MetroFlow

Coupled Models for the Calumet and Mainstream Des Plaines TARP System

User Manual

January 2014

Developed By:

TARP Research Group, Ven Te Chow Hydrosystems Laboratory
Department of Civil & Environmental Engineering
University of Illinois at Urbana-Champaign

Champaign, IL

Under a Contract With:

Metropolitan Water Reclamation District of Greater Chicago *and*
United States Army Corps of Engineers, Chicago District

Chicago, IL

TABLE OF CONTENTS

TABLE OF FIGURES

FOREWARD

MetroFlow provides a way for a suite of models to interact in a sequential fashion to simulate a complex urban stormwater/wastewater system such as the Metropolitan Water Reclamation District of Greater Chicago's (MWRDGC's) Tunnel and Reservoir Plan (TARP). The MetroFlow interface allows the user to perform piecewise analysis of: the transformation from rainfall to runoff in overland regions, the hydraulics of combined sewer systems, the hydraulics of interceptor sewer systems (including the ability to model pumping and connecting structures), and the complex hydraulics of deep tunnel and reservoir systems (including the ability to simulate transients and track shocks).

Data can be imported or created, both of which serve as inputs into a scenario builder that allows the user to run a model or combination of models. In MetroFlow, there is a list of available working layers such that each layer corresponds to a hydrologic or hydraulic model included within the suite of models. The current working layer can be queried for results or edited. In sum, MetroFlow is a sophisticated interface that facilitates user-specified analysis of complex urban stormwater / wastewater systems.

It should be noted that MetroFlow was developed specifically to model MWRDGC's TARP system. This User Manual has been written in reference to MetroFlow as applied to TARP. However, MetroFlow's capabilities and framework can be applied to other systems.

The objectives of this user manual are:

1. To provide the user with an understanding of the capabilities and limitations of what MetroFlow can simulate (section 1);
2. To allow the user to easily install MetroFlow (section 2);
3. To provide the user with a guide to the MetroFlow interface, identifying the function of each window, toolbar and button (section 3);
4. To provide the user with instruction to allow them to quickly run and analyze a scenario in MetroFlow (section 4).

This manual is not designed to provide detail on the methodologies behind each of hydrologic/hydraulic engines that can be used in MetroFlow. Additional references are provided and referenced that will allow the user to have a better understanding of such methodologies.

Principal Investigator
Professor Marcelo H. Garcia
2535b Hydrosystems Lab
205 N Mathews Ave
Urbana, IL 61801

Email: mhgarcia@illinois.edu
Telephone: 217-244-4484
Web: http://vtchl.illinois.edu/

Co-Investigators
Dr. Arthur R. Schmidt
2524 Hydrosystems Lab
205 N Mathews Ave
Urbana, IL 61801

Email: aschmidt@illinois.edu
Telephone: 217-333-4934

DISCLAIMER OF LIABILITY

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE CONTRIBUTORS, COPYRIGHT HOLDERS, THE UNIVERSITY OF ILLINOIS, OR THE METROPOLITAN WATER RECLAMATION DISTRICT OF GREATER CHICAGO BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS WITH THE SOFTWARE.

1. METROFLOW SPECIFICATION

1.1 Purpose

MetroFlow provides a framework so that different models, which reflect different purposes, scales, and levels of complexity, can be consolidated into a single system. MetroFlow allows for:

- Simulation of hydraulic response of TARP to the expected range of hydrologic and operational conditions;
- Simulation at precision and complexity consistent with the needs being addressed.

1.2 Scope

MetroFlow currently has the ability to run the following modules for the TARP system:

- Rainfall
  - Design Storm Event
  - Historical Rainfall Event
  - Extended Period Simulation (e.g. average water year)
  - User defined rainfall hyetograph (e.g. predicted rainfall event)
- Hydrologic Model (Overland Flow and Combined Sewer Systems)
  - SWMM 5 (e.g. lumped hydrologic model for Calumet)
  - Illinois Urban Hydrologic Model (Mainstream & Calumet)
  - City of Chicago InfoWorks model (Mainstream only)
- Hydraulic Models (Interceptor and TARP Systems)
  - SWMM 5 for the Interceptors, connecting structures between interceptors and TARP, and CSS and interceptor model (Calumet only)
  - SWMM 5 for the TARP system (Calumet only)
  - Illinois Conveyance Analysis Program – ICAP (Mainstream & Calumet)
  - Illinois Transient Model – ITM (Mainstream & Calumet)
  - InfoWorks model for simulating connecting structures and TARP (Mainstream only)

1.2.1 Capabilities Not Included

The models are run in a modular fashion with each module taking the input from the previous model, converting it to an output file that becomes the input file for the next module. At present the model does not allow real time feedback among modules. For example, backwater impacts in the combined sewers that may result from conditions in the interceptor and TARP system are not fed back into the IUHM/Lumped hydrologic model seamlessly. A free overfall (i.e. critical depth) is assumed as the downstream boundary condition for IUHM/Lumped hydrologic model.

1.2.2 Example Scenarios

MetroFlow is capable of being used in many different ways. For example the following scenarios may be assessed using MetroFlow:

1. In September 2008 a large storm hit Chicago causing wide spread flooding in the Chicagoland area and combined sewer overflows in nearby waterways. The Metropolitan Water Reclamation District of Chicago (MWRDGC) wishes to investigate this event further and determine how the system would have reacted if the TARP reservoirs were online. In addition, residents in some locations reported geysering. MWRDGC wishes to investigate measures to mitigate such events.

2. For planning purposes, the MWRDGC would like to know the number of CSO's and percentage of CSO captured during the average water year.
3. Under the current agreement between the United States Army Corps of Engineers and MWRDGC, a percentage of the capacity of Thornton Reservoir is reserved for inflows from Thorn Creek for flood control. MWRDGC wishes to investigate how the Calumet TARP system behaves under different initial conditions in the reservoir and inflows from Thorn Creek.
4. Based on prediction of rainfall, what outfalls will discharge?

1.3 Inputs

There is an ability to input and/or change the following:

- Design storm rainfall hyetograph;
- User-defined rainfall hyetograph (variable in time and space);
- Rainfall hyetograph from precipitation-gage data;
- Characteristics of service areas and combined sewer system (e.g. IUHM input files);
- Characteristics of interceptor system (e.g. infiltration and inflow, sanitary flows, etc.);
- Tunnel geometry (e.g. diameter, length, roughness etc.);
- Control rules (e.g. gate openings, pump controls, initial water elevations etc.).

1.4 Functions

The model allows the following:

- Transformation from rainfall to runoff;
- Simulate unsteady flow phenomenon for TARP tunnels;
- Operation of control gates and pumping stations;
- Operation of isolation gates in the TARP system;
- Allow testing of modifications to the system;
- Run in a timely fashion;
- Be used by District staff.

1.5 Outputs

The following outputs are available:

- Output file showing continuity errors and mass balance;
- Error warnings (and guidance on what they mean and how to fix them);
- Time series plots for nodes and links showing head, flow etc. (use SWMM as an example);
- Summary table for combined sewer overflows, identifying location, duration, frequency, and excess flow from TARP and connecting structures;
- Animation showing profile plot and HGL changes with time (use SWMM as an example);
- Color coded map interface to view changes in system over time;
- Ability to export tables and graphs;
- Mechanism for highlighting different hydraulic modes (e.g. steady, unsteady, transient).

2. INSTALLING METROFLOW

2.1 Software/Hardware Requirements

The minimum hardware requirements to run MetroFlow are:

- Hard Drive: 80 GB minimum (1 GB for local installation of programs, 10 GB minimum for data);
- Processor: Intel Core 2 Duo, 2 GHz or better;
- RAM: 2 GB minimum;
- Monitor: 21-inch or larger (preferably widescreen aspect ratio);
- Operating System: Windows XP or higher; 64-bit OS are supported;
- Security: Administrative access for installation of programs.

The program is to be open source and available to the District and its consultants.

2.2 Installing MetroFlow

Installation Guide

To install MetroFlow on a local computer, Internet access on the computer that will run MetroFlow is required. After installation, Internet access is not required (but is desirable in order to obtain software updates). If no Internet access is available, Nils Oberg can be contacted via email (noberg@illinois.edu) to obtain a standalone copy of the program. Installation is performed via a hybrid approach. Two libraries are required for running MetroFlow: the MATLAB R2011a MCR runtime and the MapWindow 4.7 ActiveX control; these require administrator access. The actual MetroFlow program is installed via ClickOnce technology and does not require administrator access. In addition, no administrator access is required to run the program. It is safe to ignore all warnings that are presented during the installation process.

The prerequisites are obtained from the following links:

http://hydrolab.illinois.edu/software/metroflow-1.7/MCRInstaller.exe [163 MB]

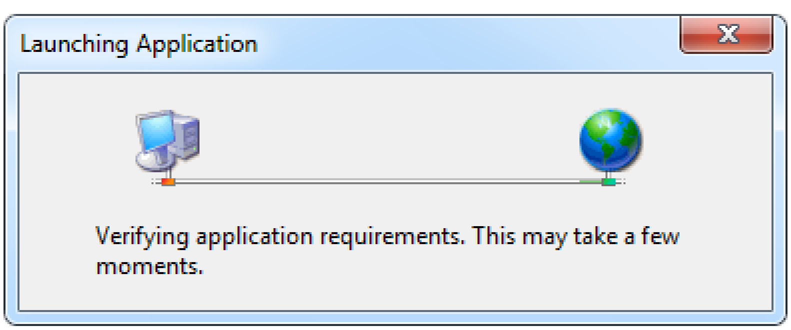
http://hydrolab.illinois.edu/software/metroflow-1.7/MapWinGIS47SRa-x86-Setup.exe [25 MB]

These programs must be installed prior to launching the MetroFlow installation. They require Administrator access in order to be installed, and all default options should be selected.

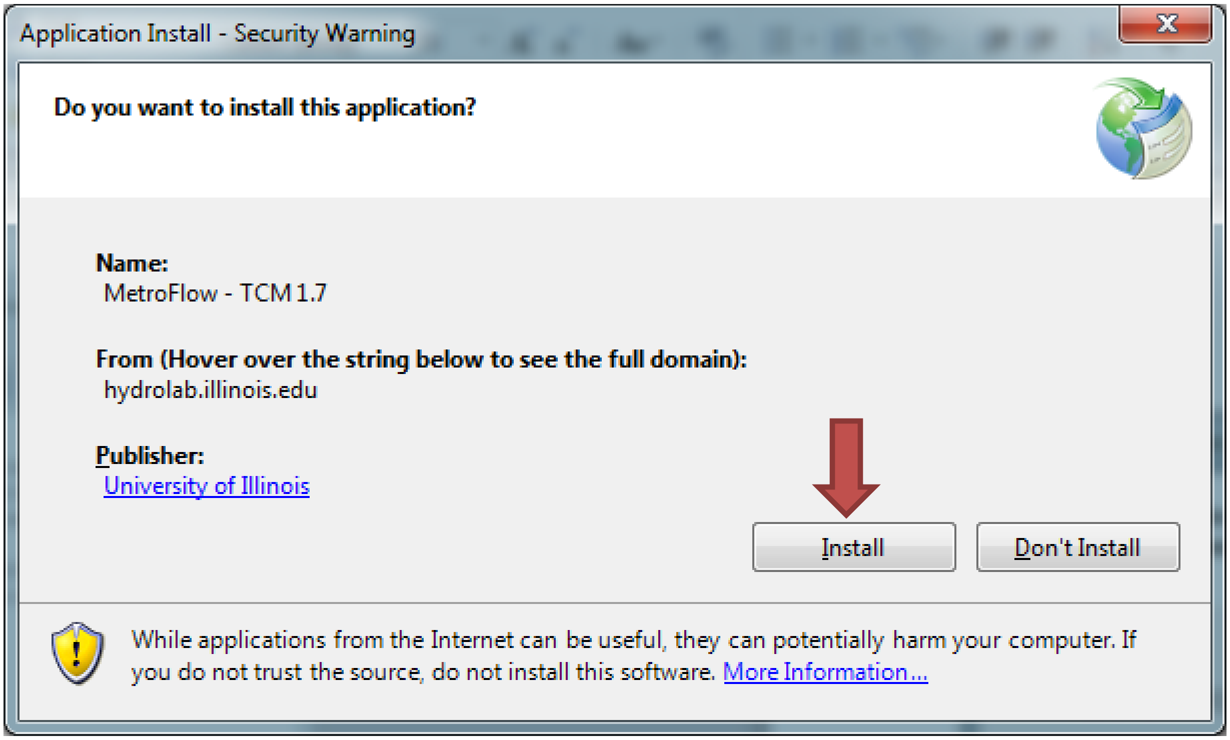
The ClickOnce installer is downloaded from the following URL (and may also be included on any applicable media that was received with this manual):

http://hydrolab.illinois.edu/software/metroflow-1.7/MetroFlow-1.7.application [< 6 MB]

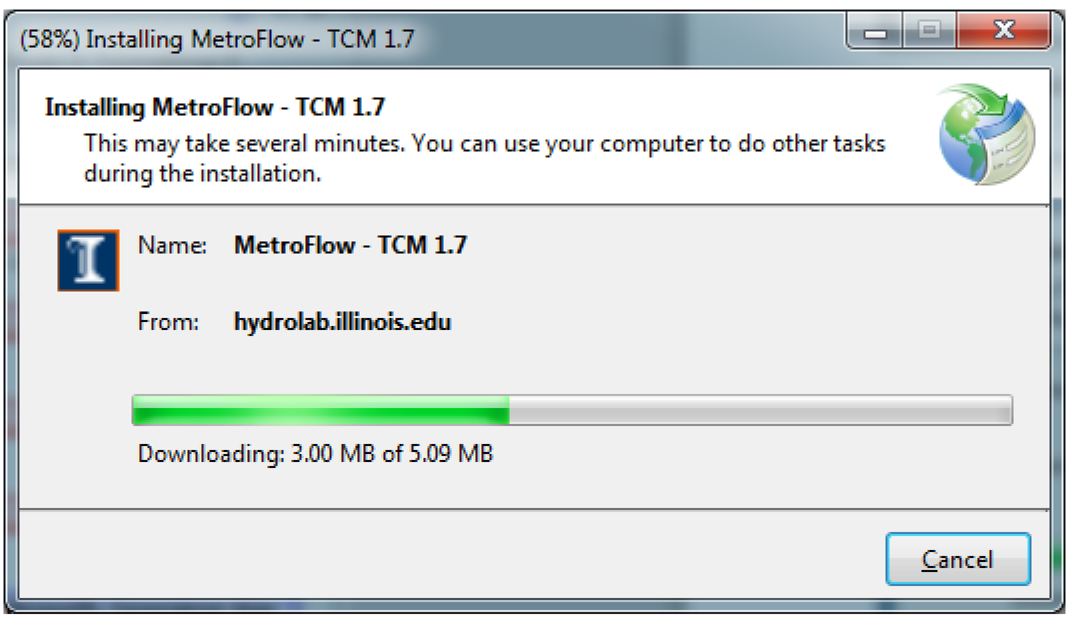
Download the MetroFlow-1.7.application file and install it (in Internet Explorer the installation will take place once the link is visited). The following dialog will be displayed.



Then a window will appear asking the user if they wish to install the program.



The warning can be safely ignored, and the **Install** button should be pressed. The installation will show a progress bar:



Once the download and installation completes, MetroFlow will automatically start and ask the user for a package to load. Packages are simply directories that end in with the *.mwp* extension, and contain a large number of files. A package must be obtained via DVD media due to the potentially large amount of data that is stored in it. The package folder should be copied to the local computer. It will not work to read the package off of a DVD media. The package can be stored on a network share but loading and accessing data in the package take significantly more time.

The MetroFlow website is located at http://hydrolab.illinois.edu/metroflow-1.7/. This website contains download links for the prerequisites, installer, user manual, and tutorial videos.

If errors or bugs are encountered while running MetroFlow, it is recommended that the error be logged using the error reporting tool that appears for fatal errors. Comments can be submitted at the following web form: http://goo.gl/ymRpC .

Interface Guide

3. METROFLOW INTERFACE

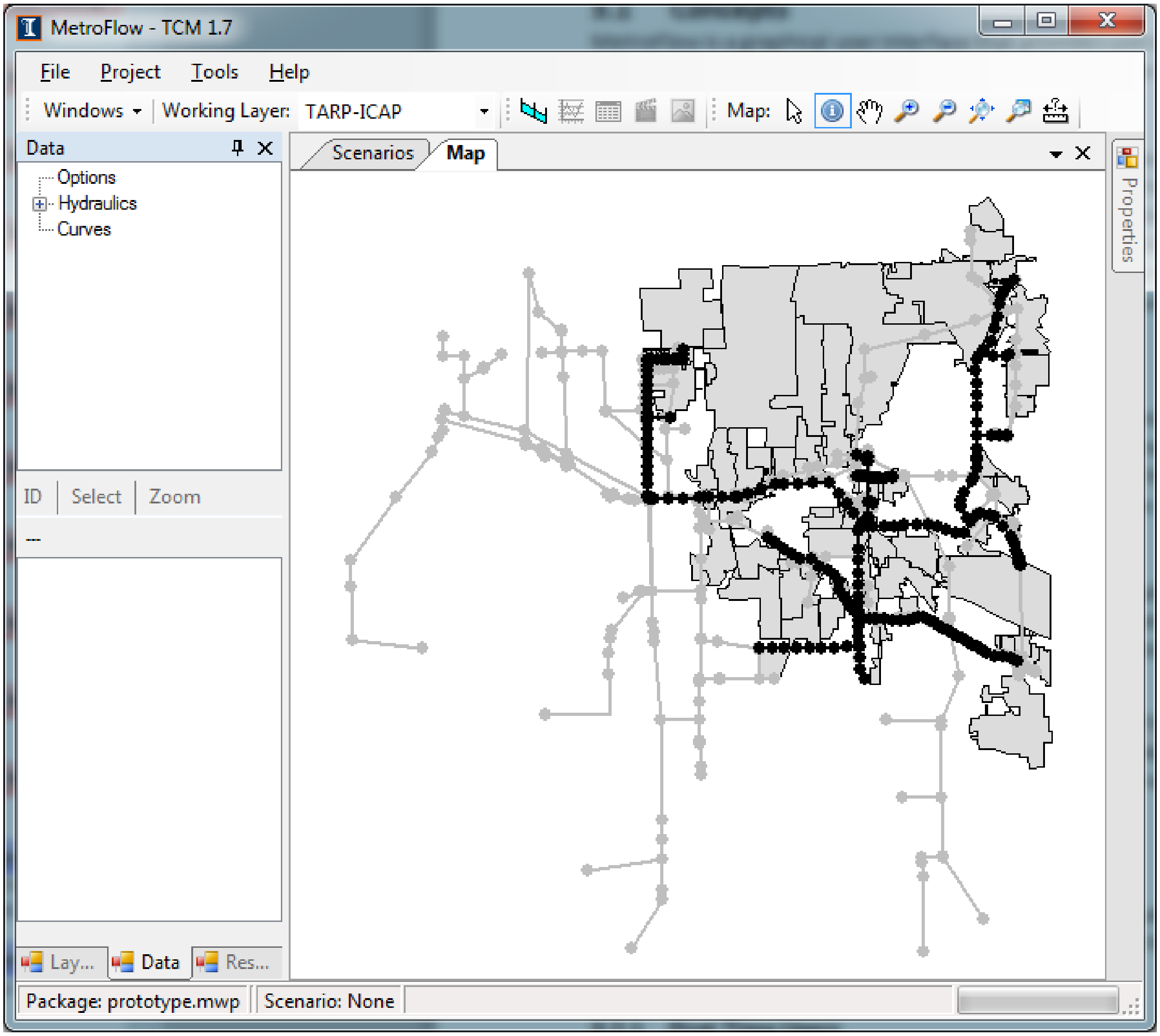


Figure 1: The MetroFlow main screen when the interface is first opened.

3.1 Concepts

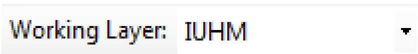
MetroFlow is a graphical user interface that provides users with the ability to run the various models in a combined sewer system and inspect results. Some of the key concepts are summarized as follows.

3.1.1 Packages

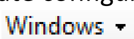
All model data, model executables and libraries, model geometries, timeseries input, and other information necessary to running the program are stored within a *package*. A package is a directory on the computer that contains many sub-directories and files. The package directory should not be manually modified. MetroFlow uses the information in the package to deliver a "one-stop" user interface for running models, analyzing simulations, and exporting results. Since packages are self-contained, multiple packages may coexist on the same computer with each package having different models and information.

3.1.2 Modeling Layers and the Working Layer

Each model in the package is stored as a separate layer and contains its own geographically-referenced map data and geometry and parameters necessary for modeling. Since multiple models may be included in a single package, the *working layer* is the current modeling layer that is selected. If a user is visualizing results from a simulation, only results for the current working layer are available for visualizing and reporting on with graphs and tables (which are discussed in following sections). The selected working layer will be drawn on top of all of the other visible layers. The working layer is selected by using the *Working Layer* drop-down menu on the toolbar.



3.1.3 Main Window

The primary feature of the MetroFlow main screen when first loaded is a geographically-referenced map of the spatial features of the various models included in the package. The user interface features windows and toolbars that are dockable or can be "torn off" to be placed in separate windows or alternate configurations. Windows can be brought to the foreground by using the items in the *Windows* menu 

option.

> If buttons within a window or toolbar seem to be missing, the program window may be too condensed causing the tools to appear hidden. To view the hidden buttons, try expanding the program window, or select the drop down arrow in the desired window.

Each window contains specific functionality. The following windows are available from the *Windows* menu in the *Standard Toolbar* and are described in following sections:

- Properties
- Layers
- Data
- Results Animation
- Timeseries Sets
- Controls
- CSO Analysis
- Scenarios
- Map
- TS Mappings

By default the Map, Scenarios, Properties (collapsed), Data, and Layers windows are opened.

When tables, graphs, profile plots, and reports are created, they appear as tabs listed along the top of the center portion of the main window. To toggle between windows, the user may click on the desired tab or click the arrow at the right of the tab bar to select one of the available tabs from the list.

The bottom of the main window contains a status bar with multiple features.

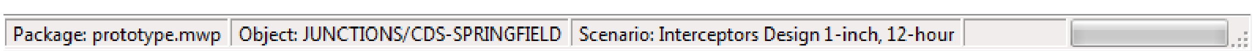
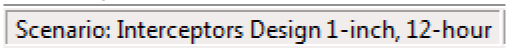


Figure 2: Main window status bar

Multi-function Progress Bar

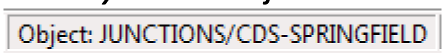
When plots, tables, and other data are being loaded or when scenarios and timeseries sets are being exported or imported, this indicates the progress of the task. When a scenario is executing and the controller window is minimized this indicates the progress of the scenario.

Currently-loaded Scenario



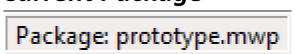
This field displays the name of the currently-loaded scenario. If no scenario is loaded, this field is empty.

Currently-selected Object



This field displays the name of the currently-selected object. Selected objects have their properties displayed in the *Properties* window and can be added to a results selection box for plotting or tables.

Current Package

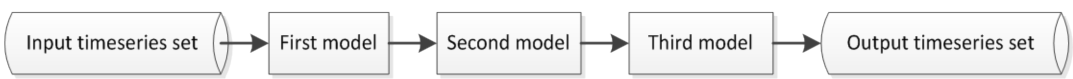


The current package field displays the name of the current package that is loaded. Clicking on the field will display the full path to the package.

3.1.4 Simulation Workflow

Models require input *timeseries sets* to run. A timeseries set is simply a collection of named timeseries. These timeseries either come from imported data (e.g. historical events, DSS data files, external hydrographs) or from design storms. Creating timeseries sets are described in section 3.10.

After timeseries sets are available, scenarios can be created. A scenario consists of a timeseries set input and one or more models to execute. If multiple models are present in a scenario, they are run in sequence, with the output set from each model providing the input set for successive models:



The intermediate output is also stored as a timeseries set, with the *Intermediate* flag set. These intermediate timeseries sets can be used just like normal timeseries sets. The final output timeseries set may or may not be present depending on the model; if for example the last model is a TARP tunnel model, then no output timeseries set is present.

Once a scenario has been created it can be run, which will execute each of the models in the scenario in sequence. After the run has completed, results can be viewed by using the Table, Graph, Profile Plot, Animator, and CSO Analysis Window tools. If multiple models were in a given scenario then only results for the current working layer are able to be queried.

3.2 Opening a MetroFlow Package

A "package" is the term for the suite of models including all of the input files for all models (e.g. IUHM, ICAP, ITM), as well as all of the user-produced scenarios, time series/results, etc. MetroFlow works as an interface that draws upon the package data. When MetroFlow is loaded, the *MetroFlow Package Selector* window appears (Figure 3) which allows the user to select a package to open.

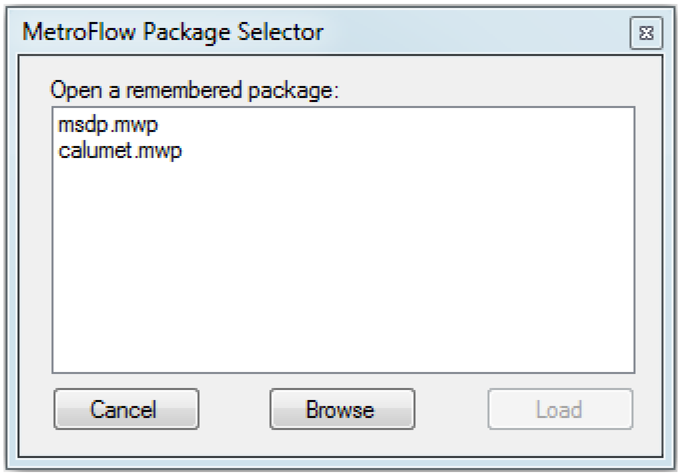


Figure 3: MetroFlow package selector window

If this is the first time the program has been run on the computer, no packages will be listed. Click the *Browse* button and navigate to where the package folder is located, select the folder ending with *.mwp*, and press *OK*. The main MetroFlow window will appear.

3.3 Main Menu Bar

The title bar includes menus *File*, *View*, *Project*, and *Tools*.

3.3.1 File

The *File* menu allows the user to use the *Exit* menu option which closes the MetroFlow program.

3.3.2 Tools

The tools menu contains four entries. The *Advanced Rainfall Processor* provides a user with the ability to create design storms and process historical storms and output to a file (whereas the MetroFlow timeseries tools provide a subset of this functionality). The *Package patcher* allows a user to manually update the package with an offline package update. The *Time convertor* converts between simulation and real time (see section 4.4.1). In addition, the *CSO display name tool* allows the user to change the display names of CSO locations (see section 3.17).

3.4 Toolbars

3.4.1 Map Navigation Toolbar

In addition to scrolling the mouse, which zooms in or out about the position of the cursor, these tools help the user navigate the map as follows:

The *Cursor* allows the user to select nodes, links, and system features.

When a feature is selected with this *Identify* tool, the *Properties window* opens to provide the user with the feature's name and key properties. This tool is also used for the *Profile Plot*, *Graph*, and *Table* tools.

The *Pan* enables the user to move manually around the map in any direction.

When this *Zoom-In* feature is selected, the user can click once on the map to perform an automatic zoom-in centering about the point of the cursor, or the user can click and hold to create a box around a particular area to which the map will zoom. Scrolling with the mouse wheel also zooms in and out.

When this *Zoom-Out* feature is selected, the user can click once on the map to perform an automatic zoom-out, or can click and hold to create a box around a particular area that will become the center of the zoomed-out area.

Zoom to full extent causes the map window to display the full extent of the system, which is determined by a sum of all the visible layers.

Zoom to layer causes the map to zoom to only the present working layer.

With this *Measure* tool, the user can determine approximate geographic measurements by selecting the two desired endpoints and reading the number following "Measured" along the bottom of the MetroFlow window. Note that sequential clicks add new segments to the measurement, which computes summarily. The measurement may be cleared by double-clicking followed by selection of the new first endpoint.

3.4.2 Results Toolbar

These tools help the user navigate the analysis of scenarios. With the exception of the profile plot tool, they are only enabled when a scenario is loaded that contains a run for the selected working layer.

3.4.3 Standard Toolbar

The standard toolbar, appearing the upper left corner of the MetroFlow window, contains two important items. The first is the *Working Layer* box and the second is the *Windows* menu which allows the user to bring to the forefront any of the primary windows (such as Map, Scenarios, Timeseries Sets, Timeseries Mappings, CSO Analysis, Results Animator, Controls, Data, Layers, and Properties).

3.5 Map Window

Using the Map

The map window displays the geographical features of the current working layer. The map window is interactive, and is controlled by the Map Navigation Toolbar (see section 3.4.1). Additionally, the user can navigate with his or her mouse by using the mouse scroll. The position of the cursor within the map window becomes the center of the next frame. Scrolling forward zooms in, and scrolling backwards zooms out.

The user can query map coordinates by clicking on the map. A box will appear with the latitude and longitude coordinates of the point at which the map was clicked.

3.6 Data Window

Using the Map

The Data window is opened by default on the left section of the MetroFlow main screen. It shows the current list of objects available in the current Working Layer. This window can be used to locate specific features and explore their properties, providing an alternative to selecting component attributes using the Map Navigation Toolbar. At the top of the window, navigate the Hydraulic directory to find the desired feature. Features will be listed in the window below the directories. The *Options* directory opens the *Properties window*, but instead of describing an individual feature, it displays the *Module options* correlating to the scenario as seen in the *Scenario builder*. Finally, the *Curve* directory is listed as well and for some scenarios may contain a list of curves such as stage-storage relationships.

For example, if ICAP is the working layer, the user can navigate to any node or link in the ICAP model. Double-clicking on the object name will highlight it on the map temporarily, as will selecting *ID* among the three buttons in the Data window. The button *Select* marks an object as the currently selected object and the *Properties* window displays the properties of the object.

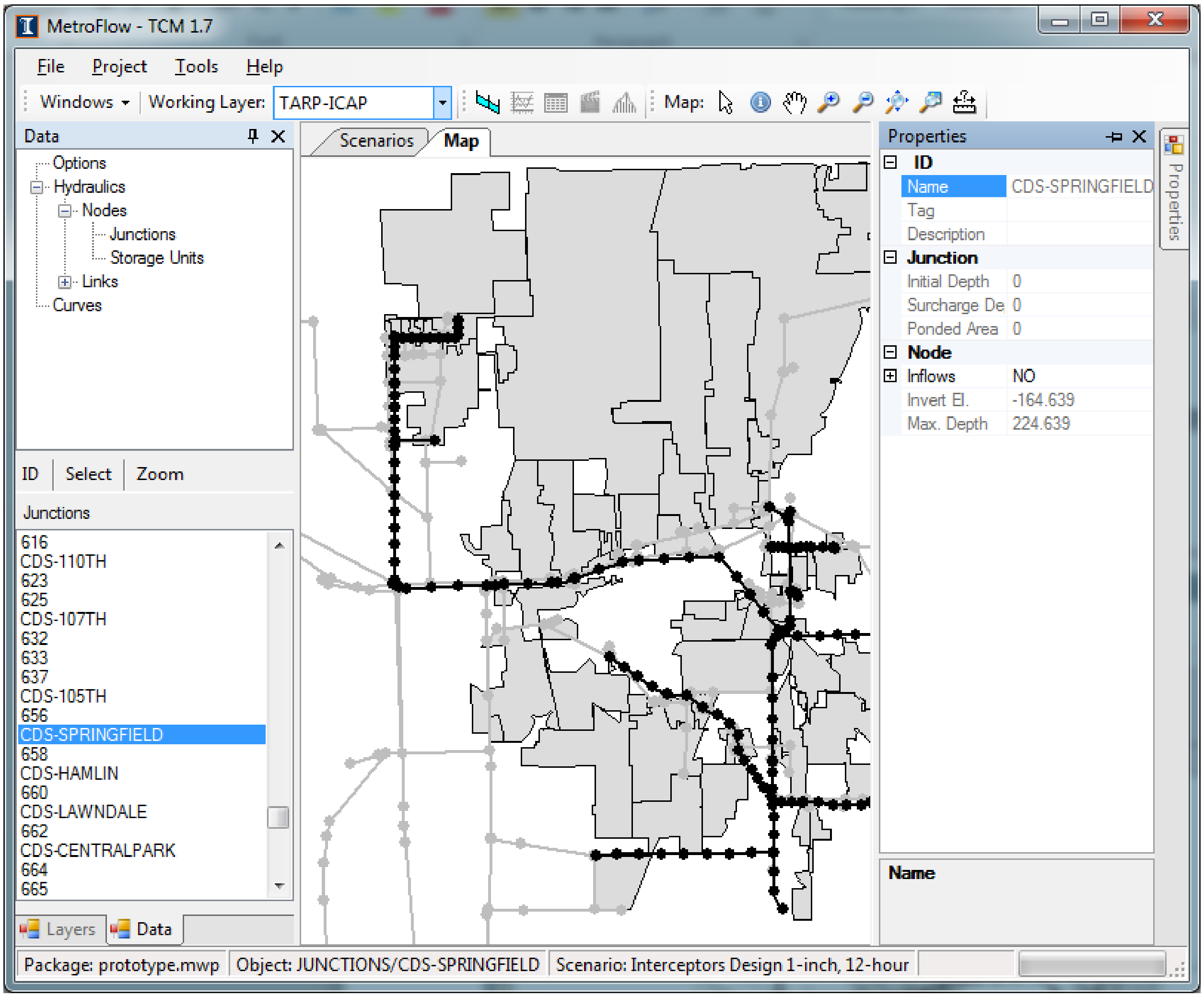


Figure 4: In the Data window, the tools Zoom and Select are used to zoom-in to the selected feature, junction CDS-59, and to display its properties

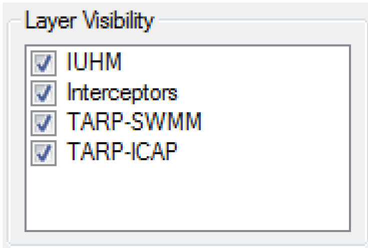
Using the Map

3.7 Layers Window

The Layers Window is also opened by default on the left side. Within the window there are three section headers: *Layer Visibility*, *Labels For Active Layer*, and *Symbology For Active Layer*.

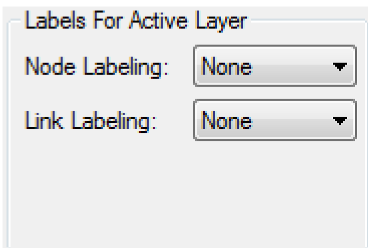
3.7.1 Layer Visibility

This section allows the user to turn on and off the visibility of the various “Working Layers” in the map window. Note that only the *Working Layer* is navigable via the map—the other layers simply display underneath the active layer.



3.7.2 Labels for Active Layer

The features in the Labeling and Symbology sections are only available when a run scenario is loaded and matches the selected working layer. In essence these tools are for use in viewing the results of a simulation. In the Labeling section, the user can select labels for the basins (applicable to the IUHM model only), or the nodes and links. The map will automatically populate with the selected label feature.



The available basin labels and their definitions are:

- None: This is the default option, where no labels show;
- Name: All basin features will be labeled by their name field.

The available node labels and their definitions are (items prefixed with ‡ are only available once a scenario was successfully ran and loaded):

- None: This is the default option, where no labels show;
- Name: All node features including junctions, outfalls, and storage units will be labeled by their name field;
- ‡ Depth: The simulated flow depth at the node;
- ‡ Head: This label displays the hydraulic head of a node, which is a combination of the elevation and water pressure at that point;
- ‡ Volume: Nodes modeled as storage units (e.g. the reservoir) will display the volume of water in the node in cubic feet;
- ‡ Lateral Flow: Lateral flow is the flow that enters a node from an external inflow (such as a hydrograph);
- ‡ Total Flow: Total flow is the sum of the flow entering a node from upstream as well as an external inflow;
- ‡ Flooding: Flooding is defined as excess overflow when the node is at full depth.

The available link labels and their definitions are (items prefixed with ‡ are only available once a scenario was successfully ran and loaded):

- None: This is the default option, where no labels show;
- Name: All link features will be labeled by their name field;
- ‡ Flow: The flow rate in the conduit;

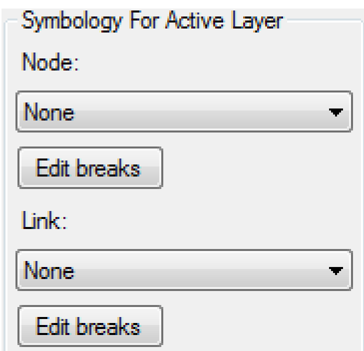
- ‡ Depth: The average water depth in the conduit for a given time;
- ‡ Velocity: The flow velocity in the conduit;
- ‡ Froude No.: The Froude number F_r indicates the effect of gravity, meaning subcritical or supercritical flow. The Froude number is determined by variables V=average velocity, g=gravity and D_h = the hydraulic depth, from the equation:

$$F_r = \frac{V}{\sqrt{gD_h}} \quad where\ F_r \begin{cases} \text{<1 indicates subcritical flow} \\ \text{=1 indicates critical flow} \\ \text{>1 indicates supercritical flow} \end{cases}$$

- ‡ Capacity: The Capacity is the ratio of depth to full depth.

3.7.3 Symbology for Active Layer

This section works in conjunction with the Animator tool and allows the user to see variables at specific objects on the map change as the scenario plays back. The symbology options are only visible when a successfully-ran scenario has been loaded. The user selects a criterion from the Nodes and/or Link fields that they would like to see animated. The options for Nodes are Depth, Head, Volume, Lateral Flow, Total Flow, Flooding, and None (nodes will not be animated). The options for Links are Flow, Depth, Velocity, Froude Number, Capacity, and None (links will not be animated). These variables are discussed in the *Labels for Active Layer* section.



Once a variable has been selected, the *Edit Breaks* buttons allow the user to specify the color scheme of the node or link. The default color scheme is Standard hot-to-cold, where red indicates the highest values (e.g. overflow) while blue indicates the lowest values (e.g. no flow). More information on the *Breaks Editor* is included in section 3.15.

3.8 Properties Window

Changing Object Properties

This window is located on the right side of the MetroFlow main screen and displays information about the selected node, link, basin, or system information. It is hidden on the right of the screen by default, but it will expand when the user selects an object on the map with the identify tool or moves the cursor over its tab name. The only fields that are editable in the property window are the McCook/Thornton reservoir curve selection, and this only option is only available once a TARP-SWMM, TARP-ICAP, or TARP-ITM scenario has been loaded (for this to work, the scenario need not to have been run).

> Once a property has been changed, the save button in the Scenarios window must be pressed to register the changes with the scenario.

ILLINOIS

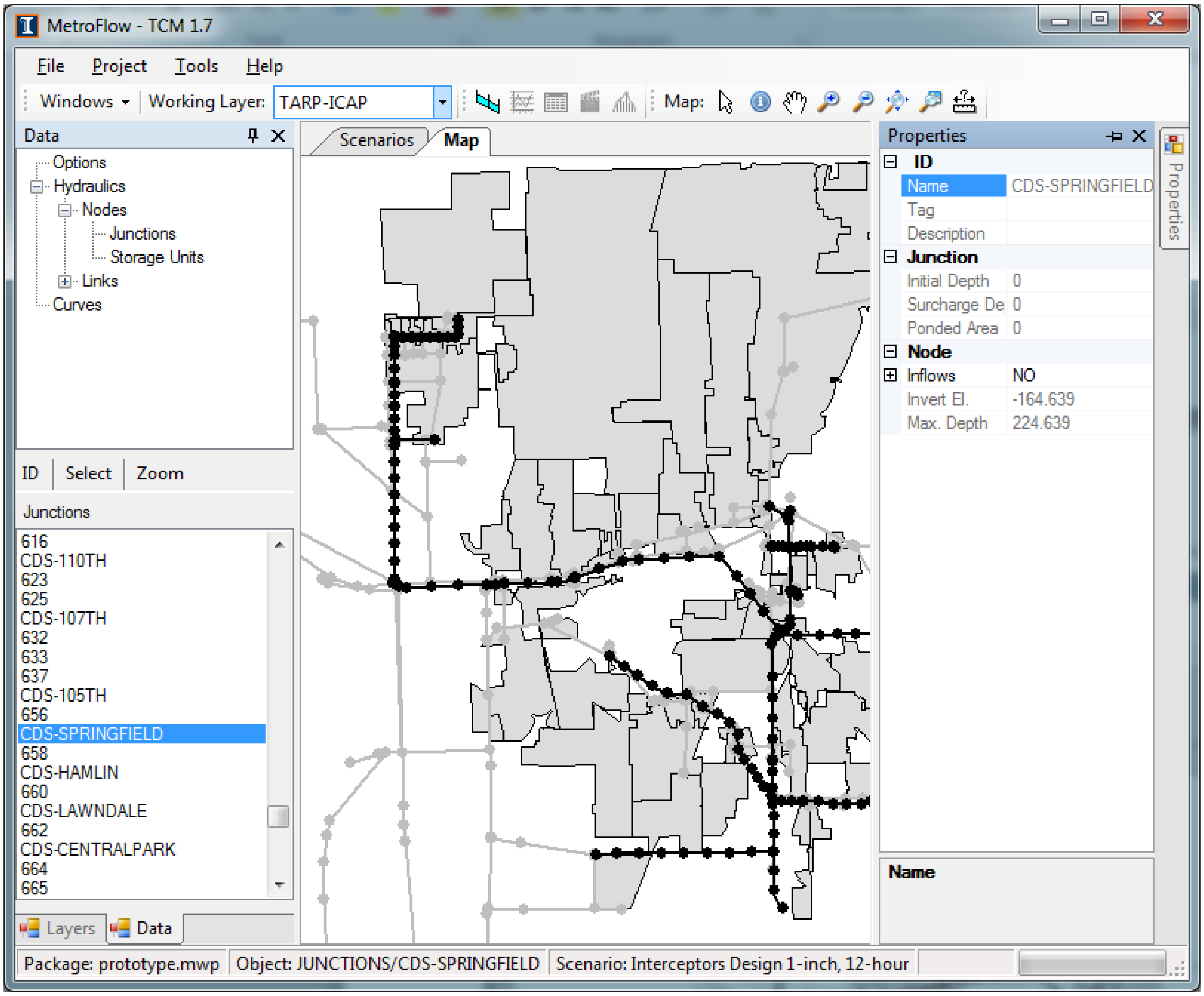
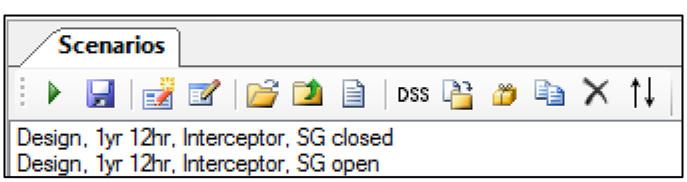


Figure 5: The Properties window displays at the right and lists feature-specific information.

3.9 Scenarios Window

This window is where the user produces and executes scenarios using the various models and model combinations. The scenario manager by default loads in the primary window area. Within this window, the user can create or import a new scenario, or edit or execute an existing scenario. If multiple scenarios are selected, they will be batch executed in sequence.



Here are the available tools:

 Run the currently selected scenario(s).

 Save the current scenario.

 Create a New scenario (see explanation following tools details).

Edit the details of the current scenario.

Load the current scenario (only works for scenarios that have been run).

Unload the current scenario (if one is currently loaded).

View the report from the scenario run. If the run was unsuccessful, this report will show a log of the errors with the run. If the run was successful, the report will detail the output of the run.

DSS Saves the loaded scenario to a HEC-DSS file (see *Exporting to DSS* below)

Import an external scenario from a “compressed scenario (*.sczip)” file

Export a scenario as a “compressed scenario (*.sczip)” file.

Duplicate the selected scenario. The duplicate will appear underneath the original scenario in the Scenarios window labeled with (copy).

Before duplicating, be sure to unload the selected scenario by clicking the Unload button.

Delete the selected scenario.

Creating a Scenario

3.9.1 Creating a Scenario

To begin, click the *New scenario* button to launch the *Scenario Builder*. The following window appears.

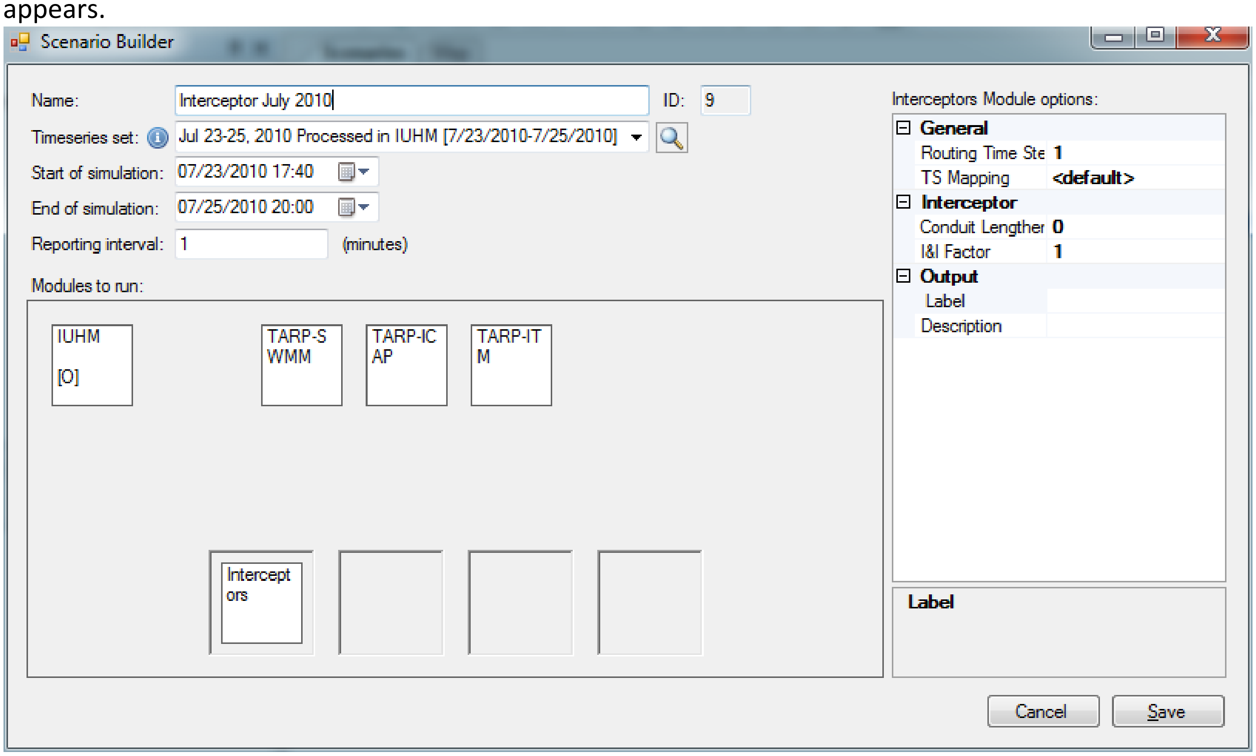


Figure 6: This Scenario builder window shows an example scenario of the Interceptor model for historic storm July 23-25, 2010

The Scenario Builder allows the user to drag models into an execution sequence. A valid scenario requires one, two, or three models as permitted by the following combinations (left is for Calumet, right is for Mainstream/Des Plaines):

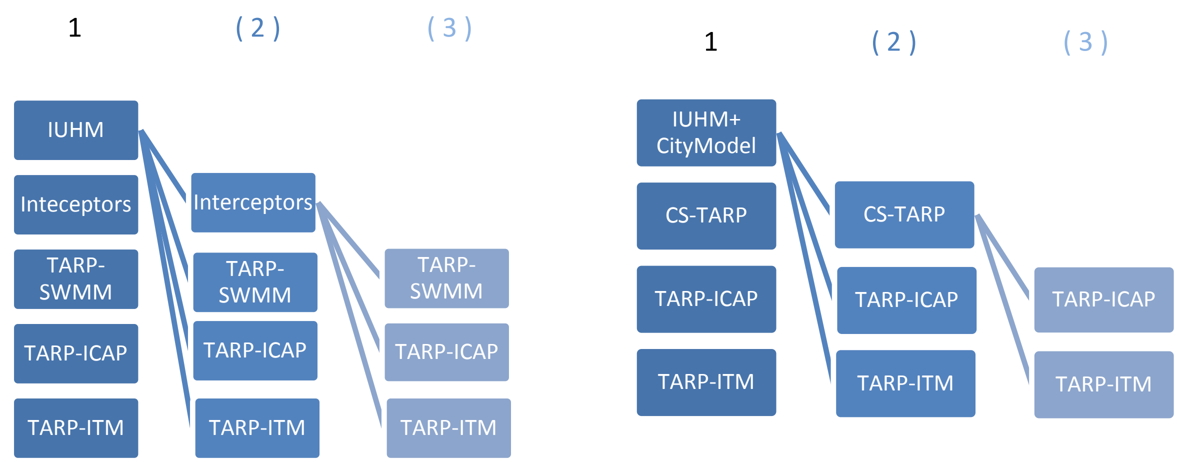


Figure 7: A logic figure of the possible execution sequence of models

At the top of the Scenario Builder window, the user should indicate the name of the scenario in the Name field.

Only letters, numbers, spaces, dashes, and underscores are admissiblenaming characters.

The first model is selected by dragging the desired model's square icon in the top left portion of the "Modules to run" box into the first square slot (the start of the execution sequence) located at the bottom right portion of this box. After this first selection, the Timeseries set field becomes active, and the user can indicate which dataset he or she would like to use. If the first model is the hydrologic model IUHM, then rainfall sets will be shown. Otherwise, hydrograph sets will be available.

The selection of a Timeseries should automatically change the Start and End of simulation fields to match the dataset. The Reporting interval has a default value of 1 minute (which can be changed) that is shared across all of the models within the scenario.

Additional models can be dragged down to form an execution sequence. Figure 7 provides an illustration of the possible model paths. When multiple models are given, the intermediate timeseries are stored as a timeseries set in the timeseries set database and window.

When a model is dragged down into the execution sequence, the options for that model will be displayed on the right side of the window under *[Model] Module options*. The user can edit the following module options which are described further in section 8:

MODEL	TIMESERIES SET AVAILABLE	MODULE OPTIONS
IUHM (Calumet)	Rainfall	Label for Output, Description for Output
IUHM+CityModel (MS/DP)	Rainfall	Routing Time Step, Parallel Execution, Label for Output, Description for Output

Interceptor (Calumet)	Set of Hydrographs (e.g. from IUHM)	Routing Time Step, Conduit Lengthening Step, I&I Factor, Label for Output, Description for Output
CS-TARP (MS/DP)	Set of Hydrographs (e.g. from IUHM)	Routing Time Step, Label for Output, Description for Output, Use Pumping, Use Reservoir
TARP-SWMM (Calumet)	Set of Hydrographs (e.g. from Interceptor runs)	Routing Time Step, Global Head
TARP-ICAP (Calumet, MS/DP)	Set of Hydrographs (e.g. from Interceptor runs)	Routing Time Step, ETP Threshold, Inflow Threshold, Pumping Rate
TARP-ITM (Calumet, MS/DP)	Set of Hydrographs (e.g. from Interceptor runs)	Maximum Time Step, Initial Condition File, Initial Condition Type, Max. # of Cells per Pipe, Min. # of Grid Points, # of Cells Per Pipe to Plot, Pressure Wave Celerity, Units

Figure 8: This table identifies the Timeseries set and module options pertaining to each of the models

After dragging down each model, it is optional to name the *Output Label* field in each models options, if the model supports output (e.g. the TARP tunnel models do not provide output). This will override the default naming convention which may be more desirable. If no *Label* is provided the default naming convention is to name the output timeseries set *SCENARIO_NAME (FIRST_MODEL_NAME)* (e.g. *Design 5-year, 12-hour (IUHM)*).

The *Reporting interval* value is simply the value at which results will be reported to the program. The *Reporting interval* value should typically be 1 minute for historical events such as individual storms but should be larger for longer term simulations. For water year events the interval should be 60 minutes due to the large volume of data that is stored.

Reporting intervals should be set before changing the routing time step module option. If the routing time step (given in seconds) is larger than the reporting interval (given in minutes), then the routing time step will automatically be clamped to the reporting interval value.

Timeseries Mapping

Using Timeseries Mappings

Timeseries sets contain multiple individual timeseries that are used as inputs to nodes (inflow locations) in the models that are inside of MetroFlow. Each inflow location expects a specifically-named timeseries, so how timeseries are named—or alternatively mapped—are very important. This is discussed in greater detail in section 3.11. When a scenario is created, the user has the option of selecting the timeseries mapping (TS Mapping in the module options for the scenario builder, see Figure 9). When a timeseries set is selected, the selected mapping (or the default mapping, if none is selected) is compared to the selected timeseries set to see if it contains every timeseries that the model expects. If it is missing one or more timeseries, a warning message is displayed in the Scenario Builder. The user can click the warning icon for more information on the timeseries inconsistencies. If the user chooses to execute the scenario with the mismatched timeseries, it will be created and will run, excluding the missing timeseries. For custom mappings, the user can go to the Timeseries Mapping window and create or edit a new mapping. After the mapping has appropriately added or modified, then the user may go back to the Scenario Explorer, edit the scenario, and select the mapping that was created or edited.

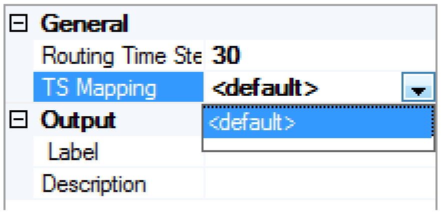


Figure 9: Module options showing timeseries mapping menu

Saving a Scenario

To register the scenario, select *Save*. The scenario that was just created should now appear on the list of scenarios in the *Scenario window* with its given scenario name. For further details of scenario analysis, see section 3.9.2 or the tutorial, section 4.1.2.

3.9.2 Executing a Scenario

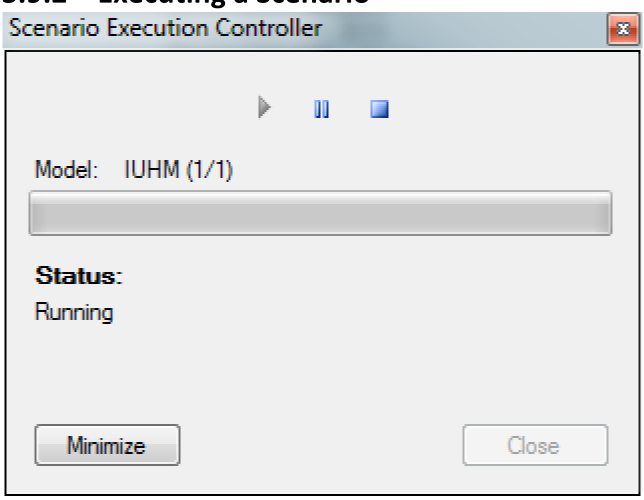


Figure 10: Scenario execution controller window

When the run scenario button is pressed the selected scenario(s) are executed. The *Scenario Execution Controller* window is displayed. During scenario execution, most of the MetroFlow user interface is disabled although most windows can be opened but not interacted with. The map can be navigated during scenario execution.

The controller window will appear above all other windows opened on the user's computer. However, it can be minimized by pressing the *Minimize* button. This will cause that window to be hidden and a progress bar will appear on the main window on the lower right. Clicking on the progress bar in the main window will cause the controller window to reappear.

The scenario can be temporarily paused with the button. This may be useful in older computers where there is only a single processor available; pausing will allow other programs to utilize the processor. To resume scenario execution press the button again. The scenario execution may also be stopped with the button which terminates the simulation.

A scenario may be stopped before completion, and results are typically available for the simulation up to the point at which the simulation was stopped.

During execution, the *Model* line indicates the currently-running model and provides an indication of where in the model sequence the current model run is. This is primarily relevant for multi-model scenarios; for example, for a scenario with IUHM and Interceptors the *Model* line would read *(1/2)* for IUHM and *(2/2)* for Interceptors. For single-model scenarios the indicator will read *(1/1)*. Below the *Model* line is the progress bar and below that is the *Status* of the scenario. For some models the elapsed time is indicated in the format DD:HH:MM (DD: days; HH: hours; MM: minutes).

If multiple scenarios are selected before the Run button is pressed in the Scenario Explorer window, they will be executed one-by-one. The window will appear as in Figure 11.

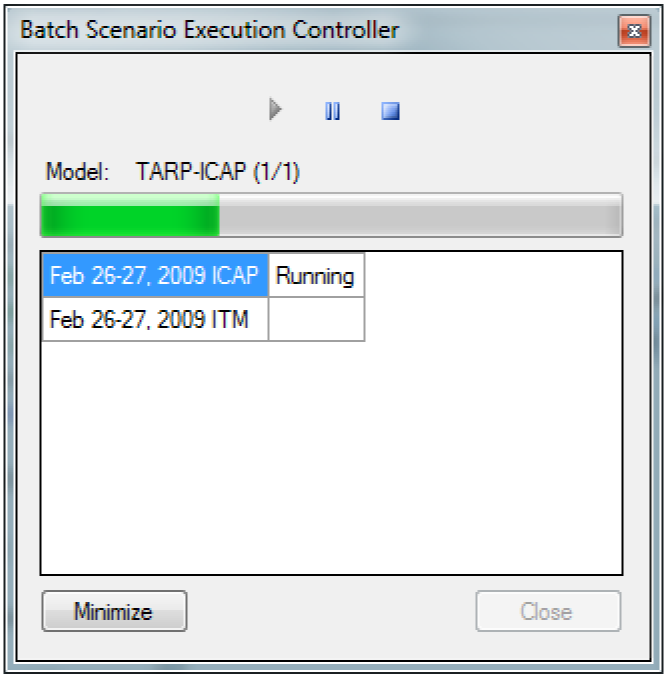


Figure 11: Multiple scenario execution window

3.9.3 Exporting to DSS

Exporting Scenarios to DSS

MetroFlow has the ability to store model outputs in DSS files (for information on DSS see section 7). This is done with the *Save to DSS* tool. After selecting and saving the file name, every parameter in the file will be exported. The parameter type will be placed into Part F, such as depth and flow. Also, when naming the file, be sure to include the name of the model from which it was exported. Otherwise, there is no way of knowing the source of the data. Following is a list of the way that data is exported:

Part A: “MWRD”

Part B: LINK_{ID} or NODE_{ID} where {ID} is the link/node name

Part C: Units [FEET (depth, head), FPS (velocity), CFS (flow), FEET3 (volume), NONE (Froude No.)]

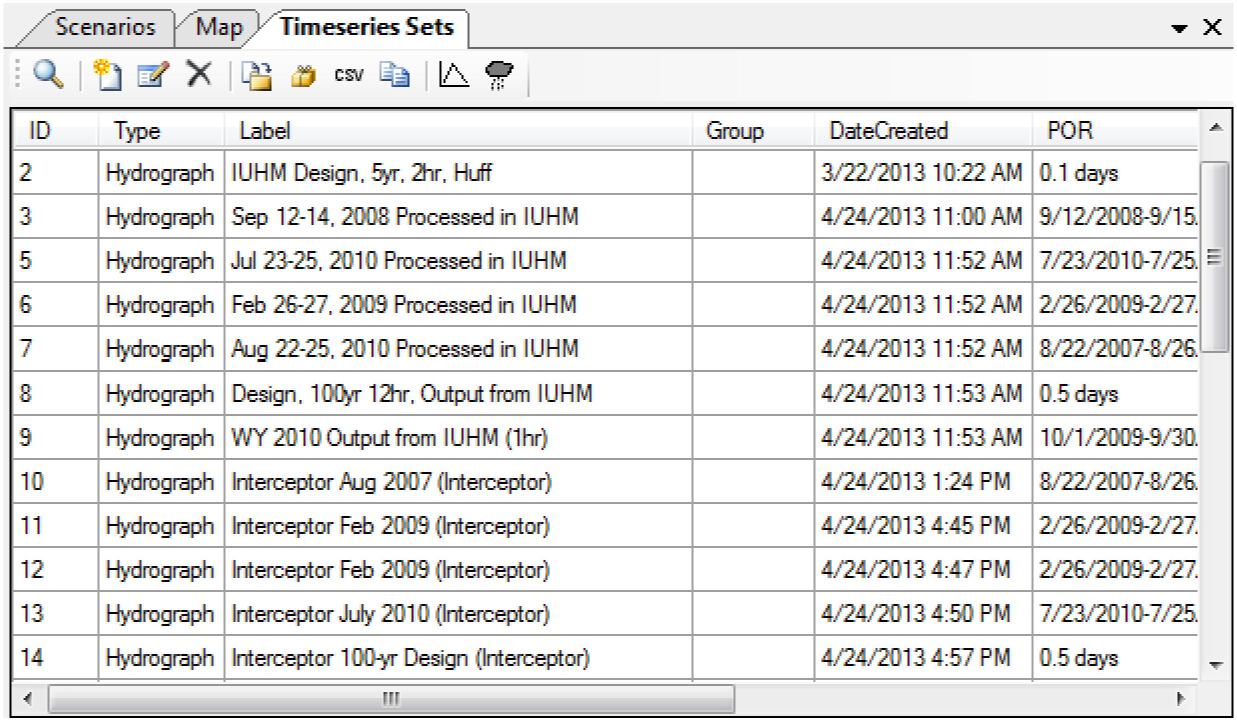
Part D: Date

Part E: Data interval (e.g. 1MIN, 1HOUR)

Part F: Data type (for nodes: DEPTH, HEAD, LATERAL-FLOW, TOTAL-FLOW, VOLUME, FLOODING; for links: FLOW, CAPACITY, DEPTH, VELOCITY, FROUD-NO.)

Note that exported data files sizes may range up to hundreds of megabytes, depending on how long a simulation was done for and what the time step was.

3.10 Timeseries Sets Window



ID	Type	Label	Group	DateCreated	POR
2	Hydrograph	IUHM Design, 5yr, 2hr, Huff		3/22/2013 10:22 AM	0.1 days
3	Hydrograph	Sep 12-14, 2008 Processed in IUHM		4/24/2013 11:00 AM	9/12/2008-9/15.
5	Hydrograph	Jul 23-25, 2010 Processed in IUHM		4/24/2013 11:52 AM	7/23/2010-7/25.
6	Hydrograph	Feb 26-27, 2009 Processed in IUHM		4/24/2013 11:52 AM	2/26/2009-2/27.
7	Hydrograph	Aug 22-25, 2010 Processed in IUHM		4/24/2013 11:52 AM	8/22/2007-8/26.
8	Hydrograph	Design, 100yr 12hr, Output from IUHM		4/24/2013 11:53 AM	0.5 days
9	Hydrograph	WY 2010 Output from IUHM (1hr)		4/24/2013 11:53 AM	10/1/2009-9/30.
10	Hydrograph	Interceptor Aug 2007 (Interceptor)		4/24/2013 1:24 PM	8/22/2007-8/26.
11	Hydrograph	Interceptor Feb 2009 (Interceptor)		4/24/2013 4:45 PM	2/26/2009-2/27.
12	Hydrograph	Interceptor Feb 2009 (Interceptor)		4/24/2013 4:47 PM	2/26/2009-2/27.
13	Hydrograph	Interceptor July 2010 (Interceptor)		4/24/2013 4:50 PM	7/23/2010-7/25.
14	Hydrograph	Interceptor 100-yr Design (Interceptor)		4/24/2013 4:57 PM	0.5 days

Figure 12: The Timeseries Sets window

In versions of MetroFlow more recent than version 1.0, the Timeseries Sets window (and timeseries database) has changed.

The *Timeseries Sets* window displays timeseries sets which are the basic unit of input and output between models. At its core is a data grid, with sortable columns, along with a toolbar that allows specific actions to be taken on selected timeseries sets, as well as the importing and export of timeseries sets. Existing timeseries sets can be edited to add additional timeseries or remove existing timeseries within the set. Also, metadata such as Label, Group, and Read-only status can be edited.

The data grid is sortable, both in ascending and descending order. By clicking once on a given column header (e.g. Label) the grid is sorted by the Label in ascending order. Clicking again on the Label column header sorts the grid by Label in descending order. Sort order does not affect how the models utilize timeseries as they are linked by the set ID which is not changeable. The fields that are available are as follows:

- ID: fixed identification number;
- Type: Hydrograph or Rainfall;
- Label: the name of the timeseries set;
- Group: field for grouping sets;
- Date Created: the date that the timeseries set was created;

- POR: the period of record for the timeseries set;
- Is Relative: checked if the timeseries set has relative or absolute times;
- Source: the model which originated the set (e.g. IUHM, Interceptor).

The Group field is of particular interest, as it can be used to group sets. For example, if the user edits the Group field for five different sets and sets the value to be “WY 2012”, then the Group column header can be clicked in the data grid and the grid will be sorted by group and all sets with “WY 2012” for the group will be grouped together.

Timeseries Sets can be duplicated. A duplicated set does not duplicate all of the timeseries. It only duplicates timeseries that are edited within the set. For example, if a set has 100 timeseries within it, and it is duplicated, the duplicate will also contain 100 timeseries that are pointers to the timeseries in in the original set. If a timeseries in the duplicated set is altered, then a copy of the original timeseries will be made and the copy altered. Also, additional timeseries that are added or removed from or to the duplicated set do not affect the original set.

The *Timeseries Set* window has a toolbar with the following tools:

View a timeseries set.

New timeseries set.

Delete a timeseries set.

View and edit properties of the set.

Import an external time series set from a compressed time series (*.tszip) format.

Export a time series set as a compressed time series (*.tszip).

Export a time series set as a CSV file (*.csv).

Duplicate a timeseries set.

Create a design storm. See below for details on how to create a storm.

Import external rainfall data. See the *Theory of Importing External* Rainfall Data section below for details on the preparatory work for and use of this tool.

3.10.1 Time References

Time references in sets can be absolute or relative. Absolute time references utilize a date/time stamp and correspond to historical data such as individual historical storms or entire water years. For example, timeseries in a given set may start at 1/10/2003 00:00. When the scenario is created, the start date/time for the scenario is set to be the start of the timeseries. It may also start later or earlier (if earlier, zeros are used for timeseries that don’t have data points at the given time) for absolute timeseries sets. Not all timeseries need to have the same date/time period, although having each timeseries have the same starting and ending date would be the conventional thing to do.

Relative time references use a time stamp that is always relative to the simulation start date/time. For example, design storms start at 00:00. When a scenario is created that uses a relative time-referenced timeseries set, the timeseries data always begins at the start date of the scenario.

3.10.2 Viewing a Timeseries Set

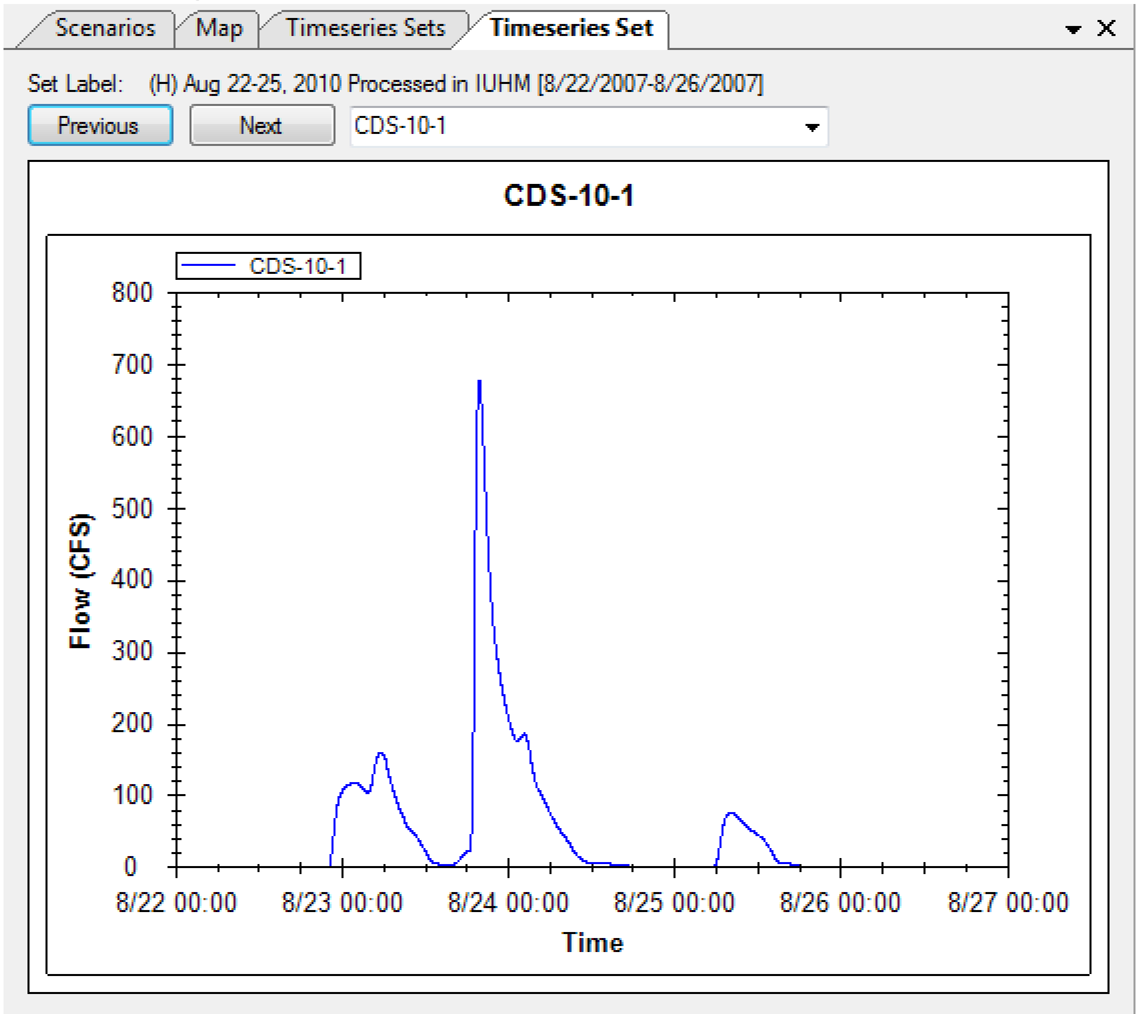


Figure 13: Viewing a timeseries set

Selecting this tool will open the plot(s) associated with the selected timeseries set.

A rainfall set will contain a set of hyetographs. A historic storm that is weighted to the hydrologic model (i.e. IUHM) will have a hyetograph for each subcatchment, whereas a design rain storm will have a single hyetograph that applies to all of the subcatchments.

A hydrograph set will contain a set of hydrographs. These sets are the output of the IUHM or Interceptor models and are used as inputs to successive models.

Editing Timeseries Sets

3.10.3 Viewing and Editing Properties

Clicking on the *View and Edit Properties* button opens the *View and Edit Timeseries Set* window. If multiple timeseries sets are selected, then the *View and Edit Timeseries Set* window is in a special mode that allows the user to edit the *Group* field for all of the selected timeseries sets.

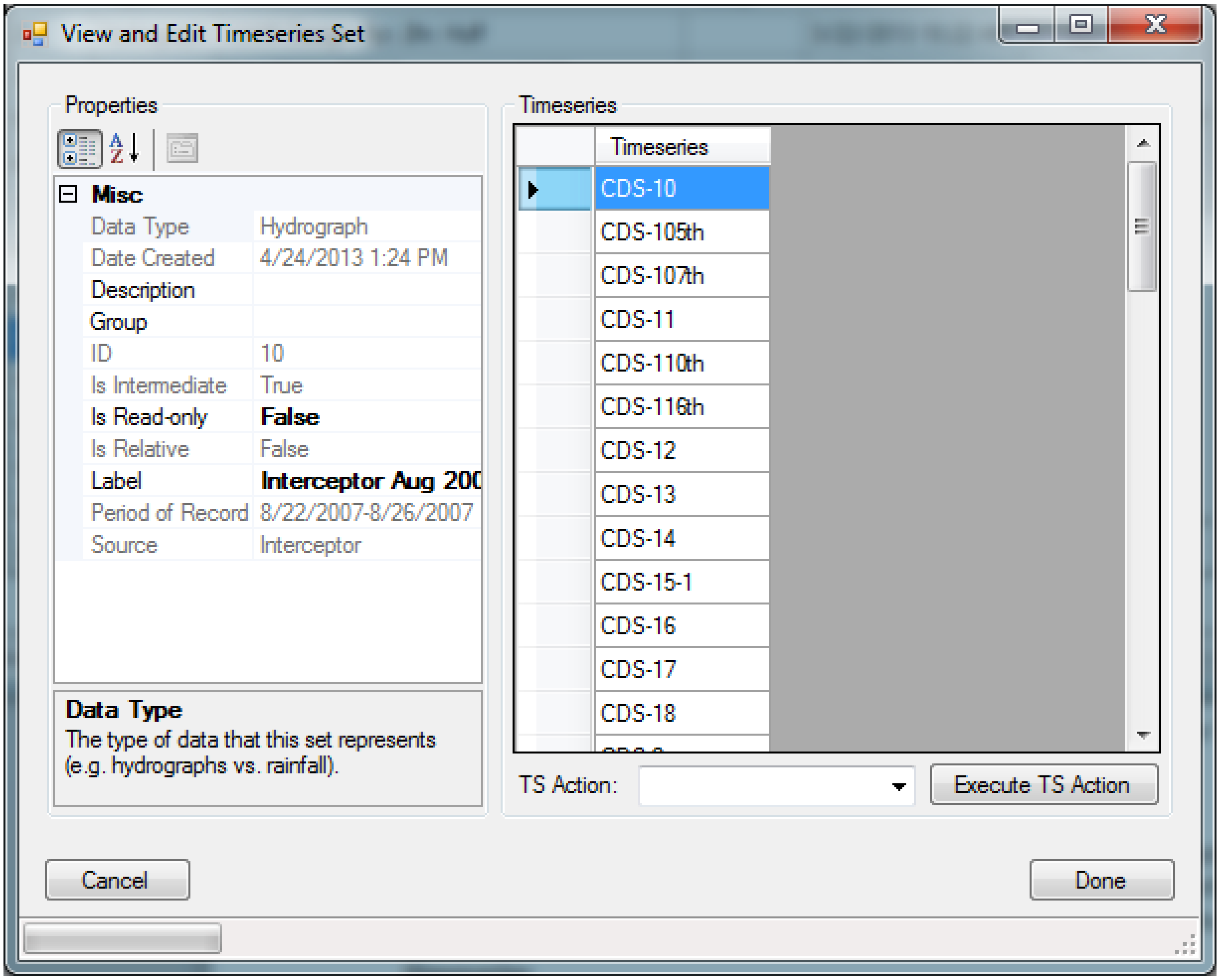


Figure 14: The View and Edit Timeseries Set window for timeseries set properties

In this window, the user may view each timeseries in the set, edit properties, and perform actions on individual timeseries (*TS Actions*) within the set.

Properties
The Properties section displays various characteristics of the selected Timeseries set. Editable features include Description, Group, Is Read-only, and Label. To view more information about a certain property, click on the respective cell and a description will appear below the Properties section.

Timeseries
The Timeseries data grid lists the timeseries that are contained within the set. Double-clicking on one of the timeseries names allows the user to view the contents of the data file.

TS Action (Timeseries Actions)
The TS Action drop-down menu allows the user to do the following actions: *Import single TS, Import multiple text TS, Import from DSS, Import from CSV, Delete TS, Rename TS,* and *Export Selected to DSS*. Select action and click *Execute TS Action* to perform selected action.

The *Import from DSS* and *Export Selected to DSS* actions allow the user to import and export DSS data into and from a timeseries set. These features are documented in section 3.10.8.

The *Import single TS* and *Import multiple text TS* options allow the user to import either individual or multiple text timeseries files, and is described in section 3.10.7.

The *Import from CSV* option allows the user to import a CSV file and select the columns to be imported; it is described in section 3.10.8.

Delete TS and *Rename TS* allow the user to delete and rename individual timeseries, respectively.

3.10.4 Create a Design Storm

Creating a Design Storm

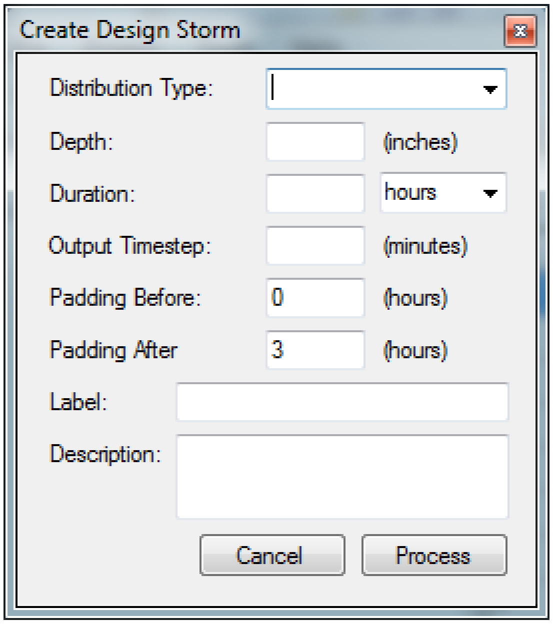


Figure 15: Design storm creation utility

The Create Design Storm tool enables users to create a generic storm. The design storm will have a uniform spatial distribution of rain. The user determines the depth (in inches) and duration (in minutes or hours) by entering the respective information into the Depth and Duration fields.

The intensity of the design storm is determined by the specified distribution in the Distribution Type field. There are four available distributions:

- Huff;
- NRCS;
- Uniform: The uniform distribution will apply a constant rainfall evenly across all subcatchments in the model;
- Yen-Chow.

Of the remaining fields, the *Output Timestep* field allows users to regulate the accuracy and length of the timeseries set, which will in turn affect the running speed of any dependent scenarios. One (1) minute is a standard output timestep.

The *Padding Before* and *Padding After* fields add buffer hours of zero-precipitation before/ after the storm. In models such as IUHM, excess rainfall is held until the next time step. Depending on the severity of the storm, this condition can cause the model to require extra time after the rainfall has ended to account for all of the backed-up precipitation. Thus, these “pad” values are in place to ensure that the model incorporates all of the rainfall. During the padded time, precipitation values of zero (0) are added to the precipitation sequence for the specified duration. The default duration of padding before the storm is zero (0), after the storm is three (3) hours.

Finally, the user should designate a storm name in the Label field, and enter a Description (if desired). The Process button completes the design process and registers the storm as a set in the *Timeseries Sets* window; the set uses a relative time reference.

Creating a design storm is covered in the tutorial, section 4.1.1.

3.10.5 Theory of Importing External Rainfall Data

This section will describe how to create rainfall and weighting files for use within MetroFlow. Creation of the weights file will usually be a one-time exercise unless the weights are to be modified. The exercise below uses Microsoft Excel to create CSV files (comma-separated values).

To create a rainfall file, hourly precipitation data must be available. Rainfall data can be obtained from multiple sources but typically is obtained from the Illinois State Water Survey (http://www.isws.illinois.edu/data/ccprecipnet/dataStation.asp?p=CCPN). Hourly data for long-term periods must be requested via email; the aforementioned website details that process.

MetroFlow is by default configured to process ISWS data from the Cook County Precipitation Network. The format is as follows. In the first column, list date and time by hour in the format dd/mm/yyyy hh:mm (where the time is in 24-hour format) through the duration of the available data. The hourly precipitation values in inches should then be entered in adjacent columns, with only one gage in each column under the heading of the rain gage identifier in the first row, as shown in Figure 16. Note that the program assumes that 0:00 is the start of the day, not the end of the previous day. Date/times in the format 1/1/2000 24:00 are assumed to be the start of the following day (e.g. 1/2/2000 00:00).

It is important that there be at least one hour of zero rainfall at the end of the file in order to allow ponded water in the IUHM model to be released back into the conduits. It is recommended that two additional rows be provided with zero values, the first at the next logical time increment in the original data, and the second at after the last time value at 5 hours after for hourly data and 48 hours after for daily data. Values in between will be interpolated as necessary by the various models.

Time/Date	G1	G2	G3	G4	G5
10/1/2008 1:00	0	0	0	0	0
10/1/2008 2:00	0	0	0	0	0
10/1/2008 3:00	0	0	0	0	0
10/1/2008 4:00	0	0	0	0	0
10/1/2008 5:00	0	0	0	0	0
10/1/2008 6:00	0	0	0	0	0
10/1/2008 7:00	0	0	0	0	0
10/1/2008 8:00	0	0	0	0	0
10/1/2008 9:00	0	0	0	0	0
10/1/2008 10:00	0	0	0	0	0
10/1/2008 11:00	0	0	0	0	0
10/1/2008 12:00	0	0	0	0	0
10/1/2008 13:00	0	0	0	0	0

Figure 16: Example of a rainfall file using ISWS gages

When all of the rainfall data has been entered into the spreadsheet, choose "Save As", select a .csv file extension, and give the file a descriptive filename.

It is required that the gage identifiers in the rainfall file must match the gage identifiers in the weight file (see below). If this is not the case, the program will apply the first data column in the rainfall file to all of the basins listed in the weights file (i.e. no weighting will be applied). There will be the option to cancel the processing, however, instead of proceeding with a uniform distribution.

Weights File Creation

If the user is working outside of the Cook County Precipitation Network, then a custom weight file must be created. Rain gage weighting is necessary to construct an input hyetograph for IUHM. Since some service areas lie within Thiessen polygons associated with multiple rain gages, it is necessary to weight the rainfall distribution in these areas to accurately describe the spatial variation of rainfall.

In Figure 2 below, 67% of Service Area A lies within the Gage 1 Thiessen polygon, and 33% of Service Area A lies within Gage 2's boundary. Therefore, Gage 1 would receive a 0.67 weight for Service Area A and Gage 2 would receive a 0.33 weight.

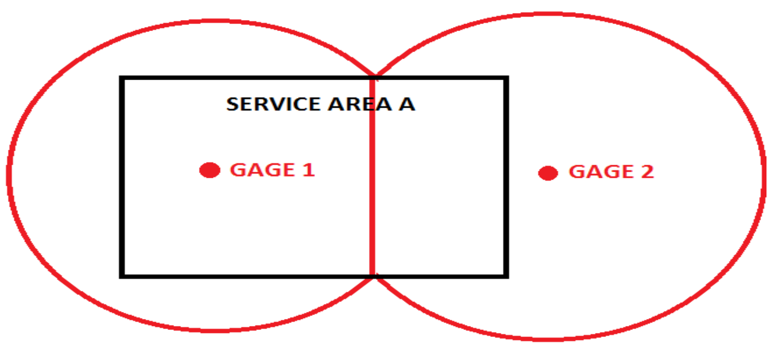


Figure 17: Illustration of the Computation of Weights for a generic service area

The weight file will have each service area listed under the heading of "Basin" in the first column. The adjacent columns will contain a rain gage identifier in the first row, with the weight of that rain gage on each subcatchment in the associated row converted to a decimal value. It is suggested that the user browse to the package directory, then enter the rainfall directory and copy the weights_iuhm.csv file to the desktop, naming it to something other than the original name. The user should then modify the copy, being careful to not delete any rows. All of the basins in the weights_iuhm.csv file should be present in the new mapping.

For example in Figure 10 below, cell N2 shows that gage G13 is associated with 25% of the surface area of CDS-10-1. Notice that because each row represents the surface area of one service area, the summation of each row will equal one, or 100% of the surface area of that basin.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T
1	Basin	G1	G2	G3	G4	G5	G6	G7	G8	G9	G10	G11	G12	G13	G14	G15	G16	G17	G18	G19
2	CDS-10-1	0	0	0	0	0	0	0	0	0	0	0	0	0.25	0	0	0	0	0.75	0
3	CDS-10-2	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0
4	CDS-10-3	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0
5	CDS-105th	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0	0
6	CDS-107th	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0	0
7	CDS-110th	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0	0
8	CDS-11-1	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0
9	CDS-11-2	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0
10	CDS-11-3	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0
11	CDS-116th	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0.73	0.27	0
12	CDS-12-1	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0
13	CDS-12-2	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	0
14	CDS-13	0	0	0	0	0	0	0	0	0	0	0	0	0.45	0.01	0	0	0	0.47	0.07

Figure 18: Excerpt of an example weight file. Adding across each row will equal one

When all of the weights have been entered into the spreadsheet, choose "Save As", select CSV file type, and give it a descriptive filename. Importing this file into MetroFlow is discussed in the following section. By default, MetroFlow includes a weights file for the Cook County Precipitation Network and names this *ISGS Gages*.

Importing Rainfall

3.10.6 Importing Rainfall into MetroFlow

Within MetroFlow, go to the Timeseries tool and click the storm button. This will bring up the import historical rainfall event tool:

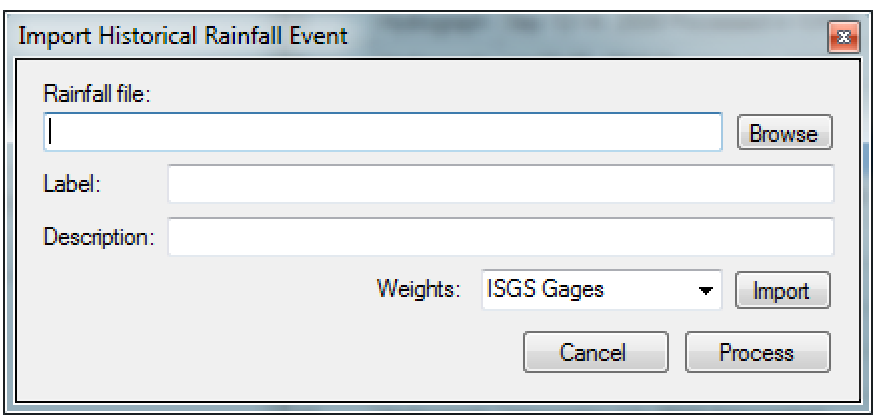


Figure 19: Utility for importing a historical rainfall event

Browse to a rainfall CSV file using the Browse button, and then input a descriptive short label to name the new timeseries; the description is optional. To use an existing weights file, select it in the Weights drop down menu and press Process. To import a new weights file, click the Import button by the weights drop down menu. This will bring up a file selection dialog. Browse to the new weights file and select it. The user will then be prompted to give the weights file a label. Input a short name. Once done, the Weights drop down menu will have a new entry. Select it, and then press Process.

A new rainfall timeseries set will appear in the Timeseries window.

3.10.7 Importing External Hydrographs

Editing Timeseries Sets

A single timeseries file can be imported into an **existing timeseries set** with the *Import single TS* action in the *View and Edit Timeseries* window (which is described in section 3.10.3, TS Action (Timeseries Actions)). This will allow the user to browse to and select a single file that allows importing of the timeseries. The format of the timeseries can be as follows, where Date/Time is in `M/D/YYYY HH:MM` format, and the columns are tab separated (where the date and time are space-separated):

```
TSNAME     DATE/TIME  VALUE
TSNAME     DATE/TIME  VALUE
TSNAME     DATE/TIME  VALUE
```

Or

```
DATE/TIME  VALUE
DATE/TIME  VALUE
DATE/TIME  VALUE
```

Or

```
TIME  VALUE
TIME  VALUE
TIME  VALUE
```

An example:

```
9/12/2000 00:00  0
9/12/2000 01:00  0.5
9/12/2000 02:00  1
9/12/2000 03:00  2
9/12/2000 04:00  3
9/12/2000 05:00  4
```

Once a file is selected it is imported into the set and the set is automatically saved. It can now be used in a scenario.

The *Import multiple text TS* option allows importing and renaming of multiple timeseries files. It can also be accessed by dragging multiple text timeseries files into the Timeseries list in the *View and Edit Timeseries Set* window. It provides a grid view whereby the user can rename the timeseries for use inside of MetroFlow. An additional option, *Automatically add dashes after first three characters* can be used to automatically correct multiple timeseries names in the case that timeseries names don't match the internal convention of CDS/MDS/DDS followed by a hyphen, then the dropshaft number. An example of this window is given in Figure 20. Files are selected by dragging them from Windows Explorer into the window.

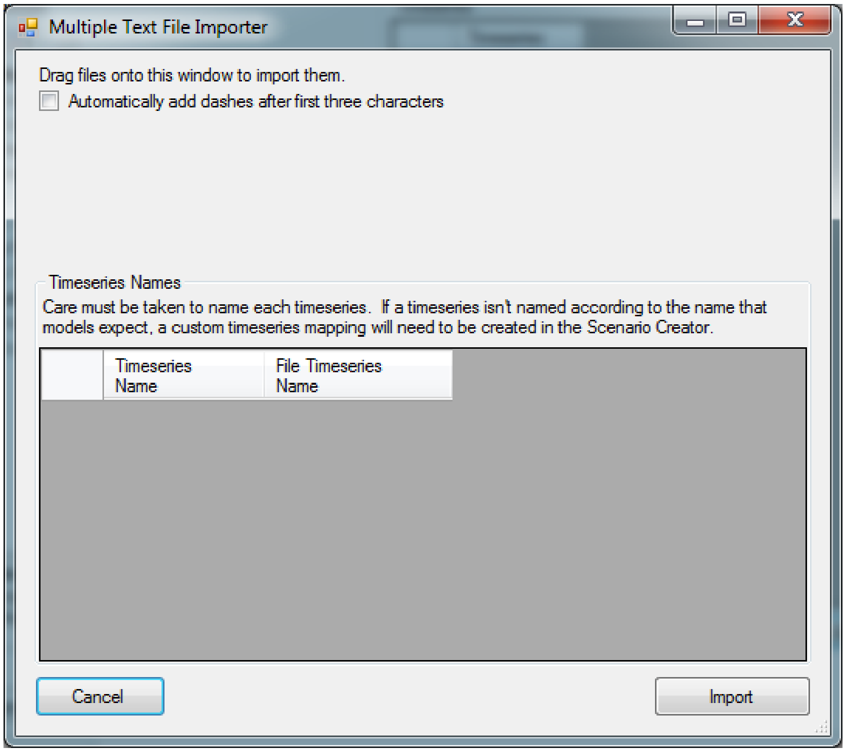


Figure 20: Multiple text timeseries import window

3.10.8 Importing from CSV Files

MetroFlow supports importing and exporting individual and bulk records from CSV files, such as those generated by external models (e.g. InfoSWMM, InfoWorks) or external data tables. This option is accessed from the *View and Edit Timeseries Set* window, under the *Import from CSV* TS Action, or can be accessed by dragging a CSV file into the Timeseries list in the aforementioned window.

The window allows import of CSV data in two steps: 1) selection of time column and records to import, and 2) custom renaming of timeseries. The first step involves selecting a time column using the *Time column* drop-down selection box. Once a time column is selected, the program attempts to guess the format of the column, which is can be overridden by the user by selecting one of the *Time format* options (*Date/time*, *Minutes*, or *Seconds*). The user then selects the columns that are to be imported by either using the *Select all* checkbox (making sure to deselect any time columns) or by clicking on the timeseries list box. This is illustrated in Figure 21.

Once the columns have been selected, the user can select a conversion factor under the *Units conversion* selection box in order to convert units if necessary. Available options are *None* for no conversion, *CMS_to_CFS* for a cubic meters per second to cubic feet per second conversion, and *MGD_to_CFS* for a million gallons per day to cubic feet per second conversion. The user then can press the *Add To Import List* button and proceed to the second step.

The second step involves setting a time period and custom timeseries names. If the CSV file contains more data than is wished to be imported, the *Time Window* can be set to determine the window of data to import. Naming is done under the *Timeseries Names* section where the timeseries can be renamed from the default column names. The second step is illustrated in Figure 22.

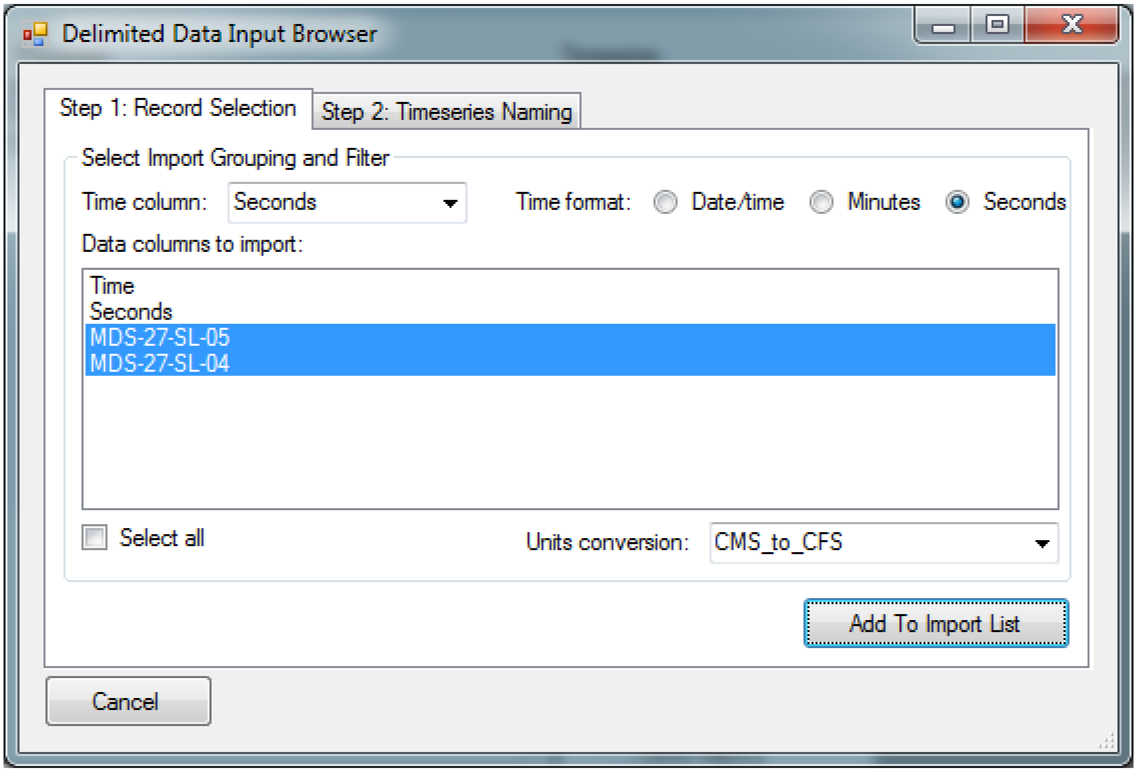


Figure 21: CSV Import window

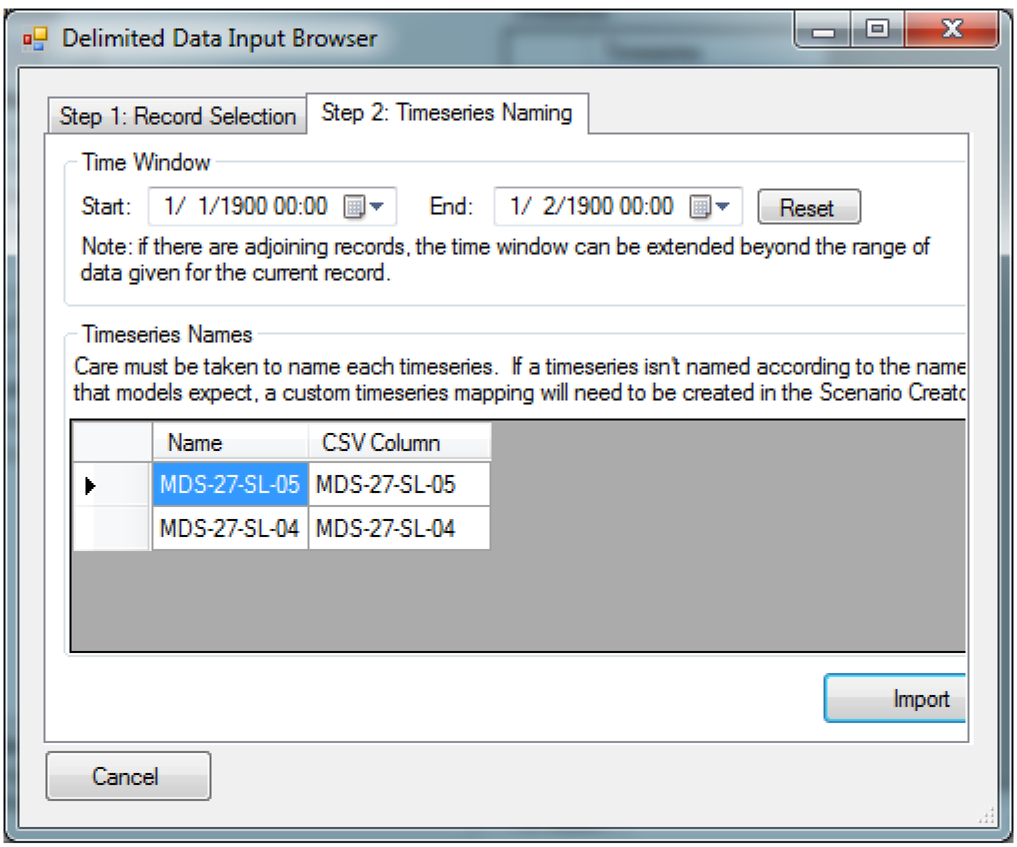


Figure 22: CSV timeseries import naming

3.10.9 Importing and Exporting from and to DSS Files

Importing/ Exporting Timeseries to DSS Files

MetroFlow supports importing and exporting individual and bulk records from and to HEC-DSS files. HEC-DSS is a data storage format developed by the U.S. Army Corps of Engineers and is described in detail in section 7.

Import Records

If the user has a DSS file with many categories of data, importing multiple records from DSS performs a bulk import based on categories of the users choosing, e.g. flow, time interval, location. To import multiple records from DSS, create a new timeseries set in the *Timeseries Set* Window, where the timeseries set is ***not*** relative. Save the timeseries, then View and Edit Timeseries Set. Choose *Import from DSS* in the TS Action dropdown menu, then Execute TS Action. After selecting the data, the DSS Input Browser window will appear. Choose Parts A, C, and E in accordance with above descriptions, then click *Add to Import List*.

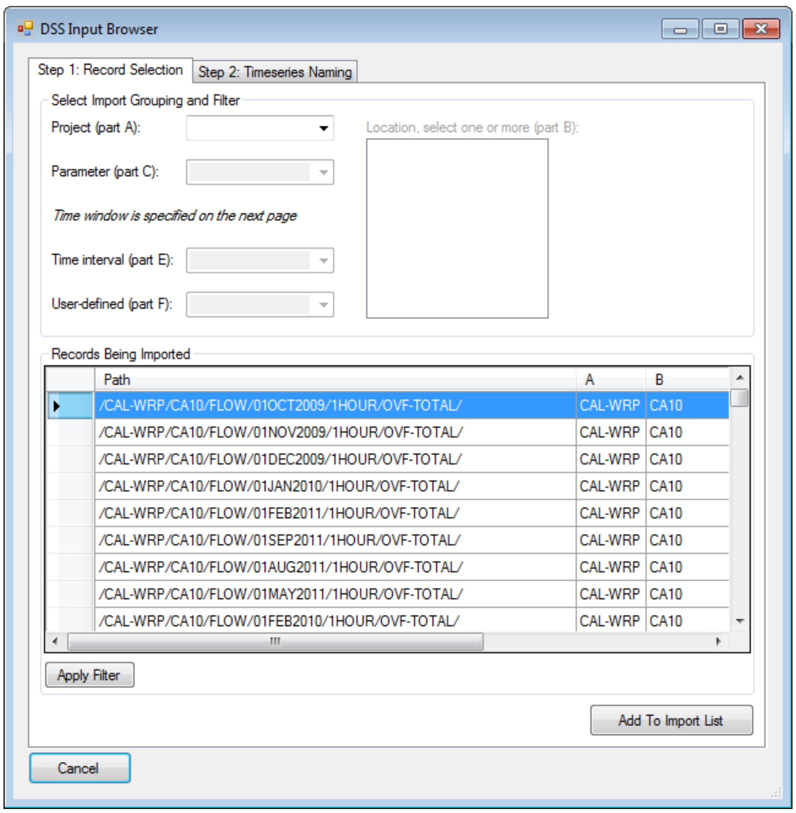


Figure 23: DSS import browser for selecting HEC-DSS file records to import into MetroFlow

Select the time window in the second tab, *Step 2: Timeseries Naming*. The time window will restrict import to the time period chosen. For example use Start: 10/1/09 and End: 9/30/2010 for Water Year 2010. The time window is correspondent to the time extents of all of the data records combined.

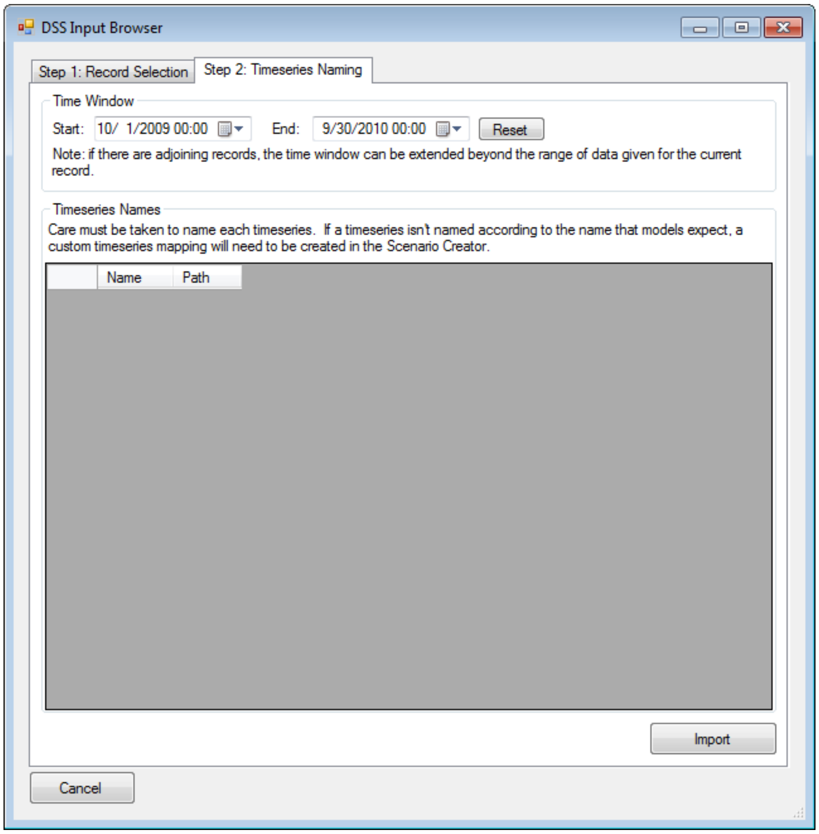


Figure 24: Step 2 in the DSS import process, timeseries naming

Each location has a different period of record. If a time window outside of the period of record is selected for a specific location, the imported data will store a zero data value.

After choosing *Import*, the timeseries will appear in the *View and Edit Timeseries Set* window. To view raw data, double click on the timeseries name cell.

This same process can be used to import individual DSS records into an existing timeseries set.

When the DSS files are initially imported, the timeseries are very likely not associated with any inflow nodes because they are not named correctly. The process of mapping timeseries to nodes is described in section 3.11.

Exporting Selected Records

The TS Action *Export Selected to DSS* allows the user to export selected timeseries to a DSS file. The DSS output browser allows the user to type in parameters for Part A, C, E, and F. In the data grid, the user alters the Part B names as desired (by default they match the *Input Timeseries* names). The time window can be specified to only export data within a specific time window.

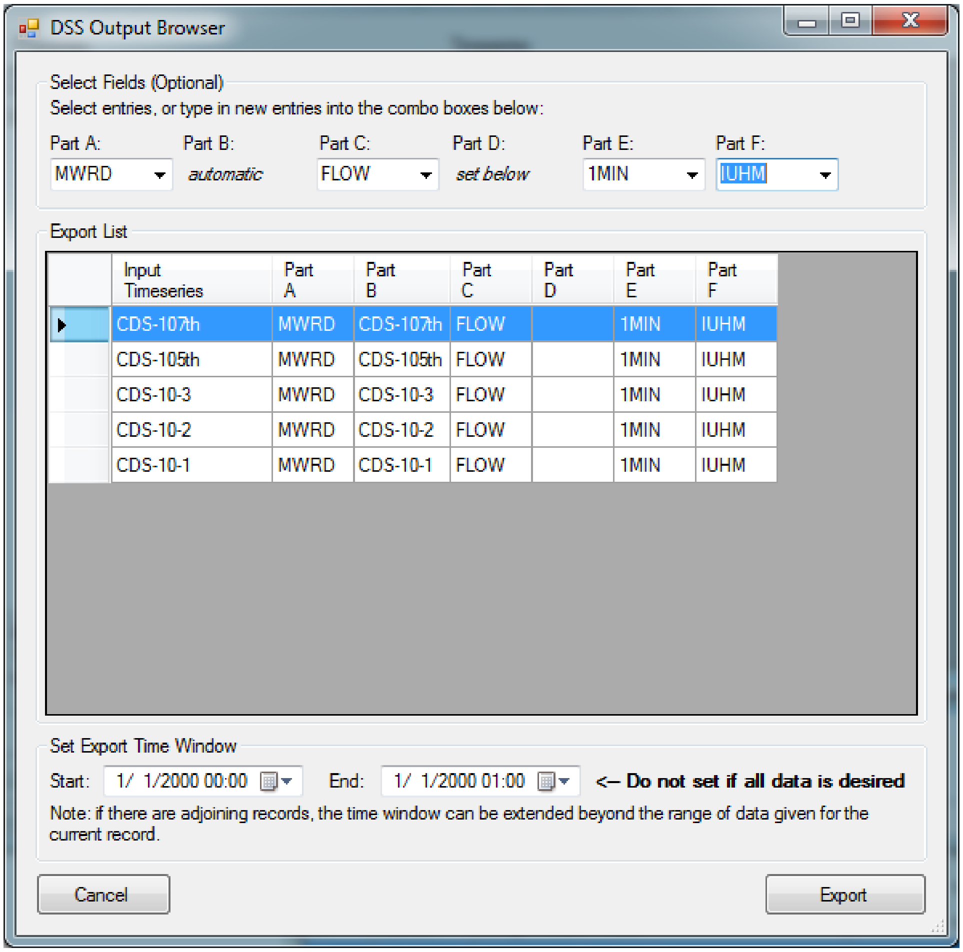


Figure 25: DSS output browser for saving timeseries to a HEC-DSS file

Using Timeseries Mappings

3.11 Timeseries Mapping Window

The various models in MetroFlow expect specifically-named timeseries at locations known as inflow nodes, so how timeseries are named—or alternatively mapped—is very important. Existing built-in mappings link timeseries names to inflow nodes, but for timeseries that are externally imported from DSS or other sources the naming may not be consistent with the mappings. The *Timeseries Mappings* window allows the user to create new mappings. (An alternative method involves renaming the timeseries inside of the *View and Edit Timeseries Set* tool, accessed from the Timeseries Sets window.)

To create a new mapping, select the *TS Mappings* window from the Windows menu. Then select the working layer that corresponds to the layer whose inflows will be remapped. The *Timeseries Mappings* dropdown menu in the *TS Mappings* window will be populated with the available mappings for the given working layer. Certain mappings cannot be altered, and these are known as the Built-In mappings.

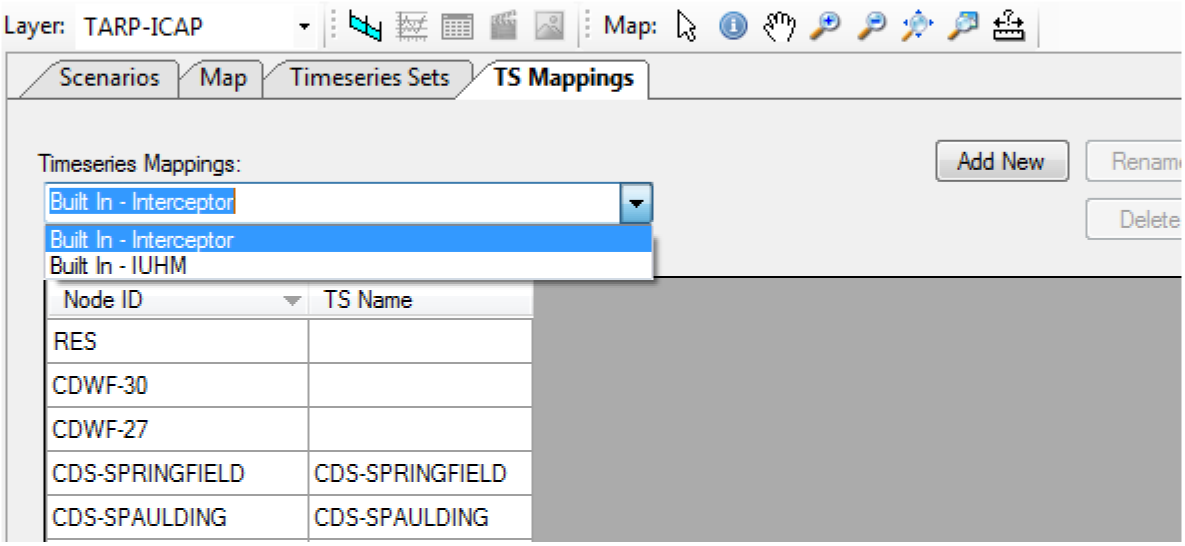


Figure 26: Timeseries Mappings window for mapping timeseries to specific nodes

A new mapping is created with the Add New button, and fields for creating the mapping are displayed.

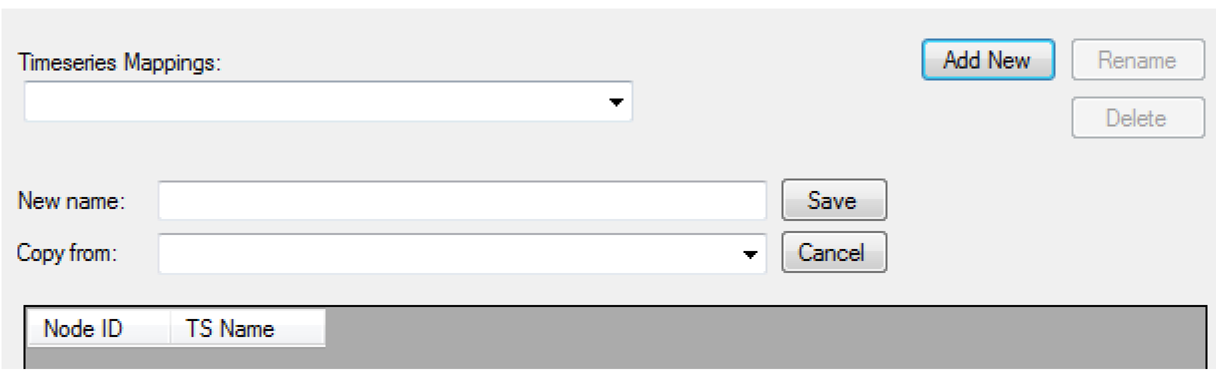


Figure 27: Timeseries Mappings window actions

A new mapping must have a unique name, and also can be copied from an existing mapping (e.g. the Built In mappings). If the new mapping is copied from another mapping, then the inflow locations will have the timeseries names from the other mapping. If no mapping is copied from, then the inflow locations will have no default timeseries names. The Save button then creates the new mapping which can at that point be edited.

To edit an existing (non-built-in) or newly created mapping, a data grid is displayed with two columns. The first column is the Node ID which is the inflow location, and the second column is the TS Name which is the timeseries that will be assigned to the inflow location. The user can type values into the TS Name field. Changes can be reverted to the original state, or they can be committed by clicking the Save Changes button. If the user desires to retrieve the mappings for use in another program, the Copy to Clipboard button can be pressed to copy the contents of the mapping to the clipboard. If the user desires to edit the mapping in another program (e.g. Microsoft Excel) the mapping can be pasted back from the other program via the Copy from Clipboard button.

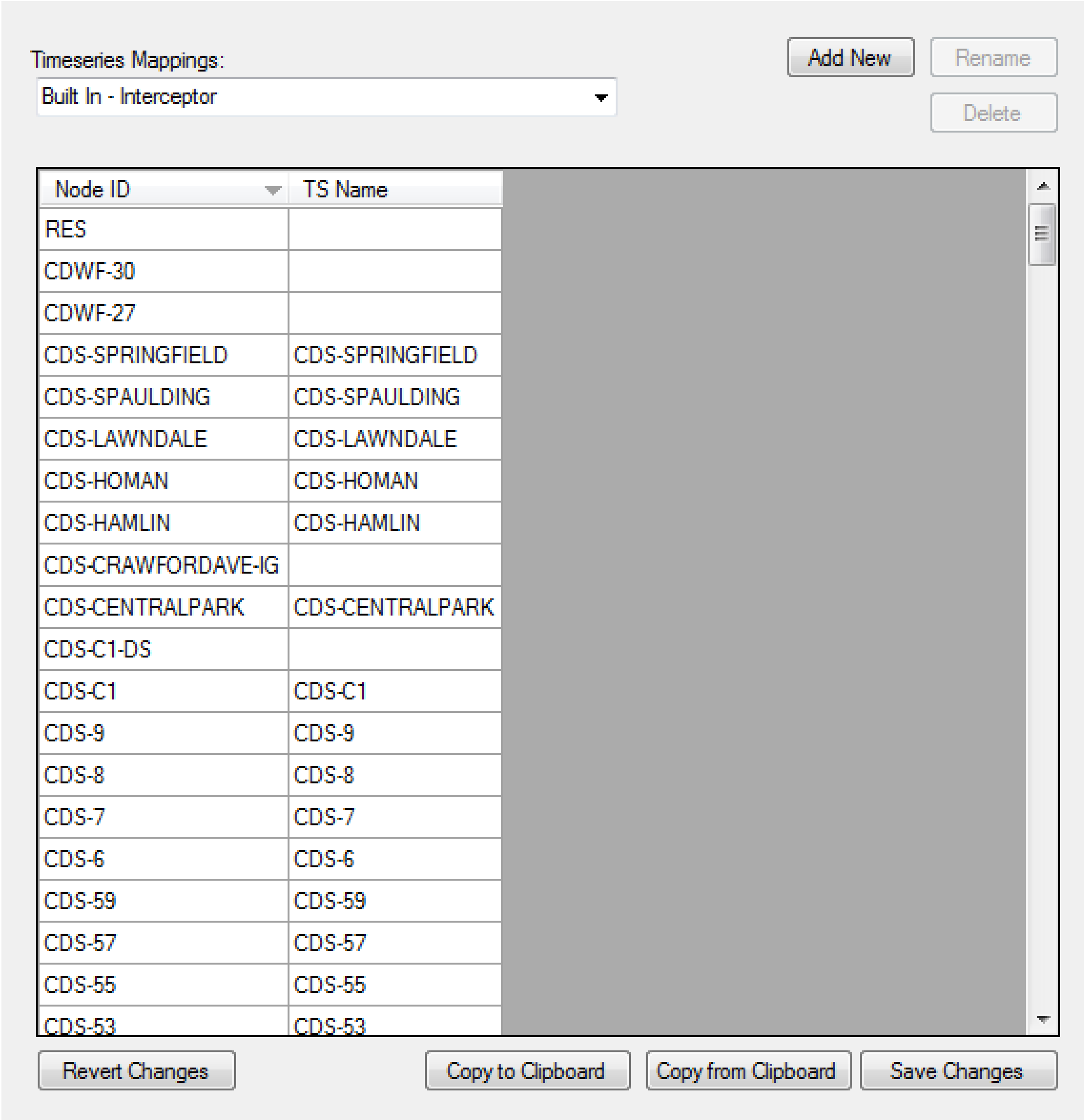


Figure 28: Timeseries mapping grid for viewing and/or altering inflow locations

Other actions that the user can take are deleting an existing non built-in mapping, and renaming one of the non-built in mappings. These buttons are enabled once a valid mapping is selected.

3.12 Results Animation Window

Once results have been loaded that correspond to the current working layer, the Animator can be utilized for analysis. The Animator allows results to be animated on the map and/or in profile plots. To open, click the Animator button on the Results toolbar to display the Animator window in the left window. The Results Animation window has three components: Calendar, Time Slider and Elapse Time, and Playback Control.

The Calendar component, shown in Figure 29, allows a user to see the actual date of the current animated step. The map can be updated to a specific date simply by clicking on a specific date. Note that the Calendar component is hidden for simulations that are less than two days in duration. Also, if a design storm simulation has been loaded, the calendar defaults to January 1, 2000. For most design storm simulations, however, the Calendar will be hidden.

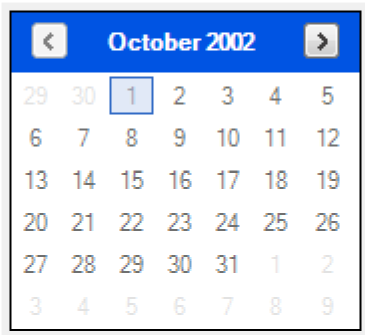


Figure 29: Calendar component in Results Animation window

The Elapsed Time field stores the elapsed time since the start of the simulation in the format DDD.HH.MM.SS where D is days, HH is hours, MM is minutes, and SS is seconds. The Elapsed time field will increment automatically during an animation, but can also be edited manually to jump to a particular time in the modeled scenario. In addition, the Time Slider, shown in the top of Figure 30, allows the user to drag a grab-bar to rapidly move the displayed simulation results to a specific time.

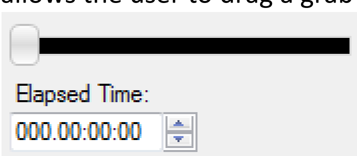


Figure 30: Time Slider and Elapsed Time components in the Results Animation window

Elapsed Time, or simulation time, is different than clock time. The Elapsed Time field will display the amount of time passed since the start of the scenario; however, it will not display the actual date and time of the scenario (if it is historic). For example, for a storm starting August 22, 2007 at 21:00, the Elapsed Time field will display 0.00:00:00 signifying the start of the scenario and NOT Aug 22, 2007 21:00, the clock time of the actual storm.

The Playback Control allows for automatic playback of the time, and the slider controls the speed. The slider at the left is the slowest speed and moving it to the right increases the speed.

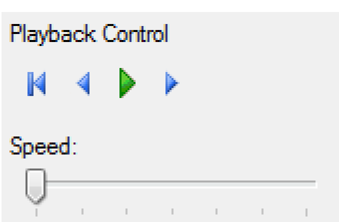


Figure 31: Playback Control component in Results Animation window

Press the green arrow to start the animation. The first symbol resets the Elapsed Time field to 0.00:00:00 whereas the backwards and forwards arrows will stop the automatic animation and allow the user to manually add or subtract one minute to the model clock.

3.12.1 Map visualization during animation

In order to observe the animation on the map, the *Symbology for Active Layer* fields in the *Layers window* must be edited. A selection of characteristics can be simulated such as flooding for nodes and flow for links. The color map options allow the user to choose the color scheme of the map visualization. The gradation of the color schemes can be edited by clicking on the Edit breaks buttons. Section 3.7 describes the variables that can be animated.

3.13 Controls Window

The *Controls window* works in conjunction with the Interceptor model and allows the user to simulate the sluice gate rules. This window will be blank until an Interceptor scenario is loaded and the *Working Layer* is set to *Interceptor*.

The suggested gate operation rules are as following:

1. Global Gate rule: when the HGL of node CALUMET_PS reaches -150ft, the system gates should be reduced to a 15% opening.
2. CDS-13 Gate rule: when the HGL of node CALUMET_PS reaches -190ft, close completely the gate at CDS-13.
3. CDS-34 Gate rule: when the HGL of the CDS-34 reaches -257ft, close completely the gate at CDS-34.

To determine the appropriate time to close the sluice gates, the recommended procedure is to run the Interceptor and TARP-SWMM model with the sluice gates open, determine the closing times from the TARP model, and then feed those times into a second set of Interceptor and TARP runs. For an example of this procedure, refer to section 4.4 in the Quickstart Guide.

Once the gate closure times are established, enter the information in the *Control* window.

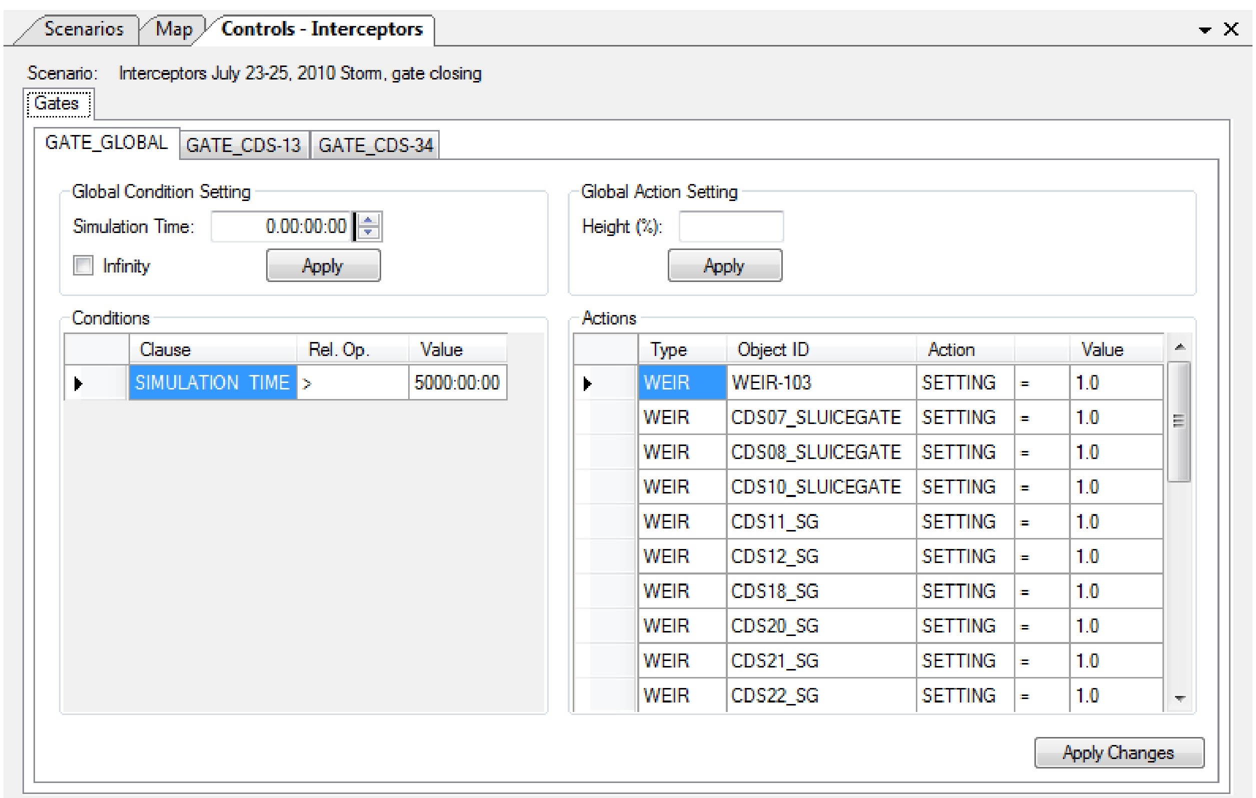


Figure 32: Controls window, used for controlling gate openings and times

It is necessary to convert "real time" to "simulation time". For example, for a storm that started at 5:40 PM on July 23, 2010, 7:00 AM the next day (July 24, 2010) occurs at 13 H 20M in simulation time, as opposed to 7H in real time. The *Tools* menu, *Time convertor* program allows the user to do this.

There is a tab for each gate rule. For each rule, enter the appropriate simulation time at which the gates should close and the percent to which they should close, clicking the local *Apply* button after each entry. To enter a non-zero gate open percentage, type in the number without the percent symbol (e.g. 15). To close gates entirely, enter 0, without the percent symbol.

3.14 Results Windows

These tools help the user navigate the analysis of scenarios. With the exception of the profile plot tool, they are only enabled when a scenario is loaded that contains a run for the selected working layer.

Results are obtained by selecting a list of objects through selection windows; these are documented in the following sections. When a results selection window is opened, the current map tool is set to be the identify tool to allow the user to click on objects to select them.

3.14.1 Profile Plot

Using Profile Plots

A profile plot shows the hydraulic grade line (HGL) for a selected sequence of conduits and dropshafts at any given time in a simulation. This is accessed from the results toolbar by clicking on the *Profile Plot* button. If no scenario is loaded, the tool will allow the user to view a profile plot of the tunnels without any HGL. If a scenario is loaded, HGL results will be displayed over the tunnel profile in conjunction with the Animator. The profile plot will be plotted based on the current working layer.

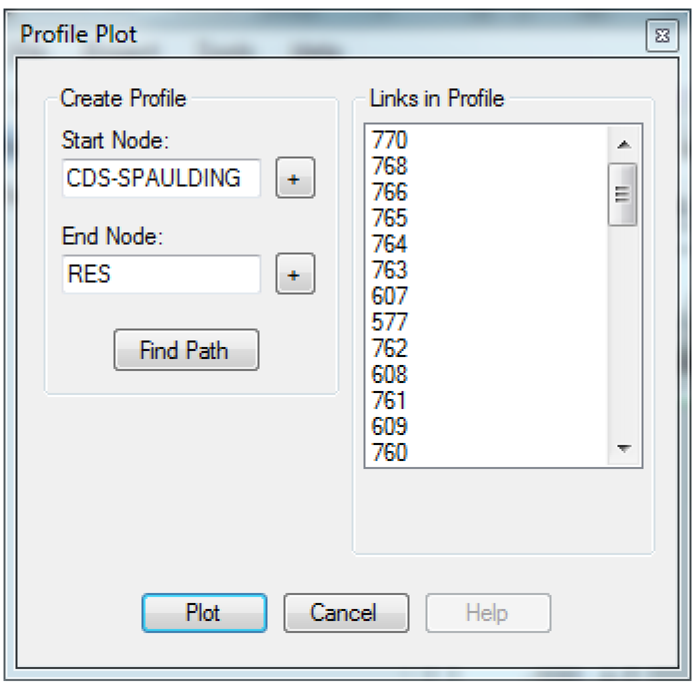


Figure 33: An example of the Profile plot window for an TARP-SWMM scenario.

Figure 33 and Figure 34 illustrate how a profile plot begins at CDS-SPAULDING and ends at Thornton Reservoir (RES). It is possible to type the start and end nodes directly, or the nodes can be selected by clicking on the map. The *Find Path* button attempts to find a path between the selected nodes. If a valid path was found, a listing of the connecting links between the selected nodes will populate the *Links in Profile* list box. Click *Plot* to continue or reselect the nodes if the path was not valid.

The results are displayed in a new tab in the main section of the MetroFlow window. The profile plot can be animated in order to visualize the profile at different times in the scenario. The *Results Animation* window provides the controls to visualize the HGL profile in the selected path at any time within the loaded scenario. The HGL in the plot will update corresponding to each change in time in the animator. The *Results Animation Window* will allow the user to animate the profile plot.

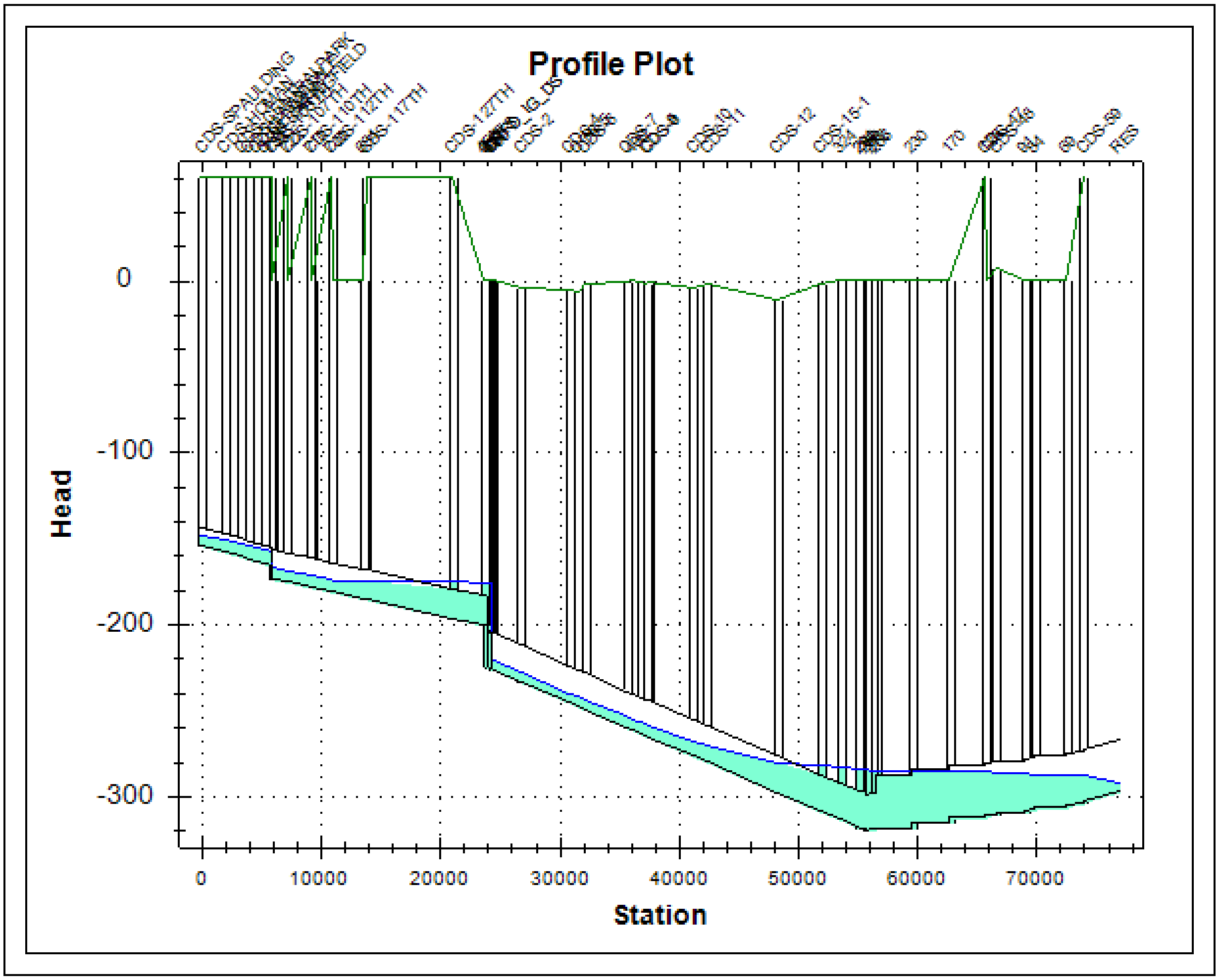


Figure 34: An example profile plot for a historic storm for the TARP-ICAP model. The Elapsed Time can be read at the left: this screen shot was taken when the storm was 6 hr 22 min into the simulated storm

Additionally, the following plot options are available by right-clicking on the plot:

- Copy: places the plot on the user's clipboard.
- Save Image As: saves the plot in a user-specified format (e.g. jpeg).
- Page Setup: opens a printing options menu where the user can specify the paper size and margins.
- Print: opens a print window, where the user can specify the desired printer, number of copies, and print the graph.

- Show Point Values: This function will cause the plot to display specific point values for the plot where the cursor meets the plot.
- Un-Zoom: This function returns the plot to the previous window range by undoing one zoom.
- Undo All Zoom/Pan: Returns the plot to its initial frame.
- Set Scale to Default: Has the same effect as the *Undo All Zoom/Pan* option.
- Plot Options: Other options for configuring the plot (such as colors, etc.).


Using Graphs

3.14.2 Graph

Use the graph tool to plot results for one or more objects which selected by clicking on the map. The Object Type menu will select the type of object that can be plotted. One plot will be generated for each object and variable in the list; for example if two objects are selected and two variables are selected then four plot windows will be displayed in the main window area. If multiple objects are selected but only one variable is selected, the *Overlay Objects* checkbox is enabled which will allow the user to plot all of the selected objects on one plot.

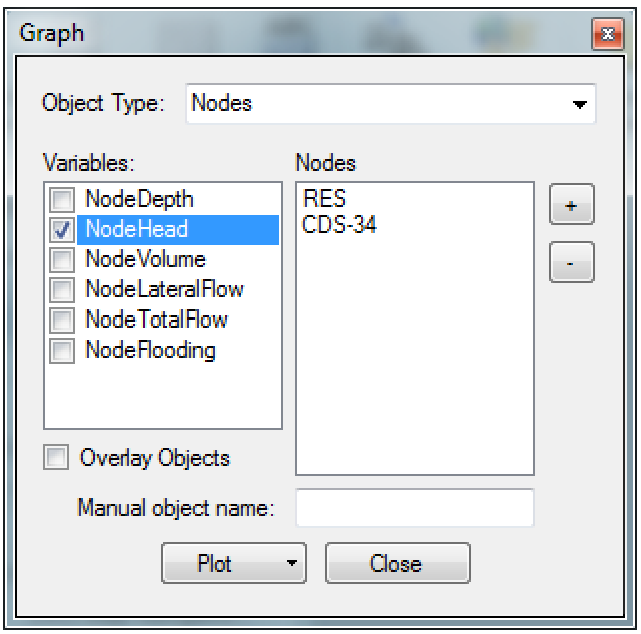


Figure 35: Graph type and object selection window

Objects are added to the select list by typing in the text box near the bottom of the window and either pressing the *Enter* key on the keyboard or the + button. The user can alternatively click on the map to select and object and then press the + button to add that object to the list. The *Variables* list box reflects the available data variables for the given object type (i.e. nodes, links, or system).

The *Plot* button is a combination menu-button. Simply clicking on the button performs the plotting actions as described previously (see Figure 36). Clicking on the arrow pointing down to the right of the button allows the user to merge an existing graph in the menu with the current data to be plotted. This is illustrated in Figure 37 and Figure 38. This feature allows comparison of the same object across scenarios. In addition, the merging capability allows plotting of different variables for the same objects within the same plot (see Figure 39). This allows a user to simultaneously see head response and flow through a node, for example. Zooming in and out of the map is synchronized with the X-axis.

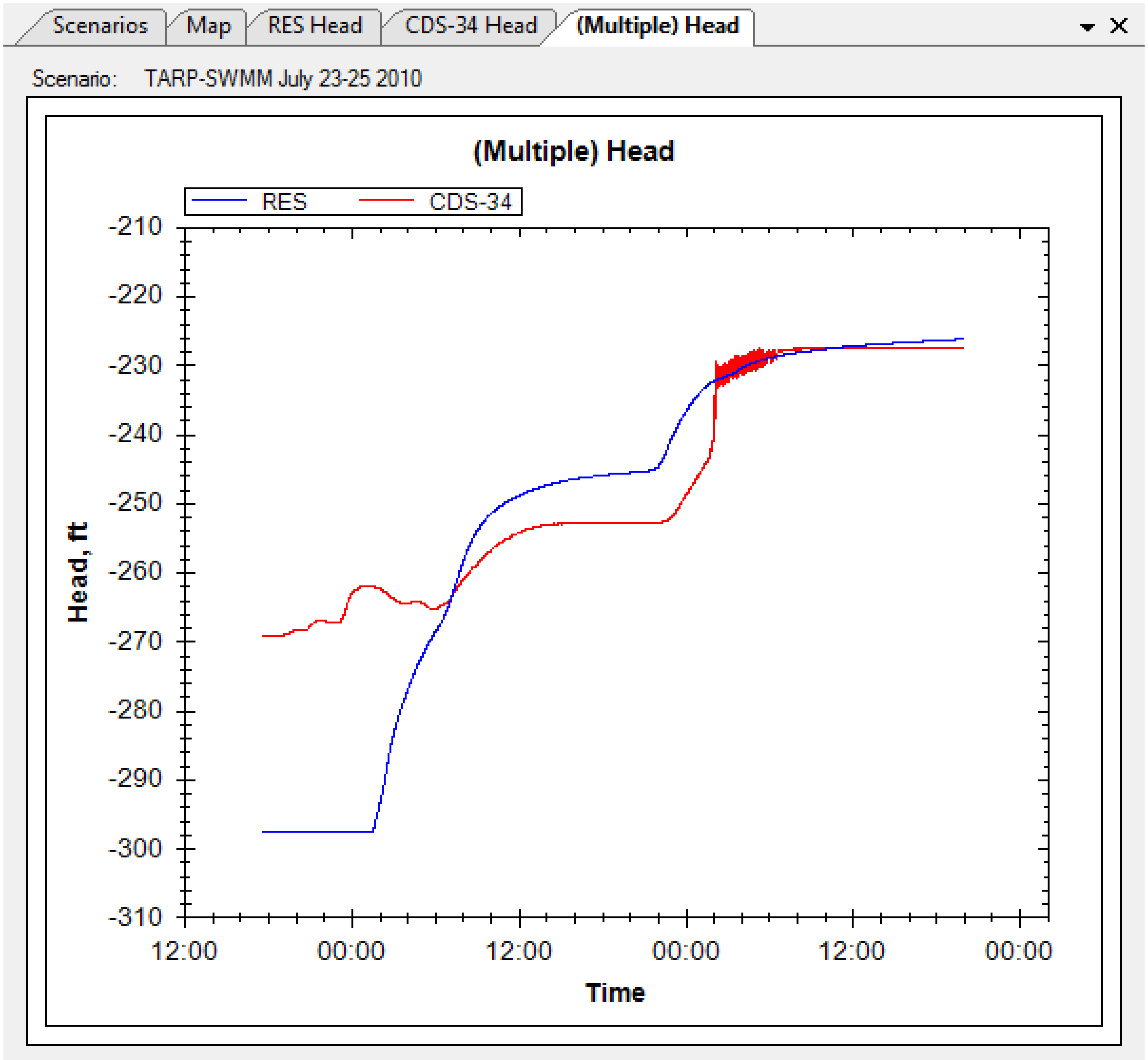


Figure 36: An example graph comparing the Node Depth for the reservoir and CDS-34 for historic storm July 2010

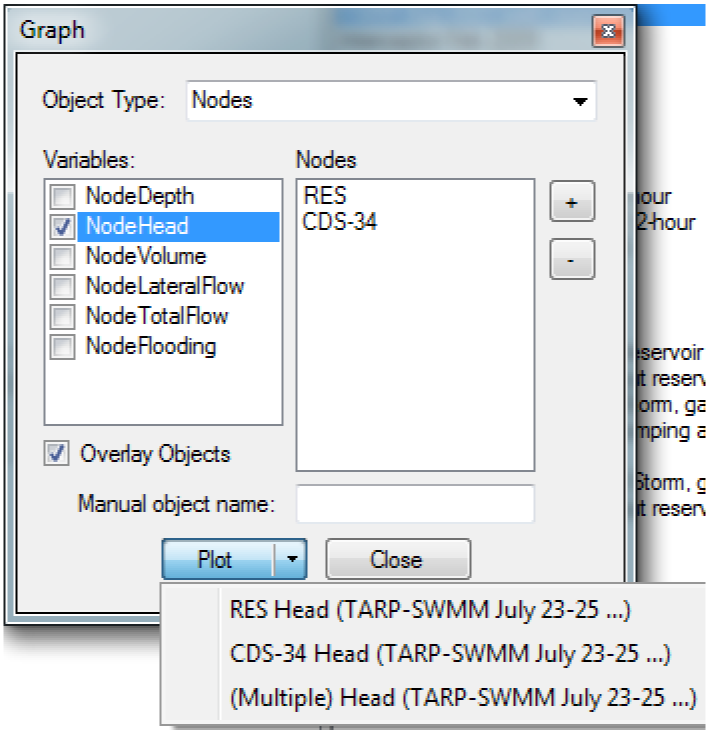


Figure 37: Example of merging plots

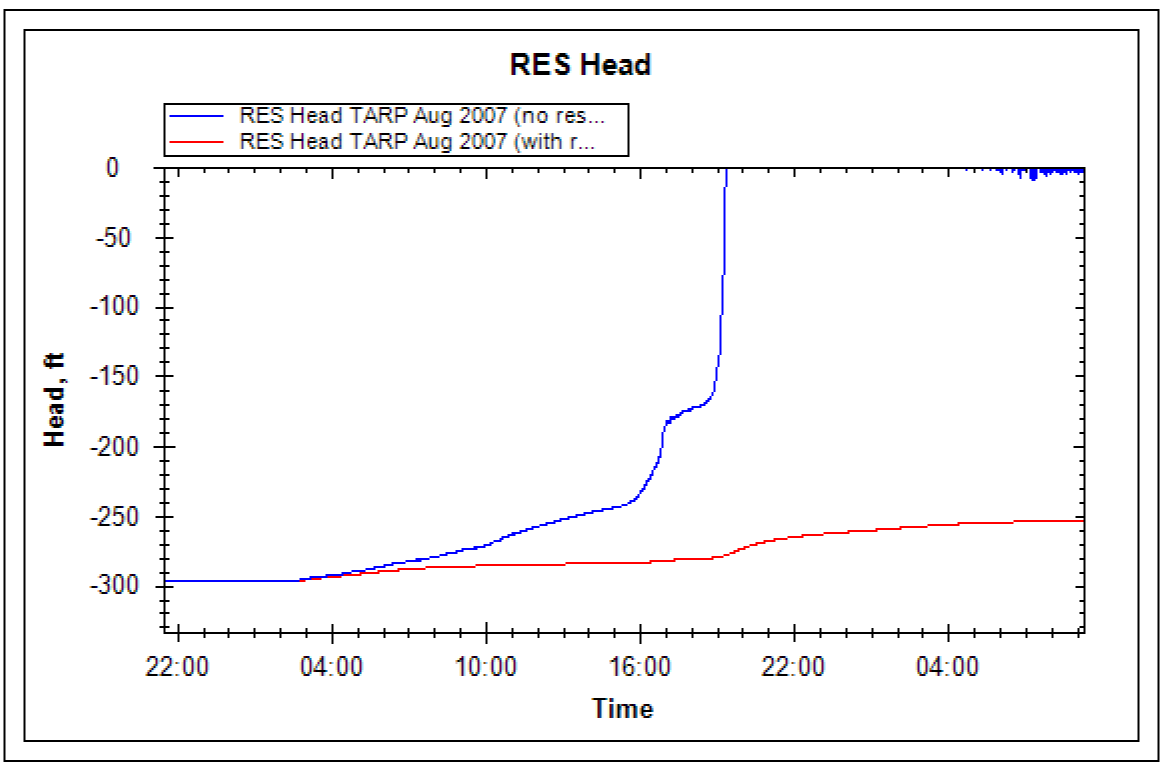


Figure 38: Example of comparing two scenarios for the same object using the merging capability of the graph tool

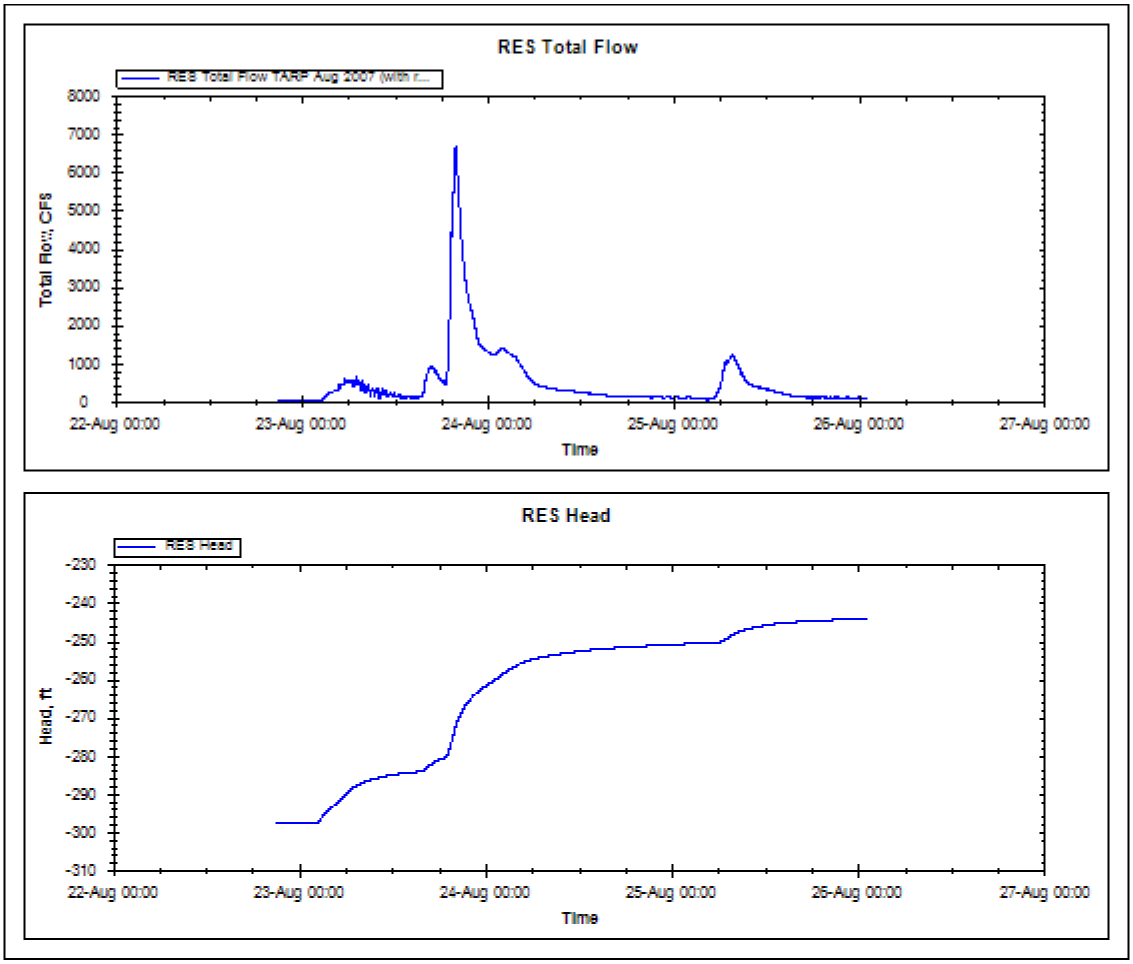


Figure 39: Example of comparing different variables for the same object using the graph merge capability

Once the plot window appears, the user may zoom-in by clicking and moving the mouse to create a box around the area of interest. The plot options, viewed by right-clicking the screen, are also displayed.

- Copy: places the plot on the user's clipboard.
- Save Image As: saves the plot in a user-specified format (e.g. jpeg).
- Page Setup: opens a printing options menu where the user can specify the paper size and margins.
- Print: opens a print window, where the user can specify the desired printer, number of copies, and print the graph.
- Show Point Values: This function will cause the plot to display specific point values for the plot where the cursor meets the plot.
- Un-Zoom: This function returns the plot to the previous frame by undoing one zoom.
- Undo All Zoom/Pan: Returns the plot to its initial frame.
- Set Scale to Default: Same effect as the *Undo All Zoom/Pan* option.
- Plot Options: shows a property grid allowing the user to change the axes extents. The X-axis is given in ordinal days since 1900 and should be modified with care. An example of this appears in Figure 40.

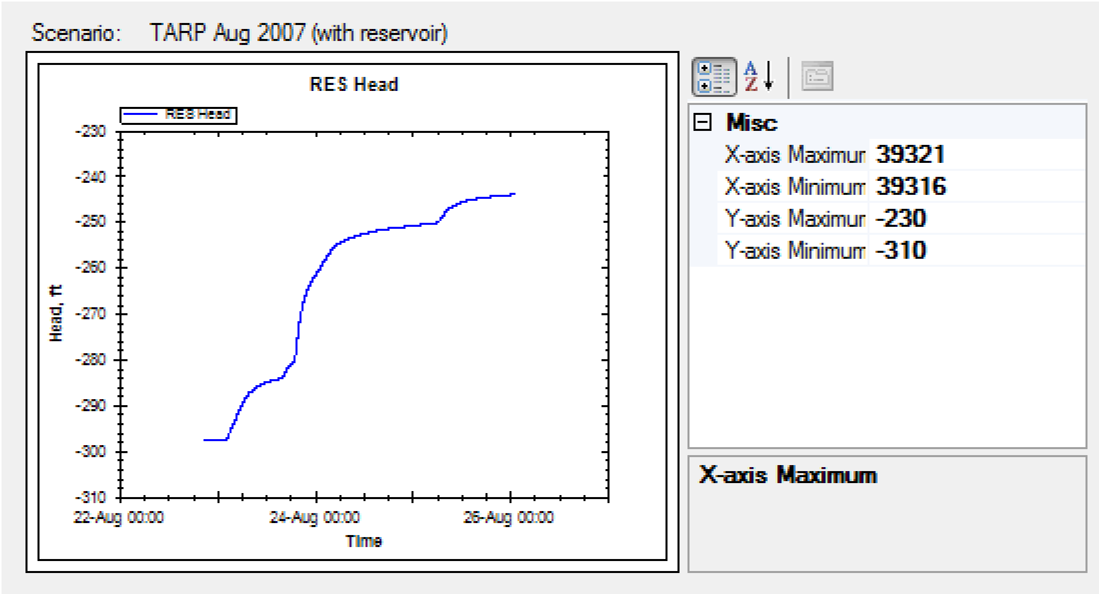


Figure 40: Example of plot options

Using Tables

3.14.3 Table

The table tool displays a tabular view of selected variables of simulation results. The table tool operates identically to the graph tool. In addition to the *Node* and *Link* object types, the *System* object type is enabled for displaying summary data of the entire modeling layer; system variables are described below.

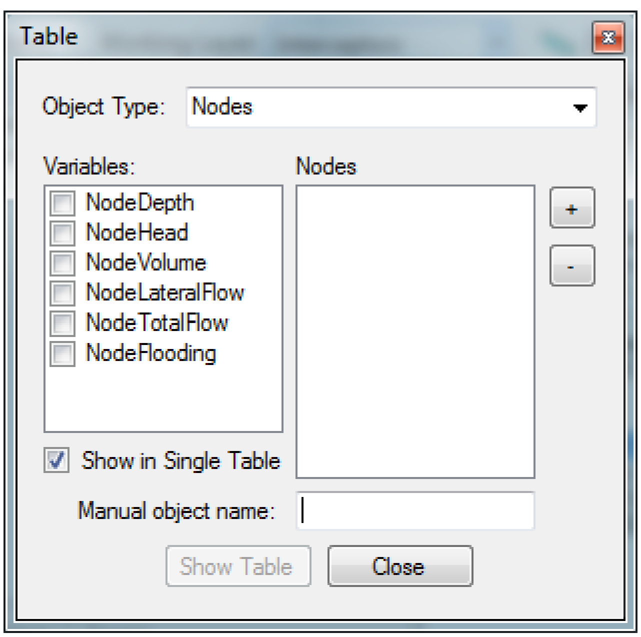


Figure 41: Table type/object selection window

It is possible to copy the contents of the table to the clipboard so that they can be used in external programs such as Excel. This is done by clicking on the *Copy to Clipboard* button in the table viewer once a table has been created. The user can also sort the tables by a column, by simply clicking once inside the column name. Clicking again reverses the sorting order.

Map | **(Multiple) Table**

Save Data   Copy to Clipboard

Scenario: July 2010, Interceptor, SG open

Date/Time	JCT-546 Depth [ft]	
7/23/2010 17:40	0	
7/23/2010 17:41	0	
7/23/2010 17:42	0	
7/23/2010 17:43	0	
7/23/2010 17:44	3.741436E-05	
7/23/2010 17:45	0.0001281419	
7/23/2010 17:46	0.0002837555	
7/23/2010 17:47	0.0005328127	

Figure 42: Operations in the table window

Depending on the model, there may be variables for the *System* object type.

Interceptor System Tables

The system variables that are available for the Interceptor model are the *Junction Flooding*, *Dropshaft Loading*, *CSO Loading*, and the *Outflow Loading* tables. The *Junction Flooding* table presents the sum of any volume of water that exceeded the flood depth of a given node (as defined by the SWMM model). The *Dropshaft Loading* table gives the user the ability to view volume and frequency of flow that went to the deep tunnels through the connecting structures. The third table, *CSO Loading*, likewise gives the volume and frequency of flows to CSO locations. Finally, the *Outflow Loading* table gives the user the ability to view the volume and frequency of flows to all outfall-type objects in the model. The loading tables summarize the flow frequency (percent of simulation that water was flowing at that node), total duration in hours, average flow and maximum flow in CFS, and total volume in MG of flows into the TARP system. As with the other table options, these tables can be copied to the clipboard for use outside of MetroFlow.

Scenarios | Map | Junction Flooding Report | Dropshaft Loading | CSO Loading | **Outflow Loading**

Save Data   Copy to Clipboard

Scenario: Interceptor July 2010

Outflow Node	Flow Freq. (%)	Total Duration (Hours)	Avg. Flow (CFS)	Max. Flow (CFS)	Total Vol. (MG)
CALUMET_WRP154	84.7	42.650	139.250	139.250	159.941
CALUMET_WRP16	94.9	47.750	295.122	539.980	379.497
CDS_105TH_DS	0.0	0.000	0.000	0.000	0.000
CDS_107TH_DS	19.1	9.617	14.411	34.001	3.732
CDS_110TH_DS	24.8	12.483	9.411	19.822	3.164
CDS_116TH_DS	70.7	35.567	37.496	159.388	35.914
CDS_C1_OUTFALL_SOUTH	7.9	4.000	5.835	14.274	0.629

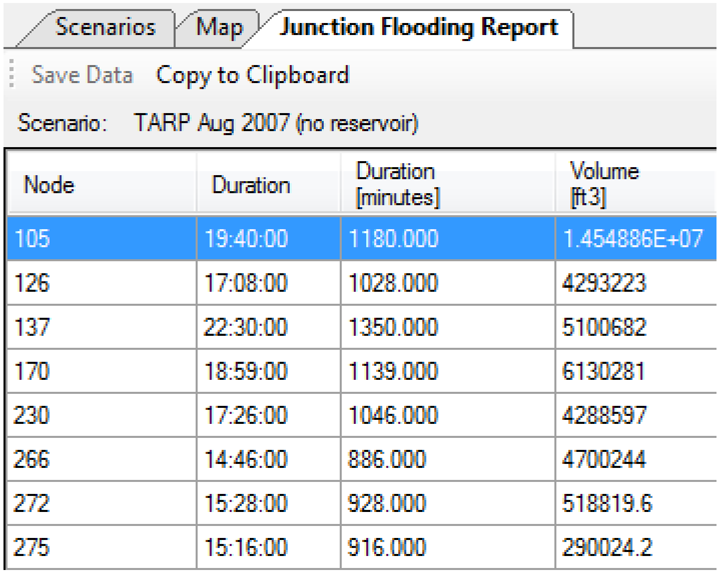
Figure 43: Example of Junction Flooding, Dropshaft, CSO, and Outflow Loading tables

CS-TARP System Tables

For the CS-TARP model, only *Dropshaft Loading* and *CSO Loading* tables are available, and they function as described in *Interceptor System Tables*.

TARP-SWMM System Tables

For the TARP-SWMM model, the only table available is the *Junction Flooding* table. This table summarizes any volumes of water that exceeded the flood depth of a given node, as defined by the EPA SWMM model. This table may be empty if no flooding occurred.



Scenarios | Map | Junction Flooding Report

Save Data Copy to Clipboard

Scenario: TARP Aug 2007 (no reservoir)

Node	Duration	Duration [minutes]	Volume [ft3]
105	19:40:00	1180.000	1.454886E+07
126	17:08:00	1028.000	4293223
137	22:30:00	1350.000	5100682
170	18:59:00	1139.000	6130281
230	17:26:00	1046.000	4288597
266	14:46:00	886.000	4700244
272	15:28:00	928.000	518819.6
275	15:16:00	916.000	290024.2

Figure 44: Example Junction Flooding table

3.14.4 Animator

See Results Animation Window, section 3.12.

3.15 CSO Analysis Window

The CSO analysis tool is a new feature in MetroFlow 1.7 and provides the user with the ability to perform graphical statistical analyses on combined sewer overflows. Figure 45 illustrates that this window appears on the left side bar of the MetroFlow window. These analyses are similar to the *DS/Outfall Loading* and *Overflow Volume* tables described in section 3.14.3, but are instead presented in multiple formats:

- Tabular, for CSO locations only:
    - TARP-SWMM dropshafts are considered to be CSO locations for the TARP-SWMM model;
    - An internal list of outfalls is maintained by MetroFlow for the Interceptor model;
- Graphically, summarized on a map by coloring and sizing CSO node locations;
- Graphically, through time stack plots: these plots use color bands to show the entire CSO hydrograph and stack these on top of each other for a graphical summary of all CSOs;
- Animated on the map, at CSO locations only; this contrasts with the other map animation option which shows variables for all nodes, not just a subset.

These options are all accessed through the CSO analysis window.

The CSO analysis tool is only available for TARP and Interceptor modeling layers. A valid scenario and working layer must have been loaded for the tool to be enabled.

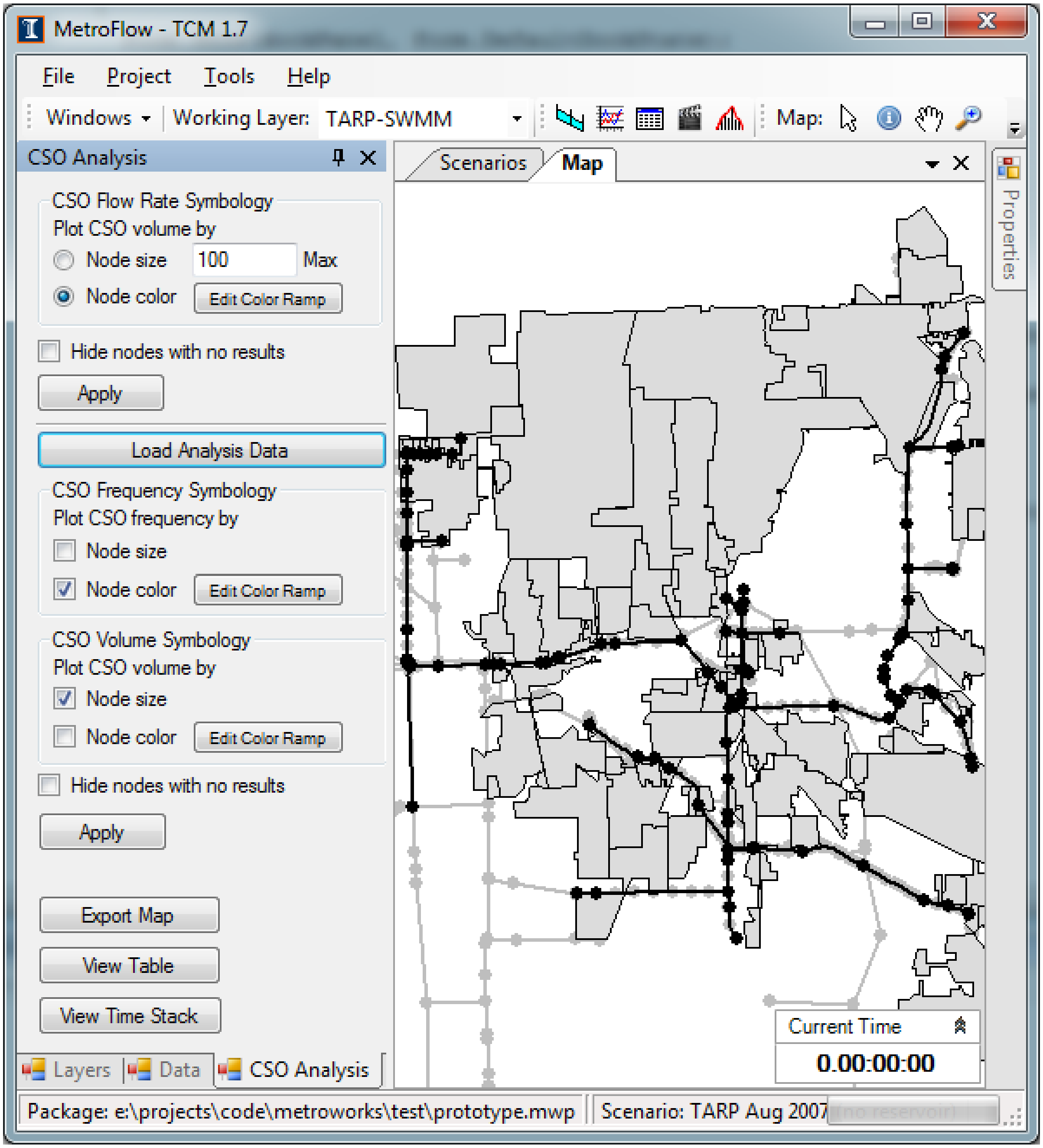


Figure 45: CSO analysis window

There are two components to the CSO Analysis window, divided in the window by a horizontal line just before the *Load Analysis Data*.

3.15.1 CSO Animation

Animating CSO Events

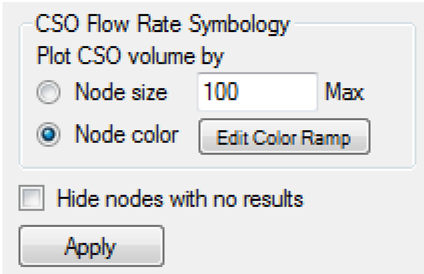


Figure 46: CSO analysis, animation of CSO events

The first component, illustrated in Figure 46, allows animation of CSO events at CSO locations. Animation of these CSOs begins when 1) the Animator tool is animating the map, and 2) when the user presses the *Apply* button. There are three options that control the graphical animation.

Node Size

CSO events can be indicated by the diameter of the node on the map by selecting the *Node size* option and choosing a value for the *Max* field (defaults to 100). Nodes will be sized from a graphical minimum to maximum diameter by mapping the CSO flow rate to a range of 0 – *Max*. CSO flow rates outside of the 0 – *Max* range will be mapped to the graphical minimum or maximum diameter respectively. illustrates this option.

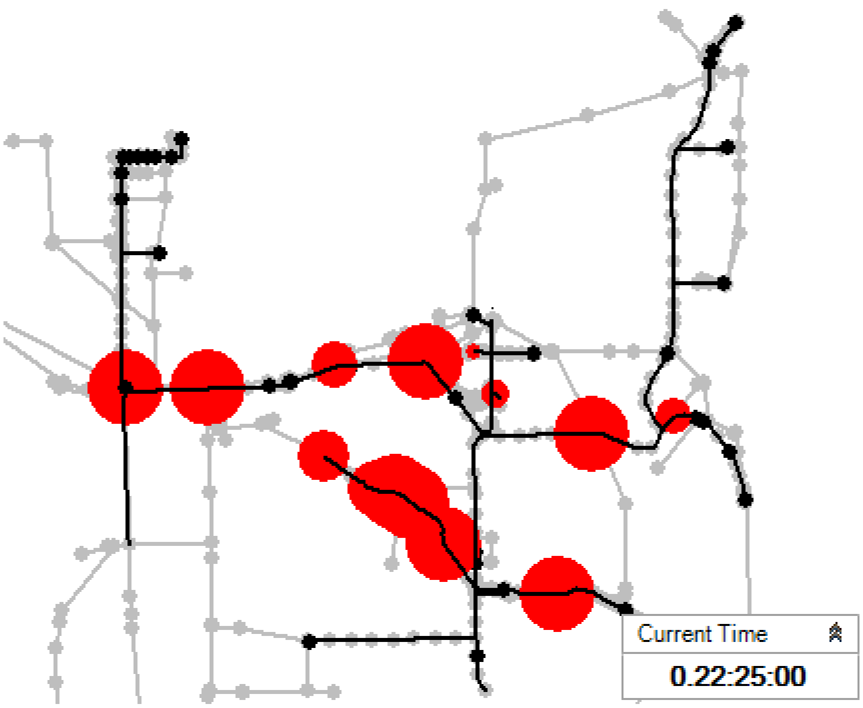


Figure 47: Node size CSO animation

Node Color

CSO events can be indicated by the color of the node (with a constant size) on the map by selecting the *Node color* option and optionally changing the color ramp with the *Edit Color Ramp* button. The default mapping is a standard hot-to-cold color ramp with blue representing low flow values and red representing high flow values with intermediate colors of yellow and green. Editing the color ramp is a powerful way to view CSO locations that exceed a particular threshold. This can be done by creating a number of breaks (5-10 is a good number) from a data minimum (e.g. 0) to the maximum threshold that is to be visualized; then by applying the hot-to-cold color ramp and closing the color ramp editor. The mechanics of this process are described in section 3.16 and in the tutorial.

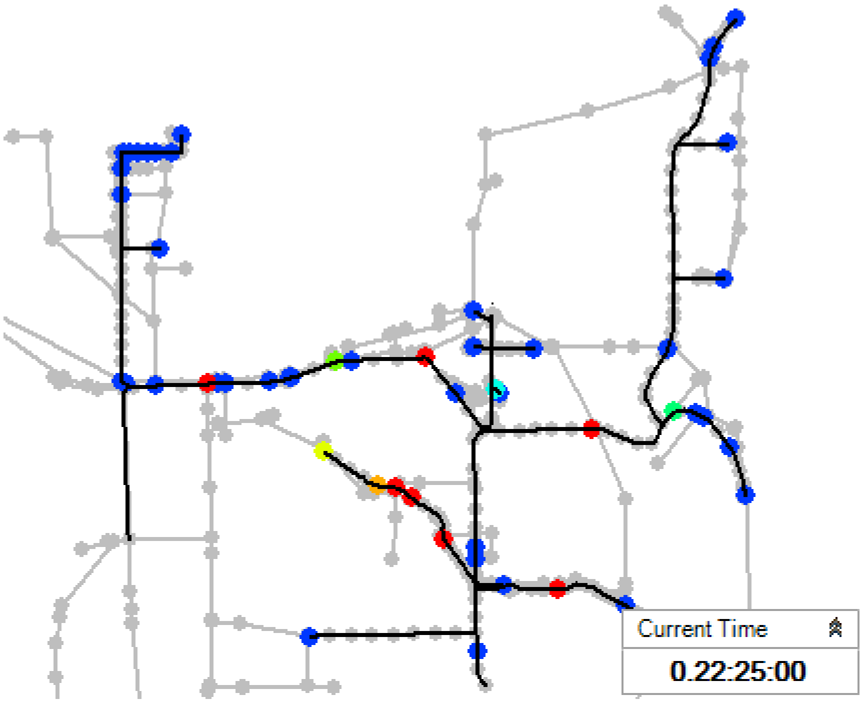


Figure 48: Node color CSO animation

Hide Nodes with No Results

To improve the clarity of results it is helpful to hide nodes on the map that are not CSO locations and the *Hide nodes with no results* checkbox does this.

Upon choosing acceptable options, and then pressing the *Apply* button the map is updated and as the animation of the simulation results progresses, colors/sizes will change according to the user selections.

Visualizing CSO Statistics

3.15.2 CSO Statistics

The second component to CSO analysis is the statistics visualization component which is illustrated in Figure 49. This component consists of tools for visualizing CSO frequency and volume statistics as well as CSO timeseries. This tool does not perform animation; it rather summarizes the entire simulation with the statistics that are displayed.

The CSO statistics tables and map visualization give volume in ***million gallons***. The CSO time stack plots give volume in ***cubic feet***.

There are two variables that are used to analyze CSO events in MetroFlow. The first variable is the CSO frequency which is simply the percentage of the simulation that the CSO location had non-zero flow. The second variable is CSO volume which is the total volume of water that overflowed at a specific location, given in million gallons. In visualizations below, these two variables can be visualized simultaneously by using the node color and size.

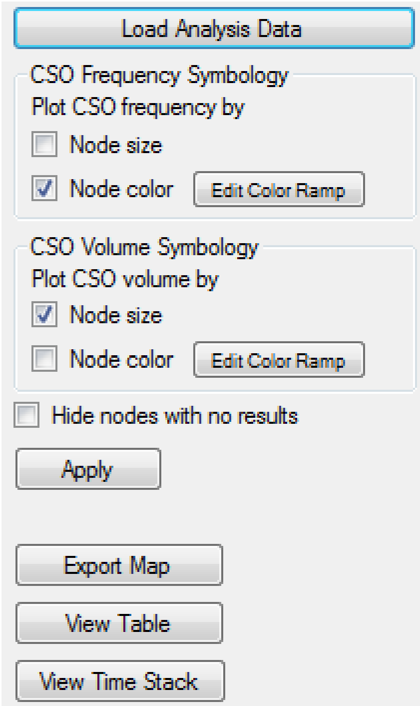


Figure 49: CSO analysis, statistics

In order to perform the statistical analysis, a valid working layer from a loaded scenario must be selected. Until this occurs, the entire CSO analysis tool is disabled. Once enabled, the only element that is initially enabled in the statistical analysis tool is the *Load Analysis Data* button. Pressing that button loads all of the necessary data and performs an analysis on it and enables the rest of the elements in the tool.

The *Load Analysis Data* button loads all of the flow data for all of the nodes in the simulation and analyzes it. The loading process may take multiple minutes but progress should be indicated in the progress bar on the lower-right corner of the main window.

Visualizing the Statistics

The two CSO variables, frequency and volume, can be simultaneously visualized by controlling the checkboxes in the *CSO Frequency Symbology* and *CSO Volume Symbology* group boxes. If a variable has the *Node size* checkbox selected, then the size of nodes on the map will be scaled by applying the specific node's value to the total range of data available. For example, if CSO frequency is symbolized by node size, and the maximum frequency is 100% and the minimum is 0%, then nodes with a frequency of 50% will have a size that is 50% of the maximum node size. Figure 50 illustrates the effect of applying the node size alone. If the *Node color* checkbox is checked then by default the color of nodes on the map will be scaled by the value of the node; the default color scheme is the standard hot-to-cold color ramp.

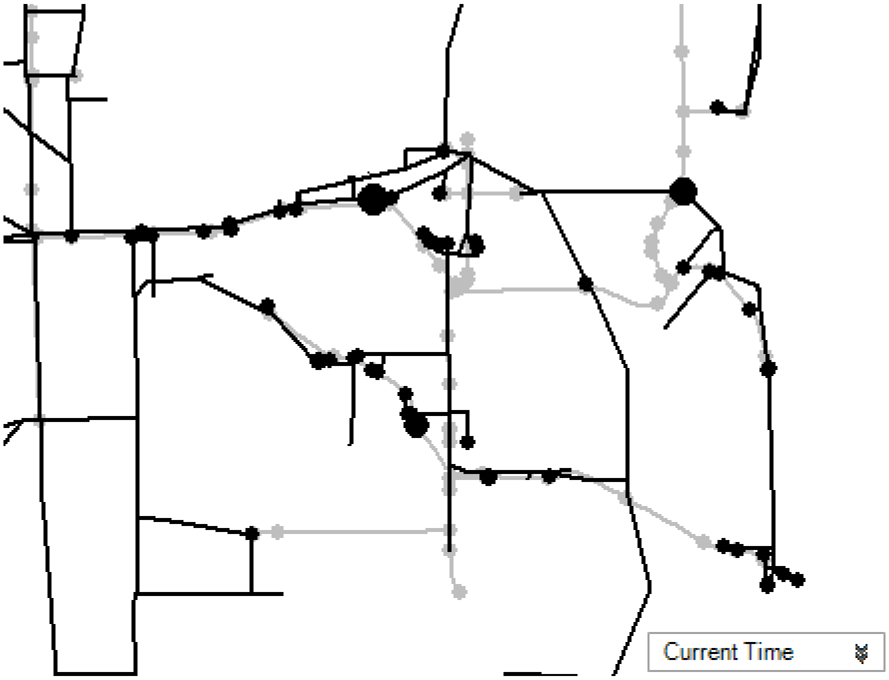


Figure 50: CSO statistics by visualizing volume alone by node size

By combining node size and color for the two variables in various ways, the user can visualize problem areas. For example, if the frequency of CSO events is important, then it is helpful to visualize the frequency variable by node size and volume by color. This is an illustration of this type of visualization. MetroFlow provides the user with the choice of size or color to denote the variable of importance to allow users to answer questions in ways that are most meaningful.

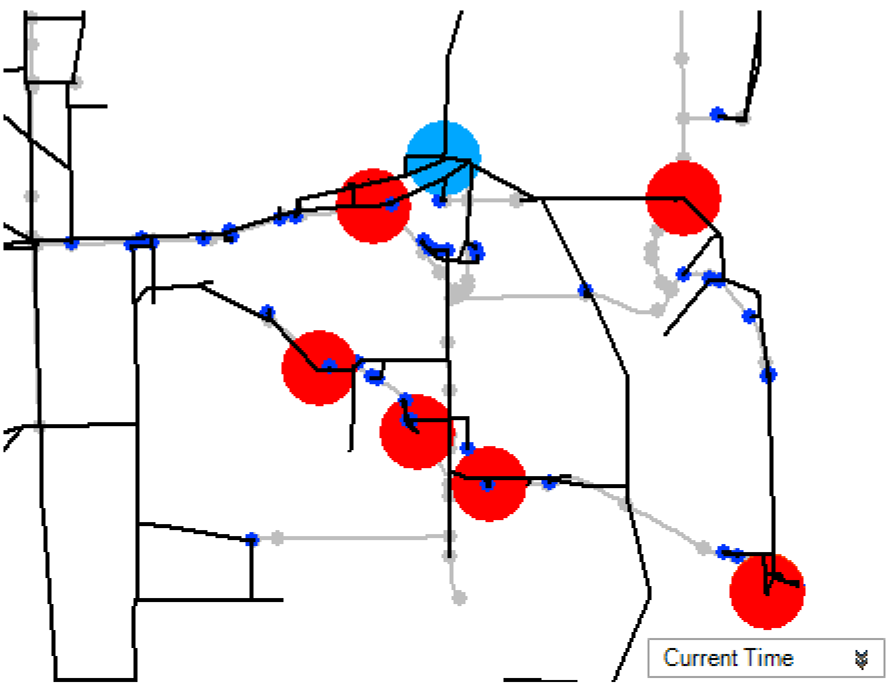


Figure 51: CSO analysis by visualizing frequency by size and volume by color.

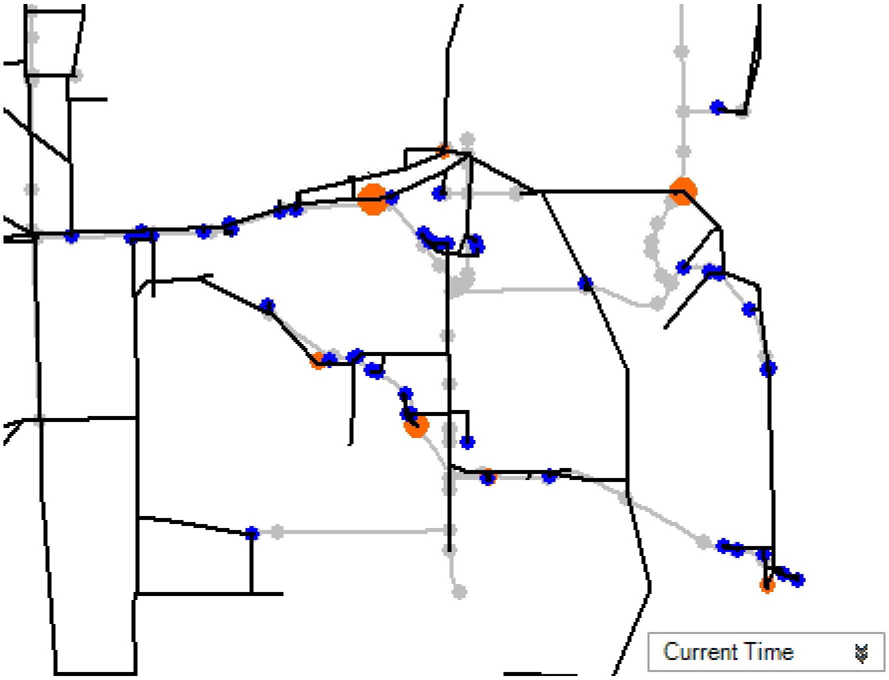


Figure 52: CSO analysis by visualizing frequency by color and volume by size.

Editing the color ramp is a powerful way to view CSO locations that exceed a particular threshold. This can be done by creating a number of breaks (5-10 is a good number) from a data minimum (e.g. 0) to the threshold that is to be visualized; then by applying the hot-to-cold color ramp and closing the color ramp editor. This will allow nodes that have values over the given threshold to be marked as red and others to be varying shades of blue, green, yellow, or orange. The process of using the breaks editor is described in section 3.16 and in the tutorial.

When CSO analysis symbologies are changed, the map is not altered until the *Apply* button is pressed.

Other Options

A tabular view of the values that are used to generate the map visualizations is available by pressing the *View Table* button. The *Hide nodes with no results* checkbox is valuable for cleaning up the visualization by hiding nodes that don't have data. Finally, the *Export Map* button allows the user to save a copy of the map as a bitmap image of 1,200 pixels wide that can be used in reports or presentations.

3.15.3 Interacting with the CSO Statistics via Time Stack Plots

Using Time Stack Plots

In addition to providing the plots used for summarizing CSO events for a simulation, the CSO analysis tool provides the user with the ability to interact with the CSO event timeseries via the *View Time Stack* button. The term *time stack* comes from the use of *time bands* to represent timeseries and the process of stacking many of these together in a column. Each individual *time band* presents an entire hydrograph by using colors from a hot-to-cold color ramp to represent magnitude (see Figure 53).

Results in the time stack plots are given in ***cubic feet***.

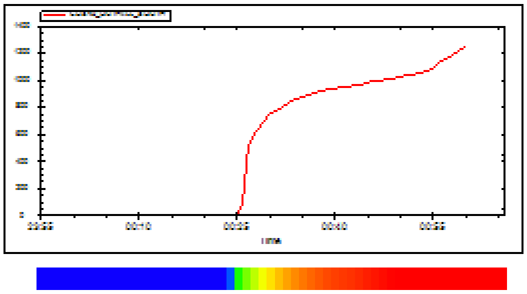


Figure 53: Time band and equivalent hydrograph

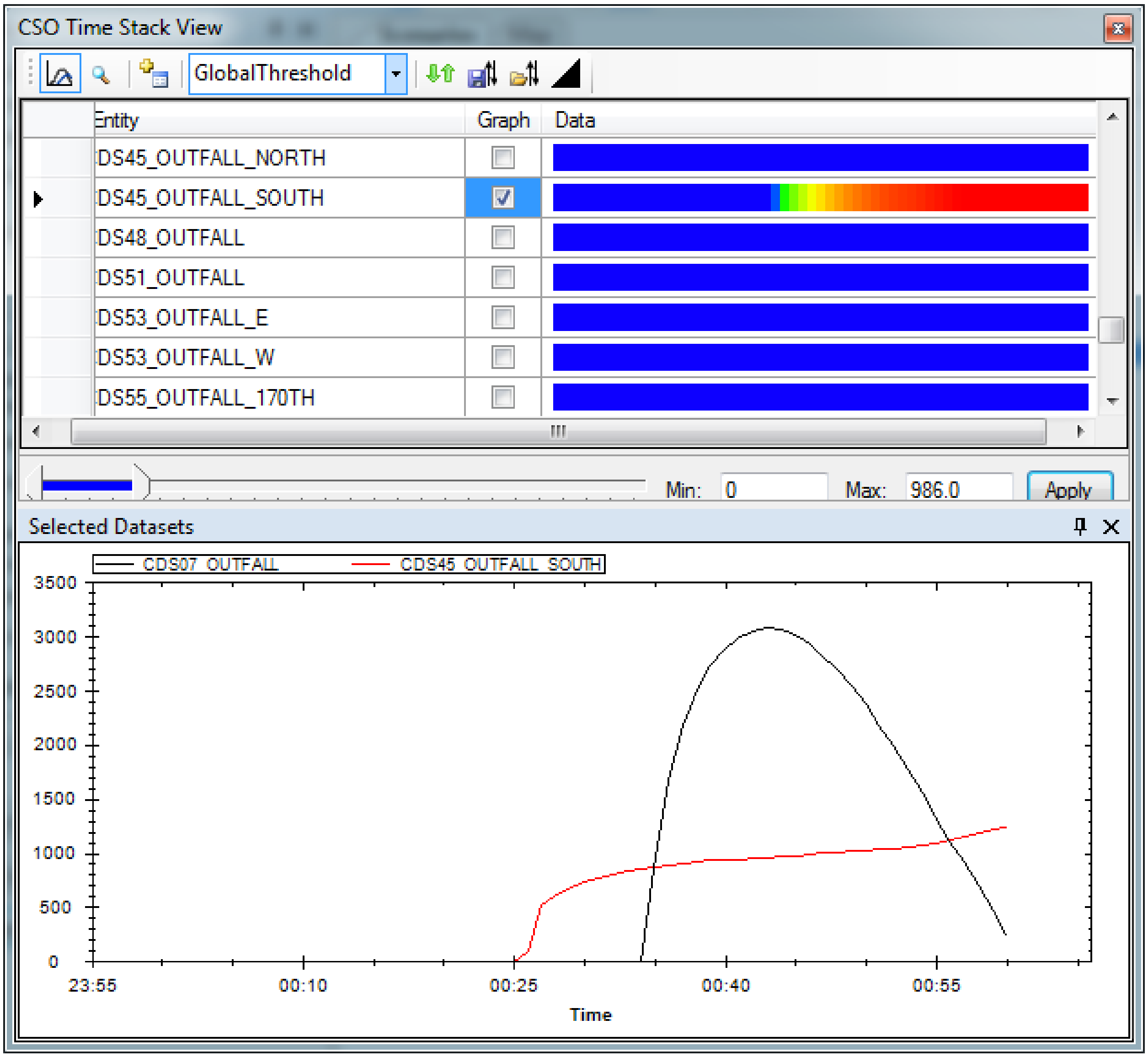


Figure 54: The CSO analysis time stack tool

Normalization

The normalization tool, which is the GlobalThreshold drop-down menu on the toolbar allows the user to normalize the visualization by the following options:

- Global: Global normalization determines the color for a given timeseries point based on the hot-to-cold ramp applied to the minimum and maximum values of all of the timeseries. This option uses the same color ramp mapping for all timeseries.
- Local: Local normalization determines the color for a given timeseries point based on the hot-to-cold ramp applied to the minimum and maximum value of the given timeseries. This option uses a timeseries-specific color ramp mapping.
- GlobalThreshold: The Global Threshold normalization is the same as global, except that maximum value for the global color ramp is determined by the user. When this option is selected the threshold slider is made visible in the middle (or bottom) of the tool window (see Figure 55).

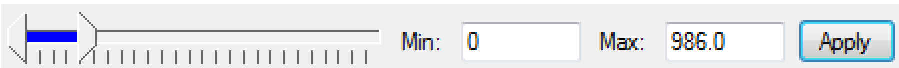


Figure 55: Global threshold slider for the time stack tool

Simultaneous Hydrograph View

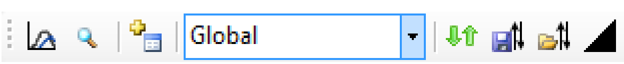
The user may elect to plot a traditional hydrograph for one or more CSO timeseries by clicking the checkbox in the *Graph* column for the desired rows. If the graphing sub-window is not already open at bottom of the tool window, it is opened. More than one CSO timeseries may be selected in which case they are superimposed on each other.

Mouse Interactivity

The user can inspect the values of individual points in the timeseries by turning on the mouse interactivity button. When this is active, an additional column becomes visible that shows the value of the time bands at the point of the mouse. The user can move the mouse across the time bands and see the values change in all of the rows simultaneously. If the graph window is open, a vertical red line is also drawn in the graph window to indicate the mouse position. This is illustrated in Figure 56.

The Toolbar and Other Tools

The time stack tool provides the user with several options for analysis of the CSO timeseries; these options are available from the toolbar at the top of the time stack tool window.



Show/hide timeseries plots window. When the *Graph* checkbox for a given line is checked, the timeseries plots window is automatically shown.

Toggles mouse interactivity. When this is active, an additional column becomes visible that shows the value of the time band at the point of the mouse.

Enable reordering of items by using the mouse; when this is on clicking and dragging a row on the row header will move that row to the position at which the mouse button is released.

If items have been reordered, then the current ordering of rows can be saved to a file for future application to the data set.

Load and apply an external ordering file for reordering the rows based on the order in the file.

Toggle blanking of zero values. Instead of drawing individual points with zero values using the color ramp, they are drawn white.

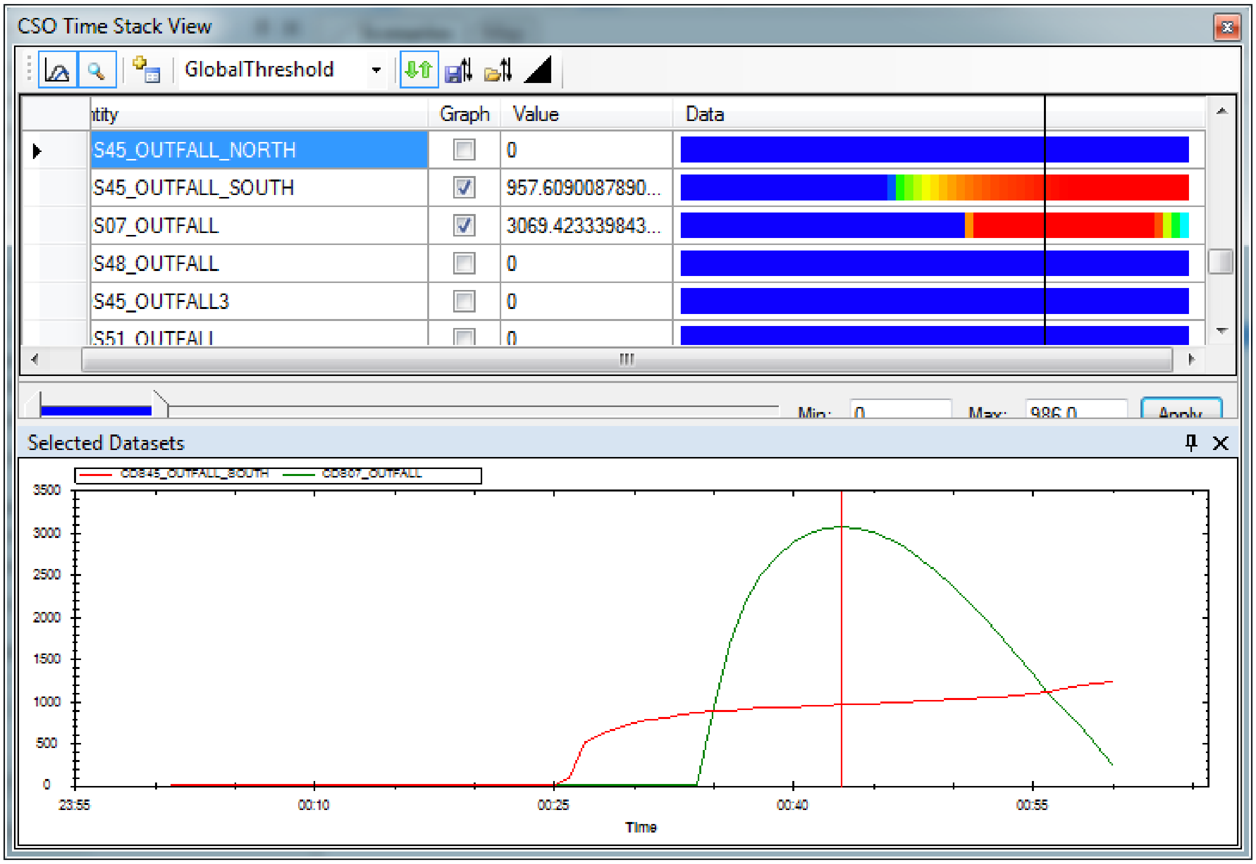


Figure 56: Mouse interactivity in the time stack tool

Using the Breaks Editor

3.16 Breaks Editor

Within the *Breaks Editor*, illustrated in Figure 57, there are many controls available for the user to adjust the color scheme to his or her preference. The preview window displays the color scheme with the associated values. The user can opt to display or make invisible individual colors by checking or unchecking the boxes next to each color. Values for each color field can be edited manually within this preview window, and each color can be edited manually by clicking on the colored box to bring up the color wheel window.

The *Color Ramps* field provides a selection of color schemes that are applied to the preview by selecting *Apply Ramp*. Clicking on the *Colors* tab of the preview window flips the current color scheme so that the colors previously associated with high values become associated with low values, and vice versa.

The *Breaks Creation* fields adjust the criteria values. The *Data Minimum* and *Data Maximum* can be entered in their respective fields. The *Number of Breaks* refers to the number of color intervals. The *Method* field remains defaulted at *Evenly Spaced* so that the range will evenly divide the range from the data minimum to maximum by the number of breaks. Selecting *Create* registers any user changes to the data range or number of intervals.

Finally, selecting *OK* registers the changes to the working map.

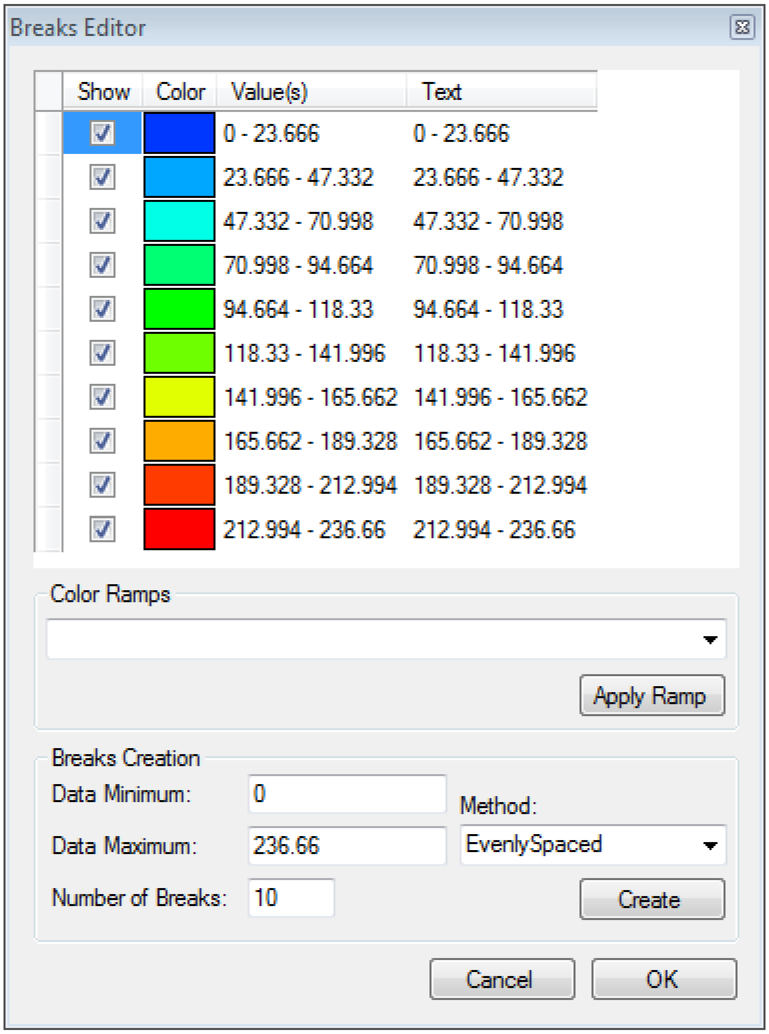


Figure 57: Breaks editor for determining breaks between color gradient

3.17 CSO Display Name Tool

Accessed from the *Tools* menu, the *CSO Display Name Tool* allows the user to change how CSO locations are named within CSO Loading reports. Activating this tool shows it in the right sidebar. Its contents are specific to a working layer and changes with the working layer. The tool displays a grid with two columns: the first column is the actual node ID and the second column is the CSO display name. Either field can be changed but care should be taken to ensure that the changed or added node ID is valid otherwise the changes will not take effect.

To edit an existing display name, click on the name to be changed and simply start typing. In addition, if the user selects an object via the map identify tool, that row will be selected in the grid. To add a new entry, go to the last row in the grid and start typing in the node ID field and then fill in the corresponding display name. To delete one or more rows (this would be used to remove objects from being displayed in the CSO loading tables) select the rows and then press the Delete key on the keyboard.

3.18 Initializing TARP-ITM from TARP-ICAP Results

The TARP-ITM model is a transient model which means that timesteps can become very small during the course of tracking pressurized conditions within the conduits. This translates into extended simulation durations. The ability to initialize TARP-ITM from an existing TARP-ICAP simulation has been included in the latest version of MetroFlow. In order to use this functionality, an ICAP simulation needs to have been run. Once the scenario is run and loaded, the *Results Animation* tool should be used to move the current time to the time at which initialization data should be exported. For example, a user would run a storm in ICAP, open a graph for the reservoir node, and find at which point the filling of the reservoir started to accelerate. The user would use the *Results Animation* tool to move to that time in the simulation.

Once an initialization export time has been established, the user navigates to the *Scenarios* window, and selects the *Export Initialization Data* tool (Figure 58) and selects the *TARP-ITM* option.

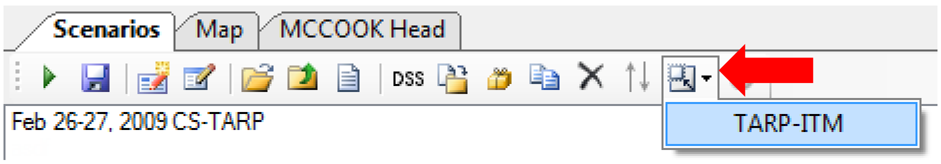


Figure 58: Scenario explorer toolbar with initialization data tool

This creates a file with initialization data in it that can be used in the Scenario Builder. The user now opens Scenario Builder, creates an ITM scenario with the same input timeseries set, and then under Module options, Hotstart, Initialize file, sets the value to the file that was saved previously. The scenario can then be saved and run.

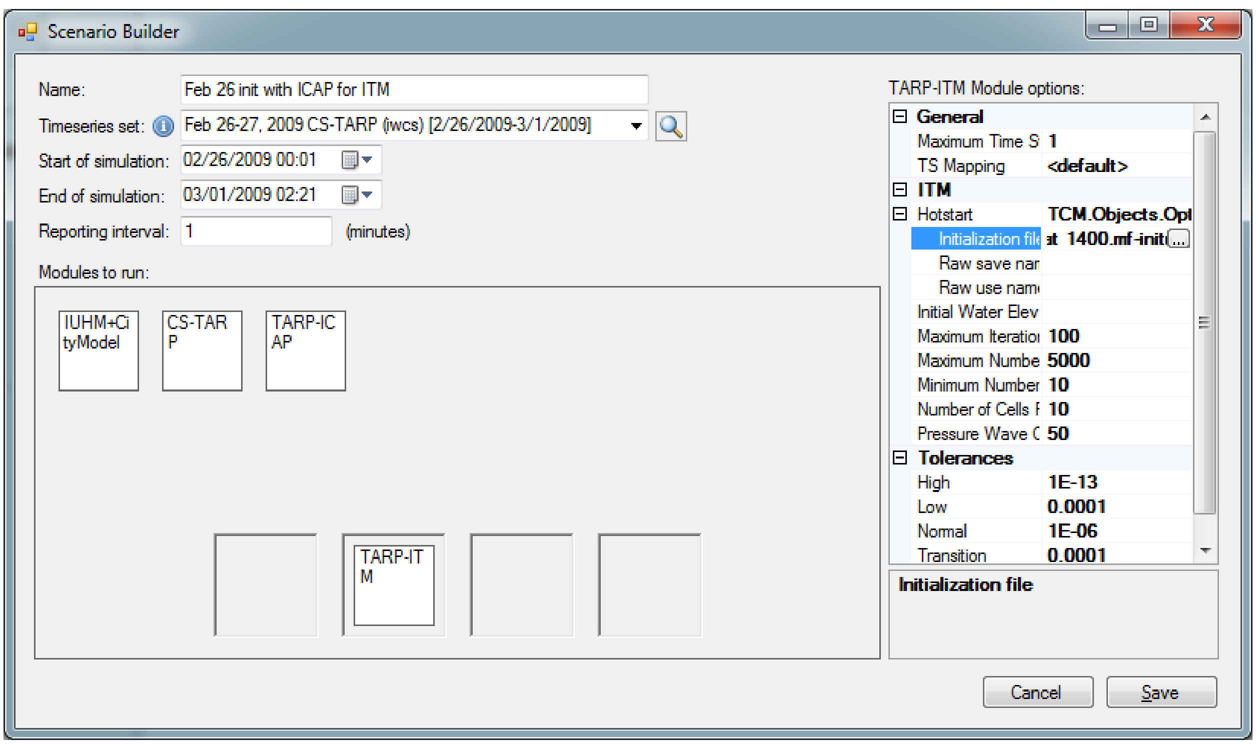


Figure 59: Scenario builder with specifying initialization data

4. QUICKSTART GUIDE

This Quickstart Guide provides a sequence of tutorials outlining the functions of MetroFlow so that the user can quickly grasp an understanding of the interface. For questions of definition or difficulty in navigation, refer back to the Navigation Basics section.

Before beginning, it is helpful to anticipate that rainfall data and output from the hydrologic models (i.e. IUHM and Interceptor models) are stored as sets in the *Timeseries Sets* window. Correspondingly, the *Scenarios* window allows the user to define simulations that will utilize the timeseries sets.

For the purposes of this tutorial, the user will be instructed on how to do the following:

- Create and analyze a generic 1-inch, 1-hour design storm;
- Import historical timeseries for single event storms and process it through the models;
- Perform analysis of conditions with and without a reservoir in place;
- Account for the effect of sluice gates in the Interceptors;
- Import and run entire water years;
- Account for Thorn Creek inflows by modifying timeseries mappings and importing timeseries;
- Account for the various features in the Mainstream/Des Plaines system.

To begin, open MetroFlow from the Start Menu and if this is the first time it has been started, select the calumet.mwp package to open. If tutorial data[1] has not yet been downloaded, it can be obtained from http://hydrolab.illinois.edu/ metroflow-1.7/Calumet-Tutorial-Data.zip.

Figure 60: Map excerpt from Bulletin 70 which illustrates the 10 rainfall frequency sections in Illinois (http://www.isws.illinois.edu/atmos/statecli/RF/fig1-rf.gif)

4.1 Example 1: Modeling a Design Storm

Modeling a Design Storm

This example will guide the user on how to generate a typical design storm and run the storm through various models in MetroFlow (e.g., IUHM, Interceptors, and ICAP).

4.1.1 Design a Generic Storm

To design a generic storm it is recommend to first understand the common intensity-frequency-duration (IDF) for storms in region to be modeled. For the state of Illinois, the Illinois State Water Survey's Bulletin 70 provides important IDF data.

Corresponding IDF values for each section depicted in Figure 60 can be found at http://www.isws.illinois.edu/atmos/statecli/RF/table10.pdf. For more information, refer to the main Illinois State Water Survey site at http://www.isws.illinois.edu/atmos/statecli/RF/rf.htm. In addition, the NOAA National Weather Service maintains IDF data via the Precipitation Frequency Data Server at http://hdsc.nws.noaa.gov/hdsc/pfds/pfds_map_cont.html?bkmrk=il.

For the purpose of this example, the user will design a 5-year; 24-hour storm. Based on IDF values from Bulletin 70 for the Chicago area (i.e., section code 02, Northeast), a 5-year, 24-hour storm has a rainfall depth of 3.80 inches.

[1] The tutorial data may be used for sections 4.1-4.6.

To create this design storm in MetroFlow, open the *Timeseries Sets* window, click the *Create Design Storm* button, and within the dialog select the following options (as depicted in the right image of Figure 61):

> *Distribution Type: Yen-Chow*,
> *Rainfall Depth: 3.80 inches*,
> *Rainfall Duration: 24 hours*,
> *Output Timestep: 1 minute*,
> *Padding Before*: *0 hours,* (leave at default value),
> *Padding After: 3 hours,* (leave at default value),
> *Label: 5-year; 24-hour*
> *Description: 5-year; 24-hour [3.80 inches]*.

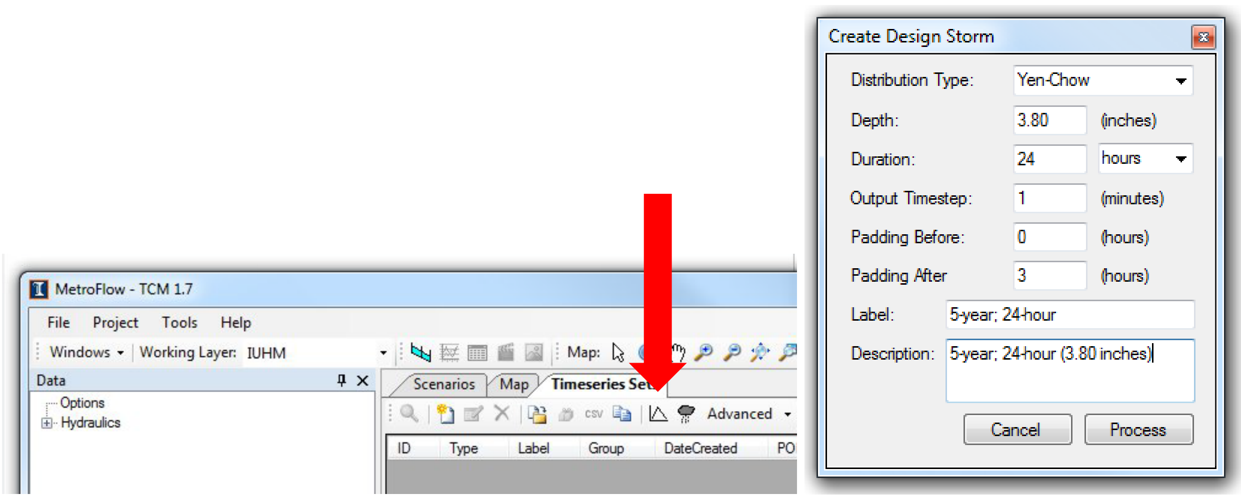


Figure 61: Creating a Design Storm: (left) Click Create Design Storm button; (right) Create Design Storm dialog.

Once all the field values have been entered, press the *Process* button to create and register the storm as a new set in the *Timeseries Sets* window, as illustrated in below in Figure 62.

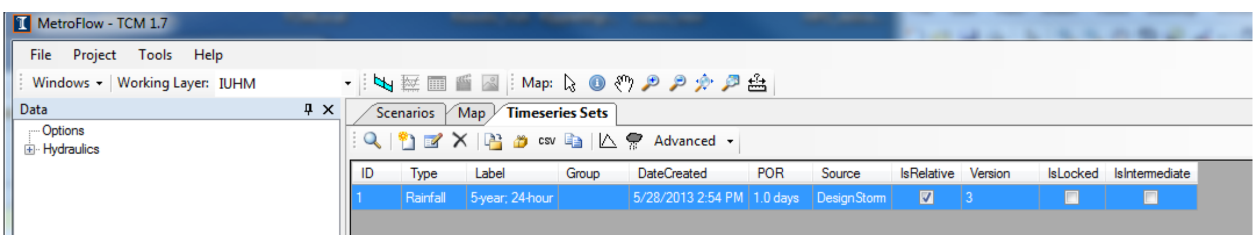


Figure 62: Timeseries Set entry of 5-year; 24-hour design storm

Creating a Scenario

4.1.2 Create a Scenario Using the Design Storm

Open the Scenarios window by the navigating to the *Windows* menu and clicking *Scenarios*. In the *Scenarios* window, click the *New* scenario button ( ) to launch the Scenario Builder. Within the *Scenario builder*, create an IUHM, Interceptors, and TARP-ICAP model scenario by selecting each corresponding icon and dragging it into the respective slot of the execution sequence, as shown in Name the scenario "Example01_5-year; 24-hour", and select the timeseries set "(R) 5-year; 24-hour [1.0 days]" from the dropdown menu; this is the design storm created in the prior subsection.

The timeseries set input will only appear after the IUHM module is dragged into the slot. Since the selected model is a hydrologic model (IUHM), the available timeseries are rainfall sets. Otherwise, sets of hydrographs would be available.

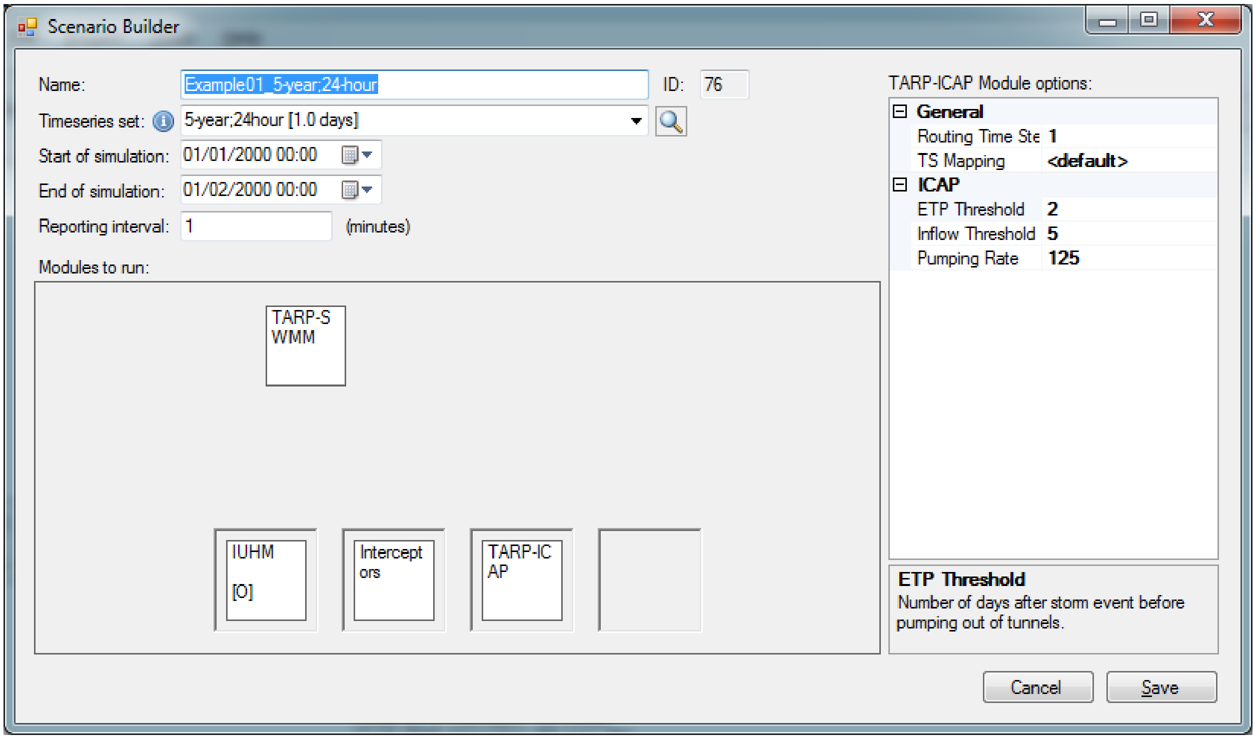


Figure 63: Scenario Builder dialog with setting to run IUHM, Interceptors, and TARP-ICAP modules for 5-year; 24-hour design storm

The selection of a timeseries should automatically change the *Start* and *End* of simulation fields to match the dataset, which in this case is the generic 01/01/2000 00:00 to 01/02/2000 0:00 for design storms. These can be manually altered to reduce the period that is being simulated. The *Reporting interval* value will be left at 1 minute for this example and does not correspond to the interval of the rainfall data that was previously ended when creating the design storm. This interval is simply the value at which resulting output will be reported by the program. The reporting interval value should typically be 1 minute for historical events such as single storms but should be larger for longer term simulations, e.g., for water year events, the interval should be 60 minutes.

Module options, located on the right side of the *Scenario Builder* window, are available for each model. These primarily control output names for models that provide output, as well as simulation options. For IUHM, copy the scenario name “IUHM Example01_5-year; 24-hour” in the *Label* and *Description* fields, and follow a similar naming convention for the label and description of the Interceptor module, e.g., “Inter Example01_5-year; 24-hour”. Set the *Routing Time Step* to 1 second, and the *Conduit Lengthening Step* to 0 seconds, but leave the *I&I Factor* at 1. The reporting interval should be left at 1 minute. Note that since both IUHM and Interceptor models were included in the simulation, timeseries sets will be generated for each of these models and the names of those sets will be given the labels as given by the user. They will also share a common *Group* field value in the timeseries set which will be the scenario name. This is to provide the user with a way to group output timeseries set outputs from model runs in the *Timeseries Sets* window.

Select *Save* to register the scenario. The scenario that was just created should now appear on the list of scenarios in the *Scenario* tab with its given scenario name as seen in Figure 64.

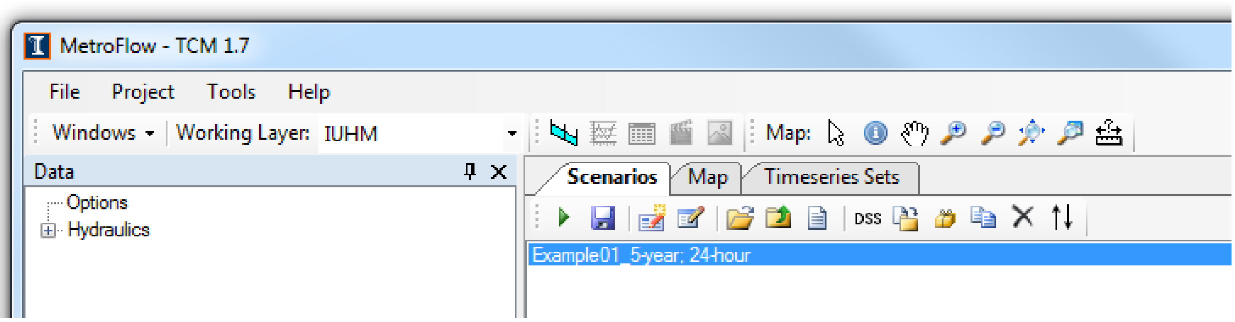


Figure 64: List of registered scenarios depicting that Example 01 was saved.

4.1.3 Running the Example 1 Scenario for the Design Storm

To run the scenario, select the scenario that was created in the previous section and click the *Run* button. A separate window will appear illustrating the progress of the run. Simulations make take anywhere from minutes to several hours to run. For example, IUHM simulations of an entire water year may take 10-24 hours; interceptor simulations of an entire water year may take a few days; ICAP simulations of the same water year make take as little as five minutes. For this example, the IUHM, Interceptors, and ICAP modules took approximately 23 min, 4.5 min, and 4 min, respectively, on an Intel i5 3.0 GHz processor. The simulation window will remain on top of all other windows on the computer; if this is not desired then pressing the *Minimize* button will hide this window and the progress indicator on the main window will display the simulation progress.

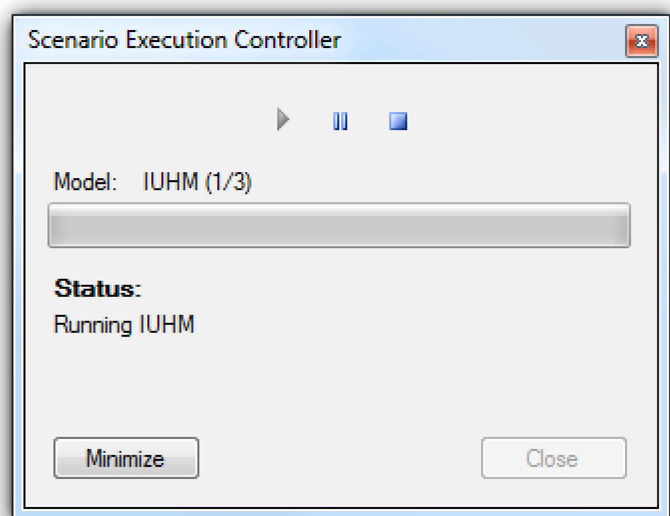


Figure 65: Scenario execution controller window

4.1.4 Run Competition and Quality Check of Example 01 Scenario for the Design Storm

Once the run has stopped, after running either successfully or unsuccessfully, click the *View Report* tool in the *Scenarios* window and select the model type in the *Model List* (i.e. iuhm). The result is shown in Figure 67. If multiple models are in the scenario the Model List will be populated with those models. If the run was unsuccessful, this will open the error log to indicate why the run did not work. If the scenario succeeded in running the model, the report will display summary information of the model run.

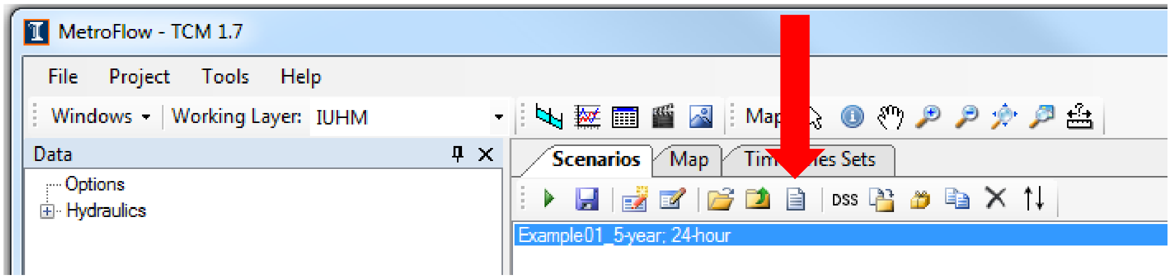


Figure 66: *View Report* tool to display results of models

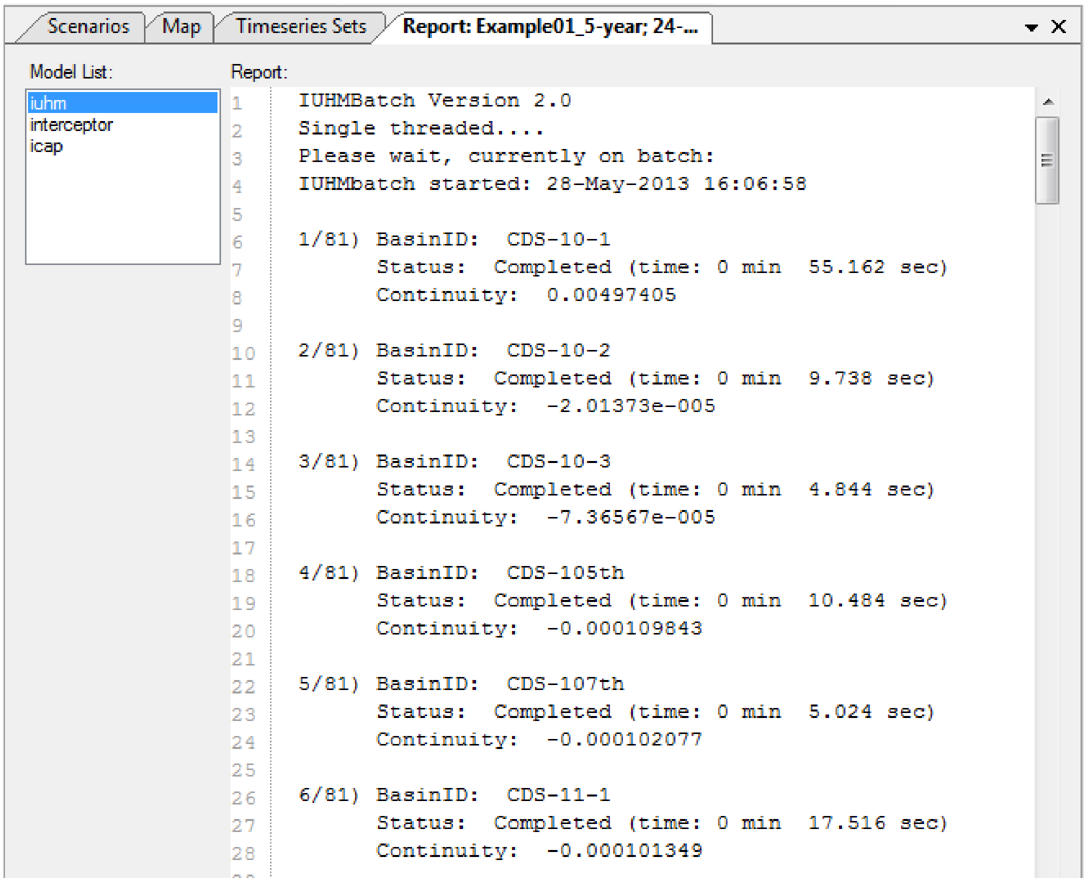


Figure 67: The simulation report window displaying IUHM model results

Quick Quality Checks for IUHM Report

The IUHM report lists each of the basins with processing time and a continuity error. A few quick quality indicators include checking for outlier continuity errors, and scrolling to the bottom of the report to check the IUHMBatch Status that 100% of the basins were processed.

Continuity error is (inflow – infiltration – outflow) divided by inflow. A continuity error of 1 means that 100% of the inflow was lost during computation. Acceptable continuity errors should be on the order of 5% or less. Excessive continuity errors don't necessarily mean that the results are invalid.

Quick Quality Checks for Interceptor and TARP-SWMM Reports

Once the run has finished, view the report to perform a quick quality check. The *Continuity Error* in the *Flow Routing Continuity* section should be lower than a magnitude of 5%, the lower the better (for example, see Figure 68). The SWMM 5 user manual, available at the EPA website http://www.epa.gov/nrmrl/wswrd/wq/models/swmm/, describes quality parameters.

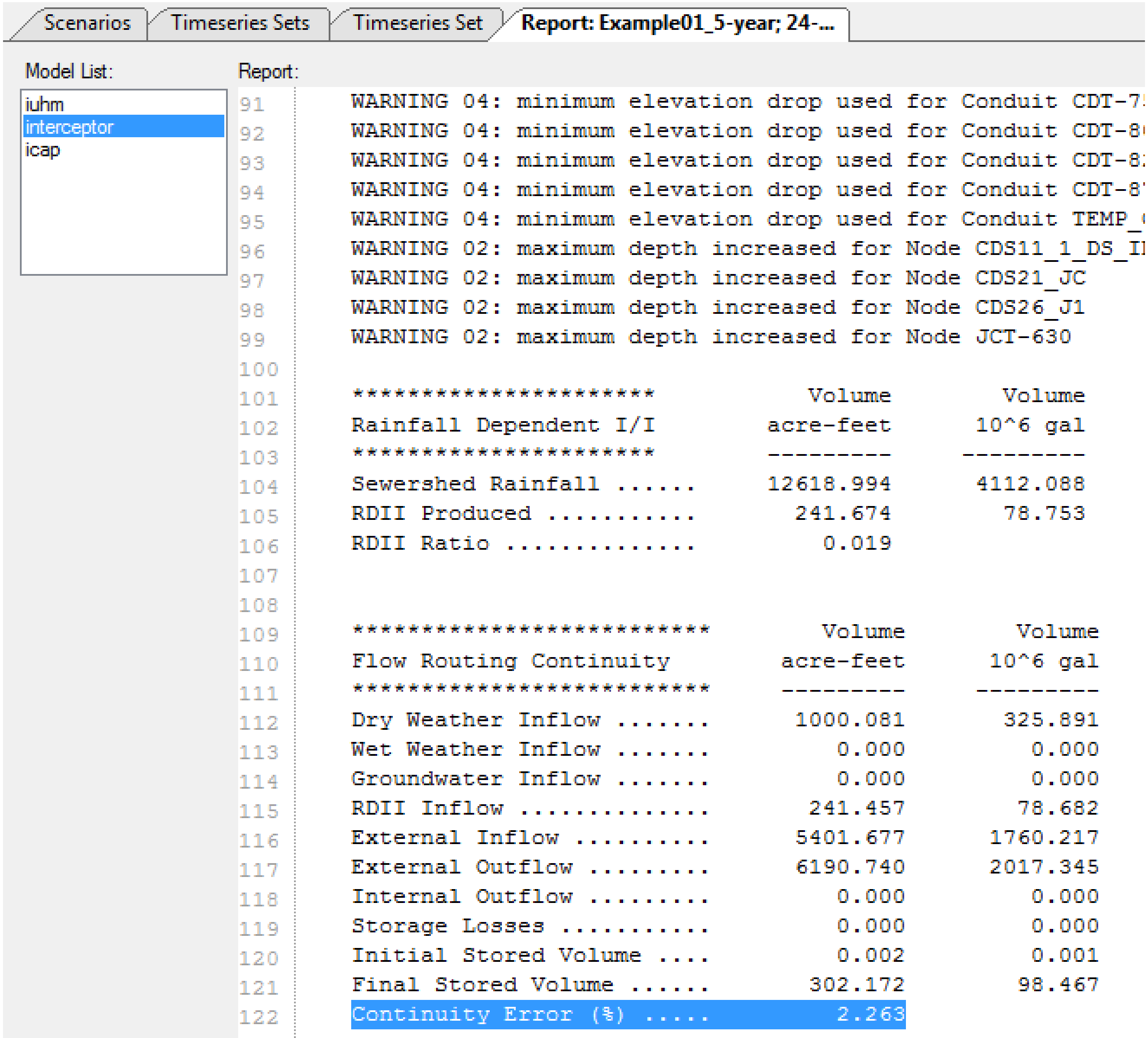


Figure 68: The simulation report window displaying Interceptor model results

Quick quality checks for ICAP report

Similar to the IUHM and Interceptor reports, continuity error is a way of quantifying the simulation accuracy. The continuity error is stored in the report file under the *Flow Routing Continuity* section. Again, 5% or less for an individual event is acceptable. Longer simulations with many pumping events may have larger continuity error.

4.1.5 Viewing Results of Example 01 Scenario for the Design Storm

The scenario that was executed will still appear listed in the scenario window. It is important to ensure the desired scenario results are loaded, check the *status bar* at the bottom left of the MetroFlow window, and use the *Load* ( ) and *Unload current scenario* ( ) buttons to ensure the desired scenario is loaded. Additionally, IUHM and Interceptors models create timeseries set listed in the *Timeseries Sets* window, in this example titled "IUHM Example01_5-year; 24-hour" and "Inter Example01_5-year; 24-hour", respectively.

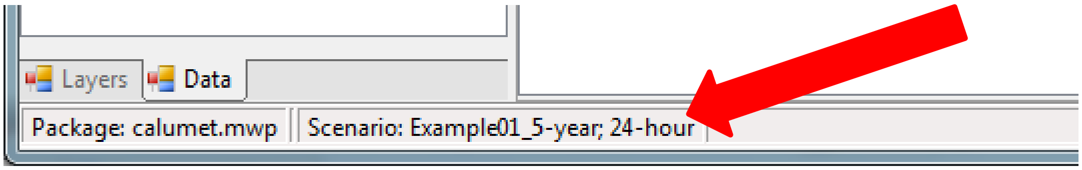


Figure 69: Status bar which denotes the current package and scenario in use

Timeseries Analysis of IUHM Results

Navigate to the *Timeseries Sets* window. There are three timeseries sets that correlate to this example scenario: the Rainfall set "5-year; 24-hour" (the input set) and the "IUHM Example01_5-year; 24-hour" and "Inter Example01_5-year; 24-hour" (the output sets from each model).

Select the Rainfall set and click *View* button ( ).

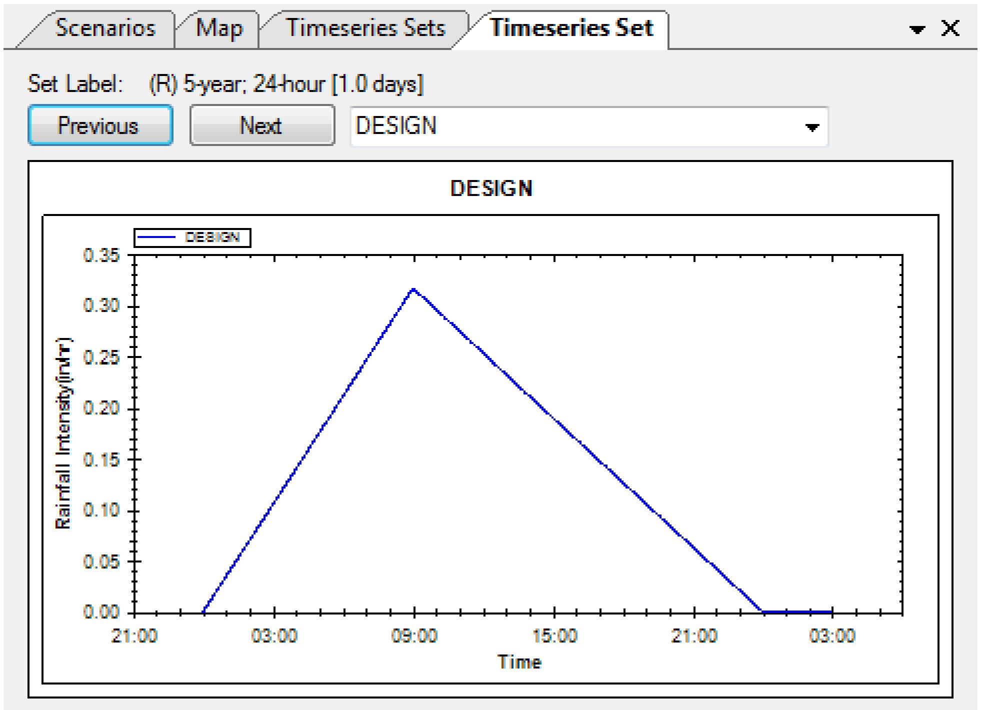


Figure 70: Graph of plot of Example 01 rainfall set for design storm

A graph will appear in the main window depicting the rainfall hyetograph. Because this is a simple 3.80 in, 24-hr design storm, the rainfall is assumed to be spatially uniform over all of the basins in the model, and therefore there is only one hyetograph titled DESIGN (as shown) for the whole model. Right-clicking the plot displays available plot options such as *Copy*, *Print*, and *Show point values*. Click on the *Show Point Values* option and then move the mouse over the peak in the hyetograph. The date of the peak and the value at the peak will be shown as the mouse moves along the plot.

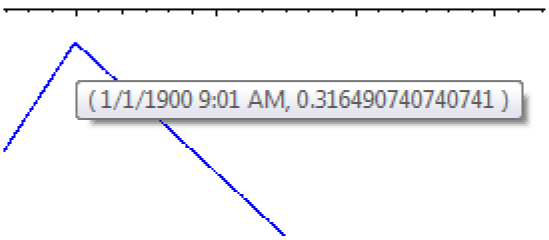


Figure 71: Show Point Values option for plots

Next, select the "IUHM Example01_5-year; 24-hour" timeseries and click *View* ( ). This time, a set of hydrographs are available from the plot drop-down menu, because this timeseries is the output of the IUHM model. There will be a hydrograph for each subcatchment in the model. Again, right-clicking on the plot displays available plot options such as *Copy* and *Print.*

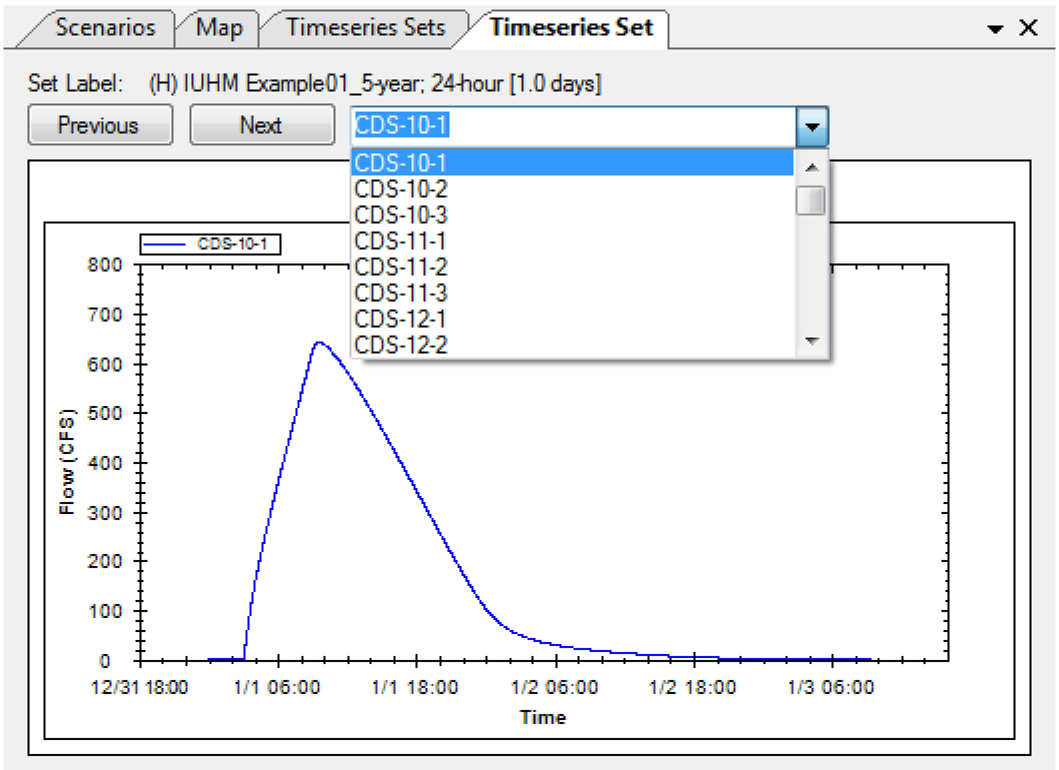


Figure 72: Output hydrograph from IUHM timeseries set

Timeseries Analysis of Interceptor Results

Set the working layer to *Interceptor*. Next, navigate to the *Timeseries Sets* window. Select the "Inter Example01_5-year; 24-hour" timeseries and click *View* button. The output from the Interceptor model will appear as a set of hydrographs in the center window, one plot for each dropshaft. Some of these hydrographs will show no flow which indicates no water went to that particular dropshaft. Again, right-clicking on the plots provides the user functional plot options such as *Show Point Values* and *Copy*.

Overview of Interceptor Model Results for CSOs

To help clarify elements on the map, we will first enable node labels. With Interceptors selected as the working layer, navigate to the *Layers* window. To make the visualization cleaner, uncheck the IUHM option under *Layer Visibility*. Then set the *Node Labeling* to *Name* to display the junctions, and set the *Symbology* and *Link* fields to *None*, as seen in Figure 73.

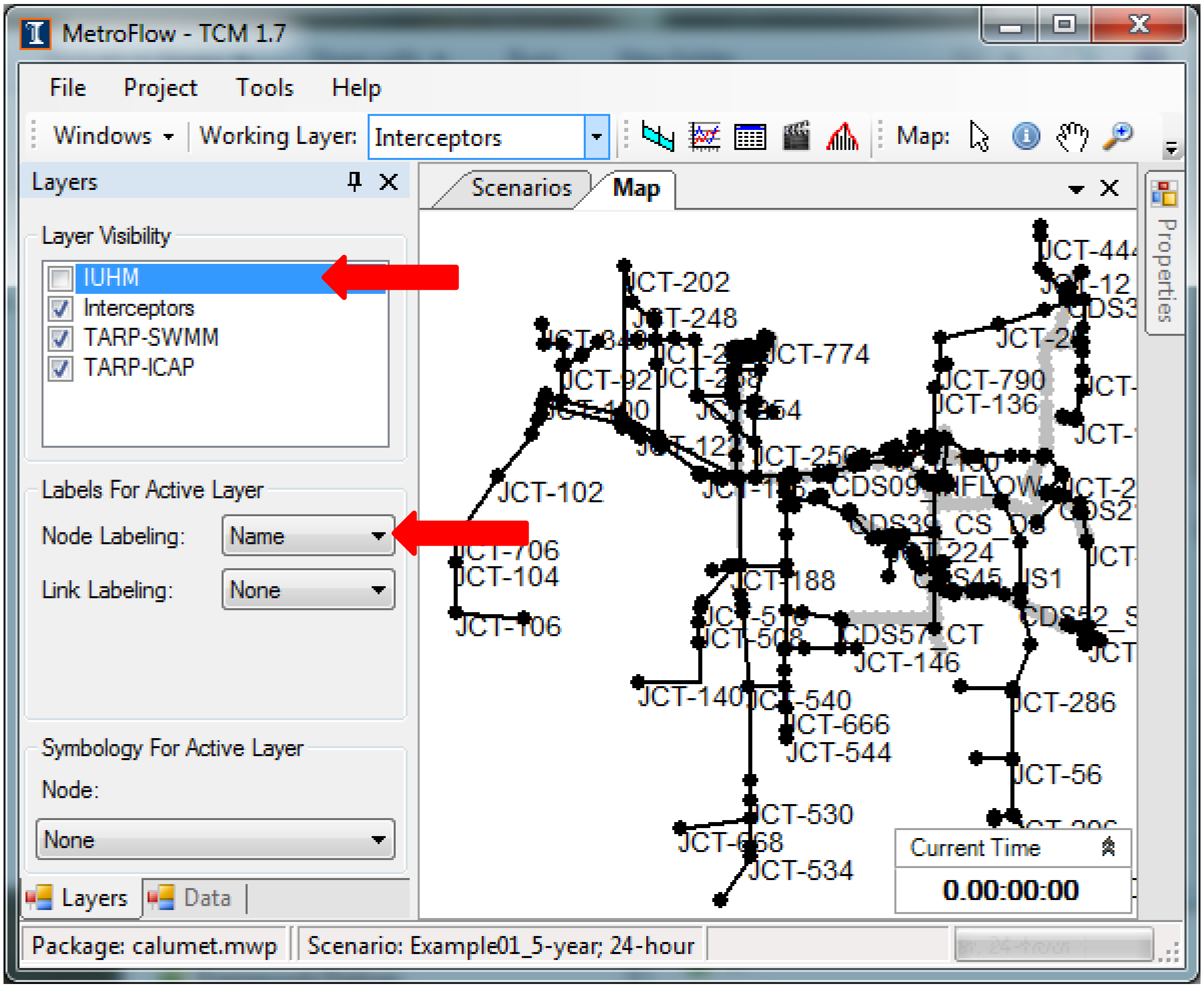


Figure 73: Enabling Node Labeling for active layer

Next, press the *CSO Analysis* icon ( ). This tool will allow graphical analysis of both frequency and volume of CSOs for the entire model duration which is useful to provide key insight into problematic areas in the system. In the *CSO Analysis* window press the *Load Analysis Data* button and set *Plot CSO frequency by Node Size* and *Plot CSO volume by Node color*. Then, press the *Apply* button and you should see results similar to Figure 74. Note that by zooming in to the map, the lighter blue circle is located at the node labeled “PARNELL_OUTFALL”.

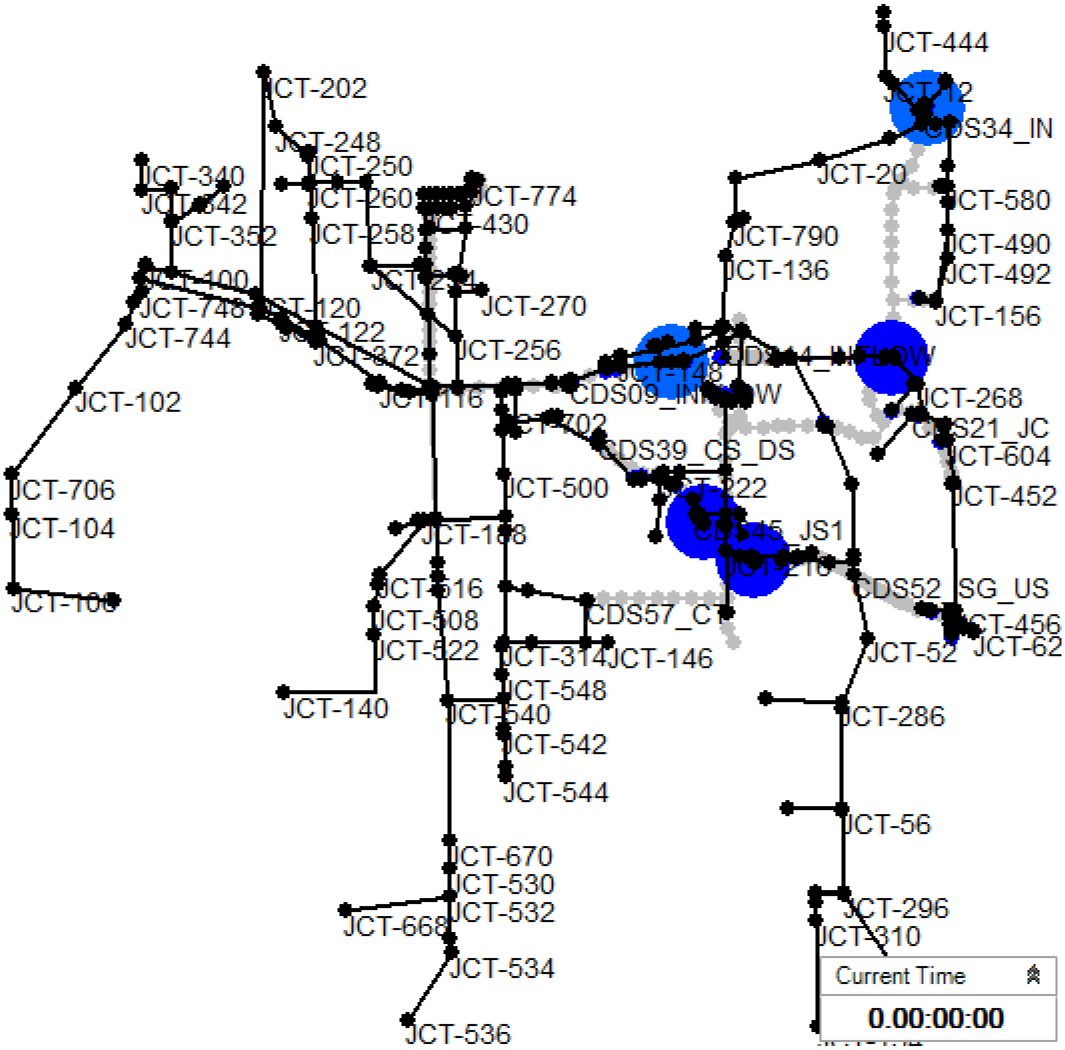


Figure 74: CSO Analysis of Example01 interceptor model

The default color ramp doesn't always provide the best mapping of colors to values. Click the *Edit Color Ramp* button in the *CSO Volume Symbology* box. In order to mark all nodes with a total CSO volume over 5 million gallons with red, put in 5 in the *Data Maximum* field and then press the *Create* button in the *Breaks Creation* box. The color grid will turn to black and will give a set of breaks between 0 and 5 evenly-spaced by intervals of 0.5. Click the *Color Ramps* drop down and select the standard hot-to-cold color ramp . Press the *Apply Ramp* button and then the OK button to return the *CSO Analysis* window. There, check the box *Hide nodes with no results* and press the *Apply* button. Figure 75 illustrates the results of these changes.

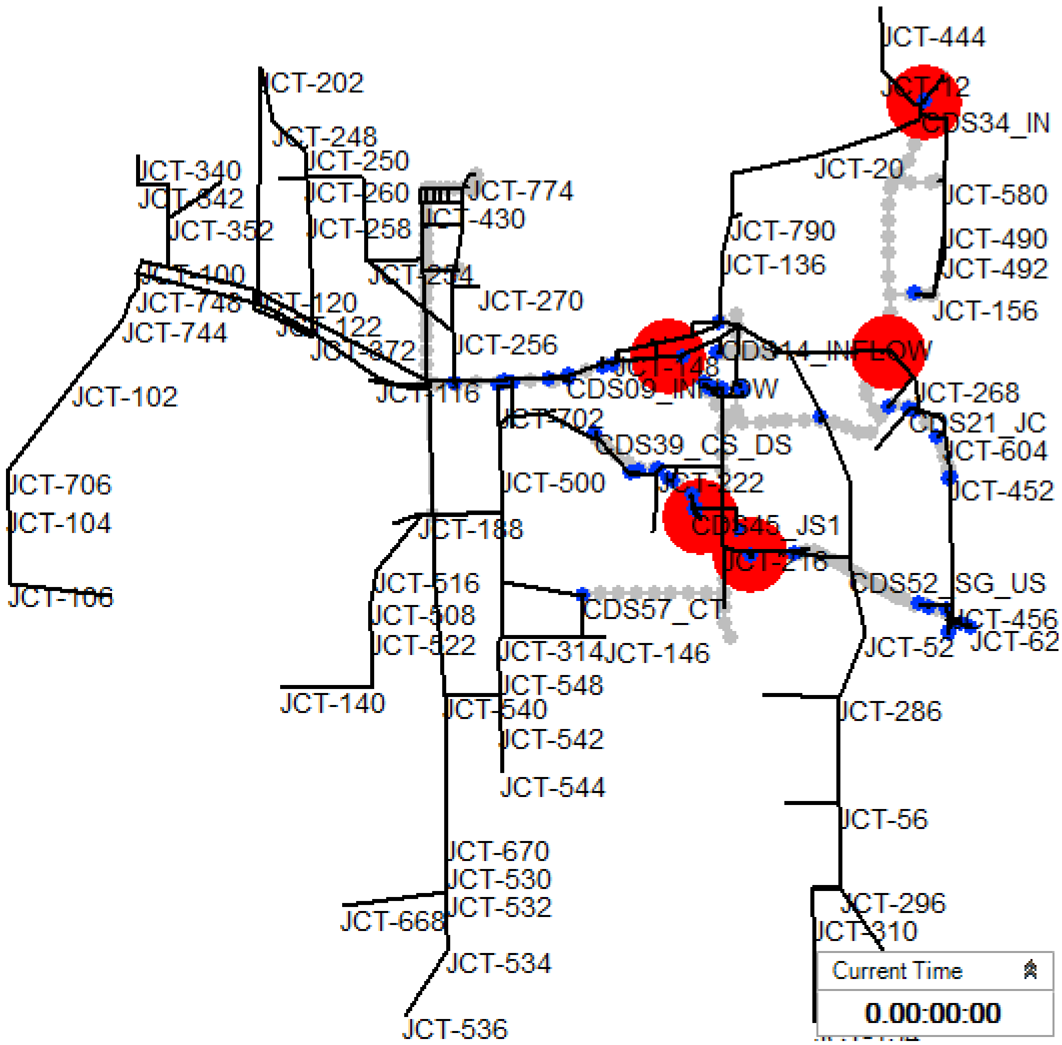


Figure 75: The CSO statistics after applying a threshold to the color ramp

To better visualize each outfall over time, press the *View Time Stack* button. In the *CSO Time Stack View* window scroll down and select the *Graph* check box for the following outfalls: PARNELL_OUTFALL, UNKOWN_OUTFALL, and CDS45_OUTFALL_SOUTH (see Figure 76). Also, select the *Mouse Preview* button (🔍) to be able to probe the data and determine the time and magnitude of the peaks, then close the *CSO Time Stack View* window.

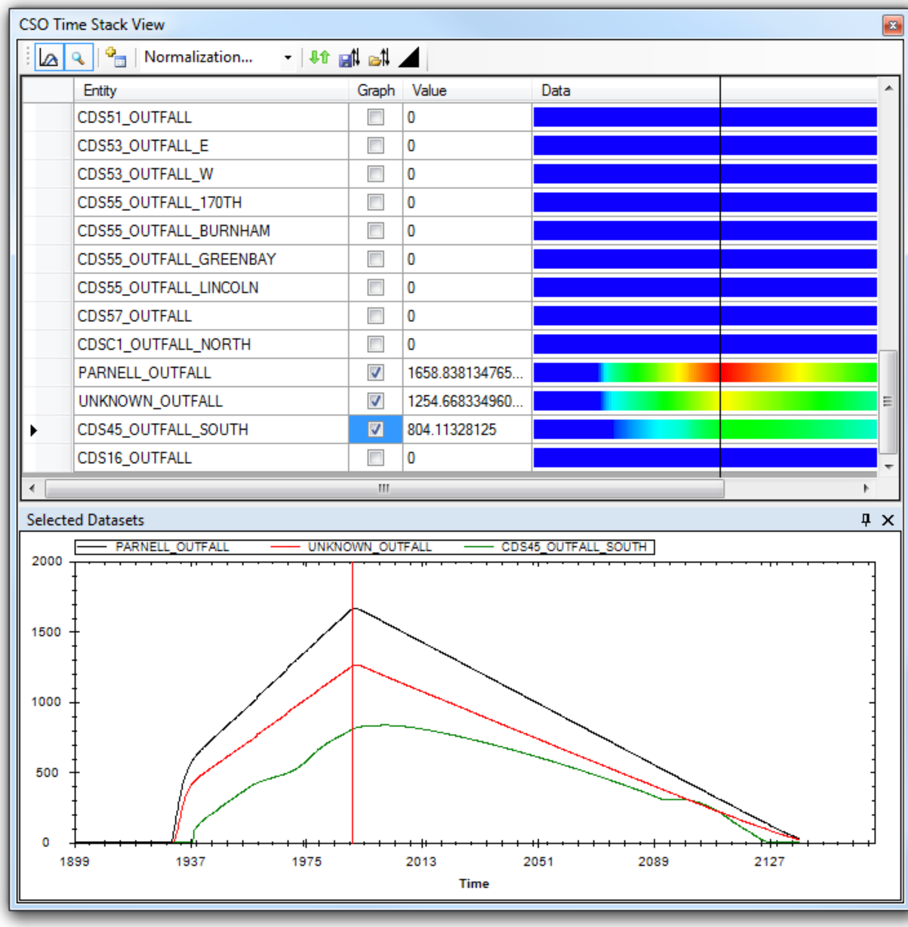


Figure 76: CSO Time Stack View window visualizing Example01 interceptor results

Further Analysis of Interceptor Results

With an overview from the CSO tool, one can now dive deeper into the analysis and probe the system to determine values associated with systems variables (e.g., Node Depth, Head, Volume, and Flow). For the purpose of this example, we will focus on the PARNELL_OUTFALL node as our region of interest. First, go to the *Map* window and zoom out to see the entire extents. Then, go to the *Data* window, and select *Hydraulics >> Nodes >> Outfalls*, then select the PARNELL_OUTFALL from the list to highlight it and then double click PARNELL_OUTFALL to see an aqua crosshair indicate its location on the map (see Figure 77). Zoom in to the PARNELL_OUTFALL node by click on *Zoom* located just above the Outfalls list (see Figure 77). Then, position you mouse in the *Map* window and zoom out a slightly to see the surrounding nodes by holding down the Ctrl key on the keyboard and scrolling your mouse wheel down. Next to the *Zoom* button, press the *Select* button (making sure the PANELL_OUTFALL node is still highlighted), this will add the node to the clipboard for use in the next subsection.

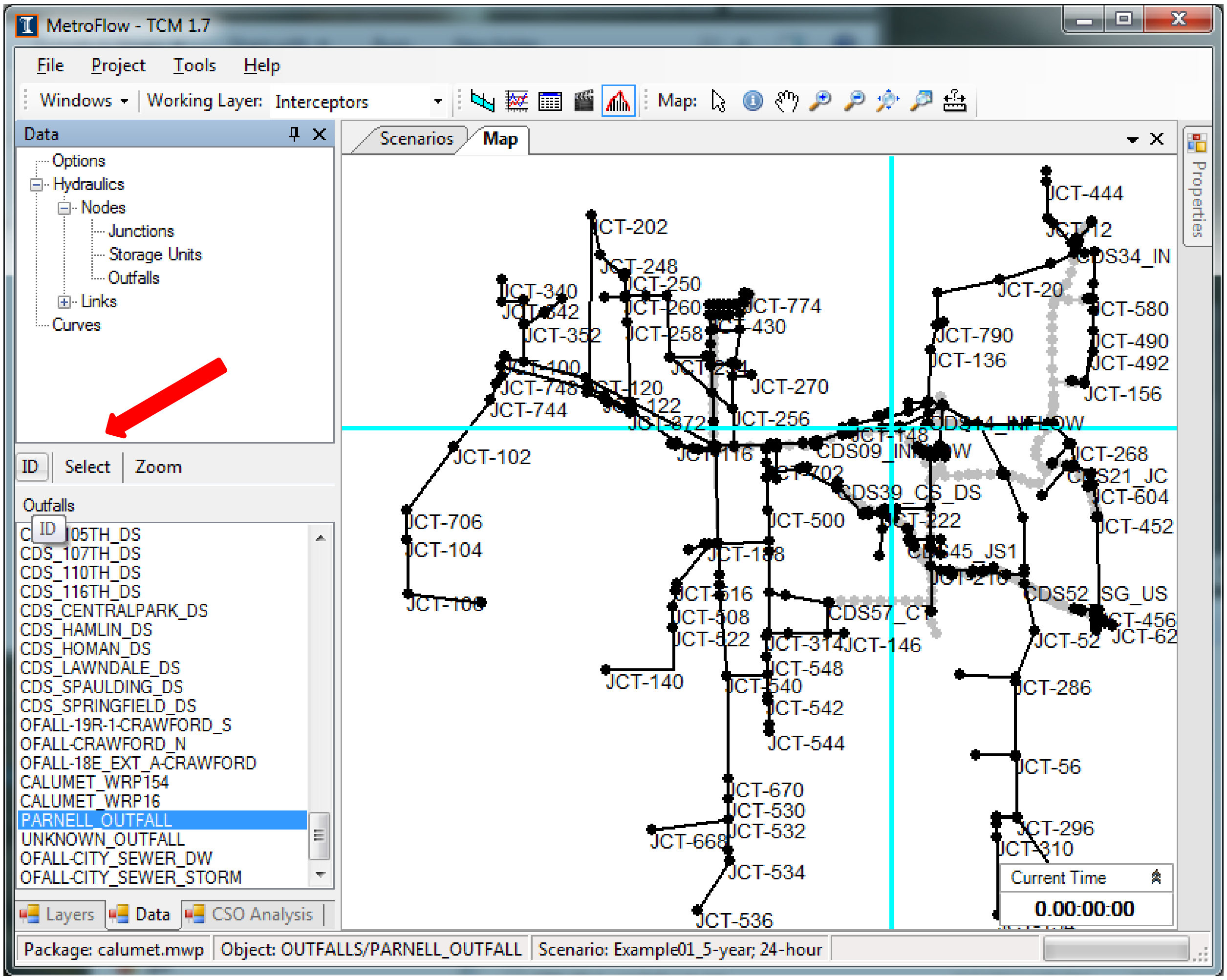


Figure 77: PARNELL_OUTFALL selected in the Data window with aqua crosshairs indicating its location. Feature buttons (ID, Select and Zoom) for selected elements indicated by the red arrow.

Tabular View

Select the *Table* tool (▦) from the *Results* toolbar. Since we previously selected PARNELL_OUTFALL using the *Select* button, click the "+" button in the *Table* window. (An alternative method of selection is to select PARNELL_OUTFALL with the cursor on the Map, and then click the plus "+" button in the *Table* window to register the node). Select *NodeTotalFlow* as the *Variable* and then click *Show Table*.

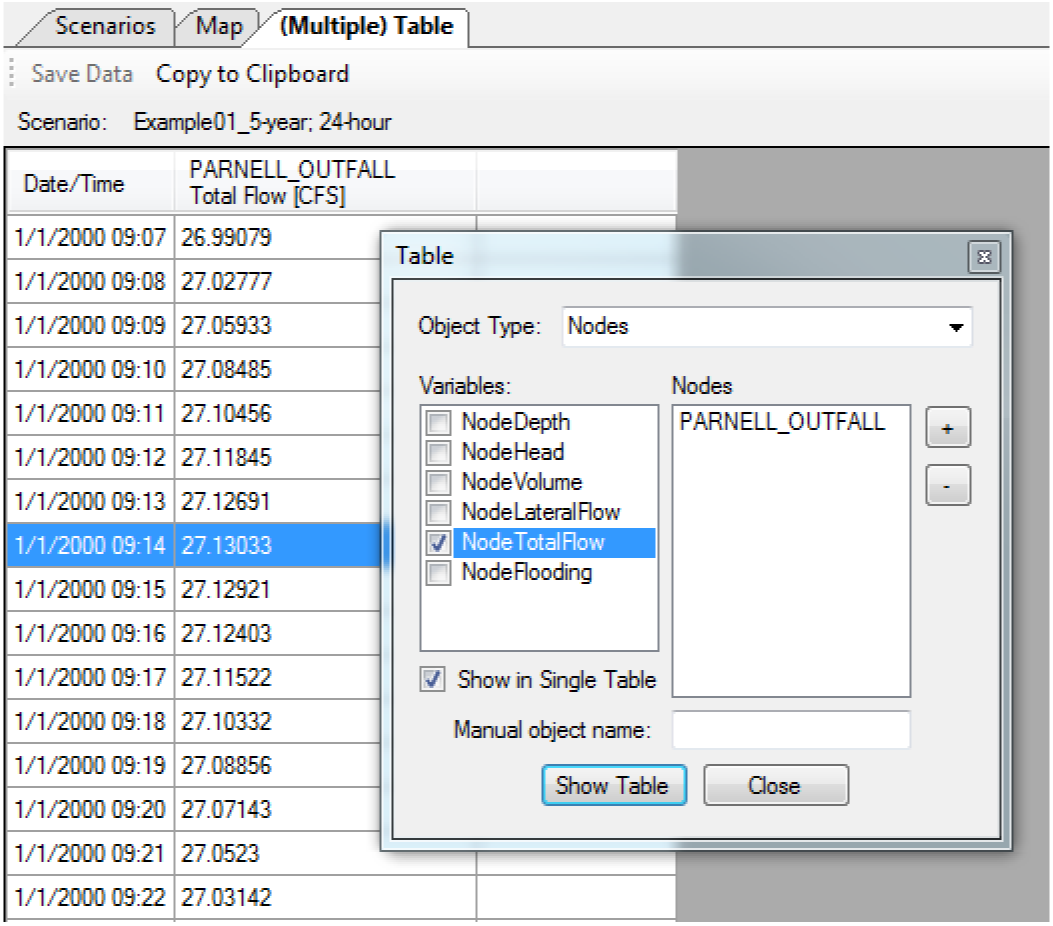


Figure 78: Tabular results window for Example01 Interceptor simulation

Within the table, the time is displayed in HH:MM format. Scroll down to 9:14. The TotalFlow value is 27.75 cfs which corresponds to the maximum value previously observed in the CSO tool.

Graphical View

Select the *Graph tool* ( ) from the Results toolbar. Select node PARNELL_OUTFALL and nearby node JCT-836.

One click on a node will open the *Properties window*, where the name of the selected feature is displayed. Some outfalls are very close in proximity to junctions, misdirecting your selection. Therefore, reference the *Properties window* to help you correctly select nodes and other features.

Again, select *NodeTotalFlow* as the *Variable.* Check the box for *Overlay Objects* to plot both of the features on the same graph, otherwise two individual graphs will appear. Select *Plot*.

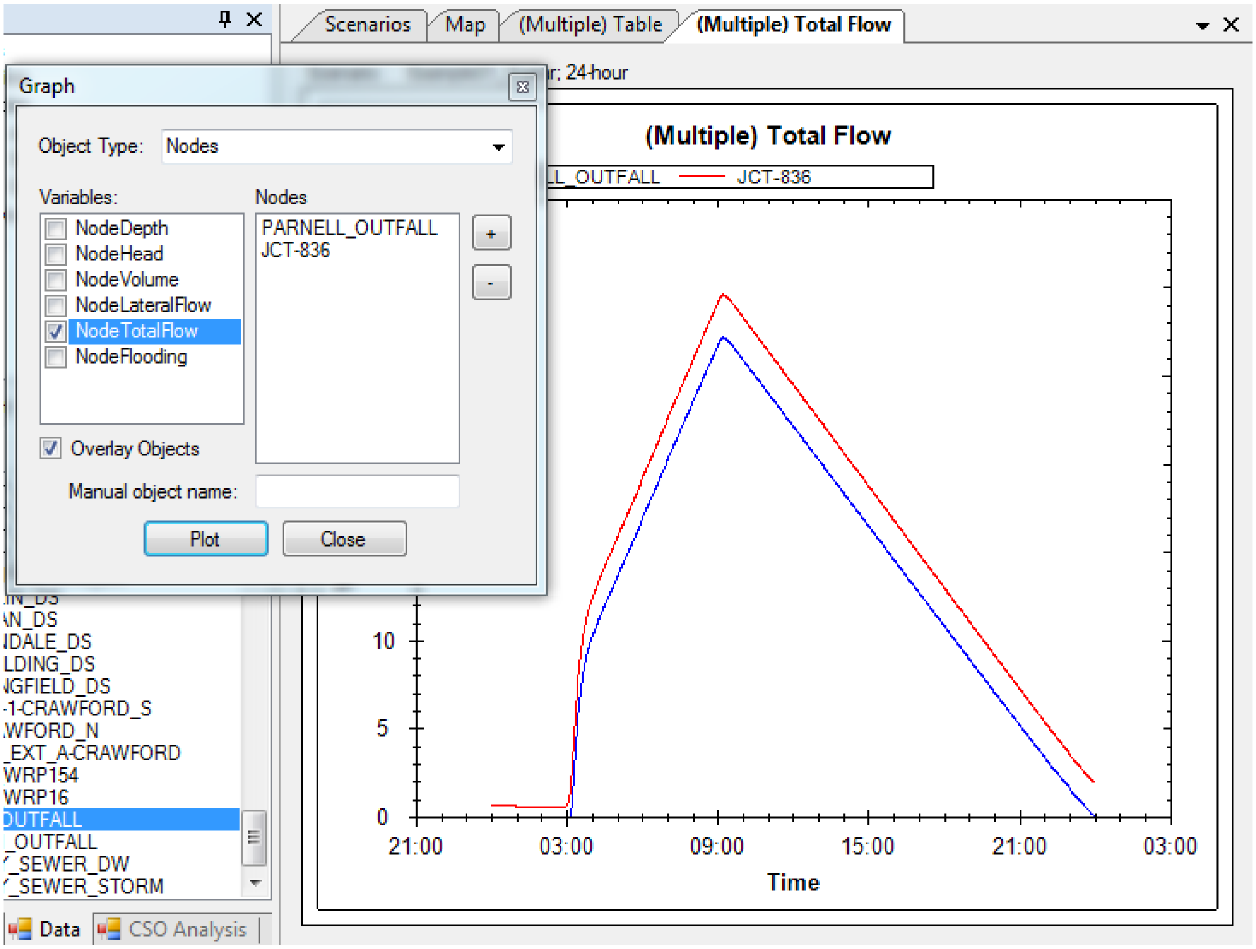


Figure 79: Graphical results window overlaying results for both PARNELL_OUTFALL and JCT-836

Profile Plot

Another analysis tool is the *Profile plot* ( ). Select the *Profile Plot* tool from the *Results toolbar*. While in the *Map window*, select the *Zoom to Full Extent* tool from the *Standard toolbar* to view the whole map. Navigate to the *Data tab* and find JCT-836. Click on JCT-836 then press *Select* to make it the active feature, and then select the top plus "+" button to register JCT-836 as the *Start node*. Next, click on JCT-76 and press Select to make it the active feature. Then select the bottom plus "+" button to register it as the *End node*. Finally, click *Find Path* to establish the intermediary nodes that will display automatically in the *Links in Profile* window. Select *Plot* and then grab the *Profile Plot (JCT-836 to JCT-76)* tab and position the window lower half the of the MetroFlow window (refer to Figure 80).

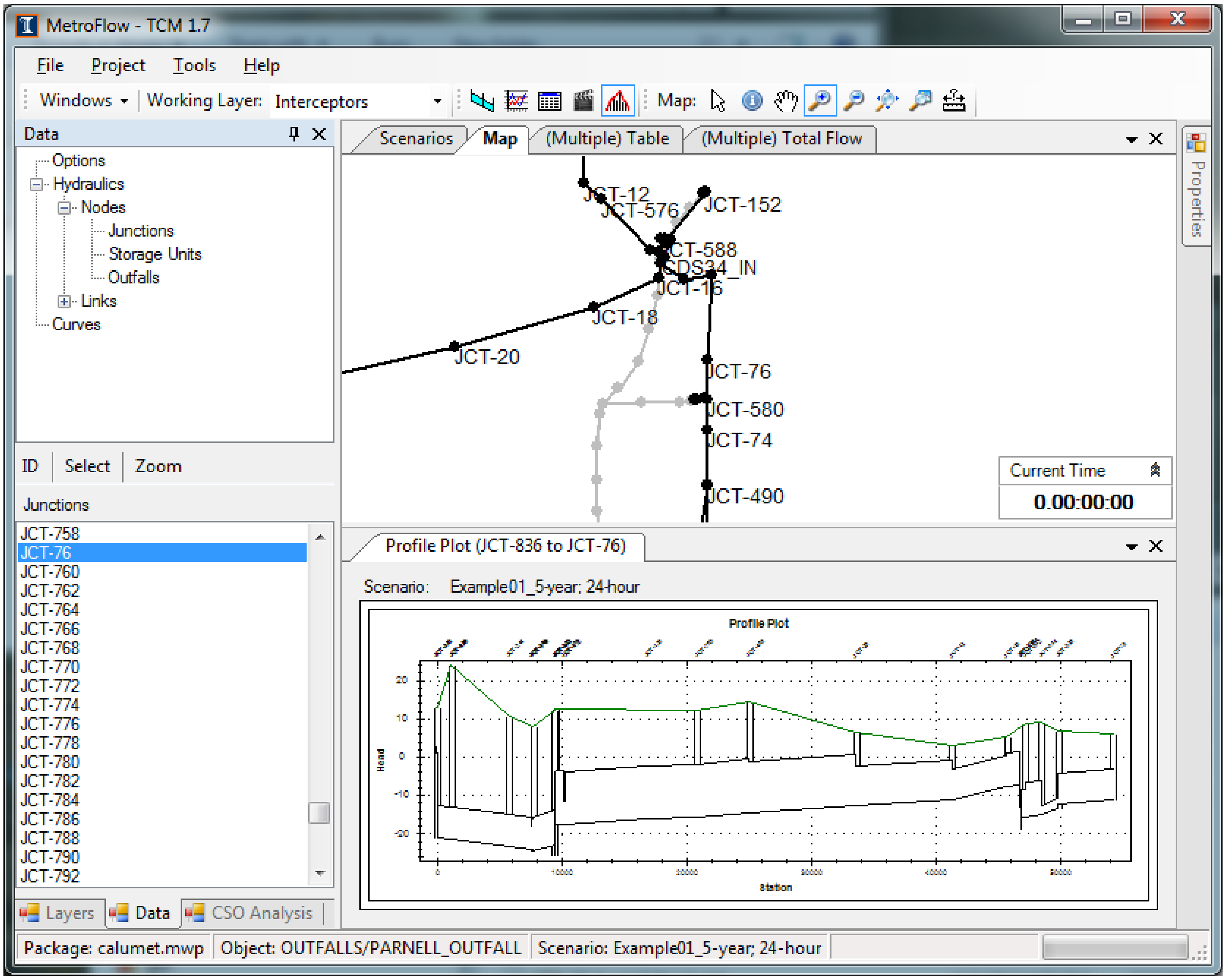


Figure 80: Profile plot visualization placed below Map

In the *Profile Plot*, to see the water level for a given time in the simulation, one needs to use the *Animator* tools mentioned in the next section.

Animator Tools

The user can animate the *Map* and *Profile Plot* windows by navigating to the *Results Animation* window in conjunction with editing the symbol visibility in the *Layers* window. First, make sure that the working layer is Interceptors and navigate to the *Layers* window. Using the nodes as the example, set the *Node Labeling* to *Name* to display the major junctions, in the *Symbology* field set *Nodes* to *Depth* and, immediately below the *Nodes* field, click *Edit Breaks*. Make sure the *Color Ramps* field is set to *Standard Hot-to-Cold* (if not select it from the pull-down menu and click *Apply ramps*), and set the *Number of Breaks* to 15 and you can change the *Data Maximum* field to 25 if you want to enhance the color contrast between nodes. Select *Create* to register the change in the number of breaks and click *OK* to terminate the editor.

Next, navigate to the *Results Animation* window. The Playback controller allows the user to animate the storm automatically or manually. Try animating the storm by pressing *Play* and adjust the speed bar to the quickest speed by dragging the handle to the right.

To visualize the animation at a specified time, manually enter value into the *Elapsed Time* dialog in the format D.HH:MM:SS where D is days, HH is hours, MM is minutes and SS is seconds, then click the *Play* button. For example, if you would like to visualize the storm at 9 hours, enter 0.09:00:00 into the *Elapsed Time* dialog, then press the *Play* button. Note the Current Time of the simulation is displayed in the lower right corner of the *Map* window, as seen in Figure 81.

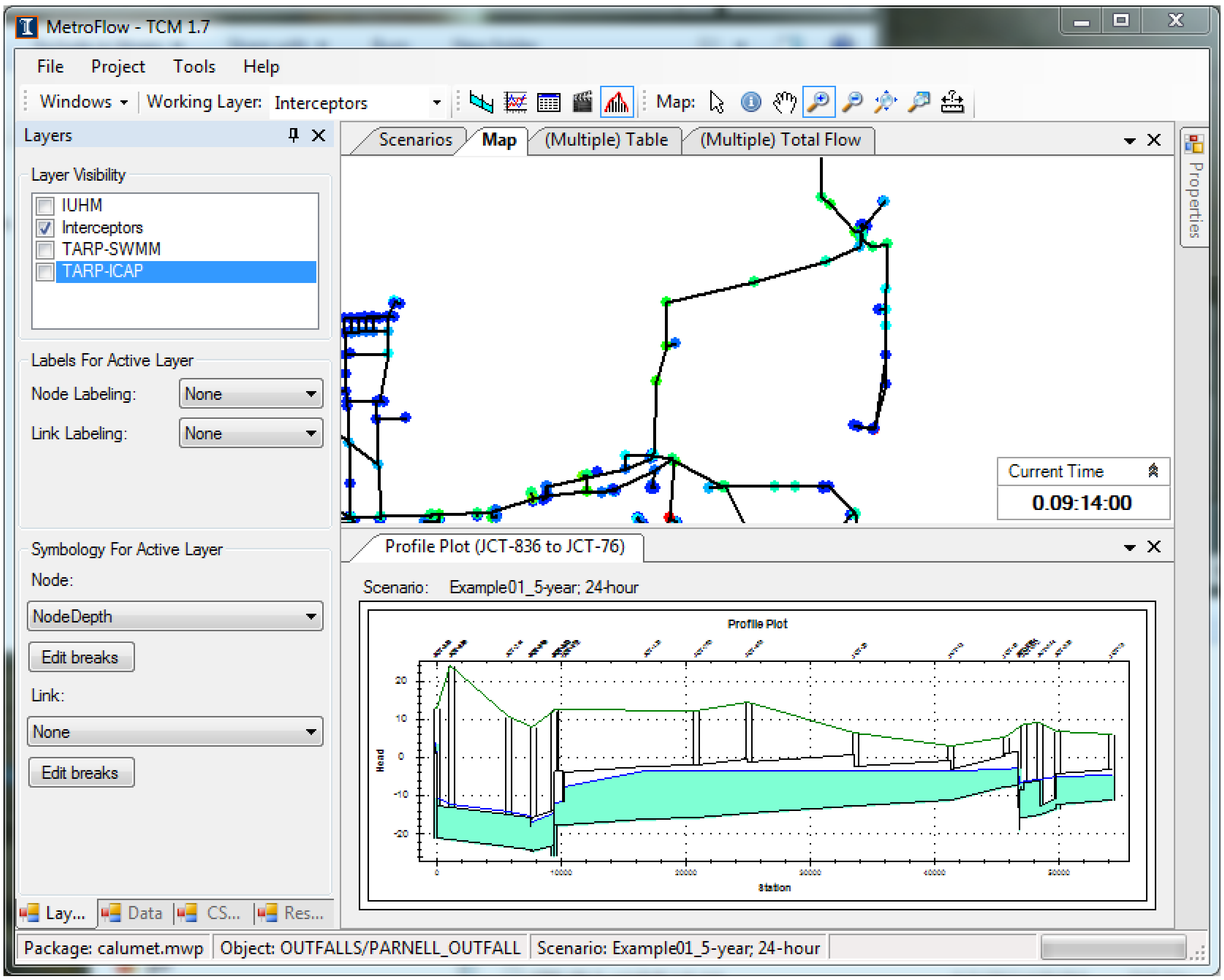


Figure 81: Animation of both Map and Profile Plot results for Example01 at 9hours

System Tables

The *Dropshaft Loading* and *Outfall Loading* table for the Interceptor working layer provides the user the ability to quantify the volume of water that was output to the dropshafts as well as combined sewer outfalls. Select the *Table* tool from the *Results* toolbar. Change the *Object Type* to *System* and select the *Dropshaft Loading* and *Outfall Loading* options under the *Variables* list box; this option is described in section 3.14.3. Press *Show Table* to view the results.

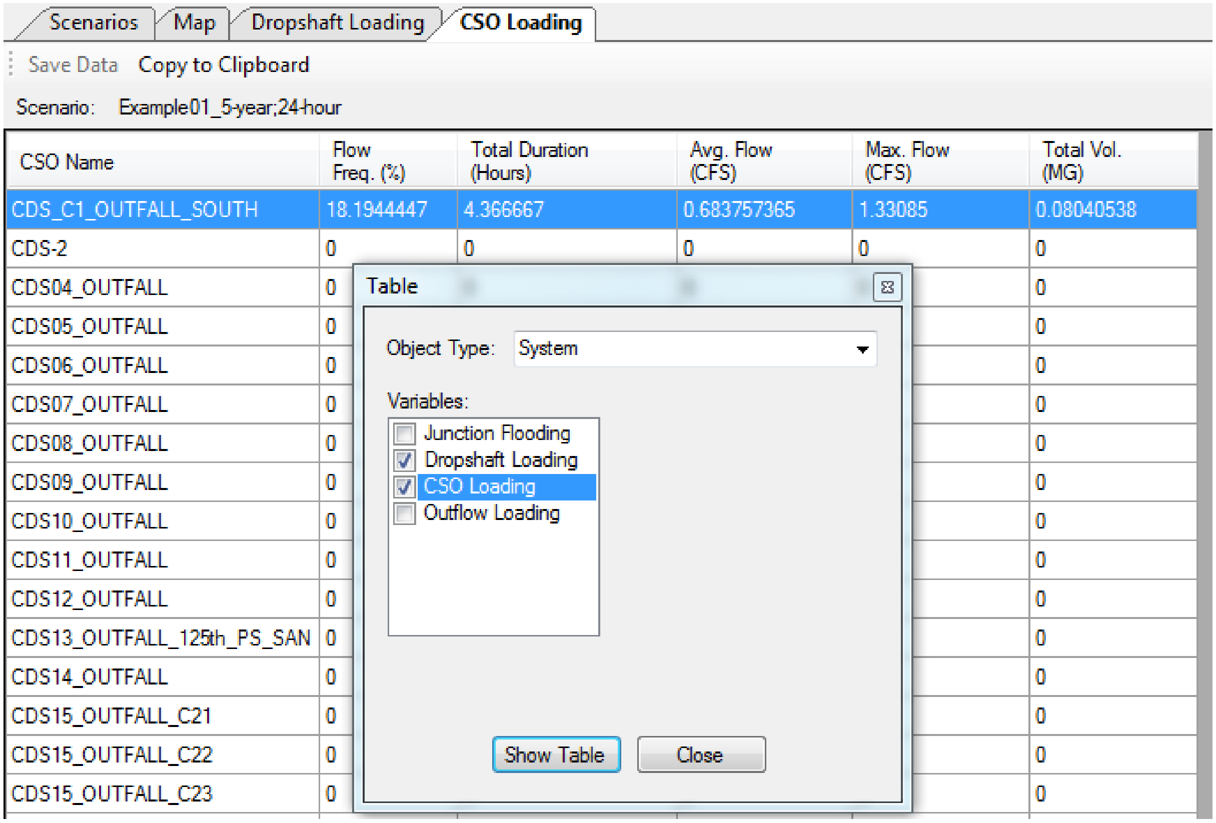


Figure 82: System tables showing Dropshaft/Outfall Loading results for Example01

For the TARP-SWMM model, a similar table called the *Junction Flooding* system table is given to provide the user a summary of any nodes that had overflow volume (as defined by SWMM). Note that due to the ponding capability in SWMM, nodes other than dropshafts may have overflow volume. This is expected behavior.

4.2 Example 2: Modeling a Historical Storm

In the same way that a design storm was processed and analyzed, MetroFlow can be used to analyze the impact of historical storms on the system. This section will be significantly shorter than the design storm tutorial because the same process that is used to analyze a design storm is used to analyze a historical event.

In order to import a historical storm, open the *Timeseries Sets* window, and click on the *Import Rainfall Event* button ( ) in the menu bar. This will display the *Import Historical Rainfall Event* window, as seen in Figure 83. Press the *Browse* button and select the desired historical rainfall file, e.g., .csv file. For this example, we will use the July 23-25, 2010 storm. Also, set the *Label* and *Description* fields to "July 23 - 25, 2010 Storm". Ensure the *Weights* option is on ISGS Gages, and press the *Process* button to register the historical storm as a timeseries set, seen in Figure 84.

Create a scenario in the *Scenarios* window that uses the IUHM and Interceptor models, and the newly-created timeseries set as the input timeseries. Ensure that the reporting interval is 1 minute, and the routing time step for the Interceptor model is 1 second. Run this scenario, and then verify successful runs via the model reports and *Timeseries Sets* window. Outputs from these scenarios will be used in following sections.

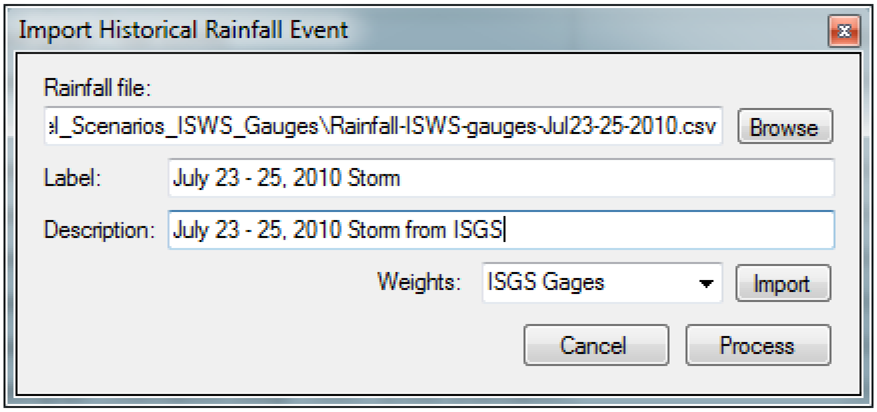


Figure 83: Import Historical Rainfall Event window with selected July 23 - 25, 2010 Storm

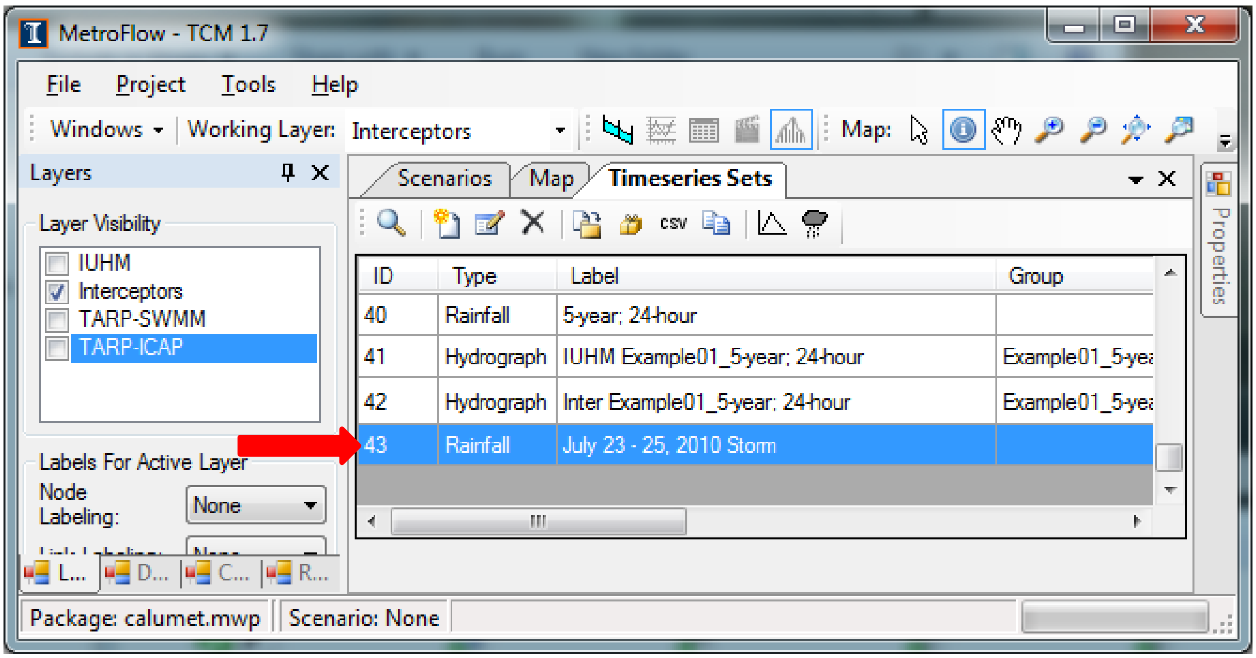


Figure 84: Registered historical storm as a Timeseries Set

4.3 Example 3: Reservoir Sensitivity with Historical Storm Scenario

A useful feature of the TARP-SWMM and TARP-ICAP models is the user's ability to run the model with no storage unit, i.e. not using Thornton Reservoir. This tutorial will illustrate how to analyze before and after cases with the TARP-SWMM model. This same process can be applied to the TARP-ICAP model.

4.3.1 Running the TARP-SWMM Model, With Reservoir

In the *Scenarios* window, create a new scenario. In the *Scenario Builder* name the scenario "SWMM July 23-25, 2010 with reservoir," and choose the TARP-SWMM model in *Modules to run*. Choose the "July 23 - 23, 2010 Storm (Interceptors)" timeseries as the input timeseries set. Leave the *Reporting Interval* at 1 minute, but in the *Module Options* ensure that the *Routing Time Step* is set to 1 second and leave the *Global Head* blank. Smaller routing time steps typically lower continuity error significantly but may add processing time particularly for long-term simulations. The *Global Head* should be left blank to start the scenario with an empty reservoir. These options are described in section 8.

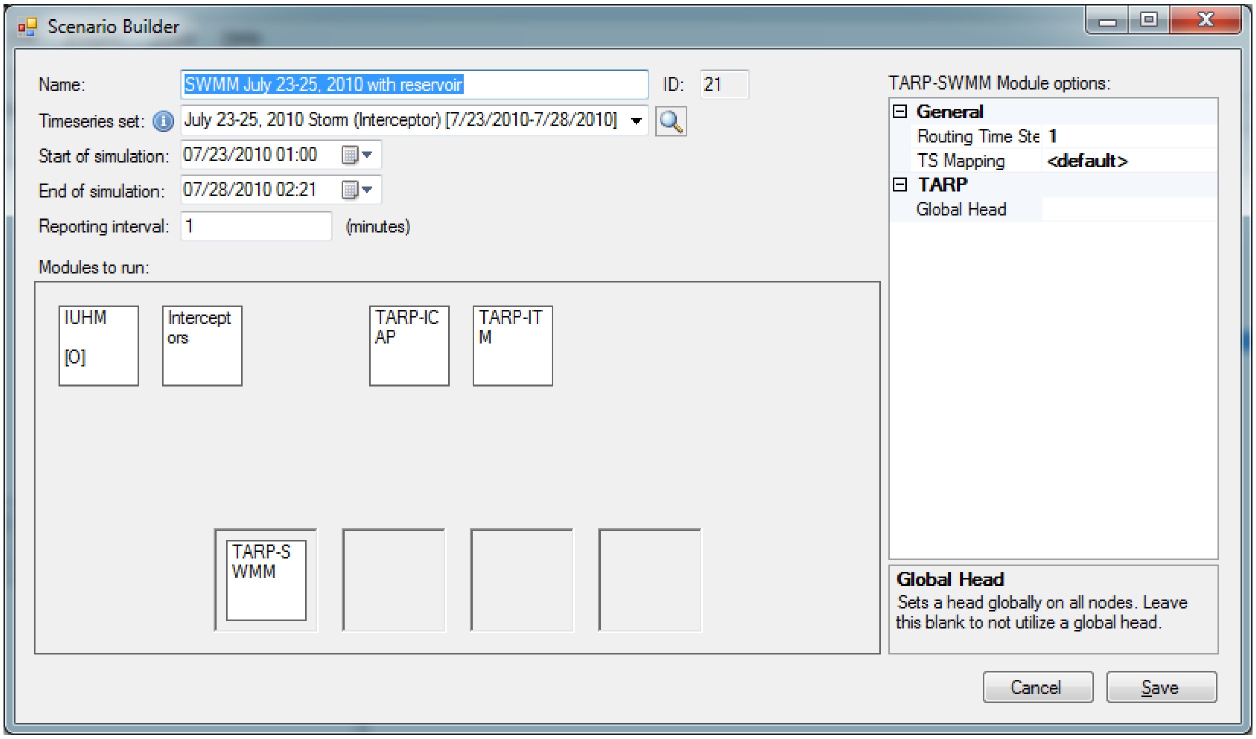


Figure 85: Scenario builder for the TARP-SWMM simulation

Viewing Results

As always, check the report in the *Scenarios* window to check for errors or to do a quality check on the results. After the run, load a system table for the *CSO Report*. This table will allow the user to quantify the CSO results. For the case with the reservoir, there should be no CSO events. The user should also check the head at the Thornton reservoir by opening the graph tool, selecting the *RES* node, and plotting *NodeHead*. The with- and without-reservoir node heads are plotted in Figure 87.

4.3.2 TARP-SWMM, Without Reservoir

Duplicate the previous scenario "SWMM July 23-25, 2010 with reservoir" by pressing the *Duplicate* button. A new scenario will appear named "SWMM July 23-25, 2010 with reservoir (copy)"; edit it to rename it to "SWMM July 23-25, 2010 without reservoir."

Select and load the *without reservoir* scenario, and set the *Working layer* to *TARP-SWMM*. Locate the reservoir and change the storage curve of the reservoir within the *Properties window*. This can be done by going to the *Data* window, opening *Hydraulics*, then *Nodes*, and then *Storage Units*. Click on the *RES* node and then press the *Select* button to load it into the properties window.

Click on the *Properties* window and use the pull-down menu next to *Storage Curve* to change the option to *NORESERVOIR*. Also, ensure that the *Initial Depth* field to zero (0). (Setting Initial Depth for the reservoir would result in the reservoir starting at the specified depth but not the tunnels which would result in the reservoir draining back into the tunnels.) Back in the *Scenarios* window, select *Save* and then *Run* the scenario. A textbox may appear that warns that rerunning will lose stored data; this message is only relevant to the copied scenario.

When changing node properties for a loaded scenario, the changes are only saved if the user presses the *Save* button in the *Scenarios* window.

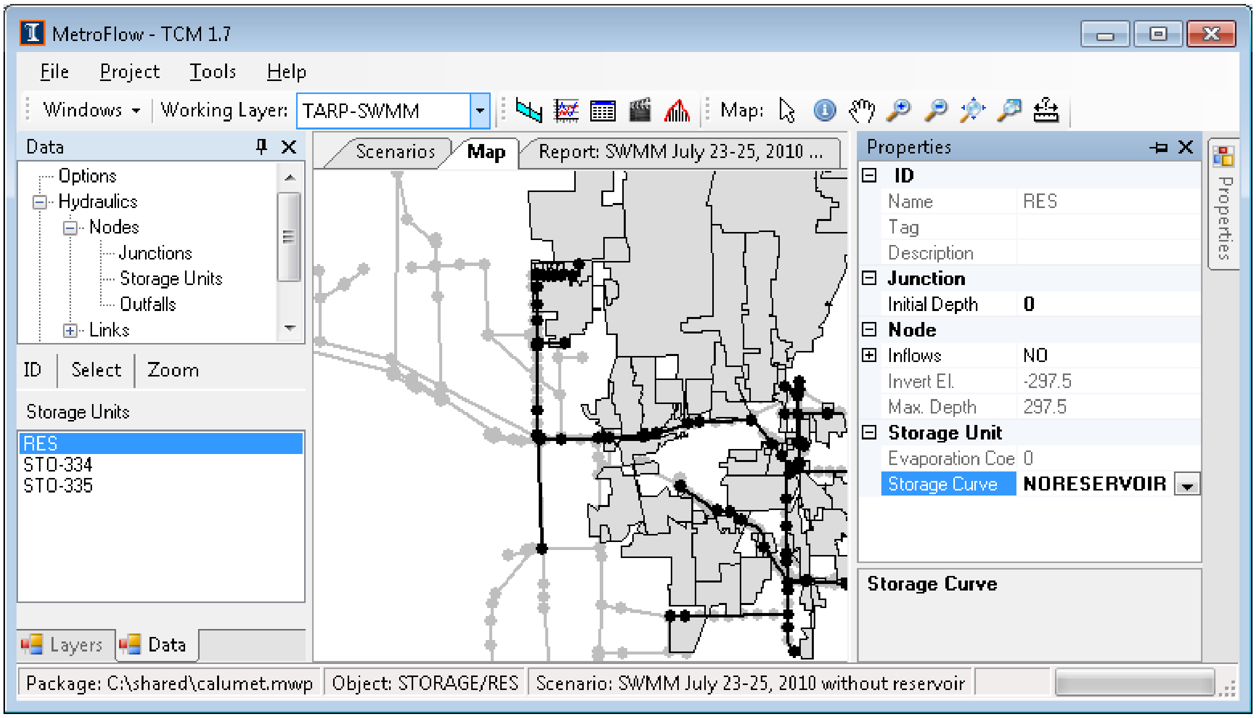


Figure 86: Modifying the properties for the reservoir node in order to simulate without Thornton in place

Once the run is complete, view the report to ensure that the run was successful. Check the continuity error under *Flow Routing Continuity* for both the with-reservoir and without-reservoir runs. After the run, load a system table for the *Overflow Volume*. By summing the volume column the user can determine the total amount of water that the TARP tunnels did not have capacity for, approximately 63 MG. The reservoir node *RES* head should be plotted and compared to the no-reservoir case, see Figure 87.

Many nodes in the TARP-SWMM may give a CSO volume but they may not be at dropshafts. This is normal behavior and is how SWMM functions.

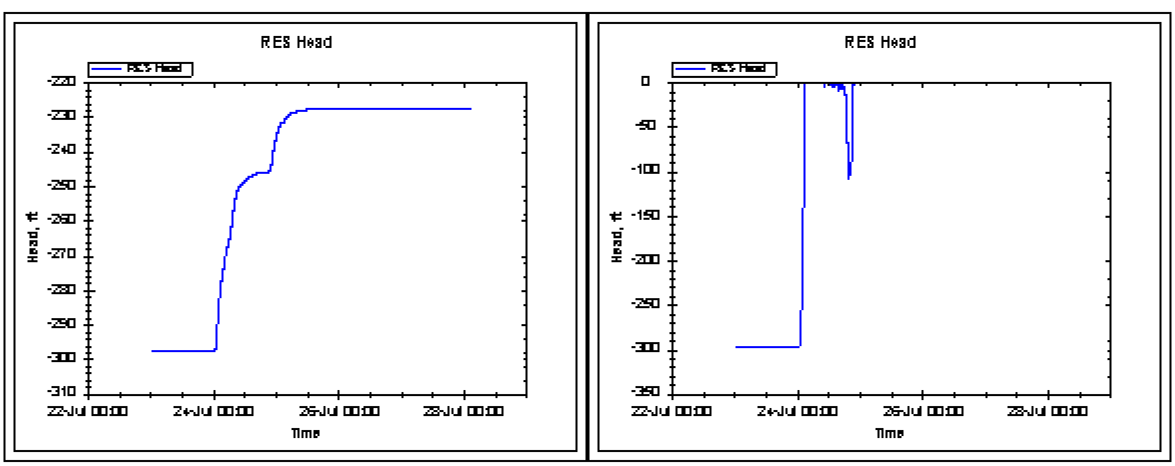


Figure 87: Reservoir node head for the with- and without-reservoir cases; note the head axes

4.4 Example 4: Incorporating Interceptor Sluice Gates into the Analysis

Using Interceptor Sluice Gates

In section 4.3.2, it was shown that by not including the Thornton reservoir, approximately 63 MG of water overflowed. In practice this will not occur, rather the sluice gates in the Interceptor system will be shut before water levels in the tunnels back all the way up the dropshafts. Since MetroFlow does not include a feedback mechanism between modeling layers, closing the sluice gates involves an iterative process of the following modeling sequence: Interceptors → TARP → Interceptors → TARP. The initial Interceptor model run with sluice gates opened generates an input for a TARP model. Running the TARP model with the Interceptor output then provides the user with an idea of when TARP capacity is exceeded. By re-running the Interceptor model and setting the gates to close at the time when TARP capacity is exceeded, a new input for the TARP model is generated that simulates what would get to TARP when the gates are incorporated. Finally, re-running the TARP model allows the user to verify that there was an actual tunnel capacity issue and not a transient. This section will illustrate how to do this and picks up where section 4.3.2 leaves off.

4.4.1 Finding the Date and Time of the First CSO Occurrence

By inspecting section 4.3.2, we can find the time at which CSOs first occur. Load *CSO analysis* window, select the TARP-SWMM layer, and then load the *without reservoir* scenario if not already loaded. Click on the *Load Analysis Data* button; when the data has been loaded the user will inspect the *Time Stack* plots. When the time stack window has been loaded, select the *Local* normalization option, and the mouse interactive button. Move the mouse over the plots, scrolling as necessary, to find the approximate first non-zero. Then check the checkbox in the *Graph* column for the CSO location with the first non-zero value. Inspect the graph by right-clicking on the plot, selecting *Show Data Points* and moving the mouse over the graph until the date and time of the first occurrence is obvious. This should correspond approximately to 7/24/2010, 2:49 AM as illustrated in Figure 89. An hour should be subtracted from this to act as a buffer, yielding 7/24/2010, 1:49 AM. Convert this to simulation time by going to the *Tools* menu in the main window and selecting *Time convertor* (see Figure 88). Type in the date and time of the first CSO occurrence into the *Real Time* field and then copy the resulting *Simulation Time* value of 1.00:49:00 out. This tool goes both ways; simulation time can be entered to obtain a real time value. The simulation time is the difference between the real time and the start time of the simulation.

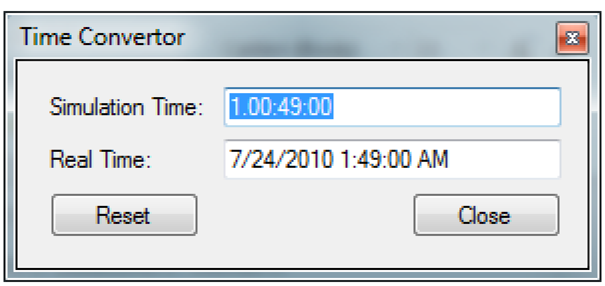


Figure 88: Time convertor for converting real time to simulation time and vice versa

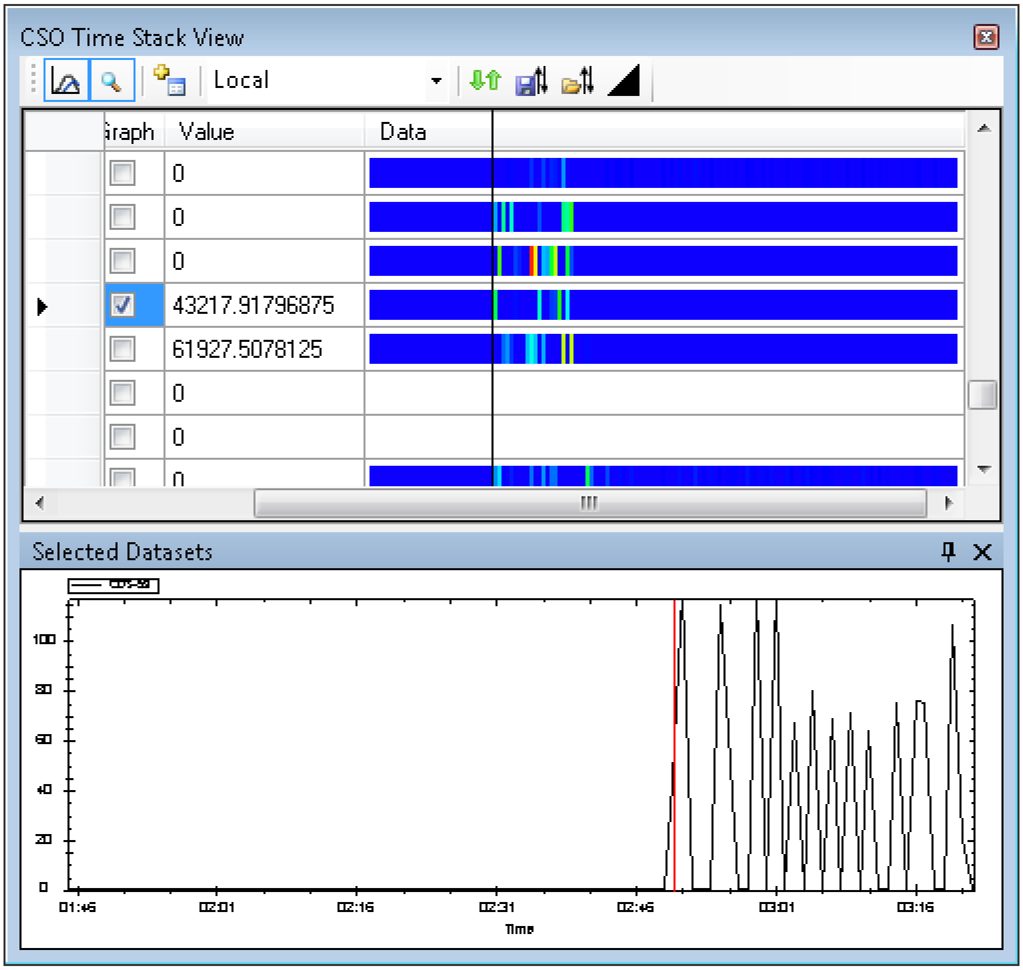


Figure 89: CSO analysis time stack plots showing the time at which a CSO event first occurs

An alternative to this approach follows the suggested gate operation rules in section 3.13. Load a plot of the head at *CALUMET_PS* (which represents the approximate pumping station location) and pick off the time at which the head exceeded -150 feet. This should be approximately 7/24/2010, 2:21 AM as shown in Figure 90 which corresponds to 1.01:21:00 in simulation time.

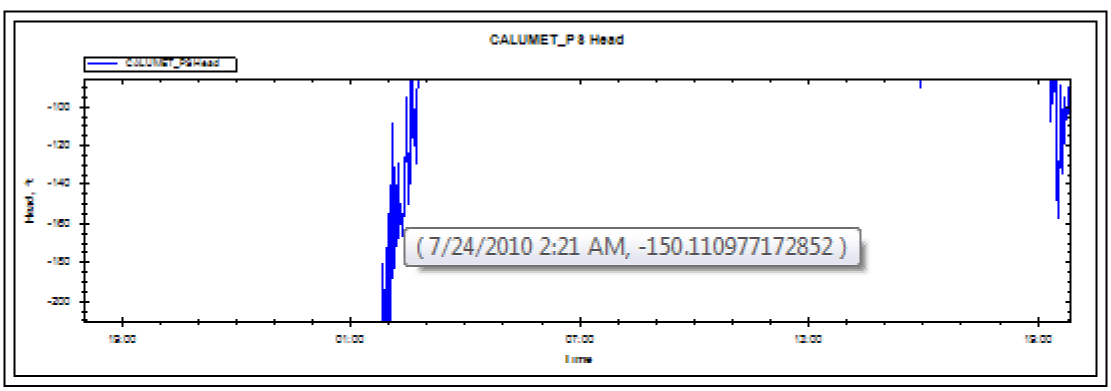


Figure 90: Head at CALUMET_PS

4.4.2 Closing the Interceptor Sluice Gates

In section 4.2 the July 23-25, 2010 storm was run through the Interceptor model. Create a new scenario using the “July 23 - 25, 2010 Storm (IUHM)” timeseries set and name the scenario “Interceptors July 23 - 25, 2010 Storm, gate closing”. Save and then press the *Load Scenario* button. Select the *Interceptors* working layer, open the *Controls* window, then under the *GATE_GLOBAL* tab, input the simulation time as determined in section 4.4.1 in the *Simulation Time* field in *Global Condition* box and press the *Apply* button. Then, in *Global Action Setting*, enter 0 for the *Height* field, and press *Apply*. All of the sluice gates in the *Actions* list will be closed at the specified simulation time. In order to save these changes, press the *Apply Changes* button, then go to the *Scenarios* window and press the save button.

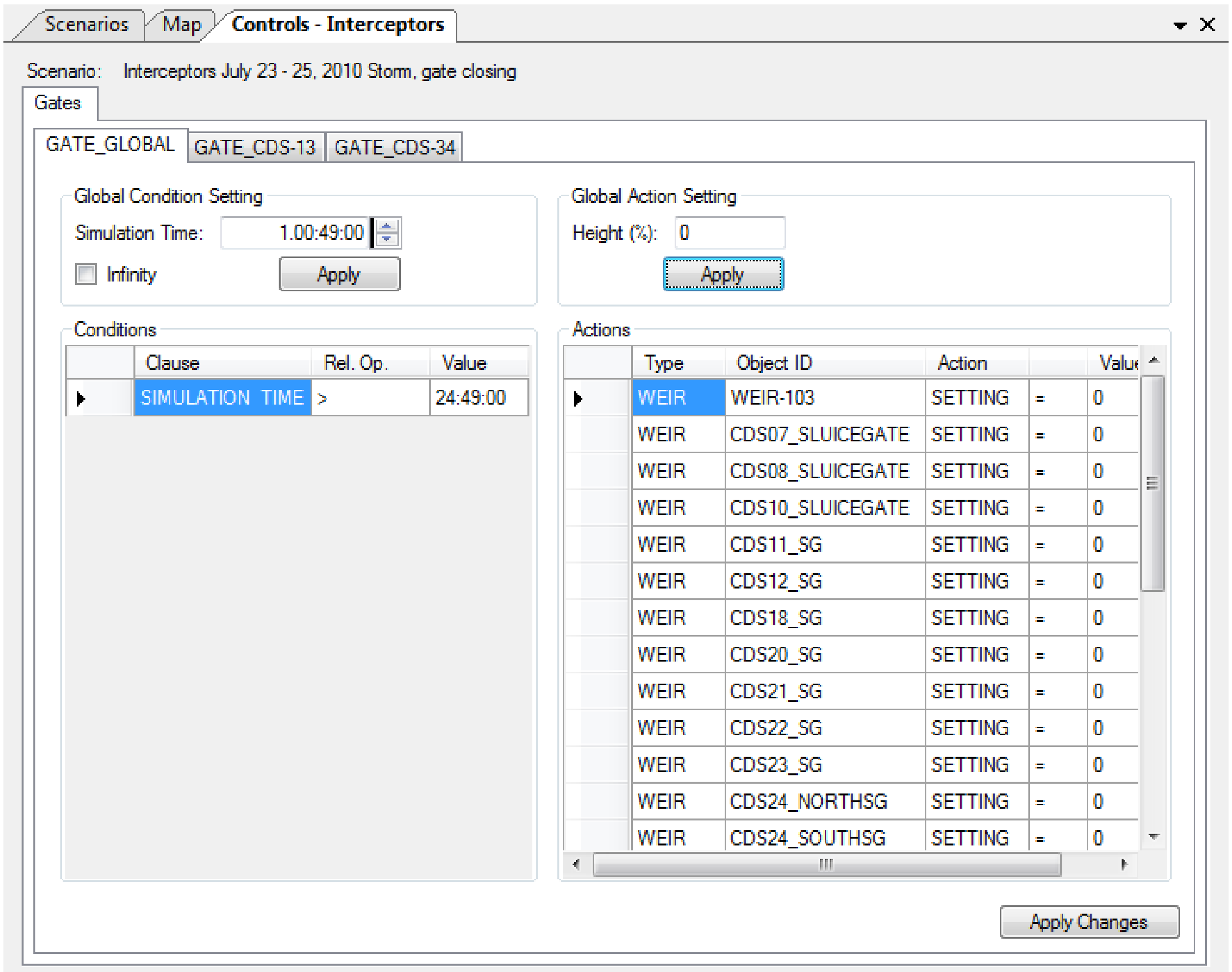


Figure 91: Setting the gate closing rules for the Interceptor model

Run the scenario and inspect the report and output timeseries set “Interceptors July 23 - 25, 2010 Storm, gate closing (Interceptors)”.

4.4.3 Running the TARP Model with the Interceptors Closed

Create a new scenario, name it “SWMM July 23-25, 2010 without reservoir, gates closed” and drag down the TARP-SWMM model. Select the “Interceptors July 23 - 25, 2010 Storm, gate closing (Interceptors)” timeseries set as the input and leave other options alone. Save the scenario, open the *RES* node properties, change the storage curve setting to *NORESERVOIR* and then save the scenario again. Now run the scenario, inspect the report (detailed in section 4.1.4), and view the *Overflow Volume* system table. Overflows will have been significantly reduced but still not eliminated since many dropshafts in the model do not have sluice gates to prevent inflow.

4.5 Example 5: Importing and Running Water Years with Pumping

Using Pumping in ICAP

Analyzing individual events such as the previous tutorials allows users to determine the ability of the system to capture individual events. However, in order to study the impact of multiple storm events in succession, with the ability to pump water to treatment facilities, it becomes necessary to import and analyze long periods of data such as those found in water years. MetroFlow has the ability to import entire water years as well as multi-year datasets. This tutorial will guide the user through the process of analyzing the effect of pumping by using the ICAP model.

> Individual models, particularly the Interceptor and IUHM models, may take extended periods of time to run. It is not uncommon for a run of IUHM and Interceptors for a water year to take 2-3 days to complete.

The process for importing and running water years is identical to that detailed in section 4.2, with the critical exception that scenarios should use a **reporting interval of 60 minutes**. Models such as the TARP-SWMM and Interceptors should use a **routing time step of 1 second**, and the ICAP model should use routing time step of 1800 seconds.

Follow section 4.2 to import the water year 2010 rainfall file and name the timeseries set "Water Year 2010". Then create and run a scenario that uses IUHM only, selecting the water year timeseries set and using "Water Year 2010, IUHM" for the output label; this will take approximately 12 hours. Using the output from IUHM, timeseries set "Water Year 2010, IUHM," create and run an Interceptor scenario, naming it "Water Year 2010, Interceptors" and setting "Water Year 2010, Interceptors" to be the output label. The Interceptor run will take at least 12 hours. The reporting interval should be 60 minutes and the routing time step should be 1 second. Verify that the scenario run was successful using the checks detailed in section 4.1.4.

4.5.1 ICAP without Pumping

After successfully running IUHM and the Interceptor model for the water year 2010 rainfall, create a scenario that models TARP-ICAP with the "Water Year 2010, Interceptor" timeseries set as input. Name the scenario "Water Year 2010, ICAP no pumping". Select 60 minutes for the reporting interval and 1800 seconds for the routing time step, in that order. Under the ICAP options, set *ETP Threshold* to be 365; this will prevent any pumping from occurring. Leave *Inflow Threshold* and *Pumping Rate* at the default values. (Descriptions of the ICAP options are given in section 8.3.1.) Figure 92 illustrates the *Scenario Builder* options for this scenario. Run the ICAP scenario and then pull up a plot of the *RES* node head. It should be something like Figure 93. Also pull up a plot of the *RES* node for total flow. Zooming in to the lower part of the plot will allow the user to see the baseline level of flow put into Calumet due to dry weather flow. In Figure 94, it can be seen that this value is approximately 150 CFS.

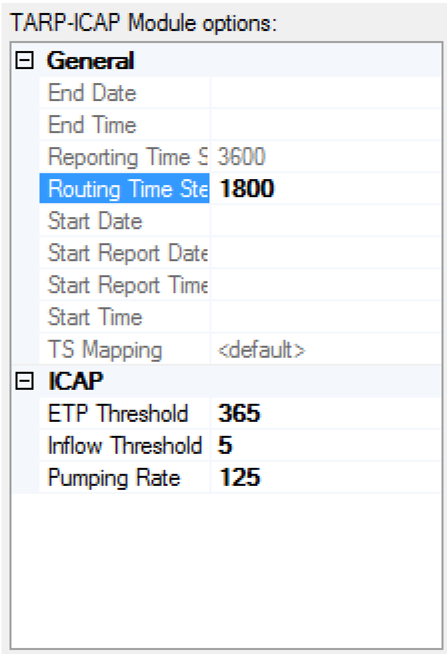


Figure 92: Scenario Builder for the ICAP water year 2010 scenario

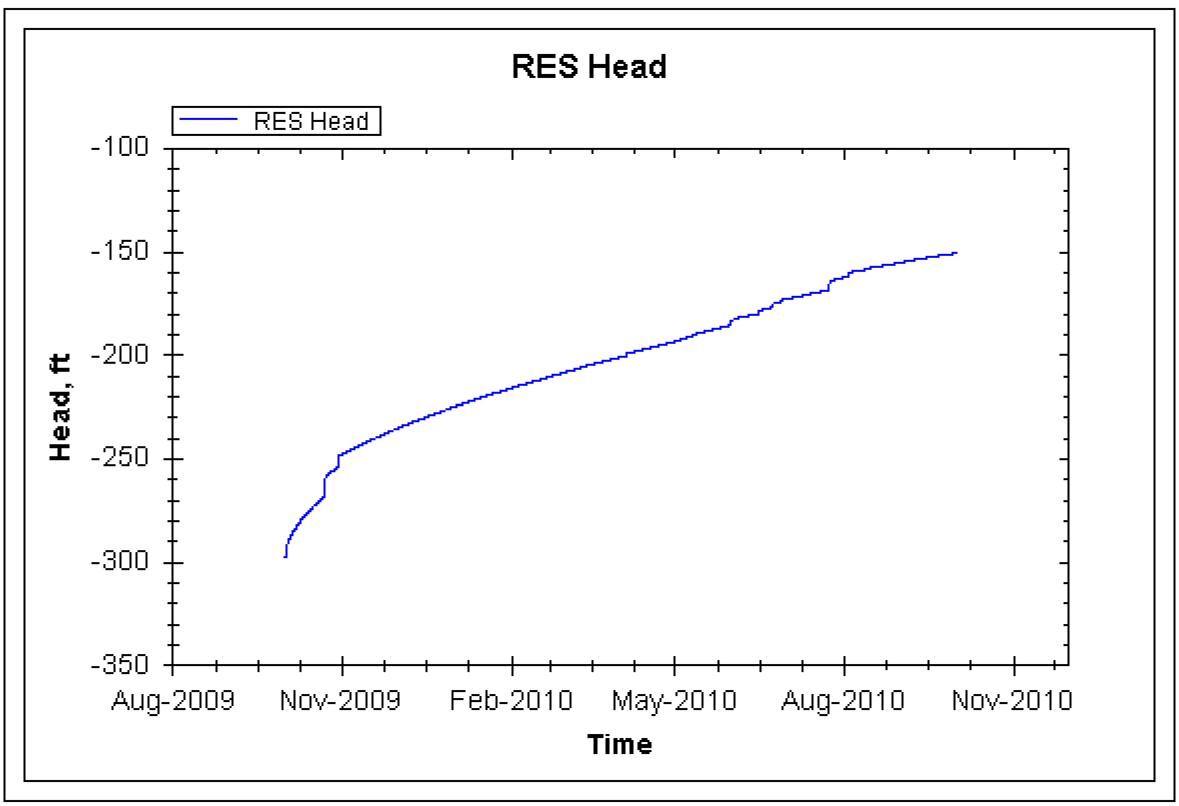


Figure 93: ICAP reservoir head for Water Year 2010, without pumping

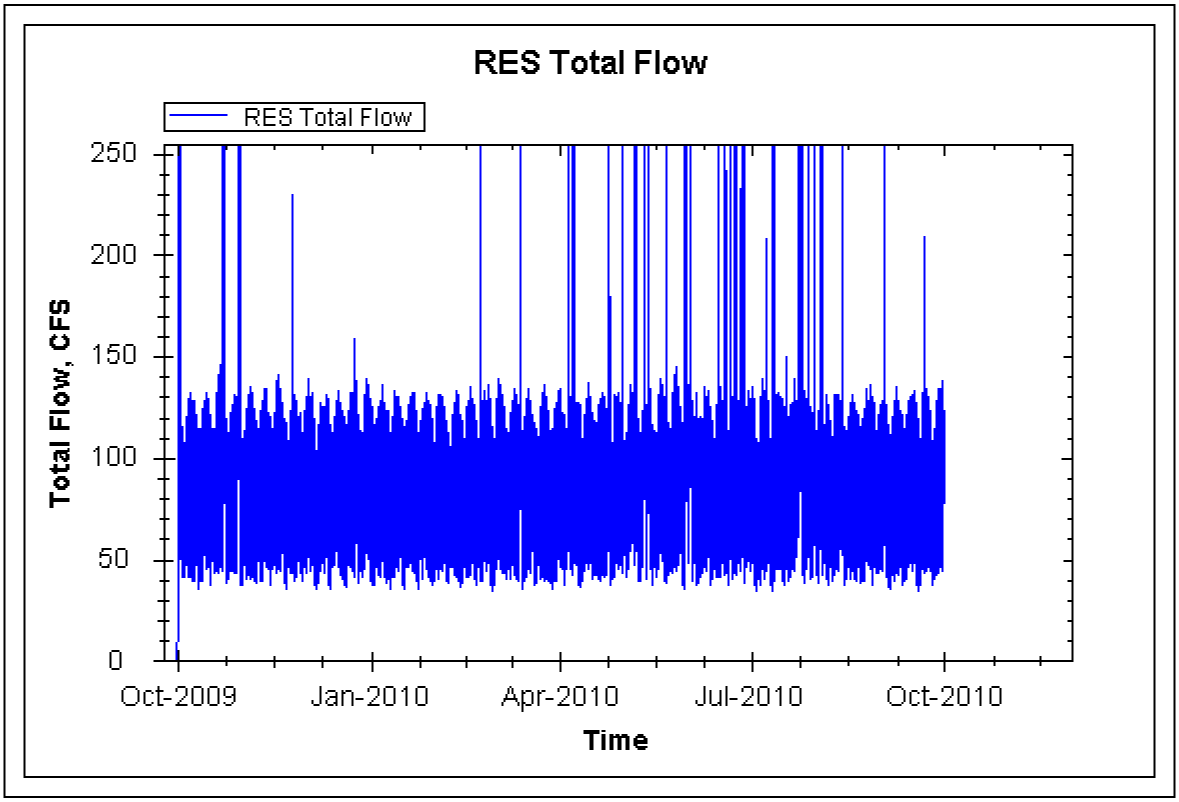


Figure 94: Total flow into the reservoir, used for determining baseline flow for pumping

4.5.2 ICAP with Pumping

As can be seen from Figure 95, the reservoir fills up part way through the water year. This is where the pumping parameters in ICAP are useful for determining the effect of pumping on a water year. Duplicate the “Water Year 2010, ICAP no pumping” scenario, edit it, and rename it to “Water Year 2010, ICAP with pumping”. Set the ICAP options to have the following options:

- *ETP Threshold* = 2, two days will have to have elapsed from the end of the event to the time at which pumping begins;
- *Inflow Threshold* = 150, 150 CFS of total inflow or lower are considered to mark the end of an event;
- *Pumping Rate* = 193.4, 193.4 CFS (125 MGD) of water will be removed from the system.

The District typically uses pumping rates in units of MGD. Useful conversions are as follows:

MGD → CFS: multiply by 1.547
CFS → MGD: divide by 1.547

Run the scenario with pumping and generate a plot of the reservoir (*RES*) head which should be similar to Figure 95.

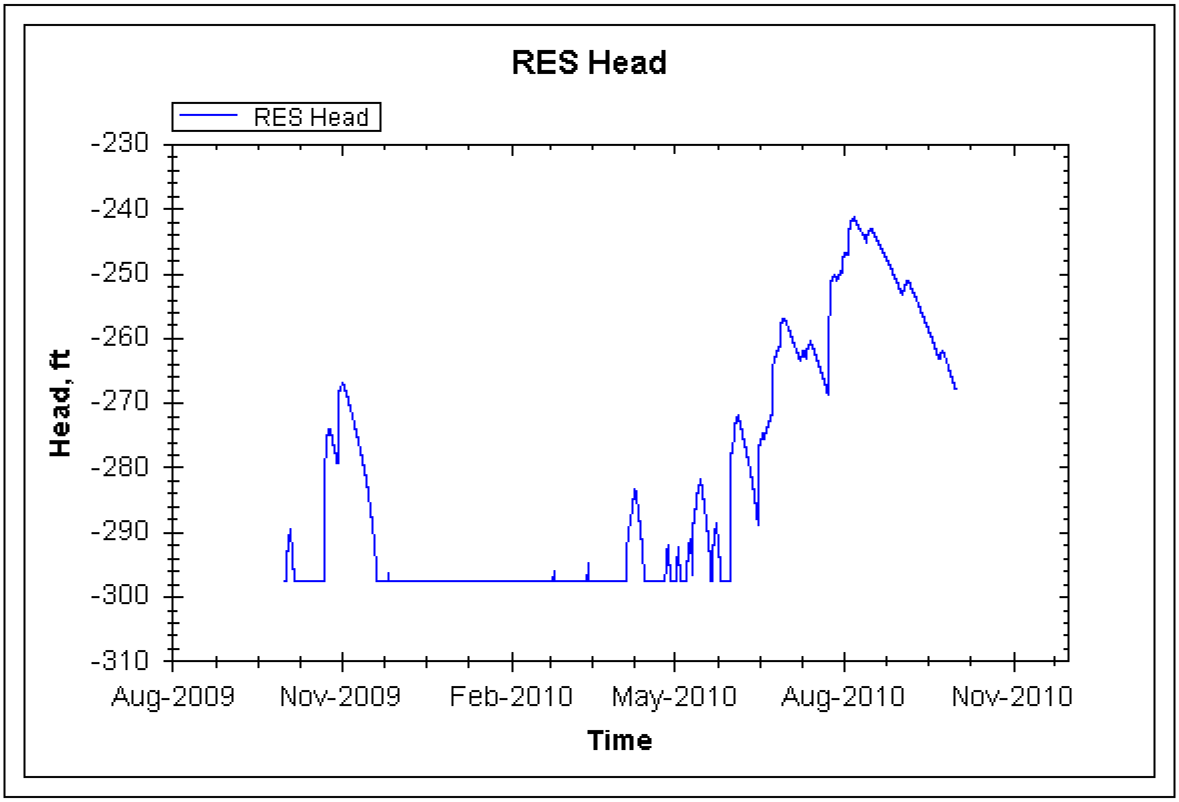


Figure 95: ICAP reservoir head for Water Year 2010, with pumping

Using Timeseries Mappings

4.6 Example 6: Simulating Thorn Creek Inflows with Custom Timeseries

In the Calumet system, flows from Thorn Creek may be diverted into the Thornton reservoir during extreme flooding events. Inflows into the reservoir from Thorn Creek may be simulated by adding a time series inflow curve to the reservoir. This tutorial summarizes the steps needed to perform this analysis. It relies on the interceptor outputs given in section 4.5.

4.6.1 Step 1: Creating a Timeseries Mapping

The first step is creating a new *Timeseries Mapping*. Select the TARP-ICAP working layer, open the *TS Mappings* window, and then click the *Add New* button. Type “Interceptor With Thorn Creek” in the *New Name* field, select “Built In - Interceptor” from the *Copy from* field, and then click the *Save* button. Figure 96 illustrates the screen up to this point.

ILLINOIS

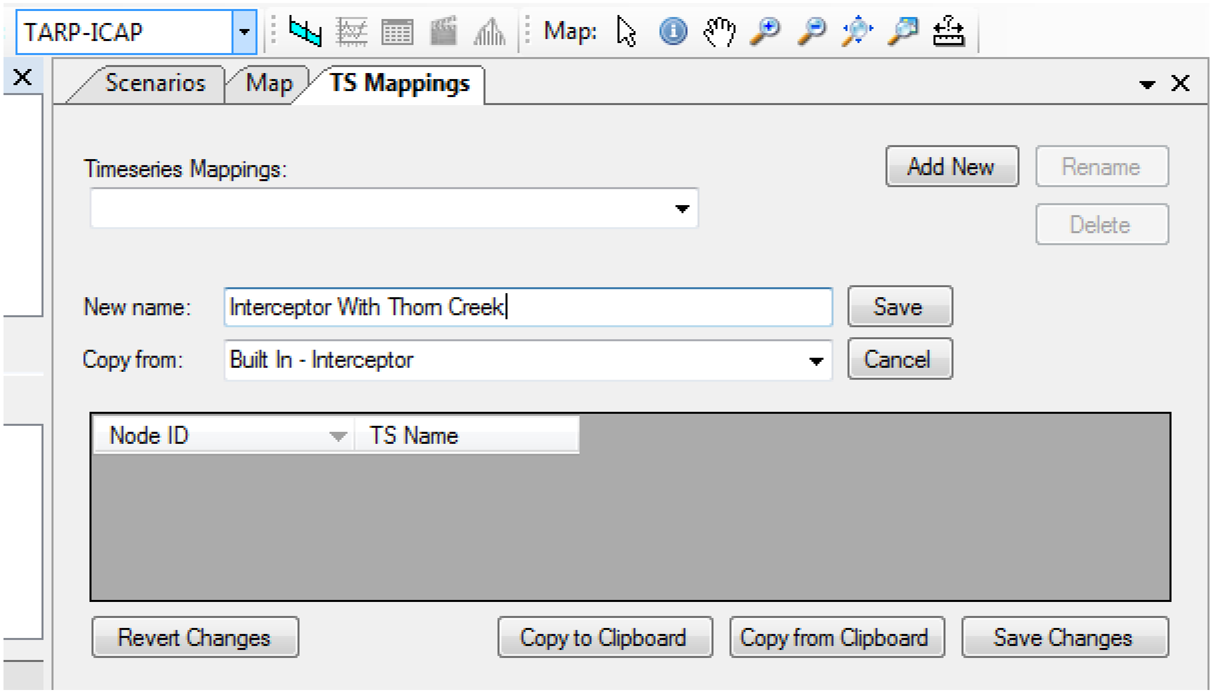


Figure 96: Initial timeseries mappings screen.

After pressing *Save*, the mapping section fills in with a list that has *Node ID* as the first column (which represents the name of the node) and *TS Name* as the second column (which represents the name of the timeseries that will be input into the node). Find the node *RES*, and type *THORN_CREEK* into the corresponding *TS Name* field. Press the *Save Changes* button.

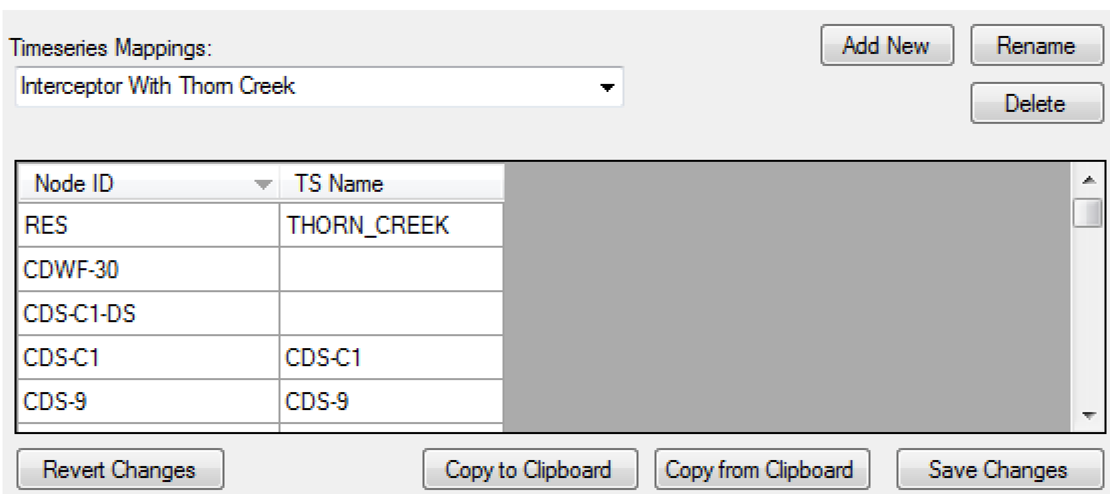


Figure 97: Changes to timeseries mapping after including Thorn Creek

4.6.2 Step 2: Converting Bulk Inflow Numbers into Hydrographs

Now the hydrograph must be prepared. If an inflow hydrograph is available, then that can be imported directly following the next section and this step can be skipped. Otherwise the directions in this section will help the user develop a hydrograph from the bulk inflow numbers that are typically given.

Thorn Creek inflows are typically given in million gallons and in order to convert to CFS, the volume of water must be divided out over the period of time that it was put into Thornton. This provides the average flow rate over the fill event period which can be used to create a simple hydrograph.

Example Fill event: Oct 24-25, 2009, 151 MG total

- ➔ 151,000,000 gallons is 20,200,000 cubic feet.
- ➔ 20,200,000 cubic feet divided by two days in seconds (2 * 86,400) is 117 CFS.

The average flow rate is 117 CFS. The equivalent hydrograph is given as follows:

10/23/2009	23:59	0
10/24/2009	0:00	117
10/25/2009	23:59	117
10/26/2009	0:00	0

The hydrograph for this period would have to be bracketed by zero values. Multiple events like this can be chained together to yield a hydrograph for an entire year. These should be placed into a text file for importing in the next step.

4.6.3 Step 3: Importing a Hydrograph into a Timeseries Set

Once a hydrograph has been prepared, it can be imported into a timeseries set for use by the models in conjunction with the modified timeseries mapping. Open the *Timeseries Sets* window, click on the set "Water Year 2010, Interceptors" and the press the *duplicate* button. This will create a new set titled "Water Year 2010, Interceptors (copy)" with the same timeseries as the "Water Year 2010, Interceptors" set; edit this set by clicking the *View and Edit* button. Change the label to "Water Year 2010, Interceptors with Thorn Creek" and then select the *TS Action* drop down menu. Select the *Import single TS* option as shown in Figure 98 and press the *Execute TS Action* button.

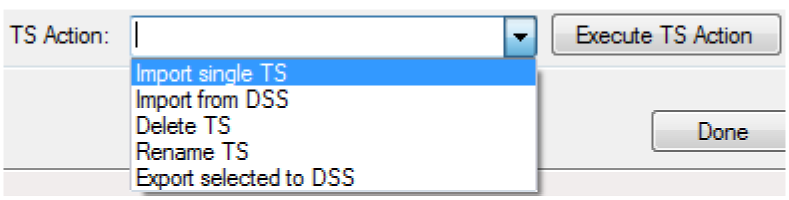


Figure 98: TS Action for importing a single timeseries

Figure 99 illustrates the *Import Single Timeseries* form that will appear. Fill in the *Name of the timeseries* field with the *TS Name* that was set in section 4.6.1, "THORN_CREEK". Click the *Browse* button to find the hydrograph file created earlier and select it. Once both fields are filled in, press the *Import* button. The new timeseries will appear in the timeseries list. Double-clicking on the timeseries will show the imported data, shown in Figure 100. Press the *Done* button in the *View and Edit Timeseries Set* window and then view the timeseries with the *View* button. The hydrograph will appear to have a square shape, like that in Figure 101; this is normal.

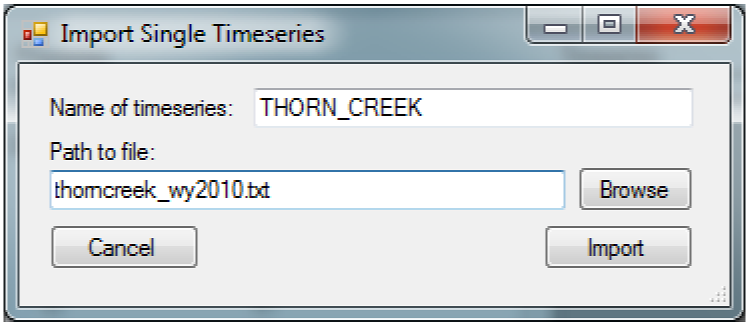


Figure 99: Importing a single timeseries option

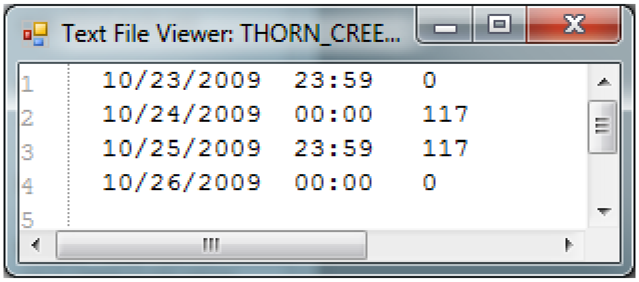


Figure 100: Imported timeseries file, viewed in the text file viewer

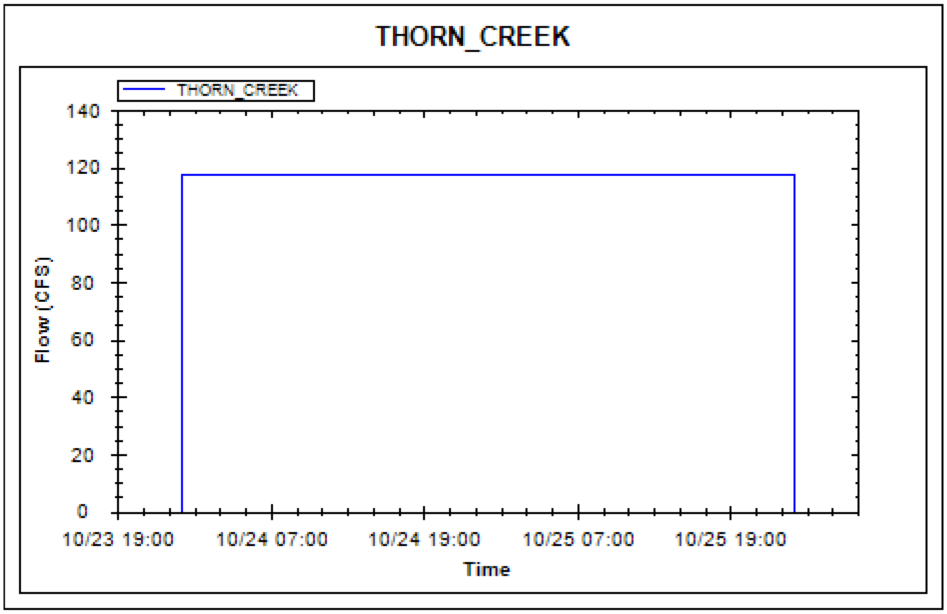


Figure 101: Imported Thorn Creek hydrograph

4.6.4 Step 4: Running a New ICAP Scenario with Thorn Creek Inflows

The final step in simulating Thorn Creek inflows is to duplicate the "Water Year 2010, ICAP with pumping" scenario, editing it, and renaming it to "Water Year 2010, ICAP with pumping and Thorn Creek". Also, select the timeseries set "Water Year 2010, Interceptors with Thorn Creek" as the input timeseries set instead of the "Water Year 2010, Interceptors" set. Finally, the timeseries mapping must be changed. In the *ICAP Module Options* section, the field *TS Mapping* should be set to "Interceptor With Thorn Creek", the mapping that was created in Step 1. Leave the options as they were given in section 4.5.2. Once this has been changed *Save* the scenario. Figure 102 illustrates the settings necessary.

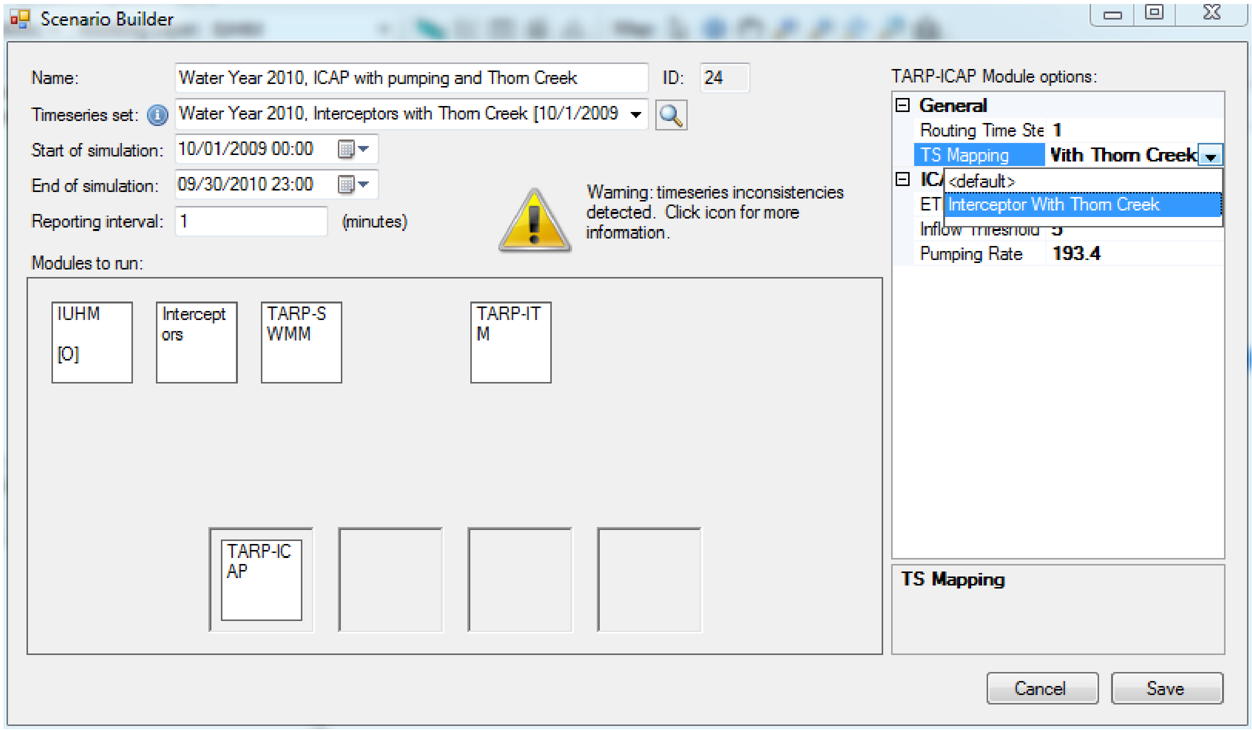


Figure 102: ICAP scenario settings for pumping and Thorn Creek inflows

The scenario should be run . Once a successful run has taken place, open a graph for the *RES* node, and plot *NodeLateralFlow* and *NodeTotalFlow*. The *NodeLateralFlow* plot should match the Thorn Creek inflows hydrograph developed in previous steps. Thus inflow events from Thorn Creek are accounted for. Figure 104 shows a plot of the head for the reservoir; this can be compared to Figure 95 to see how inflow events affect the performance of the reservoir. It can been seen that near the November 2009 mark, the peak for the Thorn Creek inflows case reaches -260 feet while for the case without Thorn Creek inflows, the head reaches -266 feet.

4.7 Accounting for Model Differences in Mainstream/Des Plaines

In the Mainstream/Des Plaines package, the CS-TARP model is an InfoWorks model and works differently than the TARP-SWMM and Interceptor layers in the Calumet system. The primary differences are as follows:

- The hydrologic model is a combination of the InfoWorks City of Chicago model for the City sewers and IUHM for Des Plaines;
- Pumping is controlled via timeseries and a scenario option;
- Reservoir/no reservoir condition is controlled via scenario option;
- There are no controls available for MS/DP unlike in Calumet.

4.7.1 Water Year Simulations and Pumping

For year-long simulations it is important to enable pumping in the CS-TARP model. This is done in a two-step process. The first step is a one-time step that needs to be done the first time a water year scenario is run, and that is creating a new timeseries mapping. In order to do this, follow the steps in section

4.6.1 to create a new timeseries mapping. Name the mapping "Water Year with Pumping" and find the node *DUMMY1*; in the *TS Name* field type in "PUMPING". Now the timeseries set that is input into CS-TARP must be modified to add in the pumping record. The pumping record is a CSV file with two columns: date/time and pumping value. The pumping value column is zero for hours in which no pumping occurred and is 300 (MGD) for hours in which pumping occurred (see Figure 103 for an example). This file must be imported into the timeseries set that is to be input to the CS-TARP model, per the directions given in section 3.10.8. **It is very important that the MGD->CFS conversion factor be selected during the import and that the imported timeseries be named "PUMPING".**

It will be most effective to not chain scenarios together for water years; in other words, run the IUHM+CityModel independently of the CS-TARP scenario.

Once the timeseries mapping has been created, and the pumping record input into the timeseries set, a CS-TARP scenario can be created. The *Use Pumping* option must be selected, and the "Water Year with Pumping" timeseries mapping must be selected in the options panel of the Scenario Builder. The scenario can then be saved and run. Wherever the timeseries "PUMPING" has a 300 MGD value, pumping out of McCook will occur.

	A	B	C
1	P_DATETIME	PUMPING	
2	1/1/2000 1:00	0	
3	1/1/2000 2:00	0	
4	1/1/2000 3:00	300	
5	1/1/2000 4:00	300	
6	1/1/2000 5:00	300	
7	1/1/2000 6:00	300	
8	1/1/2000 7:00	300	
9	1/1/2000 8:00	300	
10	1/1/2000 9:00	0	
11	1/1/2000 10:00	0	
12	1/1/2000 11:00	0	
13	1/1/2000 12:00	0	
14	1/1/2000 13:00	0	

Figure 103: Example CSV pumping record file

4.7.2 Simulating Reservoir/No-Reservoir

By default, CS-TARP runs with a 10 BG McCook reservoir in place. In order to simulate the no-reservoir condition, the *Use Reservoir* CS-TARP option in the Scenario Builder must be set to "False". Once the scenario is run, no reservoir will be simulated.

4.8 Exporting Results to Other Programs

It can be useful at various times to export results to other programs for further analysis. For example, if the user desired to compare the reservoir head for two different scenarios, the data would need to be exported by opening a table of the desired object and variable (e.g. *RES*, *NodeHead*), selecting the *Copy to Clipboard* option, and then pasting into Excel for example. This could be done for other scenarios, and then Excel plots could be used to generate a comparison of multiple scenarios.

Plots can also be saved to a variety of formats. If a plot is to be used in a report, it is recommended that the plot option *Save Image As* be selected from the right-click menu on a graph, and the EMF option be chosen. This image format is a vector format and will print out better than the other formats. For presentations, PNG is a good alternative. Note that if the user selects the *Copy* option, then a PNG version of the image as the user sees it is placed on the clipboard for pasting directly into Word, PowerPoint, or email programs.

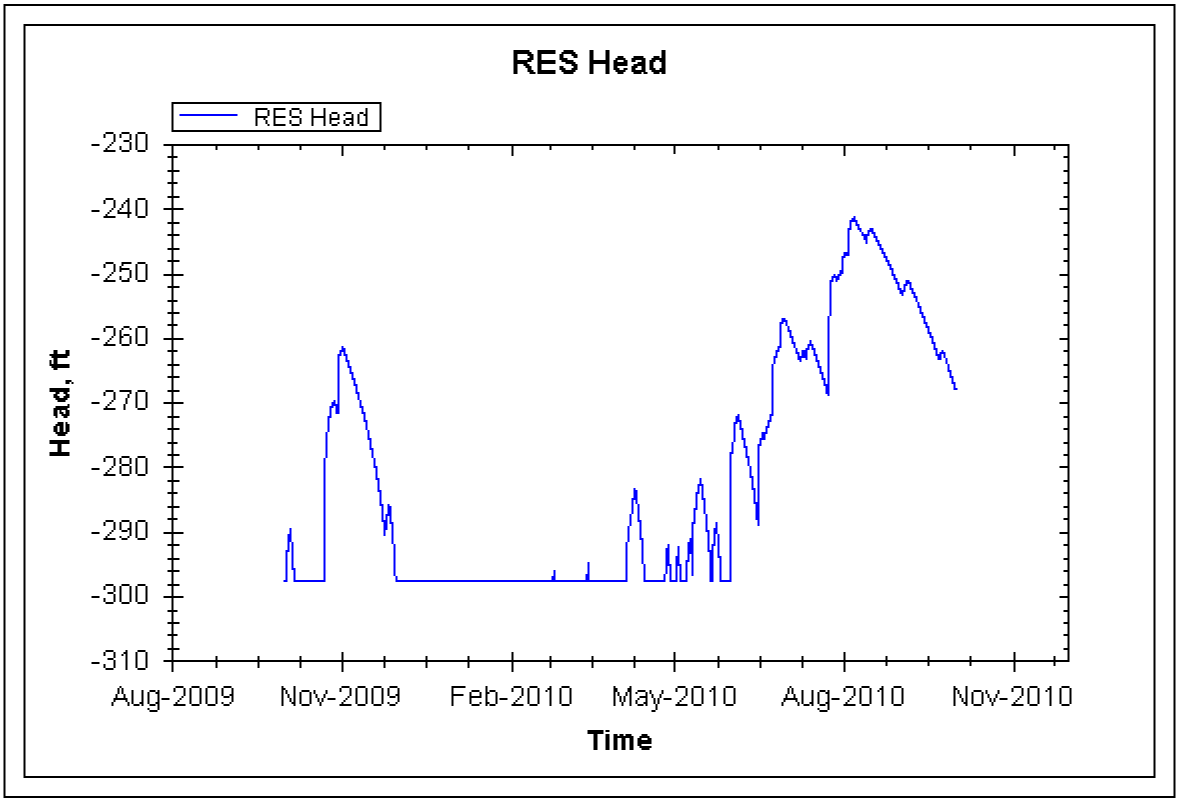


Figure 104: Reservoir head plot including one Thorn Creek inflow event

5. UPDATING METROFLOW

MetroFlow will check for program updates from a remote server every time it is launched, provided there is an internet connection available. If an update to the program is available, the user will be prompted to download the update.

In addition to new features for MetroFlow, updates to packages are occasionally necessary. In previous versions of MetroFlow, these updates were also known as package patches but are now referred to as package updates. These updates may be necessary to update geometry, individual models (e.g. IUHM), or to provide new features. Updates can be automatic or manual.

5.1 Manual Updates

For MetroFlow, it is possible to update a package without creating an entirely new package. This can be done using the *Package Patcher* program that comes with MetroFlow. This program is accessed from the *Tools* menu in the main MetroFlow interface. Once that menu option is selected, the user will be prompted to select the package update file, which always ends in the *.editpkg* file extension. The update will be applied upon restarting MetroFlow.

5.2 Automatic Updates

MetroFlow 1.5 and later include a feature for automatically updating packages. MetroFlow will check a remote repository when the program is loaded, display a list of available package updates to the user, and automatically apply the selected updates to the package that is being loaded. Note that each package will have its own remote repository and updates are only applied from the repository that they are linked to. (If a package is copied the updates will come from the same repository as the original package.) The user can hover over each entry in the package list to see a description of what the update contains. In addition, only the checked updates will be applied to the package when the *Apply Selected Updates* button is pressed. The user may also press the *No Thanks, Leave My Package Alone* button to ignore any updates and proceed as usual.

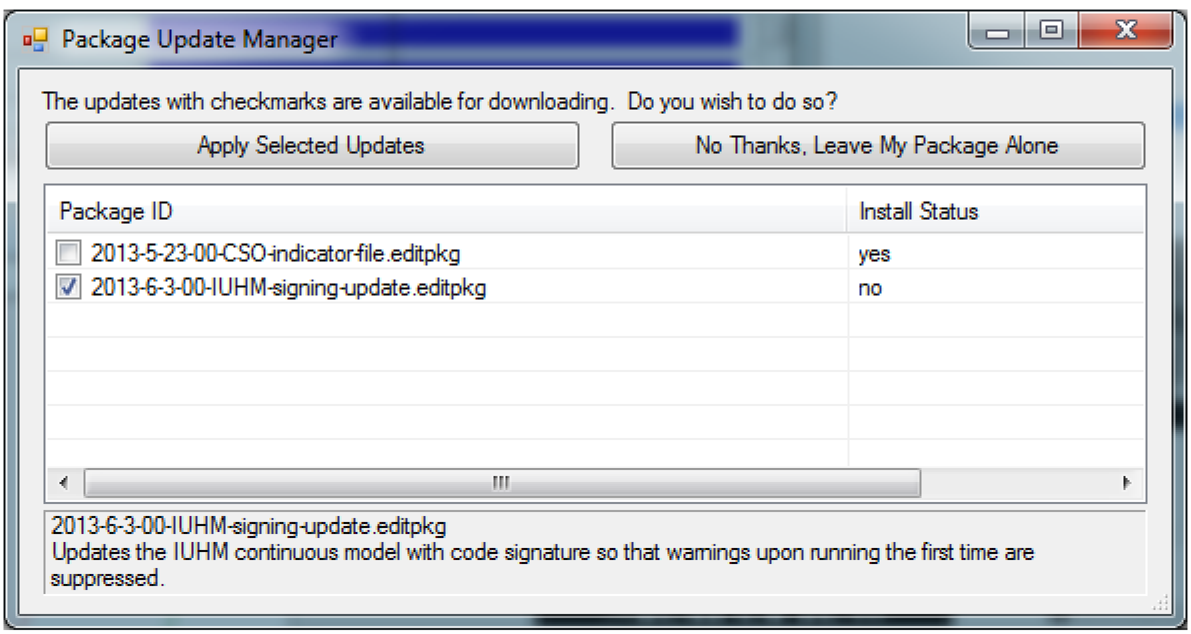


Figure 105: Automatic package update manager.

6. UPGRADING FROM METROFLOW/TCM 1.0 TO VERSION 1.7

MetroFlow 1.7 introduced several new features as well as changed existing windows and procedures. These changes were introduced to 1) fix program crashes, 2) include new features, and 3) support an enhanced timeseries database. The primary changes are as follows:

- The Timeseries Sets window has changed to better support simultaneous model runs, and sorting and organizing;
- The way timeseries mappings are created has moved from the Scenario Builder window to a separate primary Timeseries Mappings window;
- The Scenarios and Timeseries Sets windows are no longer by default on the sidebar but are included as main windows;
- A CSO Analysis tool has been created that allows for animations of CSOs as well as a graphical statistical summary of CSOs;
- Table and graph selection tools are easier to use;
- A progress bar in the main screen is included for all loading and export/import operations; previously the program hung;
- Automatic program updates;
- Automatic package updating;
- Support for a new version of ITM that is significantly faster.

The aforementioned features are documented in the preceding sections.

The new timeseries database does not by default import existing, version 1.0 timeseries sets. This must be done manually through the *Advanced* 

menu on the Timeseries Sets window. This menu is present only when the old database is present; when the old database has been imported the old database is disabled (although not physically removed from the hard drive) and the menu will not be present on future loads of the MetroFlow program. Clicking on the drop down error on that menu will reveal the *Import version 1 sets* menu item which when activated displays a form for the user to select which timeseries sets to import.

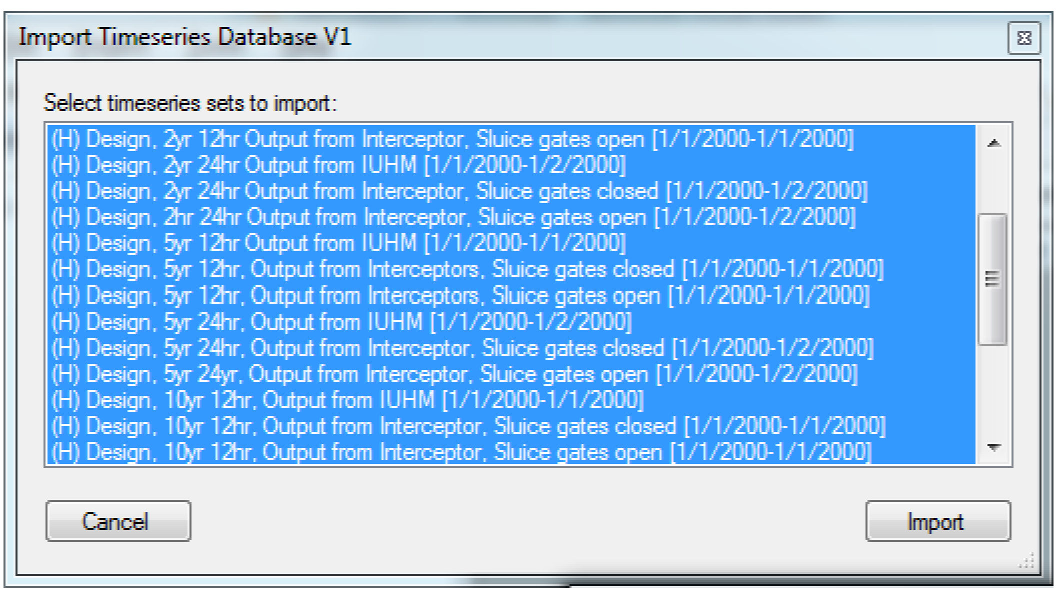


Figure 106: TCM version 1.0 timeseries database import form

By default all items are selected but clicking on a list item will unselect it from being imported. Pressing the Import button will import the selected timeseries sets into the new timeseries database.

In order for automatic package updates to work, a special patch needs to be applied to each package that is to be auto-updated. To apply this patch, download the patch file from the Hydrolab server:

> http://hydrolab.illinois.edu/software/metroflow-1.7/pkgupdates/2013-5-23-00-Enable-Auto-Package-Updates.editpkg

Inside of MetroFlow is the *Tools* menu, select this and then *Package Patcher*. The user will be prompted to select the patch file, and the downloaded patch file should be selected. MetroFlow should then be closed and reopened and automatic package updates will begin to be applied when the program is loaded.

7. HEC-DSS SUPPORT

The HEC Data Storage System, or HEC-DSS, is a database system designed to efficiently store and retrieve scientific data that is typically sequential such as time series data. The system was designed to make it easy for users and application programs to retrieve and store data. In MetroFlow, DSS import and export functions are provided for time series management, and DSS export is provided for scenario output.

HEC-DSS data files are organized by pathnames. Pathname may consist of up to 391 characters and is, by convention, separated into six parts, which may be up to 64 characters each. Pathnames are automatically translated into all upper case characters. They are separated into six parts (delimited by slashes "/") labeled "A" through "F," as follows:

/A/B/C/D/E/F/

For regular-interval time series data, the part naming convention is:

Part A: Project, river, or basin name (e.g. CAL-WRP)
Part B: Location (e.g. CA-10)
Part C: Data parameter (e.g. FLOW)
Part D: Starting date of block, in a 9 character military format
Part E: Time interval
Part F: Additional user-defined descriptive information

8. MODEL SIMULATION OPTIONS AND VARIABLES

Each model has several options that control simulation and output. They vary from model to model. Models that provide output to additional models all have the *Label for Output* and *Description for Output* fields. These provide MetroFlow with the name and description for output timeseries sets from that model. If they are not specified then they default to the scenario name followed by the name of the model in parentheses. In the following sections the model options are described.

8.1 IUHM

IUHM contains the *Label for Output* and *Description for Output* fields.

Output hydrographs are provided at each subcatchment.

8.2 Interceptor

The Interceptor model contains the *Label for Output* and *Description for Output* fields as well as the following options for controlling the simulation:

- Routing Time Step: This is the time step, in seconds, to use for performing dynamic wave flow routing in the pipes and junctions. Smaller routing time steps typically lower continuity error significantly but may add processing time particularly for long-term simulations.
- Conduit Lengthening Step: This is a SWMM5 option. In general a value of 0 gives the smallest continuity errors.
- I&I Factor: Inflow and Infiltration (I&I) flows are by default accounted for. I&I flows are determined by using the average rainfall at CDS-24, CDS-34-1, and CDS-55-1 as input to an East Rainfall Derived Inflow and Infiltration (RDII) area, and the average rainfall at CDS-6, CDS-10-1, and CDS-Spaulding as input to a West RDII area. These RDII areas are used by the SWMM engine of the Interceptor model to compute I&I.

8.2.1 Node Variables

The Interceptor model provides the following outputs at all nodes:

- Depth: The simulated flow depth at the node;
- Head: This label displays the hydraulic head of a node, which is a combination of the elevation and water pressure at that point; these values are given in CCD;
- Volume: Nodes modeled as storage units (e.g. the reservoir) will display the volume of water in the node in cubic feet;
- Lateral Flow: Lateral flow is the flow that enters a node from an external inflow (such as a hydrograph);
- Total Flow: Total flow is the sum of the flow entering a node from upstream as well as an external inflow;
- Flooding: Flooding is defined as excess overflow when the node is at full depth.

8.2.2 Link Variables

The Interceptor model provides the following outputs at all links:

- Flow: The flow rate in the conduit;
- Depth: The average water depth in the conduit for a given time;
- Velocity: The flow velocity in the conduit;

- Froude No.: The Froude number F_r indicates the effect of gravity, meaning subcritical or supercritical flow. The Froude number is determined by variables V=average velocity, g=gravity and $D_h =$ the hydraulic depth, from the equation:

$$F_r = \frac{V}{\sqrt{gD_h}} \quad where\ F_r \begin{bmatrix} \text{<1 indicates subcritical flow} \\ \text{=1 indicates critical flow} \\ \text{>1 indicates supercritical flow} \end{bmatrix}$$

- Capacity: The Capacity is the ratio of depth to full depth.

8.3 TARP-SWMM

The TARP-SWMM model contains the following options for controlling the simulation:

- Routing Time Step: This is the time step, in seconds, to use for performing dynamic wave flow routing in the pipes and junctions. Smaller routing time steps typically lower continuity error significantly but may add processing time particularly for long-term simulations.
- Global Head: A non-zero global head will initialize the TARP tunnels and reservoir to the head level that is provided. Note that this is not a depth but is rather water surface elevation, CCD.

8.3.1 Node Variables

The TARP-SWMM model provides the following outputs at all nodes:

- Depth: The simulated flow depth at the node;
- Head: This label displays the hydraulic head of a node, which is a combination of the elevation and water pressure at that point; these values are given in CCD;
- Volume: Nodes modeled as storage units (e.g. the reservoir) will display the volume of water in the node in cubic feet;
- Lateral Flow: Lateral flow is the flow that enters a node from an external inflow (such as a hydrograph);
- Total Flow: Total flow is the sum of the flow entering a node from upstream as well as an external inflow;
- Flooding: Flooding is defined as excess overflow when the node is at full depth.

8.3.2 Link Variables

The TARP-SWMM model provides the following outputs at all links:

- Flow: The flow rate in the conduit;
- Depth: The average water depth in the conduit for a given time;
- Velocity: The flow velocity in the conduit;
- Froude No.: The Froude number F_r indicates the effect of gravity, meaning subcritical or supercritical flow. The Froude number is determined by variables V=average velocity, g=gravity and $D_h =$ the hydraulic depth, from the equation:

$$F_r = \frac{V}{\sqrt{gD_h}} \quad where\ F_r \begin{bmatrix} \text{<1 indicates subcritical flow} \\ \text{=1 indicates critical flow} \\ \text{>1 indicates supercritical flow} \end{bmatrix}$$

- Capacity: The Capacity is the ratio of depth to full depth.

8.4 TARP-ICAP

The ICAP model provides the user with the following options for controlling simulations:

- Routing Time Step: This is the time step, in seconds, to use for performing step-wise steady flow routing in the pipes and junctions. Routing time steps should be approximately half of the input timeseries; for water years, 1800 seconds, for historical/design storms, 15 seconds.
- ETP Threshold: The **E**vent-**T**o-**P**umping threshold is used for water year simulations and controls the number of days (fractional days allowed) between when an event ceases and pumping begins. The default value for this is 2 days. Events are considered to have ceased when the total inflow to the system drops below the *Inflow Threshold*.
- Inflow Threshold: The inflow threshold determines at what point the system is considered to be having a storm event and is given in CFS. The default is 5 CFS. This is used for water year simulations.
- Pumping Rate: For water year simulations, the pumping rate is the constant rate at which water is pumped out of the system and reservoir, once the ETP threshold has been exceeded. **This is given in CFS**.

8.4.1 Node Variables

The ICAP model provides the following outputs at all nodes:

- Depth: The simulated flow depth at the node;
- Head: This label displays the hydraulic head of a node, which is a combination of the elevation and water pressure at that point; these values are given in CCD;
- Volume: Nodes modeled as storage units (e.g. the reservoir) will display the volume of water in the node in cubic feet;
- Lateral Flow: Lateral flow is the flow that enters a node from an external inflow (such as a hydrograph);
- Total Flow: Total flow is the sum of the flow entering a node from upstream as well as an external inflow.

8.4.2 Link Variables

The ICAP model provides the following outputs at all links:

- Flow: The flow rate in the conduit;
- Depth: The average water depth in the conduit for a given time;
- Velocity: The flow velocity in the conduit;
- Froude No.: The Froude number F_r indicates the effect of gravity, meaning subcritical or supercritical flow. The Froude number is determined by variables V=average velocity, g=gravity and $D_h =$ the hydraulic depth, from the equation:

$$F_r = \frac{V}{\sqrt{gD_h}} \quad where\ F_r \begin{bmatrix} <1 \text{ indicates subcritical flow} \\ =1 \text{ indicates critical flow} \\ >1 \text{ indicates supercritical flow} \end{bmatrix}$$

- Capacity: The Capacity is the ratio of depth to full depth.

8.5 TARP-ITM

The ITM model provides the user with various parameters for modifying how a simulation takes place. In the current version of MetroFlow, ITM version 1.4 is supported which runs significantly faster. The key to fast runs with ITM is to keep the *Pressure Wave Celerity* parameter low, typically around 50. For more accuracy a celerity value of up to 1000 may be used but this drastically increases computation time. In testing, it does not appear that a celerity value of 50 makes significant difference in most results and increases stability. The following parameters are provided:

- Minimum Number of Grid Points: the minimum number of cells to divide a pipe up into
- Maximum Number of Cells per Pipe: the maximum number of cells in a pipe
- Number of Cells to Plot: the number of values to plot within each pipe
- Pressure Wave Celerity: the celerity, given in meters per second
- Tolerance Normal: a tolerance that is used in places where computation needs a medium level of accuracy
- Tolerance Low: a tolerance that is used in places where computation does not need a high level of accuracy
- Tolerance High: a tolerance that is used in places where computation needs a high level of accuracy
- Tolerance Transition: a tolerance that is used in places where transition occurs between free surface and open channel
- Maximum Iterations: the maximum number of iterations to use when solving
- Maximum Time Step: ITM varies the time step but will never increase above this value (can be less than a second)

8.5.1 Node Variables

The TARP-ITM model provides the following outputs at all nodes:

- Depth: The simulated flow depth at the node;
- Head: This label displays the hydraulic head of a node, which is a combination of the elevation and water pressure at that point; these values are given in CCD;
- Volume: Nodes modeled as storage units (e.g. the reservoir) will display the volume of water in the node in cubic feet;
- Lateral Flow: Lateral flow is the flow that enters a node from an external inflow (such as a hydrograph);
- Total Flow: Total flow is the sum of the flow entering a node from upstream as well as an external inflow;
- Flooding: Flooding is defined as excess overflow when the node is at full depth.

8.5.2 Link Variables

The TARP- ITM model provides the following outputs at all links:

- Flow: The flow rate in the conduit;
- Depth: The average water depth in the conduit for a given time;
- Velocity: The flow velocity in the conduit;
- Froude No.: The Froude number F_r indicates the effect of gravity, meaning subcritical or supercritical flow. The Froude number is determined by variables V=average velocity, g=gravity and $D_h =$ the hydraulic depth, from the equation:

$$F_r = \frac{V}{\sqrt{gD_h}} \quad where\ F_r \left[\begin{array}{c} \text{<1 indicates subcritical flow} \\ \text{=1 indicates critical flow} \\ \text{>1 indicates supercritical flow} \end{array} \right.$$

- Capacity: The Capacity is the ratio of depth to full depth.

In addition, depth and flow are provided at intermediate locations throughout each conduit. This helps to visualize transient conditions.

8.6 IUHM+CityModel

The IUHM+CityModel layer is a combination of the City of Chicago InfoWorks model with subcatchments simulated by IUHM in regions not serviced by the City of Chicago sewers. It provides the following options for controlling the simulation:

- Routing Time Step: This is the time step, in seconds, to use for performing hydraulic and hydrologic computations. It should never be larger than 30 seconds.
- Parallel Execution: Allows IUHM and the City of Chicago InfoWorks model to execute in parallel which will save simulation time.

Node and link results for this layer are not typically available for much of the City of Chicago InfoWorks portion of the model; they are only available for nodes and links directly connected to CSO and dropshaft locations.

8.6.1 Node Variables

The IUHM+CityModel model provides the following outputs selected nodes:

- Depth: The simulated flow depth at the node;
- Head: This label displays the hydraulic head of a node, which is a combination of the elevation and water pressure at that point; these values are given in CCD;
- Total Flow: Total flow is the sum of the flow entering a node from upstream as well as an external inflow;

8.6.2 Link Variables

The IUHM+CityModel model provides the following outputs selected links:

- Flow: The flow rate in the conduit;
- Depth: The average water depth in the conduit for a given time;
- Velocity: The flow velocity in the conduit;

8.7 CS-TARP

The CS-TARP layer is an InfoWorks model that combines connecting structures, dropshafts, and the TARP tunnels for the Mainstream and Des Plaines systems. It provides the following options for controlling the simulation:

- Routing Time Step: This is the time step, in seconds, to use for hydraulic computations. It should never be larger than 30 seconds.
- Use Pumping: This option defaults to False but can be set to True for extended period simulations such as water years. It enables pumping out of the McCook reservoir. This option must be coupled with a historical pumping record timeseries (see section 4.7.1) in order to function.
- Use Reservoir: This option defaults to True. It can be set to False to simulate the TARP tunnels without the presence of McCook.

8.7.1 Node Variables

The CS-TARP model provides the following outputs at all nodes:

- Depth: The simulated flow depth at the node;
- Head: This label displays the hydraulic head of a node, which is a combination of the elevation and water pressure at that point; these values are given in CCD;

- Total Flow: Total flow is the sum of the flow entering a node from upstream as well as an external inflow;

8.7.2 Link Variables

The CS-TARP model provides the following outputs at all links:

- Flow: The flow rate in the conduit;
- Depth: The average water depth in the conduit for a given time;
- Velocity: The flow velocity in the conduit;

9. DOCUMENT REVISION HISTORY

9.1 June 2013

- Changes made to bring documentation in sync with MetroFlow version 1.7

9.2 September 2013

- Documented the following new features:
  - Combination menu-button for plotting (section 3.14.2)
  - Plotting options (section 3.14.2)
  - CSO Display Name tool (section 3.17)
- Updated documentation for the following features:
  - System table changes (Interceptor System Tables)
  - Noted that the CSO statistics tools give output in million gallons, not cubic feet (section 3.15.2)
  - Results Animator tool (section 3.12)

9.3 January 2014

- Documented the following new features:
  - Package selection dialog at startup
  - Filtering and validation of timeseries sets in Scenario Builder
  - Additional timeseries import abilities (section 3.10.3, section 3.10.7, section 3.10.8)
  - ITM initialization files (section 3.18)
  - InfoWorks layers (section 8.6 and section 8.7)
- Removed the “Open package” and “Recent packages” File menu options

9.4 February 2014

- Documented the features in the CS-TARP model
- Added tutorials for Mainstream/Des Plaines