this option to verify the impact of logic sets on the hydraulic model by enabling or disabling logic functions prior to a model run.
- **Pressure Unit** - Units to be used for pressure measurements (psi, m, kPa).
- **(Relative) Viscosity** - Ratio of the kinematic viscosity of the fluid to that of water at 20 deg. C (1.0 centistokes or 0.94 sq ft/day) (dimensionless).
- **Vapor Pressure** - Vapor pressure of the water being modeled. The higher the temperature, the higher the vapor pressure. Water at 70° F has a vapor pressure of 0.84.
- **Hydraulic Usage** - Used to indicate whether or not to save the hydraulic solution for the next simulation or to use a previously saved hydraulic solution in the upcoming simulation. If the Use or Save option is selected, the user must specify the Hydraulic File (.hyd) path and name, below, that is to be used.
- **Hydraulic File** - The external path and file name for the Hydraulic File (.hyd) file that is to be used for the Hydraulic Usage option selected above.
- **Working Folder** - The working folder only applies when the hydraulic usage is set to None. It represents the network directory that is to be used for writing the model report. If troubles occur during a model run, look for the output model report in the directory specified by the working folder.

Demand Options

- **Global Demand Multiplier** - The Global Demand Multiplier is used to assign a multiplier by which all demand is to be multiplied. For example, a MAXDAY model scenario may have a global demand multiplier of 2.5. All demands would be multiplied by a factor of 2.5.

- **Emitter Exponent** - The default value to be assigned as the pressure exponent (γ gamma) for the emitter equation $$q = C\ p^r$$. See Emitter Methodology

- **Demand Pattern Type** - Select the type of pattern to be used by InfoWater. Between time-steps, two options are available for representing usage factors between regular fixed periods in time:
    - *Stepwise* – InfoWater assumes a constant usage factor from one fixed period of time to the next.
    - *Continuous* – InfoWater linearly interpolates variations in the pattern factor between fixed time periods.
    - To learn more, select Pattern Representation Options.

- **Default Demand Pattern** - The default demand pattern that is to be applied to the simulation. Use the Browse [...] button to select a pattern.

Quality Options

Simulation Options

ID	Description
BASE	Base Simulation Option

General · **Demand** · **Quality** · **Energy** · **Advanced**

Option

- ( ) None
- (●) Chemical / Temp
- ( ) Water Age
- ( ) Source Trace
- ( ) Multi-Trace
- ( ) Advanced

Trace Node: [   ] [<]

Multi-Trace on: [All Storage ▾] [DISCONNECTED_NODES, Disconnect ▾] [...]

[ ] Temperature Modeling   WQ Tolerance: [0.001]

Chemical Name: [Chlorine]   Mass Unit: [mg/L ▾]

Global Bulk: [0]   Global Wall: [0]

Diffusivity (ratio): [1]   Limiting Potential: [0]
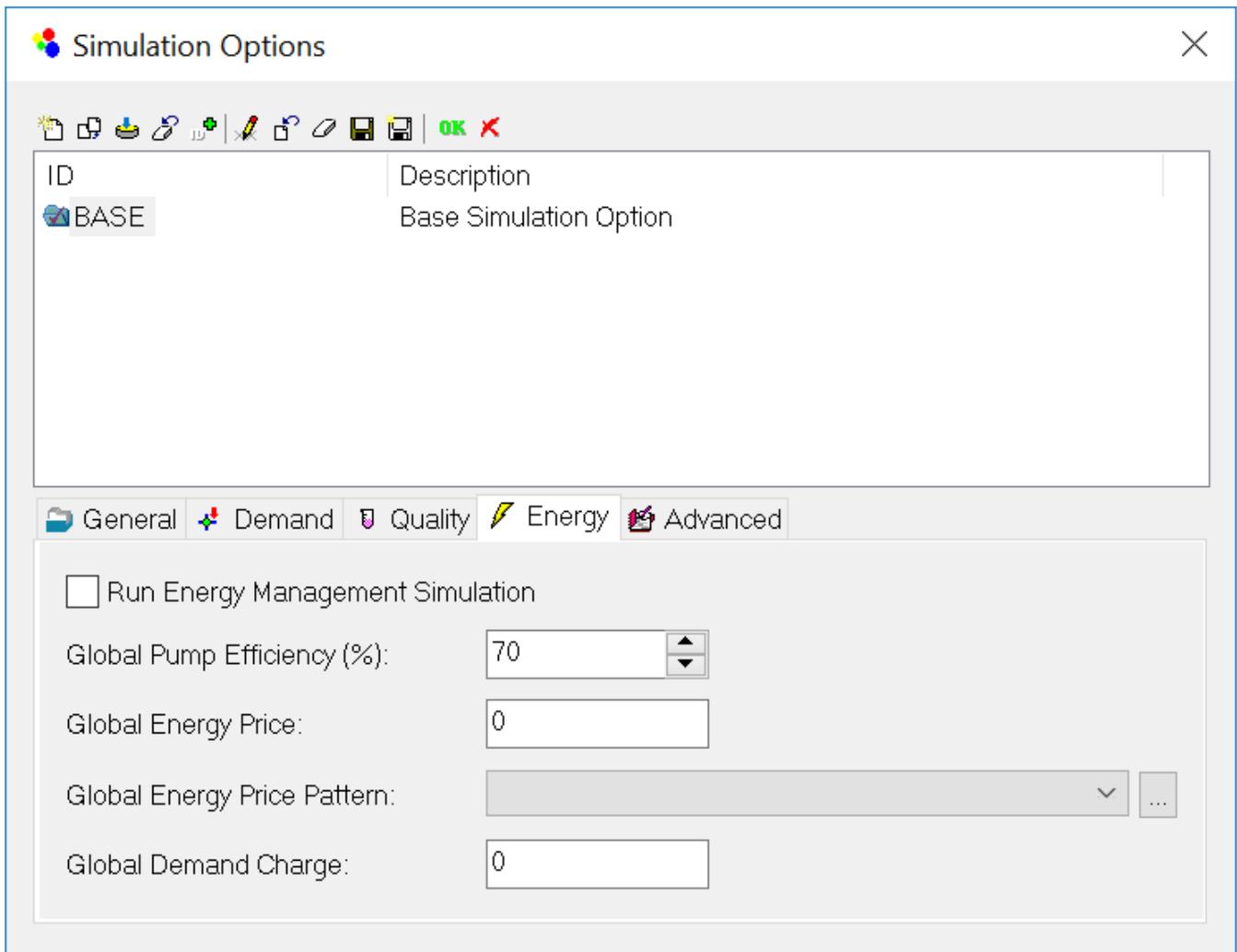
Global Pipe Bulk Reaction Order: [1]

Global Pipe Wall Reaction Order: [1]

Global Tank Reaction Order: [1]

Roughness Correlation Coefficient: [0]

Energy Options

- **Run Energy Management Simulation** - This box must be checked to run an energy management simulation.
- **Global Pump Efficiency** - Use this value to assign a global efficiency to all pumps.
- **Global Energy Price** - The energy rate is defined as the cost per unit of energy usage ($/kWh). Enter a $ value to represent the unit price of electricity.
- **Global Energy Price Pattern** - The pattern defines the cost per unit of energy based on time of day. Specify the pattern for which energy is being purchased (electricity price schedule). Use the Browse [...] button to select a pattern.
- **Global Demand Charge** - Demand (or capacity) charge is defined as the cost associated with the maximum power consumed within the charging (billing) period ($/max. kW). This is a one-time charge due to peak power used during each billing period. Specify a cost for which energy penalties are being levied during peak conditions

Note - In order to run an Energy Management Simulation you would need to check the Run Energy Management Simulation check box.

Advanced Options

- **Status Check Frequency** - The Status Check Frequency setting will check status of all control elements at every Check Frequency iterations (default value is 2).This sets the number of solution trials that pass during hydraulic balancing before the status of pumps, check valves, flow control valves and pipes connected to tanks are once again updated. The default value is 2, meaning that status checks are made every other trial. A value equal to the maximum number of trials would mean that status checks are made only after a system has converged. (Whenever a status change occurs the trials must continue since the current solution may not be balanced.) The frequency of status checks on pressure reducing and pressure sustaining valves (PRVs and PSVs) is determined by the Damping Limit option (see below).
- **Max. Status Check Iteration** - The program will stop checking status of all control elements after Maximum Status Check Iterations setting is reached. The default value is 10. This is the number of solution trials after which periodic status checks on pumps, check valves flow control valves and pipes connected to tanks are discontinued. Instead, a status check is made only after convergence is achieved. The default value is 10, meaning that after 10 trials, instead of checking status every Status Check Frequency trials, status is checked only at convergence.
- **Relaxation Factor** - This is a relaxation factor parameter for determining the new solution (flow rate) at each iteration as follows:
  - New Q = previous Q + Relaxation Factor * Delta Q
  - The default value is 1.

Note: The role of relaxation factor.

The new solution (flow rate) at each iteration of the solution engine is:

New Q = Previous Q + Relaxation Factor * Delta Q (Q increment).

Based on the Newton-Raphson (NR) solution method, this is exactly same idea with relaxation factor = 1. On comparison to the other numerical method, the main advantage of NR method is its speed and efficiency to converge upon a solution. However, it also has a disadvantage of a high level of divergence (where it can not find a solution). As a numerical aid to reduce the chance of divergence, we can add a relaxation factor (usually 0.6 to 1). This simply decreases the Delta Q (increment). By adding a small relaxation factor (decreasing Q increment), the model tends to provide a more stable solution, but, obviously, with a slower computational speed. Therefore, the relaxation factor is a trade off between computation speed and model stability. The reason we changed the default of relaxation factor as 1 is to provide faster computation times. However, it may increase the chance of reaching a simulation failure (in most cases, it should be fine). Currently, the EPANET engine uses 0.6 as a relaxation factor.

To summarize, if you have a simulation failure that used to run in a previous version, please consider changing the relaxation factor into 0.6. Also, consider doubling the Status Check Frequency to four and change the Maximum Status Check Iteration to a value between 4 and 20 to reach a solution.

- **Damping Limit** - Must be equal to or greater than zero. This is the accuracy value at which solution damping and status checks on PRVs and PSVs should begin. Damping limits all flow changes to 60% of what they would otherwise be as future trials unfold. The default is 0 which indicates that no damping should be used and that status checks on control valves are made at every iteration. Damping might be needed on networks that have trouble converging, in which case a limit of 0.01 is suggested.
- **Remarks:**
  - Frequent status checking; tends to produce solutions in the least number of iterations.

    Example: Check Frequency = 2, Max Status Check = 10, Damping Limit = 0

  - Less frequent status checking; might be needed for networks that have difficulty in converging.

    Example: Check Frequency = 10, Max Status Check = 100, Damping Limit = 0.01

  - Status checks made only after convergence is achieved; might produce convergence when other settings fail.

    Example: Check Frequency = Max trials, Max Status Check = Max Trials, Damping Limit = Convergence Accuracy

  - **Maximum Allocated Threads for WQ Simulation** - This option controls the management of shared-memory and the synchronization of parallel computation processes (threads) for the water quality simulation. Multiple processing units (processors or cores) can now share the intensive computational demand often required by a water quality simulation. For computationally intensive simulations, the saving in computation time can be more than 30% for a dual-core processor machine. The Default setting allows the water quality simulation to utilize as many parallel threads as are currently available. The user can limit the thread allocation by specifying the desired maximum number of the computational threads, from 1 to 8, in order to reserve the desired processing power for other applications.
  - **Allow Disconnection Run:** When this option is checked on, InfoWater removes demands of disconnected junctions when disconnection occurs at certain hydraulic time step, therefore allowing the simulation run to continue. After a successful disconnection run InfoWater will report the disconnection status on pipes and demand loss on junctions. By default this option is checked off.
  - **Balance Model Element Continuity -** This option allows users to perform comprehensive flow continuity check on individual model elements at the end of each hydraulic time step
  - **Continuity Check -** Select this to enable the flow balance continuity check
  - **Flow Tolerance -** Flow balance tolerance at individual model element