Snow Melt

Uncheck this option to ignore all snow melt computations. This allows you to review the effects of snow melt on the model's hydrology. For example, a model that contained snow melt components could be first run with the snow melt computations turned on and then again with them turned off. The analysis results of the two models could then be compared to see the effect of snow melt on the model.

Hydrodynamic Analysis Parameters

The Hydrodynamic Analysis Parameters section controls how the Hydrodynamic Routing method computations are to be performed. These parameters can greatly influence the stability of the model when it encounters flow complexities. A description on how to improve model stability is included below.

These parameters have no effect for the other routing methods (i.e., Steady Flow Routing and Kinematic Wave Routing).

Inertial Terms

This drop-down list allows you to select how the inertial terms in the St. Venant momentum equation will be handled for different flow conditions. The following options are available:

Keep	Maintains the inertial terms at their full value under all conditions.
Dampen	Default value. Reduces the inertial terms as flow comes closer to being critical and ignores them when flow is supercritical.
Ignore	Drops the inertial terms altogether from the momentum equation, producing what is essentially a Diffusion Wave solution.

Note that if the model results appear to be unstable (highly variable over a short period of time) while this value is set to **DAMPEN**, then try changing this value to **IGNORE** to see if it assists in stabilizing the model.

Lengthening Time Step

This is a time step, in seconds, used to artificially lengthen channel and pipe links so that they meet the Courant stability criterion under full-flow conditions (i.e., the travel time of a wave will not be smaller than the specified conduit lengthening time step). As this value is decreased, fewer channel and pipe links will require lengthening. A value of 0 means that no channel and pipe links will be lengthened (i.e., will be using their defined length values).

Note that if the model results appear to be unstable (highly variable over a short period of time), then increasing this value will tend to make the model more stable. For example, try using a value of 60 (1 minute) and see if the model results become more stable. If the model results are stable, try reducing this value to the minimum possible while keeping the model stable. On the other hand, if the model results are not stable, then continue to increase this value up to a maximum of 300 (5 minutes). If the model is not stable at this increased value, then there is something else causing the model instability.

The ratio of the artificial length to the original length for each channel and pipe is listed in the Conduit Flow Summary Table contained in the ASCII Output Report.

Junction Surface Area

This is the surface area used at junction nodes when computing changes in water depth. If 0 is entered, then the default value of 12.566 ft^2 (1.167 m^2) is used (the area of a 4 ft diameter manhole). The value entered should be in square feet for US units or square meters for SI metric units.

Supercritical Flow Occurs When

This radio button list allows you to select the basis used to determine when supercritical flow occurs in a conduit. The following options are available:

- Water surface slope & Froude number (default)
- Water surface slope is greater than the conduit slope
- Froude number at either end of the conduit is greater than 1.0

The topmost choice, which checks for either condition, is the recommended option.

Variable Time Step

This check box allows you to indicate whether or not a variable time step should be internally computed. The variable time step is continually recomputed in order to satisfy the Courant stability criterion for each channel and pipe and to prevent an excessive change in water depth at each node. The defined **SAFETY FACTOR** is then used to compute the variable time step.

The computed variable time step will not be less than 0.5 seconds nor be greater than the specified **ROUTING TIME STEP**. If the **ROUTING TIME STEP** is set to less than 0.5 seconds, then this variable time step option is ignored.

Safety Factor

This is a safety factor, which ranges between 10% and 200%, is applied to the variable time step computed from the Courant stability criterion. A typical adjustment factor would be 75% to provide some margin of conservatism. It only applies when the **VARIABLE TIME STEP** option is checked.

Read/Write External Interface Files

The Read/Write External Interface Files section controls if and what analysis interface files should be used during the simulation. The software can use several different kinds of interface files that contain either externally imposed inputs (e.g., rainfall or inflow/infiltration hydrographs) or the results of previously run analyses (e.g., runoff or routing results). These files can help speed up simulations, simplify comparisons of different loading scenarios, and allow large study areas to be broken up into smaller areas that can be analyzed individually.

The different types of interface files that are available include:

Rainfall Interface File

The rainfall interface file is a binary file created internally by the software that can be saved and reused from one analysis to the next.

The rainfall interface file assembles each of the separate rain gage external files used in a simulation into a single rainfall data file. Normally during a simulation, the software creates a temporary file of this type when a model uses external rainfall data file(s). This temporary file is then deleted after the analysis is completed. However, if the same rainfall data are being used with many different analyses, requesting the software to save the rainfall interface file after the first run and then reusing this file in subsequent runs can save computational time.

The rainfall interface file should not be confused with a rainfall data file. The rainfall interface file is a binary file created internally by the software that processes all of the rainfall data files used by a project. The rainfall data file is an external text file that provides rainfall time series data for a single rain gage.

Runoff Interface File

The runoff interface file is a binary file created internally by the software that can be saved and reused from one analysis to the next.

The runoff interface file can be used to save the runoff results generated from a simulation run. If runoff results will not change in future runs, you can request that the software use this interface file to supply runoff results without having to repeat the runoff computations again.

RDII Interface File

The RDII Interface File is a text file that contains a time series of rainfall-dependent infiltration/inflow flows for a specified set of network system nodes. RDII (rainfall-dependent infiltration/inflow) is used in analyzing sanitary or combined sewer systems.

This file can be generated from a previous analysis run when the unit hydrographs and nodal RDII inflow data have been defined for the project, or it can be created outside of the software using some other source of RDII data (e.g., through measurements or output from a different computer program). The format of the file is the same as that of the INFLOWS INTERFACE FILE, where FLOW is the only variable contained within the file.

Hotstart Interface File

A hotstart file (sometimes called a spin-up file or a restart file) is a binary file created by the software that contains hydraulic and water quality variables for the network system at the end of a simulation run. The data contained in this file consists of the water depth and concentration of each pollutant at each node of the system, as well as the flow rate and concentration of each pollutant in each link. The hotstart file saved after a simulation run can be used to define the initial conditions for a second, subsequent run.

Hotstart files can be used to avoid the initial numerical instabilities that sometimes occur when performing hydrodynamic routing. For this purpose, a hotstart file is typically generated by imposing a constant set of base flows (for a natural channel network) or set of dry weather sanitary flows (for a sewer network) over some startup period of time. The resulting hotstart file from this run is then used to initialize a subsequent simulation run where the inflows of real interest are imposed.

It is also possible to both use and save a hotstart file in a single run, starting off the run with one file and saving the ending results to another. The resulting file can then serve as the initial conditions for a subsequent run, if desired. This technique can also be used to divide up extremely long continuous simulations into more manageable pieces.

Aside from using the Analysis Options dialog box to define a hotstart file, you can also save the results of the current simulation to a hotstart file by selecting FILE ➤ EXPORT ➤ HOTSTART FILE.