US Army Corps
of Engineers
Hydrologic Engineering Center

HEC-ResSim
Reservoir System Simulation

Quick Start Guide

Version 3.3
February 2021

CPD-82a

REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)*	2. REPORT TYPE	3. DATES COVERED *(From - To)*
February 2021	Computer Program Documentation	

4. TITLE AND SUBTITLE

HEC-ResSim
Reservoir System Simulation Quick Start Guide
Version 3.3

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)

Joan D. Klipsch, Marilyn B. Hurst

5d. PROJECT NUMBER

5e. TASK NUMBER

5F. WORK UNIT NUMBER

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

US Army Corps of Engineers
Institute for Water Resources
Hydrologic Engineering Center (HEC)
609 Second Street
Davis, CA 95616-4687

8. PERFORMING ORGANIZATION REPORT NUMBER

CPD-82a

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

10. SPONSOR/ MONITOR'S ACRONYM(S)

11. SPONSOR/ MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION / AVAILABILITY STATEMENT

Approved for public release; distribution is unlimited.

13. SUPPLEMENTARY NOTES

Also, see HEC-ResSim User's Manual, CPD-82

14. ABSTRACT

The U.S. Army Corps of Engineers' Hydrologic Engineering Center's Reservoir System Simulation (HEC-ResSim) is a computer program comprised of a graphical user interface (GUI) and a computational program to simulate reservoir operations. Included are data storage and management capabilities and graphics and reporting facilities. HEC's Data Storage System (HEC-DSS) is used for storage and retrieval of input and output time series data.

15. SUBJECT TERMS

HEC-ResSim, Reservoir Simulation, computer program

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			19b. TELEPHONE NUMBER
U	U	U	UU	49	

HEC-ResSim
Reservoir System Simulation

Quick Start Guide

Version 3.3
February 2021

US Army Corps of Engineers
Institute for Water Resources
Hydrologic Engineering Center
609 Second Street
Davis, CA 95616

(530) 756-1104
(530) 756-8250 FAX
www.hec.usace.army.mil

CPD-82a

Reservoir System Simulation, HEC-ResSim

Software Distribution and Support Statement:

Use of the software described by this document is controlled by certain terms and conditions. The user must acknowledge and agree to be bound by the terms and conditions of usage before the software can be installed or used. For reference, a copy of the terms and conditions of usage are included below so that they may be examined before obtaining the software. The software described by this document can be downloaded for free from our internet site (www.hec.usace.army.mil).

HEC cannot provide technical support for this software to non-Corps users. See our software vendor list (on our web page) to locate organizations that provide the program, documentation, and support services for a fee. However, we will respond to all documented instances of program errors. Documented errors are bugs in the software due to programming mistakes not model problems due to user-entered data.

Terms and Conditions for Use of HEC-ResSim:

Table of Contents

List of Figures

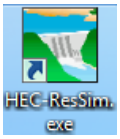
Introduction

Welcome to the HEC-ResSim Quick Start Guide. This document will take you through the steps needed to develop a reservoir simulation model using HEC-ResSim. Use this document to get a quick overview of the program and its basic features. For more detailed information, refer to the HEC-ResSim User's Manual *[references to the HEC-ResSim User's Manual are given in this type of formatted text to provide you with a cross reference for locating additional information]*. This guide provides basic steps for using the Graphical User Interface (GUI) to enter data, run simulations and view results. "New" features (from Version 3.1 to Version 3.3) are indicated as highlighted text.

Starting the Program

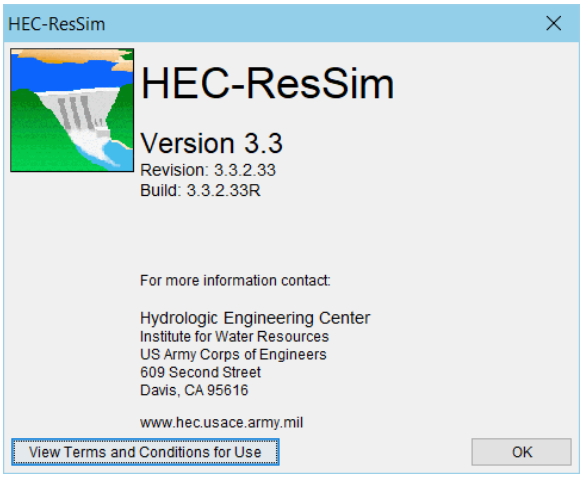
Once HEC-ResSim has been installed on your PC, start the program as follows:

From Microsoft Windows -



On your desktop, double-click on the HEC-ResSim icon or, from the **Start** menu, select **All Programs → HEC → HEC-ResSim → HEC-ResSim 3.3**.

When you first start the program, you will see an introductory screen containing the version and contact information. This screen will disappear after a few moments. However, the same information is available from ResSim's **Help** menu by selecting the **About** option.



Exiting the Program

To exit the program, from the **File** menu on the Main Window, select **Exit**. The program will prompt you to save all files.

First, Some Terminology…

What is a Watershed?

In HEC-ResSim terminology, a **Watershed** is a collection of data associated with a particular reservoir system or study *[Chapter 2]*. Types of data include Watershed Setup (Configurations) *[Chapter 3]*, Reservoir Networks (including Operation Sets *[Chapter 10]* and Alternatives *[Chapter 14]*), and Simulations *[Chapter 15]*. After performing a Simulation, which will include one or more Alternatives, you can view and print results in both graphical and tabular form *[Chapters 16 & 17 and Appendix E]*.

What are Watershed Locations?

A **Watershed Location** is the place on disk where you can store watersheds; also called a **Shortcut**, model directory, or working directory *[Section 3.2.1]*. You can have multiple watershed locations, although a given watershed cannot span multiple locations. Each watershed location should be given a descriptive name, such as *My Watersheds* or *General Projects*, and a path specification that identifies the directory where you want to create and edit your watershed data.

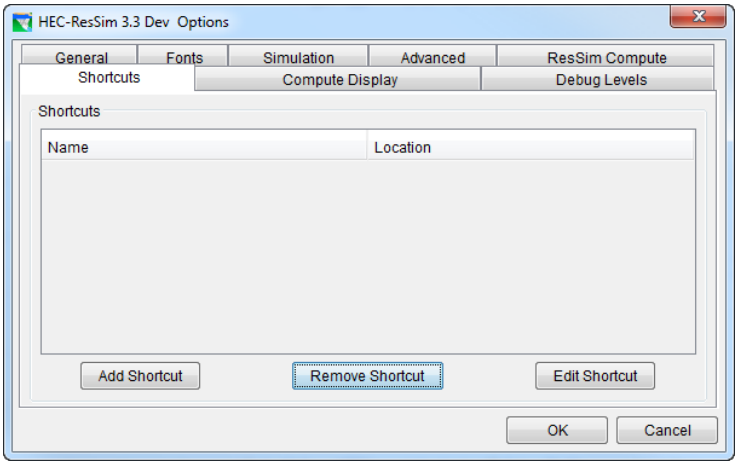


Figure 1. Tools Menu, Options Editor, Watershed Locations

Before you can create or open a watershed, you must define at least one watershed location. Do so by selecting **Options** from the **Tools** menu. The Options Editor will appear as shown in Figure 1. The **Shortcuts** tab of the Options editor is used to define Watershed Locations. To add a new location to the list, press the **Add Shortcut**… button. The **Add Shortcut** dialog will appear (Figure 2). In the **Location** field, you can enter the full path to the working directory or you can **Browse** to the area on disk where you want to build your watershed. Next, enter the name you are giving this location in the **Name** field and press **OK**.

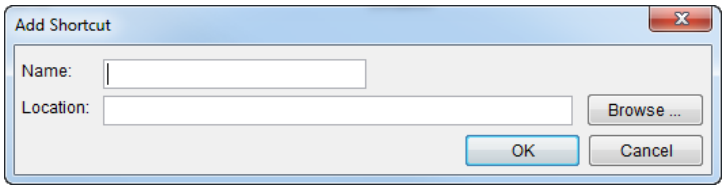


Figure 2. Add Shortcut (Watershed Location) dialog

What are Modules?

The ResSim program is divided into separate sets of functions called **modules** *[Section 2.2]*. Each module provides access to specific types and directories of data within the watershed data tree. The three modules within the ResSim program are:

- **Watershed Setup** *[Section 2.2.1 Chapters 3—5]*
- **Reservoir Network** *[Section 2.2.2 and Chapters 6—14]*
- **Simulation** *[Section 2.2.3 and Chapter 15]*

Figure 3 provides a graphical illustration of concepts and typical workflow through the three modules that make up HEC-ResSim. Refer back to this figure as you learn more about the function of each module to help keep the concepts organized.

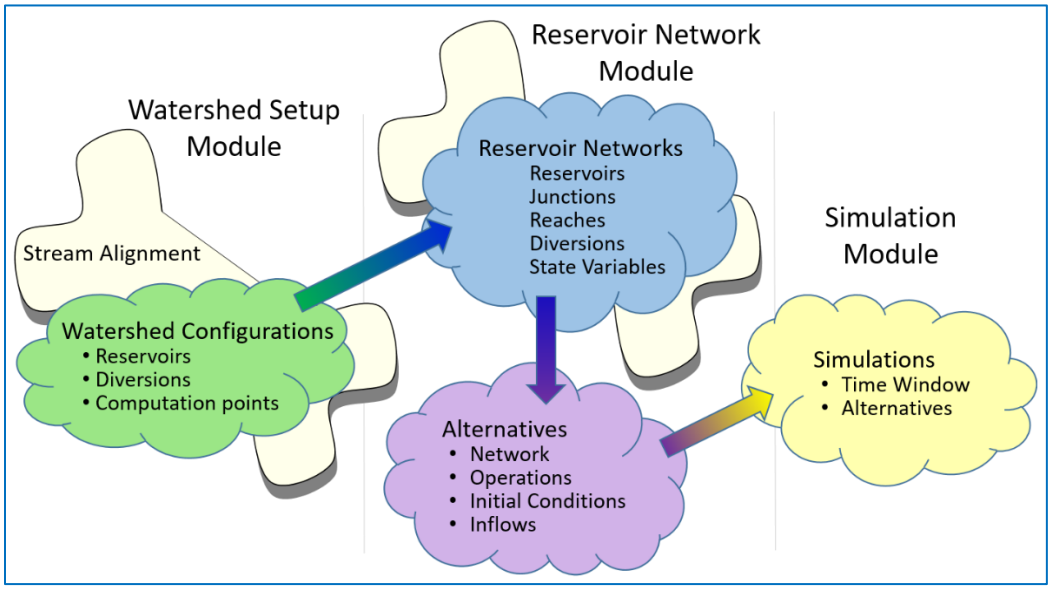


Figure 3. ResSim Module Concepts

A. Watershed Setup Module

The opening window of the **Watershed Setup Module** is shown in Figure 4. As you can see, most menus and tools are inactive (grayed-out). They will remain so until a watershed is created or opened, which can be done by selecting **New Watershed** or **Open Watershed** from the **File** menu in the Watershed Setup Module. When a watershed is created, ResSim constructs a directory tree where

all files pertaining to the watershed will be stored. Those files specific to the ResSim model (and not shared by other models like HMS, FIA, etc.) are stored in the watershed directory tree in a subdirectory named **rss**.

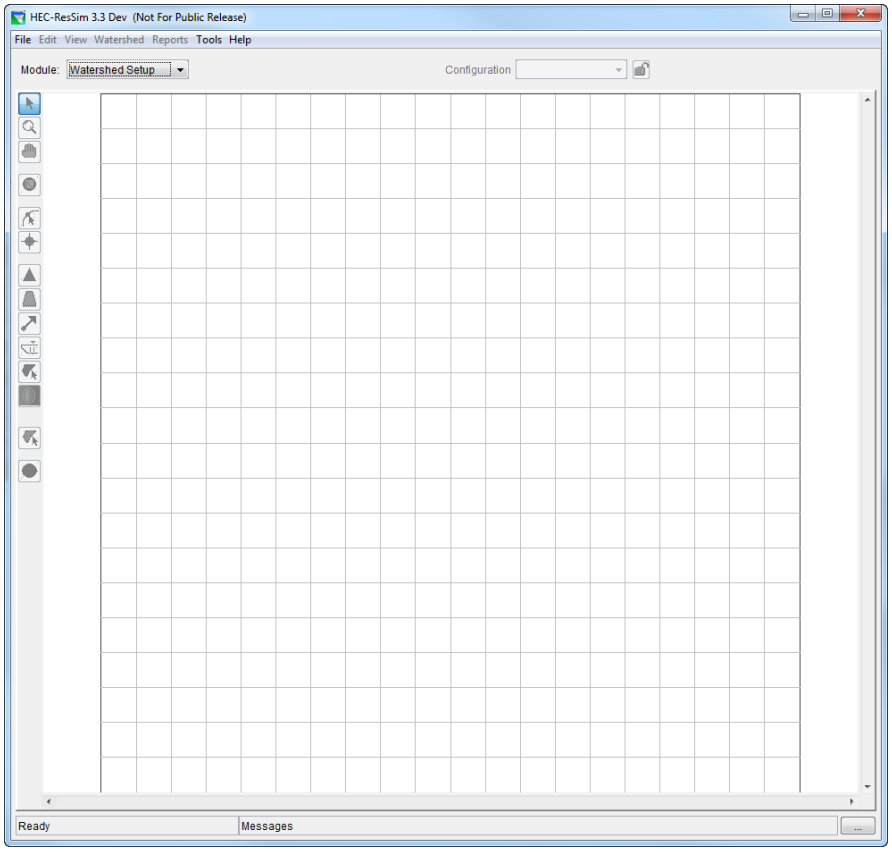


Figure 4. Watershed Setup Module, Opening Window

The **Watershed Setup Module** main window has the following menus:

File This menu will appear in all modules and most editors. In the main window, it always includes the options **Open Watershed**, **Save Watershed**, **Save Map As**, and **Exit**. In the Watershed Setup Module, the option **New Watershed** is also included. These options relate to the overall watershed data.

Edit This menu will appear in all modules and most editors. This menu is module specific. The edit options will be relevant to data elements currently available for modification. The edit options available in the Watershed Setup module include: **Watershed Data**, **All Impact Areas**, **Projects**, **Drawing Properties**, **and Allow Editing**.

View View options relate to all modules. Options include adding **Layers** that are displayed in the map region, selecting the **Unit**

System for *viewing*, and selecting whether or not to show the **Grid Lines** in the map region.

Watershed This menu is only available in the Watershed Setup Module. Watershed options relate to the entire watershed, as well as to the individual configurations and the stream alignment. Options include **Configuration Editor**, **Update Computation Points**, **Import** and **Export** (of Stream Alignment), and **Save Configuration**.

Reports Predefined reports and tables are listed in this menu. This menu appears in all modules, but its content is module specific. In the Watershed Setup and Reservoir Network Modules, most of the reports detail input data. In the Simulation Module, the reports detail simulation results.

Tools User setting options, properties, and extended tools such as **HEC-DSSVue** and scripting are found here.

Help This menu provides access to the user documentation in pdf form. It also allows for the installation of example watersheds, after a watershed location has been defined *[Section 3.2.1]*. This option also gives the ResSim version information.

Figure 5 shows the main window of the **Watershed Setup Module** after opening an existing watershed (HayesBasin). A map region occupies the majority of this window, where the user must place the project elements (e.g., reservoirs) onto the stream alignment (all configurations that will be defined for this watershed will be on this alignment). A background map can be added (but is not required) to the map region to help identify the streams for the alignment.

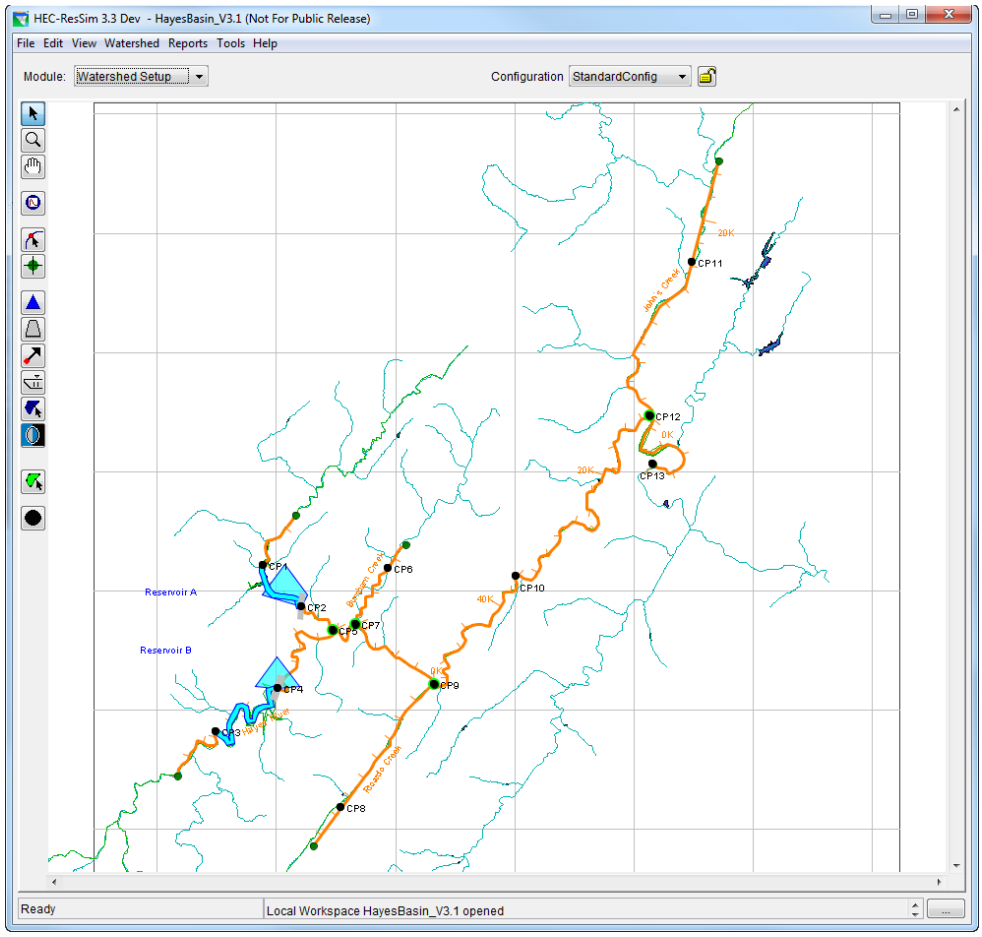


Figure 5. Watershed Setup Module, Opened Watershed, HayesBasin

B. Reservoir Network Module

The **Reservoir Network Module** is used for editing element (e.g., reservoir, computation point, and routing reach) data and placing additional elements onto the stream alignment. Figure 6 shows an example of the Reservoir Network Module window for HayesBasin. This example shows a Network named "NormalNetwork2" which was created by selecting **New** from the **Network** menu and selecting "StandardConfig" from the Configuration list. A map region occupies the majority of this window, where the user must place Routing Reaches (required for connectivity) and Diverted Outlets (if appropriate) onto the stream alignment.

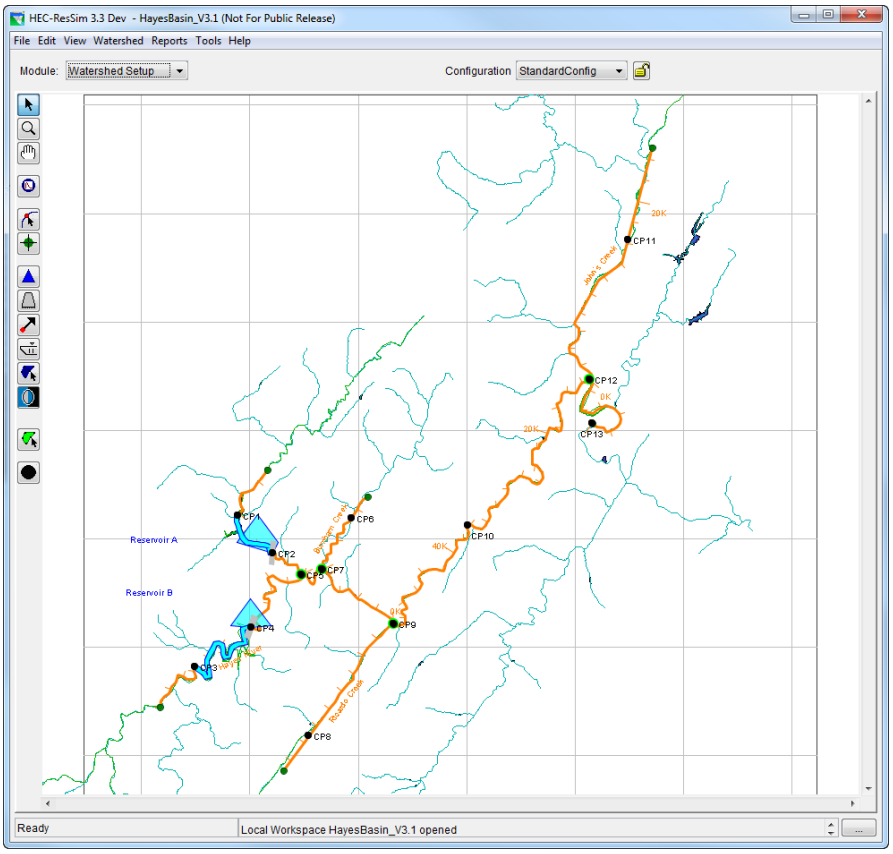


Figure 6. Reservoir Network Module, HayesBasin

Element data is edited within the Reservoir Network Module. First, select the appropriate tool from the tool bar on the left side of the window, then right-click on the element. A shortcut menu appears with options to edit data defining the element. Alternatively, select the general pointer tool from the top of the toolbar on the left side and right-click on any element to view the shortcut menu.

The menus that are new or unique to the **Reservoir Network Module** are listed and described below:

Edit This menu will appear in all modules and most editors. The edit options available for reservoir networks include: **Reservoirs**, **Reaches**, **Junctions**, **Diversions**, and **Reservoir Systems, State Variables, Import Element Properties**, and **Allow Network Editing**.

Network Network options are used in defining the network and configuration to be used. Options include **New**, **Open**, **Edit**, **Save**, **Save As**, and **Rename**. Also included are the **Update Network from Configuration**, **Import Network**, and **Delete Networks** options.

Alternative The only option in this menu is **Edit**. This opens the Alternative Editor, which will enable you to create, edit, and save alternatives.

Reports Predefined reports and tables are listed in this menu. Options include **Reservoir List**, **Reach List**, **Junction List**, and **Diversion List**. An **Advanced** option is also available that shows a summary table of the **Network** elements and their connectivity, as well as a listing of **Nodes** in the network.

C. Simulation Module

The **Simulation Module** is designed to facilitate the analysis phase of reservoir modeling. In this module you will create and run Simulations. Each Simulation is a combination of the time window and Alternatives that you want to analyze.

A view of the Simulation Module is shown in Figure 7. This view shows an open Simulation (Oct_1999_Event) with one of the Alternatives (StandNorm2) of the Simulation set as active (bolded alternative).

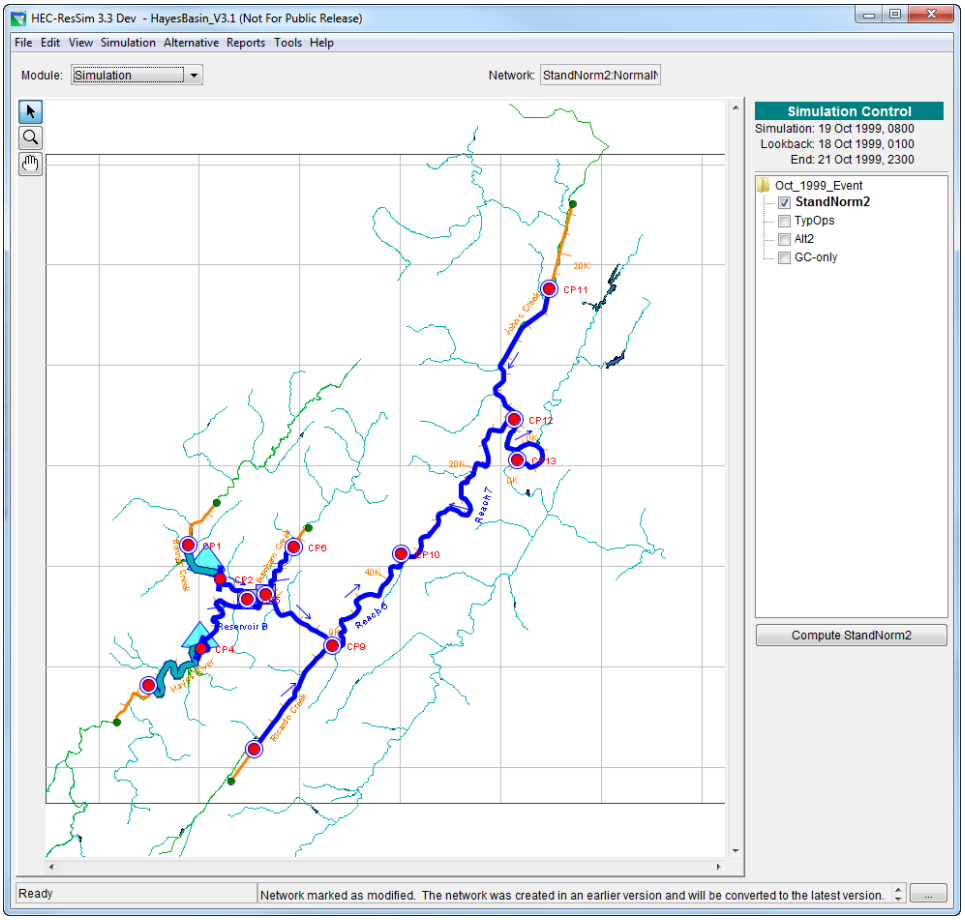


Figure 7. Simulation Module, HayesBasin

 The checkmark on the left side of the Alternative indicates that the user requests that the results are to be shown (plots, tables, etc.). Results for multiple alternatives can be shown for comparison purposes by checking all alternatives of interest. However, only one alternative can be set to **active** (**bold**) at a time and that alternative's data is reflected in the map region and is, therefore, available for editing.

The menus that are new or unique to the **Simulation Module** are listed and described below:

Edit This menu will appear in all modules and most editors. The edit options available for the Simulation Module include: **Script List** selection, **Set Active Alternative,** and editors for **Reservoirs**, **Reaches**, **Junctions**, **Diversions**, **Reservoir Systems** and **State Variables**. Also available is the **Run** editor for the active alternative.

Simulation Simulation options are provided for creating, opening, editing, saving, and running simulations. Options include **New**, **Open**, **Re-Open**, **Close**, **Simulation List**, **Replace from Base**, **Edit**, **Save**, **Delete**, **Info**, and **Run Manager**. Also, **Release Overrides** can be defined from this menu.

Alternative The only option in this menu is **Edit**. This opens the Alternative Editor, which will enable you to edit and save alternatives.

Reports Predefined reports and tables are listed in this menu. Options include **Reservoir Summary**, **Flow Summary**, **Power Summary**, **Gates Summary**, **Stage Summary** and **Release Decision** report. Also available from this menu are **User Reports**, **Compute** logs, and **Network** Element Lists (**Reservoir**, **Reach**, **Junction**, and **Diversion**, as well as the **Advanced Network Connectivity** Summary).

Some Other Useful Terms

The following is a list of terminology commonly used throughout the ResSim Quick Start Guide. This page may be useful as you progress through the manual.

Alternative An alternative is a specific composition of watershed system characteristics and necessary data to perform a scenario-based simulation.

Component	In ResSim, a component, or element, refers to a piece of the watershed model, such as a reservoir, tributary, stream node, junction, or computation point and the associated physical data.
Computation Point	ResSim allows the user to add computation points on the stream alignment. These are placed at locations where the model will need information that is not generated by the placement of projects. Examples include stream gage locations, control points, confluences, and inflow points.
Configuration	In brief, a **Configuration** is the identification of a "set" of projects within your watershed that you want to model in your watershed. If more than one set of projects will be considered in a study, then more than one Configuration will be needed. While you add projects to your Configurations, a special "superset" (named Study) is developed by the program to contain all projects from all Configurations in the watershed. If multiple programs (e.g., HEC-HMS, HEC-FIA, HEC-RAS, etc) are going to be applied to your watershed, then Configurations should be developed by the modeling team before the models for an individual program are prepared.
Operation Set	In ResSim, an **Operation Set** is the operation plan or scheme upon which a reservoir bases its decisions regarding how much water to release at each time step of a simulation run *[Chapter 10]*. Key elements of an Operation Set include reservoir zones, Guide Curve selection, and rules. A reservoir can have more than one Operation set, but only one can be active within an alternative.
Projects	Within ResSim, a project primarily refers to a reservoir. However, a levee is also considered a project. A configuration reflects a collection ("set") of projects.
Simulation	In ResSim, a simulation is where the computations are performed for alternatives within a user-defined time window. Results of alternatives are available in a simulation.

Getting Down to Business…

The basic steps for developing a reservoir model are outlined below. These steps are illustrated in the sections that follow.

Steps for Developing a Reservoir Model

Developing a ResSim model entails the following steps:

A. Set Up the Watershed

1. Create/Open a Watershed *[Sections 3.2 and 1.3]*
2. Add Map files *[Section 3.3 and Appendix B]*

3. Draw a Stream Alignment *[Chapter 4]*
4. Create a Configuration *[Section 3.4]*
5. Place Project Elements into the Configuration *[Section 3.4.3]*

B. Develop a Reservoir Network

1. Import/Create/Open a Network *[Sections 6.2 and 12.9]*
2. Add Routing Reaches *[Section 7.2]*
3. Edit Junction Properties *[Section 7.1]*
4. Edit Reach Properties *[Section 972]*
5. Edit Reservoir Properties
   a. Physical Data *[Chapter 9]*
   b. Operations Data *[Chapters 10 and 11]*
   c. Observed Data *[Sections 8.3 and 14.10]*

C. Define Alternatives

1. Create an Alternative *[Section 14.3]*
2. Define the Run Control Specifications *[Section 14.4]*
3. Select the Operation Scheme(s) *[Section 14.5]*
4. Specify Lookback Conditions *[Section 14.8]*
5. Identify Time-Series data records from DSS *[Sections 14.9 and 14.10]*
6. (*Optional*) Define Hotstart Options *[Sections 14.11]*

D. Perform a Simulation

1. Create/Open a Simulation *[Sections 15.2 and 15.3]*
2. Select/Activate Alternatives *[Section 15.4.1]*
3. Compute *[Section 15.4]*
4. View Results *[Chapters 16 & 17 and Appendix E]*
5. (*Optional*) Specify Overrides *[Section 15.7.3]*
6. Manage Simulation Data *[Section 15.8]*

A. Set Up the Watershed

1. Create/Open a Watershed

In the **Watershed Setup Module**, choose **New Watershed** from **File** menu. The new watershed screen will appear, as illustrated in Figure 8. Enter a name for your watershed and a detailed description in the appropriate fields. From the **Location** list, select where you want your watershed stored. If the **Location** list is empty, press **Cancel**, and go back to page 2 to see how to add a Watershed Location. Use the **Units** list to select the unit system that the watershed will use when storing its data. The available unit systems are English and SI (i.e., US Customary and International System of Units). Then select a **Time Zone**. The watershed time zone is always specified as some offset from GMT, which does

NOT recognize Daylight Savings Time. When you have completed all fields, press **OK**.

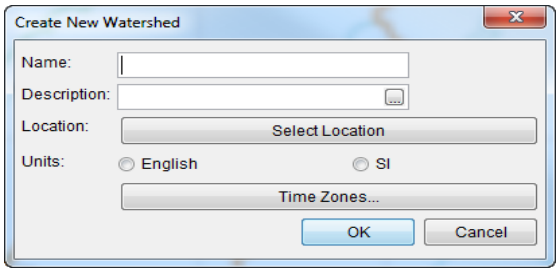


Figure 8: New Watershed

2. Add a Map

When a watershed is created, ResSim constructs a whole directory tree. One of the subdirectories in the watershed tree is the **maps** directory. Using a file system viewer (like Windows Explorer), copy the files you will use as background maps into the maps directory of your watershed:

<watershed location>\base\<watershed name>**maps**

The supported types of map files include DLG (USGS Digital Line Graphs), SHP (ArcView® shapefiles), DXF (Autocad® Digital Exchange Format), DEM (USGS Digital Elevation Models), IMG (Raster Image), ASC (ArcInfo® ASCII grid), and NET (ASCII NetTIN) *[Appendix B]*.

To display a map in the watershed, go to **View**, **Layers…** This will open the Layers Editor. First, make the layers editable by selecting **Allow Layer Editing** from the **Edit** menu. This is illustrated in Figure 9. Then, from the **Maps** menu, select **Add Map Layer**. This will open another window from which you can select the map file you want to view. You can select and deselect maps by un-checking the checkbox next to the map name.

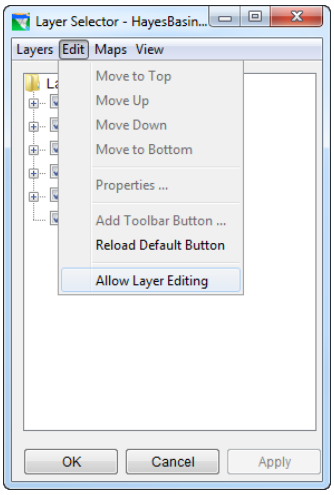


Figure 9: Allow Layer Editing

Note that it is not necessary to display a map. However, it is convenient to have the map as a guide when creating stream alignments *[Chapter 4]*.

Zoom. To zoom in or out on the map area, select the zoom tool on the left side of the window. To zoom in, click and drag over the desired viewing area. To zoom out, right-click until the desired viewing area is visible. Alternately, to zoom out to the original display, go to **View, Zoom to All**.

3. Draw a Stream Alignment

The next step is to create a stream alignment on which stream reaches and reservoirs can later be located. First, ensure that you are working in the Watershed Setup Module. If you aren't, then be sure to select Watershed Setup from the Module list. Second, from the **Edit** menu, activate **Allow Editing** (a check mark will appear to the left and the "yellow lock" in the upper right portion of the window will lock to indicate that you are in the edit mode). Alternately, the yellow lock can be pressed to allow editing.

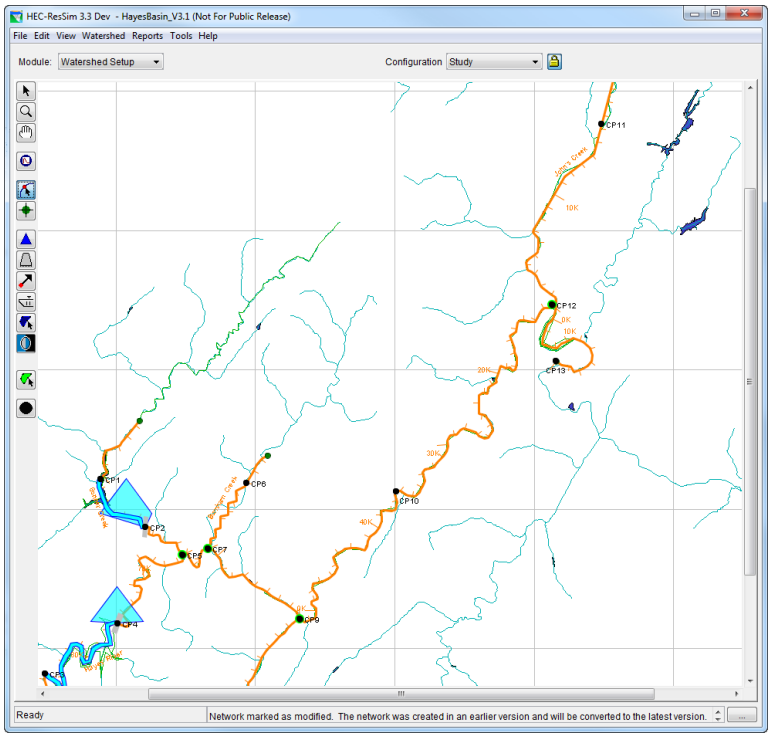


Figure 10: Drawing a Stream Alignment

- **Create a Stream Alignment Element.** The Stream Alignment must be created *from upstream to downstream*. Select the stream alignment tool from the tool bar along the left side of the map region. Hold down the control (**CTRL**) key and click to place successive points along the stream alignment. To terminate the stream element, release the **CTRL** key and click to place the last point of the stream. As shown in Figure

10, a tributary may be connected to a stream by following the same procedure as above and terminating the stream at the point of connection to the existing stream.

- **Edit a Stream Element.** Select the stream alignment tool and double click on the stream element to be edited. The stream will change color, and points will appear where each stream point is defined. These points can be moved to realign the stream by positioning the cursor over them and holding down the left mouse button and dragging. New points are added to a stream by clicking on the stream while holding down the CTRL key. Remove points from a stream by clicking over an existing point while holding down both the SHIFT and CTRL keys. An entire stream can be deleted by right-clicking on it and selecting Delete Stream Element from the shortcut menu. Streams can be renamed by right-clicking on the stream and selecting Rename Stream Element.

- **Define Stationing.** The green dots at the end of each stream in the alignment are Stream Nodes. Their purpose is to define the stationing along the stream. By default, the most downstream node will get a stream station of zero, and the node at the upstream end will get a stream station value determined by the length of the stream line in the coordinate system of the map region. However, these default values can be changed to more accurately describe the stream. Using the stream node tool right-click on a stream node and select Edit Node. In the editor that appears, uncheck the "Use Default Stationing" box then update the value of the station to represent the appropriate real world station value of that representative point. If you change the stationing of the upstream node, then you may need to edit the most downstream node to reset its station to zero (0.0). Stationing along a stream will be determined proportionally based on the station values of the bounding nodes. Interior stream nodes can be added to a stream to more accurately define distances along a stream and to locate known river station landmarks. To do so, select the stream node tool, then while holding down the CTRL key, click those locations on a stream where you want to place a stream node.

4. Create a Configuration.

Before adding reservoirs and computation points, a Configuration must be defined *[Section 3.4]*. In brief, a Configuration is the identification of a "set" of projects that you want to model in your watershed. If more than one set of projects will be considered in a study, then more than one Configuration will be needed. While you add projects to your Configurations, a special "superset" (named Study) is developed by the program to contain all projects from all Configurations in the watershed. If multiple programs (e.g., HEC-HMS, HEC-FIA, HEC-RAS, etc) are going to be applied to your watershed, then

Configurations should be developed by the modeling team before the models for an individual program are produced.

- **Make the Configuration Editable**. Before you can create or modify a Configuration, you must make it editable. To do so, select **Allow Editing** from the **Edit** menu (or click the "unlocked" yellow lock, which "locks" the configuration (allowing you to perform editing). In other words, this option turns on (or off) the ability to make changes to the Configuration.

- **Create the Configuration**. From the **Watershed** menu, select **Configuration Editor**. From the **Configuration** menu in the Configuration Editor that has opened (Figure 11), select **New**. In the new configuration window, enter a **Name** and **Description** for the Configuration and select a time step from the **Time Step** list and press **OK**. Then, click **OK** in the Configuration Editor.

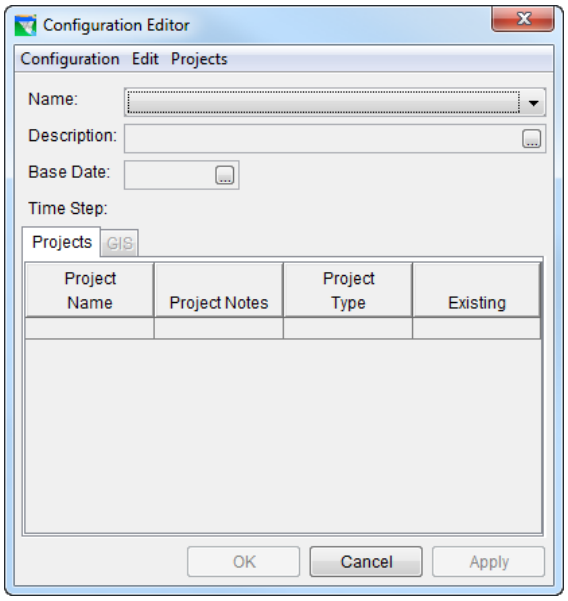


Figure 11: Configuration Editor

5. Place Project Elements into the Configuration.

The next step is to place projects into the watershed. The projects you place will be added to the current Configuration as identified in the Configuration field at the top of the window.

- **Draw Project Elements**. Verify that you have selected the appropriate configuration to which you will be adding projects. Select the appropriate tool to add projects. Each project, when placed, will create one or more computation points. When using the Reservoir Tool to place reservoirs, hold down the **CTRL** key to place the

upstream point(s) of the reservoir, then release the **CTRL** key and place the *downstream* point. A **Name New Reservoir** window will appear so that you can name the reservoir.

Another useful tool is the Computation Point Tool. Use this tool to place additional computation points by holding down the **CTRL** key and clicking on additional locations as needed. As you place additional computation points, you will be prompted to name them (when placed at the junction of two or more streams, you will need to select which stream the computation point should be placed on). Placing reservoirs on the stream alignment also creates computation points at the upstream and downstream extents of the reservoir using automatically generated names. Be sure to rename each of these points so it can be easily identified from a list when the map region is not visible. To do so, right-click on the computation point, select **Rename Computation Point** from the shortcut menu and enter a new name in the window as shown in Figure 12.

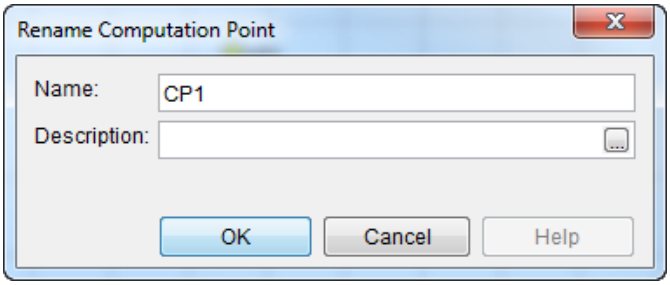


Figure 12: Rename Computation Point

- **Save the Configuration**. When you are finished adding elements to the Configuration, be sure to save it and turn off editing. Select **Save Configuration** from the **Watershed** menu to save, then "unlock" the Configuration lock (yellow lock button) to toggle off the editability of the Configuration. A fairly simple Configuration (named StandardConfig) is shown in Figure 13.

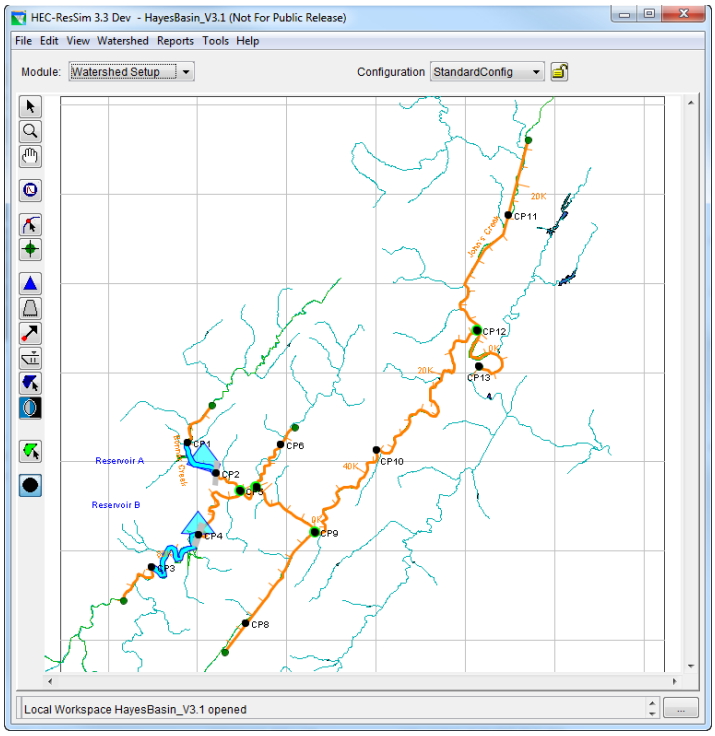


Figure 13: A Simple Configuration, StandardConfig

B. Develop a Reservoir Network

With a stream alignment in place and one or more configurations defined, a network of reservoirs and stream routing reaches can be created. A summary of the basic steps is presented in the following paragraphs. From the **Module** list, select **Reservoir Network**.

1. Import a Reservoir Network.

To use the **Import Network Wizard**, perform the following steps:

- From the **Network** menu in the **Reservoir Network** module, select **Import Network…**. The **Import Network Wizard – Step 1 of 6** will appear. Select the Watershed that contains the network *from which* you would like to import the entire network physical and operational data. The box on the left side of the dialog contains names of Watershed Locations that have been specified as Model Directories. The box on the right side of the dialog contains all the watersheds that can be found in each Watershed Location.

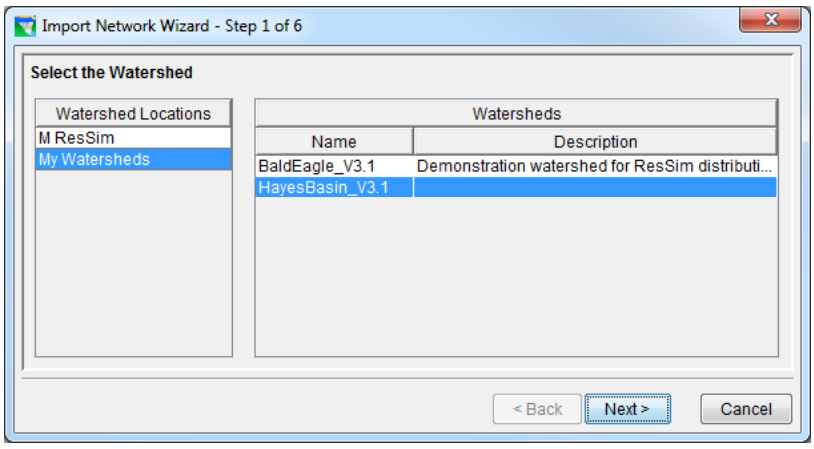


Figure 14 Import Network Wizard – Step 1 of 6 Select Watershed from which to Import

- Select **Next** and **Import Network Wizard – Step 2 of 6** will appear. Select the network *from which* you would like to import all elements by highlighting the **Network Name**.

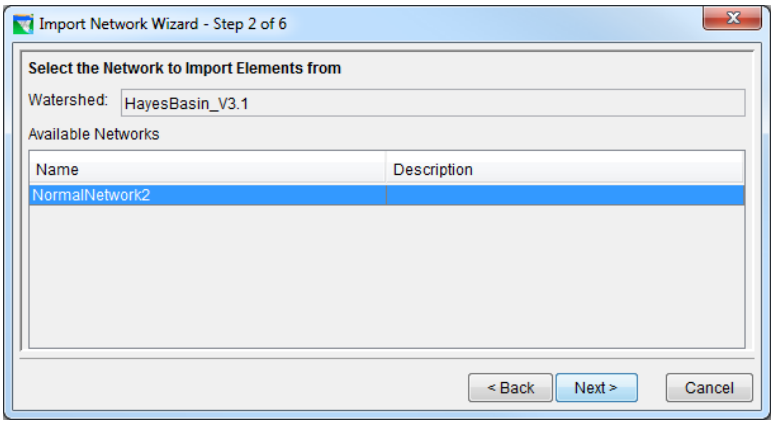


Figure 15 Import Network Wizard – Step 2 of 6 Select Network from which to Import

- Select **Next** to continue and the **Import Network Wizard – Step 3 of 6** will appear. Enter a New Reservoir Network Name and Description.

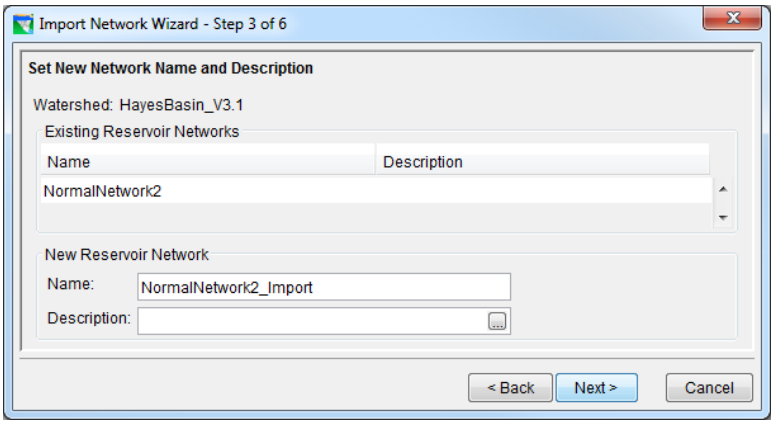


Figure 16 Import Network Wizard – Step 3 of 6 Define New Network Name and Description

- Select **Next** to continue and the **Import Network Wizard – Step 4 of 6** will appear. Assign Stream Names by drop-down menu one at a time, or all at once by selecting the **Assign Stream Names** button. Care is required in determining the connection of the Imported Element to the Stream. If Imported Elements are associated with streams that are not connected, it is possible there will be erroneous downstream control rules added to upstream junctions.

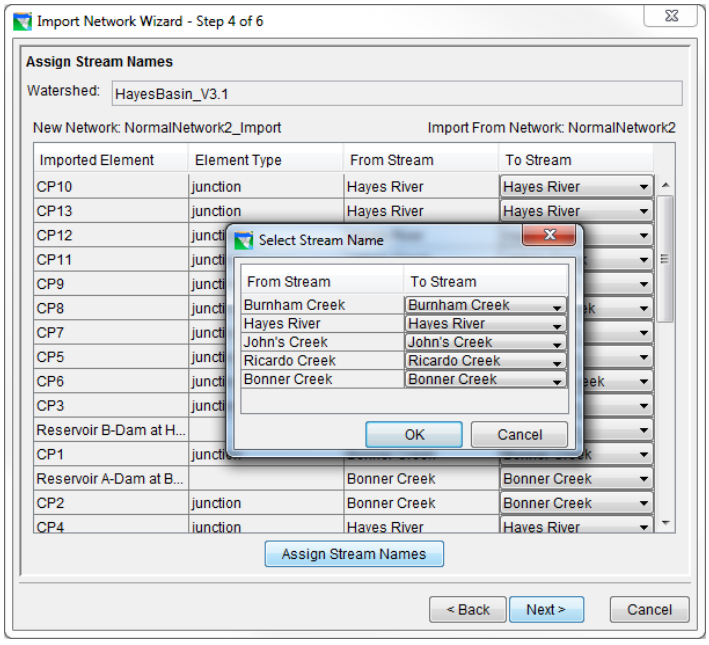


Figure 17 Import Network Wizard – Step 4 of 6 Assign Stream Names

- Select **Next** to continue and the **Import Network Wizard – Step 5 of 6** will appear. Assign the network elements to their corresponding

watershed computation points by use of a drop-down menu for each imported element.

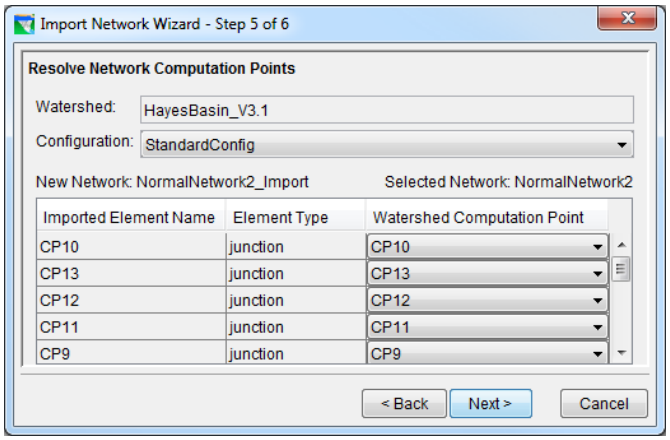


Figure 18 Import Network Wizard – Step 5 of (Resolve Network Computation Points)

- Select **Next** to continue and the **Import Network Wizard – Step 6 of 6** will appear. Review all of the information in the import summary. Check to ensure that all physical and operational data was successfully imported by reviewing the **Import** Summary and by viewing the various network element editors in the Reservoir Network module.

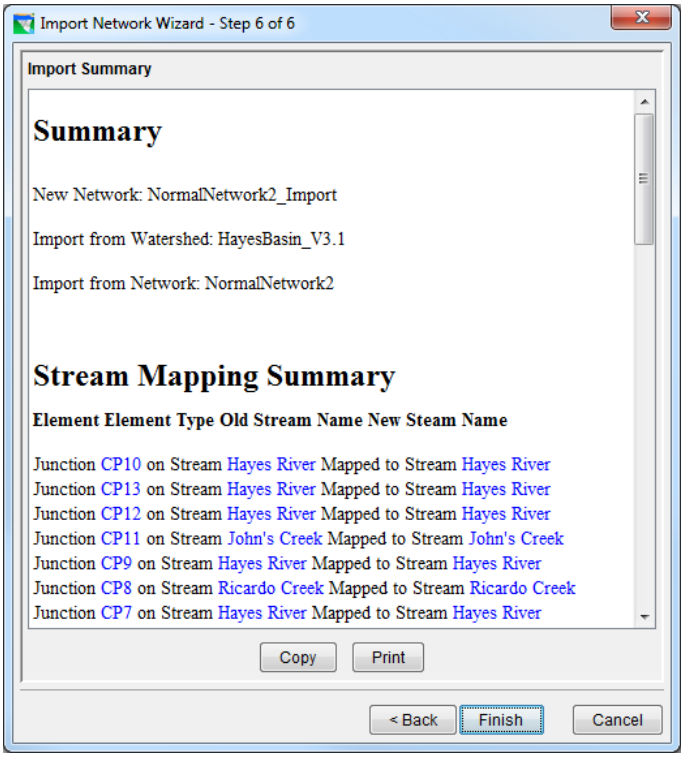


Figure 19 Import Network Wizard – Step 6 of 6 (Import Summary)

2. Create/Open a Reservoir Network.

As an alternative to Importing a reservoir network, you may also create a new network. From the **Network** menu, select **New**. The **Create New Reservoir Network** window will appear (Figure 14). Specify a name and description for the network. Then, select the Configuration on which this network will be based and click **New** *[Section 6.2]*.

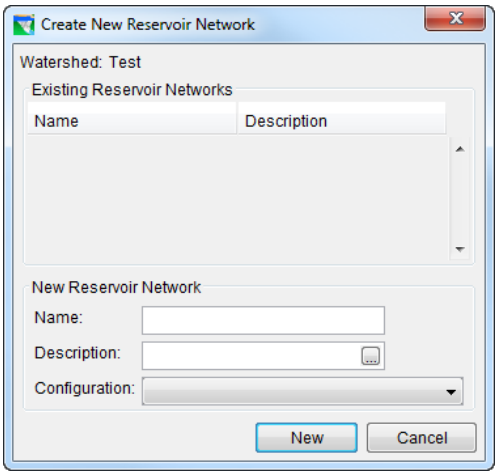


Figure 20: Create New Reservoir

Reservoirs and computation points that were part of the selected configuration will automatically be created as elements in the Reservoir Network. Reservoirs transfer directly. Computation points appear as Junctions. This is depicted in Figure 15.

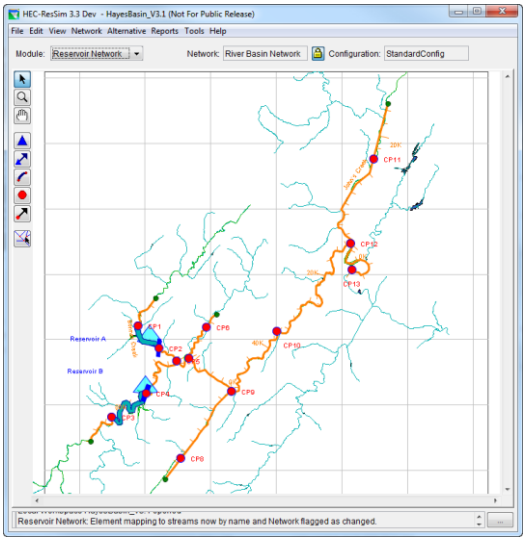


Figure 21: New Network Based on Configuration StandardConfig

Make the Network Editable. Before you can begin editing, you must ensure the network is editable. To do so, select **Allow Network Editing** from the **Edit** menu (or press the "yellow lock" button). This option turns on (or off) the ability to make changes to the network.

3. Add Routing Reaches, etc.

Draw the additional elements needed to complete the connectivity of the reservoir network schematic *[Section 7.2]*.

Draw Routing Reaches. Select the Routing Reach Tool . Hold down the **CTRL** key and click anywhere on the stream to place the upstream end of the reach. Release the **CTRL** key and move the mouse along the stream and click again at a lower station on the stream to place the downstream end, as shown in Figure 16. Routing reaches must be connected to an existing junction. If you start or end a reach where there is no junction a junction will be created, and you will be prompted to first name the new junction and then name the new reach. (Note: tributaries cannot connect to the middle of a reach; they must connect to a junction.)

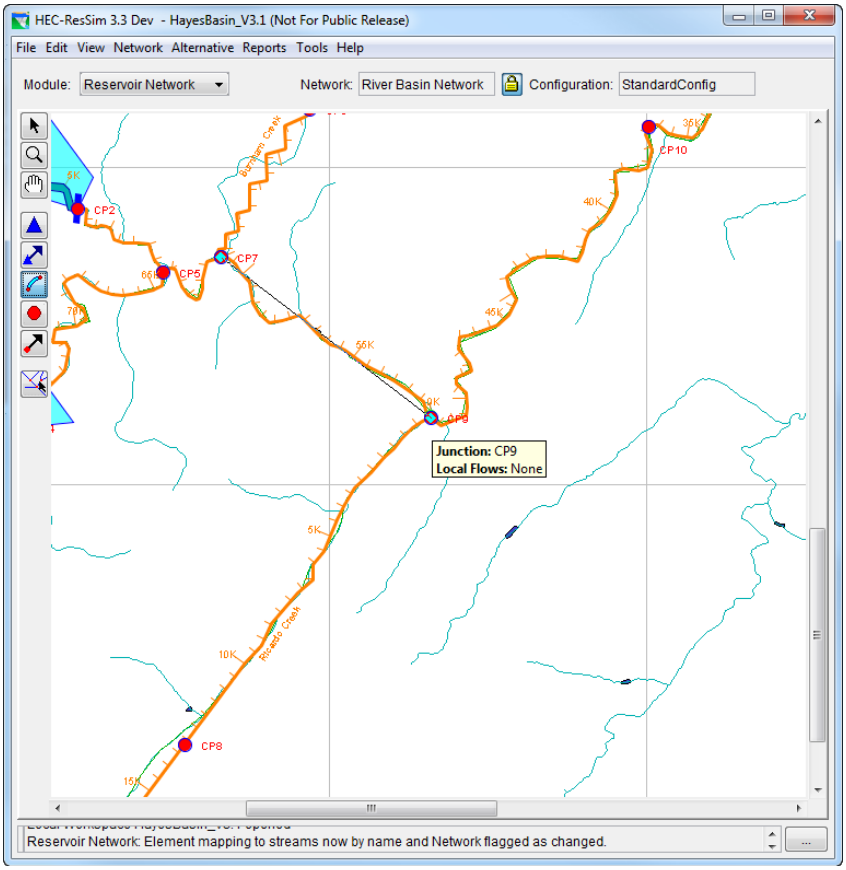


Figure 22: Drawing Routing Reaches

4. Edit Junction and Reach Properties.

With a reservoir network schematic complete, the data for the model elements can be entered and saved.

Edit Junction Properties. With the pointer tool or the junction tool selected, right-click on desired junction and select **Edit Junction Properties**. The Junction Editor will appear. Select the **Local Flow** tab and input a name and factor for each external flow entering the junction (as shown in Figure 17). The name entered will appear during the Alternative definition to identify the local flow for time-series mapping. Therefore, the name given on the Local Flow tab should be recognizable during the time-series mapping. Flows *must* be supplied to headwater junctions and are optional at other junctions.

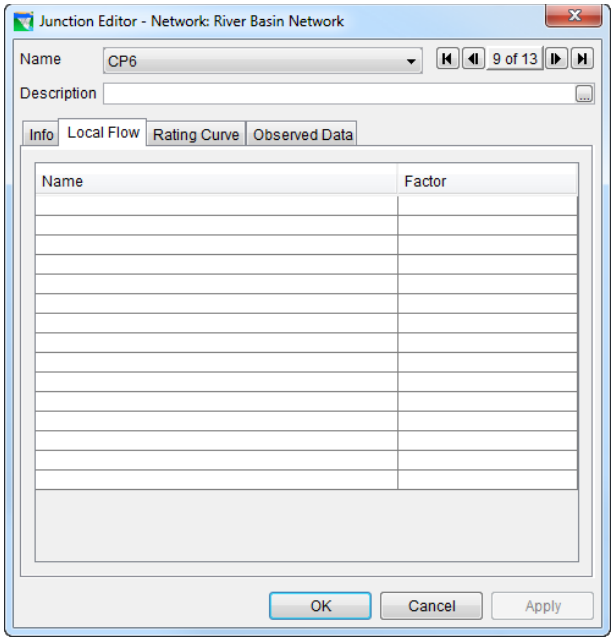


Figure 23: Junction Editor

To edit junction properties for other junctions without exiting this window, click **Apply** and then select another junction from the **Name** list or use the navigator buttons located in the upper right portion of the window. Alternately, click **OK**, then right-click and **Edit** another junction.

For each junction where you specify **Local Flows**, you will see a "halo" around the Junction element. This provides you with a visual reference for ensuring local flows have been indicated for the appropriate locations in your watershed.

Edit Reach Properties. With the reach tool or the pointer tool selected, right-click on the desired reach and select **Edit Reach Properties**. Select the routing method from the **Method** list (as shown in Figure 18) and fill in the

necessary parameters for the selected routing method. The default routing method is **Null Routing** (no route).

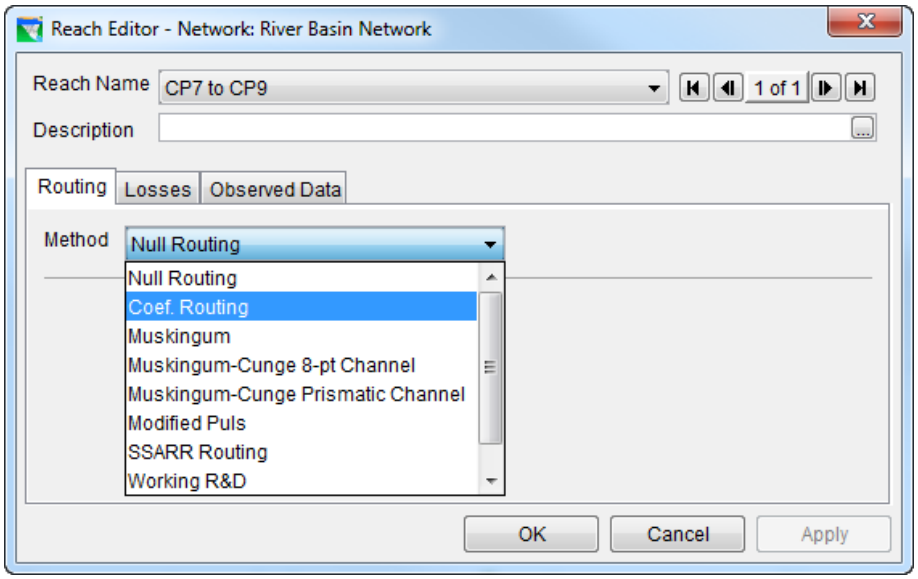


Figure 24: Reach Editor

The **Coefficient Routing** method can be used to mimic most linear routing methods by representing the method as a series of coefficients for each time step of the routing extent. Specifying a coefficient of 1.0 in the first row of the Coefficient table is the same as specifying a null routing.

To edit another reach without exiting this window, click **Apply** then select another reach from the **Reach Name** list or use the navigator buttons to select another reach. Alternately, click **OK**, then right-click and **Edit** another reach *[Section 7.2]*.

5. Edit Reservoir Physical Properties.

With the reservoir tool or the pointer tool selected, right-click on a reservoir, select **Edit Reservoir Properties** and the Reservoir Editor window will appear as shown in Figure 19. This editor has a tab panel with three tabs: **Physical** tab *[Chapter 9]*, **Operations** tab *[Chapters 10 and 11]*, and **Observed Data** tab *[Section 8.3]*.

- On the **Physical** tab, a *tree* appears in the left panel of the window. Initially, two *branches* of the tree are shown: **Pool** and **Dam**. The Pool reflects characteristics of the reservoir storage while the Dam reflects the reservoir dam.

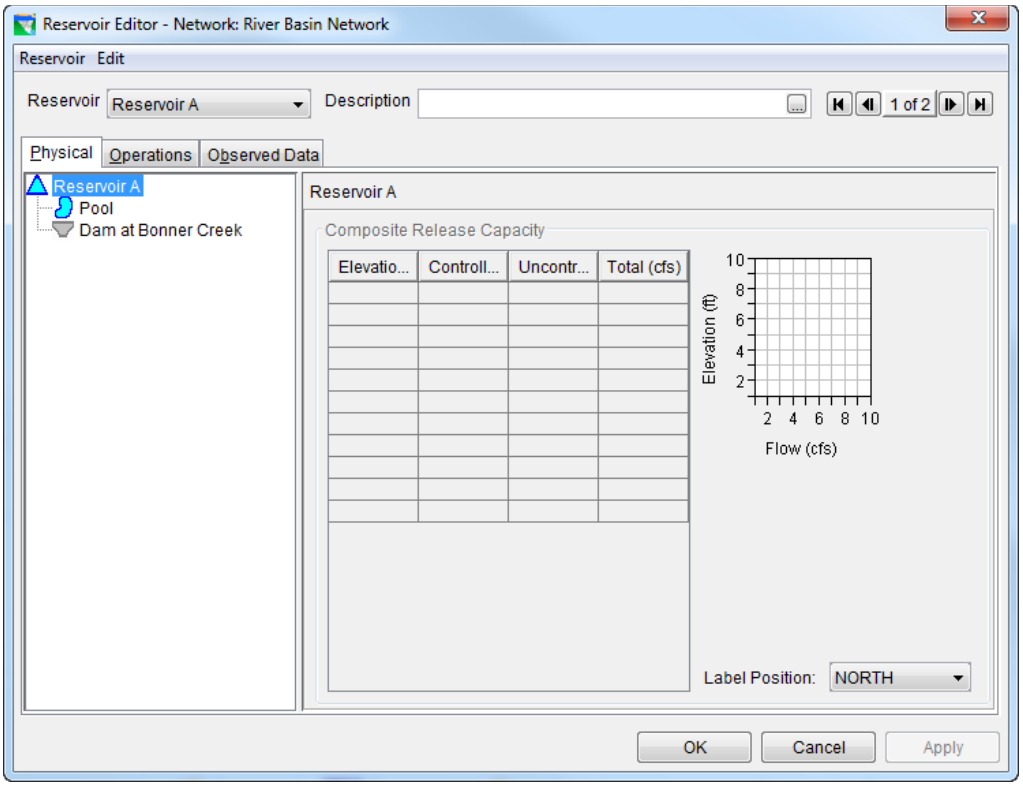


Figure 25: Reservoir Editor, Pool Data

- **Enter Pool Properties**. Click on **Pool** to enter the Elevation vs. Storage vs. Area data (as shown in Figure 19). At least two rows are required for **Elevation**, **Storage** and **Area** data, with values increasing down the columns (no duplicate values). The maximum elevation should not exceed the Elevation at top of dam, which is input when **Dam** is selected. Pool losses can be specified by right-clicking on **Pool** and selecting **Add Pool Evaporation** and/or **Add Pool Seepage** *[Section 9.3.1]*.

- **Enter Dam Properties.** Click on **Dam** in the reservoir tree. The data area to the right of the reservoir tree will change to allow you to enter the data needed to define the dam *[Section 9.4]*. Before proceeding, take a look at the name of the dam. The name is probably something like "Dam at My Main River." This is the default name the program gives to the dam. It identifies the dam based on the name of the underlying stream upon which the dam was placed. This enables the program to uniquely identify more than one dam within a reservoir. It is a rare case when a reservoir has more than one dam, so feel free to rename the dam by right-clicking on the dam name and selecting **Rename** (as shown in Figure 20). One suggestion is to name it *Dam* by removing the stream name. Another possibility is to name the dam by its actual name (many reservoirs have a name for the pool and a different name for the dam; ResSim provides the ability to name each part individually).

- Enter the elevation and length of the dam in the relevant fields. The composite release capacity table is not editable. This table is informational and will be filled automatically as you add outlets to the dam.

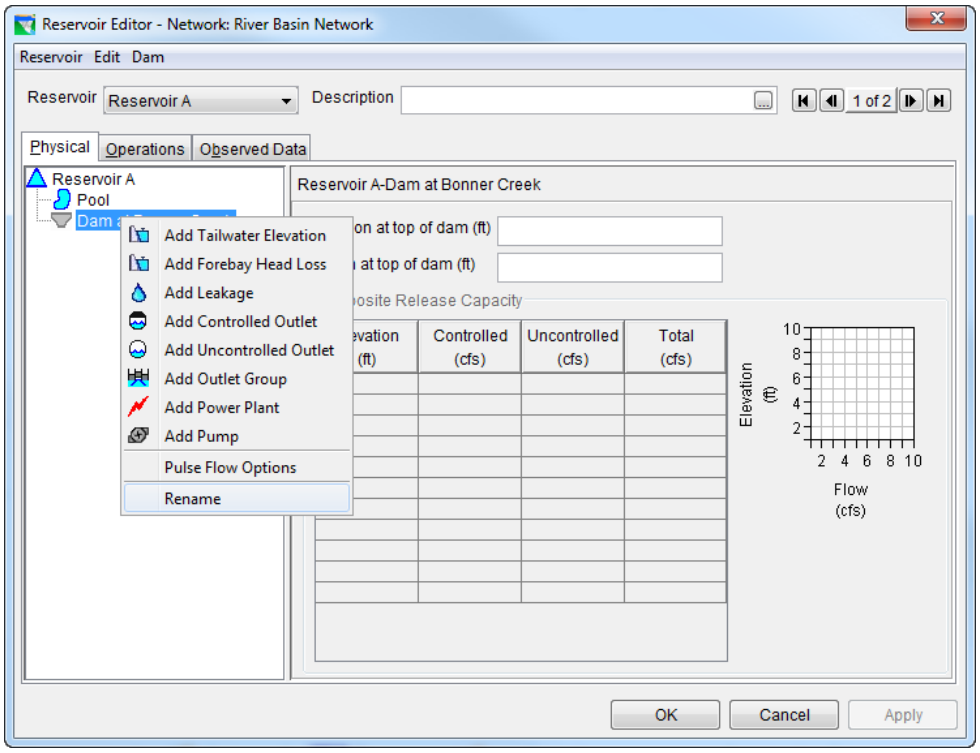


Figure 26: Reservoir Editor, Rename Dam

- **Add Outlets**. Physical **Outlets** can be added to the Dam by right-clicking on the **Dam**. Figure 20 illustrates the dam's shortcut menu. Select **Controlled Outlet** to add a gate or other control structure to the dam *[Sections 9.4.2 and 9.6]*. You can rename the outlet by right-clicking on the outlet and selecting **Rename**.

A simple controlled outlet can be defined by entering a maximum release capacity curve for the outlet. If a more detailed outlet definition is desired, press the **Edit Gate Settings** button. The Gate Settings Editor will allow you to define a set of gate openings with Elevation vs. Maximum Capacity (flow) data. When finished, the release capacity table will show several more columns (as shown in Figure 21), one for each gate opening in the set you defined.

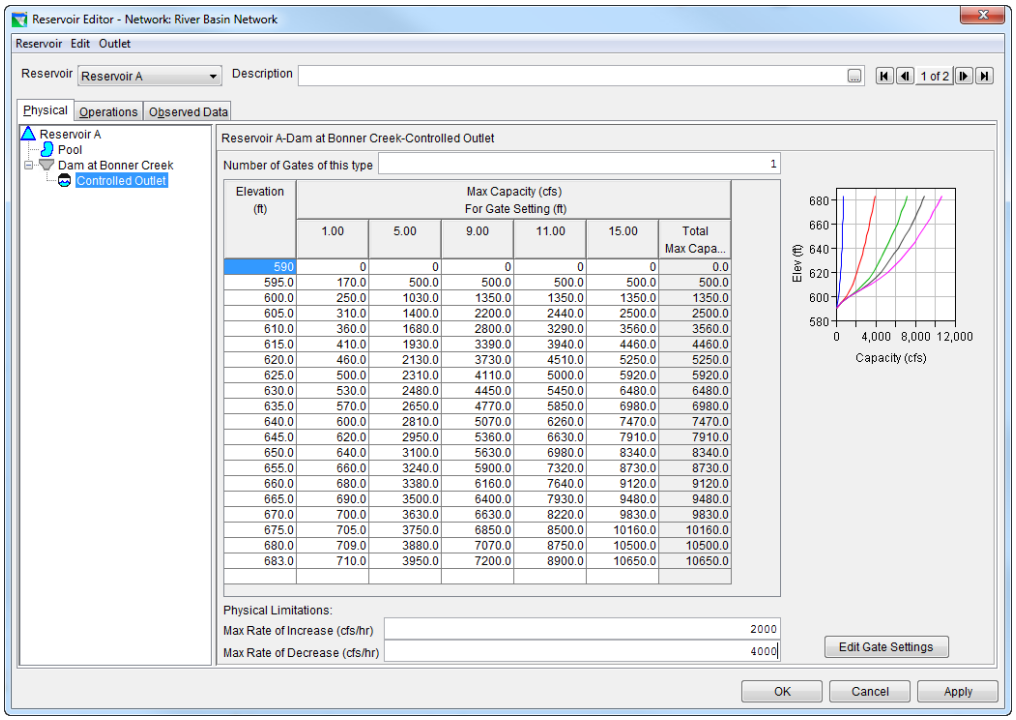


Figure 27: Reservoir Editor, Gated Outlet

Select **Uncontrolled Outlet** to add an ungated emergency spillway to the dam *[Section 9.9]*. For an Uncontrolled Outlet, the spillway length and elevation are entered along with either a **Weir Coefficient** or an **Elevation vs. Outflow** relationship. The spillway elevation must be at or below the **Elevation at top of dam** and above the minimum storage elevation. Figure 22 shows the window for entering data for an Uncontrolled Outlet.

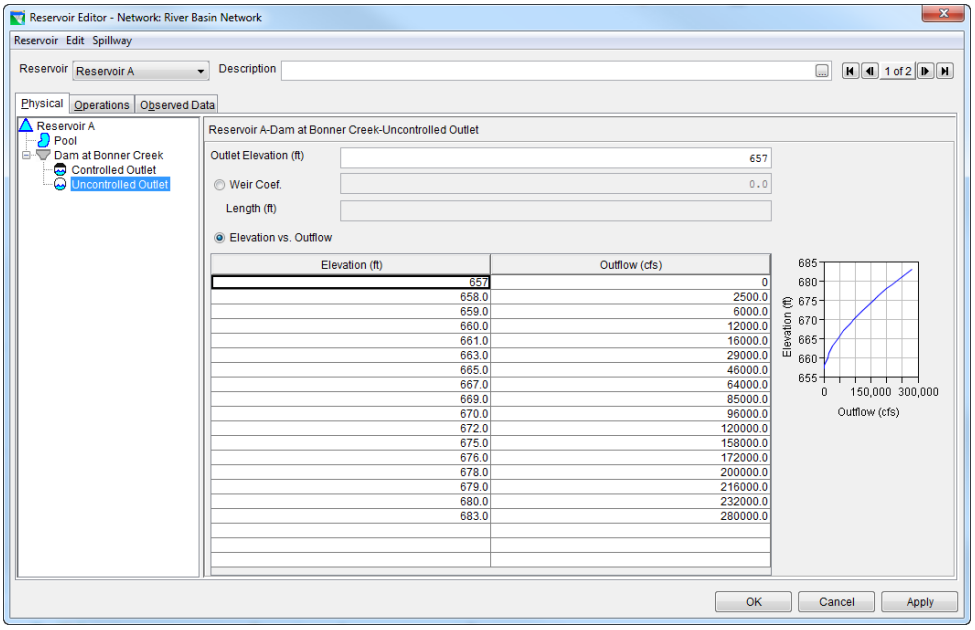


Figure 28: Reservoir Editor, Uncontrolled Outlet (Spillway)

In addition to Controlled and Uncontrolled Outlets, the following components can also be added to the Dam:

Leakage *[Section 9.4.5]*,
Outlet Group *[Section 9.4.6]*,
Tailwater Elevation *[Section 9.4.7]*,
Power Plant *[Section 9.7]*, and
Pump *[Section 9.8]*.

In addition to adding physical components to the Dam, there is an advanced option for specifying **Pulse Flow Options** *[Section 11.3.5]*.

6. Edit Reservoir Operational Properties.

After adding the reservoir physical properties, add the operational information via the **Operations** tab of the Reservoir Editor. When the **Operations** tab is initially chosen, a screen similar to Figure 29 will appear. A new operation data set must be created before operations data can be entered *[Section 10.2.1]*.

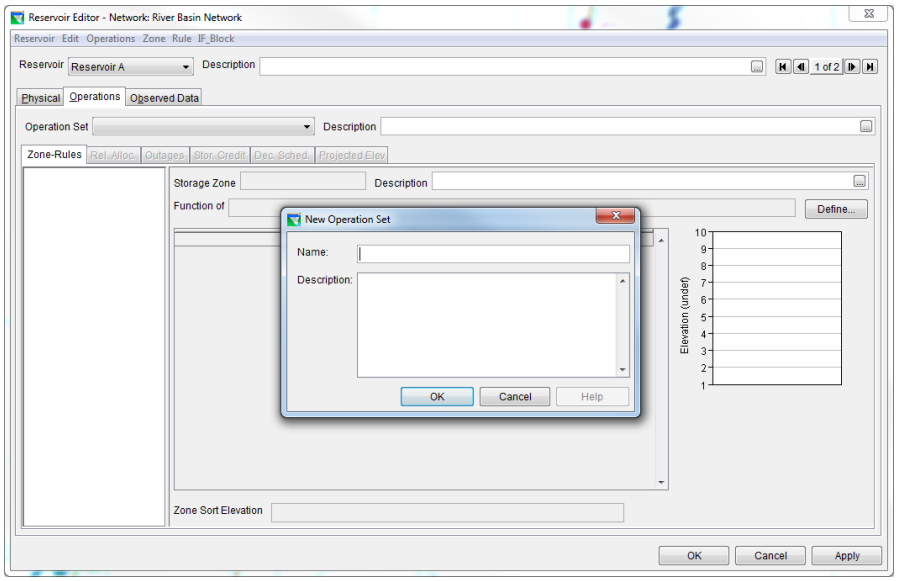


Figure 29: New Operation Set

- **Create an Operation Set**. Select **New** from the **Operations** menu and give the new operation set a name (as illustrated in Figure 23). An **Operation Set** is the operation plan or scheme upon which a reservoir bases its decisions regarding how much water to release at each time step of a simulation run *[Section 10.2]*.

- **Operation Zones**. Once an Operation Set is created, then a set of "default" reservoir zones is generated: **Flood Control**, **Conservation**, and **Inactive**. Additional zones (e.g., Top of Dam) can be added by

selecting **New** from the **Zone** menu. Top of zone curves must be entered, as no default values are available *[Section 10.3]*.

- **Operation Rules**. To add a rule to a zone, highlight the appropriate zone and select **New** from the **Rule** menu. In the new rule window (Figure 24), enter a name for the rule, then select the pool, dam, outlet, or outlet group for which the rule will apply (**Operates Release From**), then select the rule type from the **Rule Type** list, and then click **OK** *[Chapter 11]*.

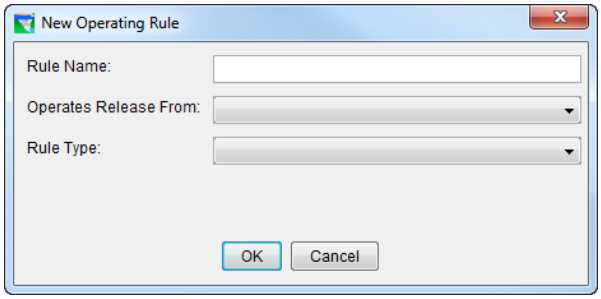


Figure 30: New Operating Rule

Rule types differ depending on where the rule is applied (pool, dam, or outlet) *[Section 11.1.1]*. The rules that apply to the Reservoir Pool, are typically relevant to *Storage* or *Elevation*, whereas the rules that apply to a Dam or an Outlet are relevant to *Flow*.

Rules that apply to **Reservoir Pool**, **Reservoir Dam**, **Controlled Outlets**, **Outlet Groups**, and **Diverted Outlets** (i.e., all 'Operates Release From' elements, except Pumps)

- **Release Function**, Maximum, Minimum or Specified Release as a function of Date, Date and Time, Model Variable, or External Variable *[Section 11.2]*.
- **Flow Rate of Change Limit**, Allowable change when increasing or decreasing release values *[Section 11.5]*.
- **Script**, User-defined scripting available that dramatically increases the flexibility of reservoir operations. *[Section 12.3]*.

Rules only for the **Reservoir Pool** include the following:

- **Downstream Control Function**, Minimum or Maximum Flow or Stage target or constraint (at a downstream location) as a function of Date, Date and Time, Model Variable, or External Variable *[Section 11.3]*.
- **Induced Surcharge**, Special flood control operation using gate regulation parameters *[Section 11.4]*.
- **Elevation Rate of Change Limit**, Allowable change when increasing or decreasing pool elevation values *[Section 11.6]*.
- **Tandem Operation**, Release based on balancing pool with a downstream reservoir *[Section 11.9]*.

Rules only for the **Power Plant** include the following:

- **Hydropower - Power Guide Curve**, Energy Requirement (expressed as % Plant Factor) is dependent on the available power storage. Optional Daily and Hourly distributions can be specified *[Section 11.7.1]*.
- **Hydropower - Schedule**, Monthly Firm Energy Requirements (specified as Monthly MWH or % Plant Factor) with optional Daily and Hourly distributions *[Section 11.7.2]*.
- **Hydropower - System Schedule**, same requirements as above except with additional options for specifying the power generation pattern and for identifying reservoirs whose power generation can be used to meet system requirement *[Section 11.7.3]*.
- **Hydropower - Time Series Schedule**, Energy Requirement as defined by user-supplied time-series data *[Section 11.7.4]*.

Rules only for **Pumps** include the following single rule:

- **Pump Schedule**, Pumpback storage operation using user defined schedule *[Section 11.8]*.
  Depending on the rule type, you will need to enter data to define the rule. A screen will open in the editor to allow you to enter the required data for each rule. The rules and storage zones are displayed in a tree on the left side of the window (as shown in Figure 25). Once a rule has been defined, it can be included in more than one zone by selecting **Use Existing** from the **Rule** menu. Similarly, a rule can be removed from a zone by selecting **Remove from Zone**. A rule can be deleted from the operation set by selecting **Delete**. However, when a rule is Deleted, then it is no longer available to any of the zones; therefore, ***caution should be used when actually deleting a rule***.

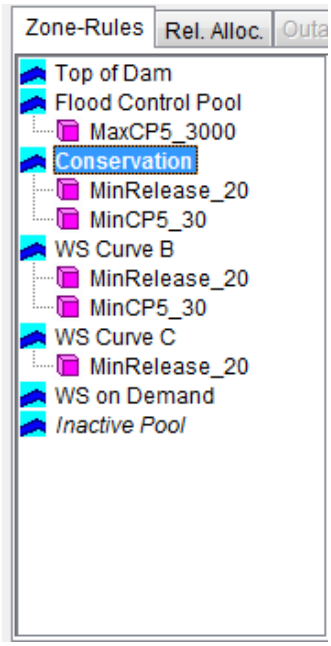


Figure 31: Zones and Rules Panel

When finished entering data for a reservoir, click **OK**. To edit a different reservoir without exiting the reservoir editor, click **Apply** and select another reservoir from the **Reservoir** list.

- The **Observed Data** tab is used to indicate that observed data is available for comparison purposes *[Section 10.23]*. If the **Observed** box is "checked" in the table (as shown in Figure 26), then there will be a corresponding entry in the Observed Time-Series mapping table when an Alternative is created *[Section 13.9]*.

5. **Save** the **Reservoir's Physical** and **Operations** data, **Junction** information, **Routing** criteria, **etc**... From the **Network** menu select **Save**. *It is a good idea to do this frequently.*

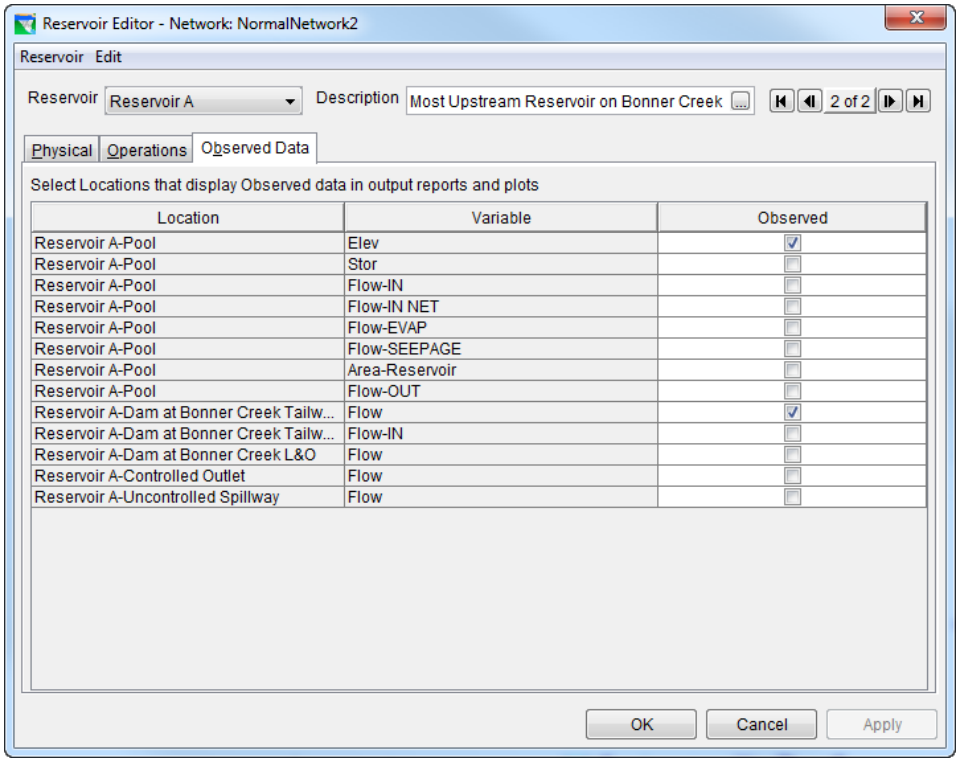
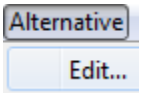


Figure 32: Observed Data Tab

C. Define Alternatives

Create an Alternative. An Alternative consists of a Reservoir Network, an Operation Set for each reservoir in the network, a definition of initial conditions, and a mapping all time-series records to identified local inflows, etc. To create an alternative:



1. From the **Alternative** menu (in the Reservoir Network module) select **Edit** *[Section 13.2]*.
2. In the Alternative editor, ensure that the appropriate **Configuration** name is displayed from the list of available configurations. Then, from the **Alternative** menu, select **New** (as shown in Figure 27).
3. In the New Alternative editor, enter a name for the Alternative in the **Name** box and a description in the **Description** box. Select a **Reservoir Network** from the list and click **OK**.

   All of the relevant fields in the Alternative Editor will become active *[Section 13.3]*.

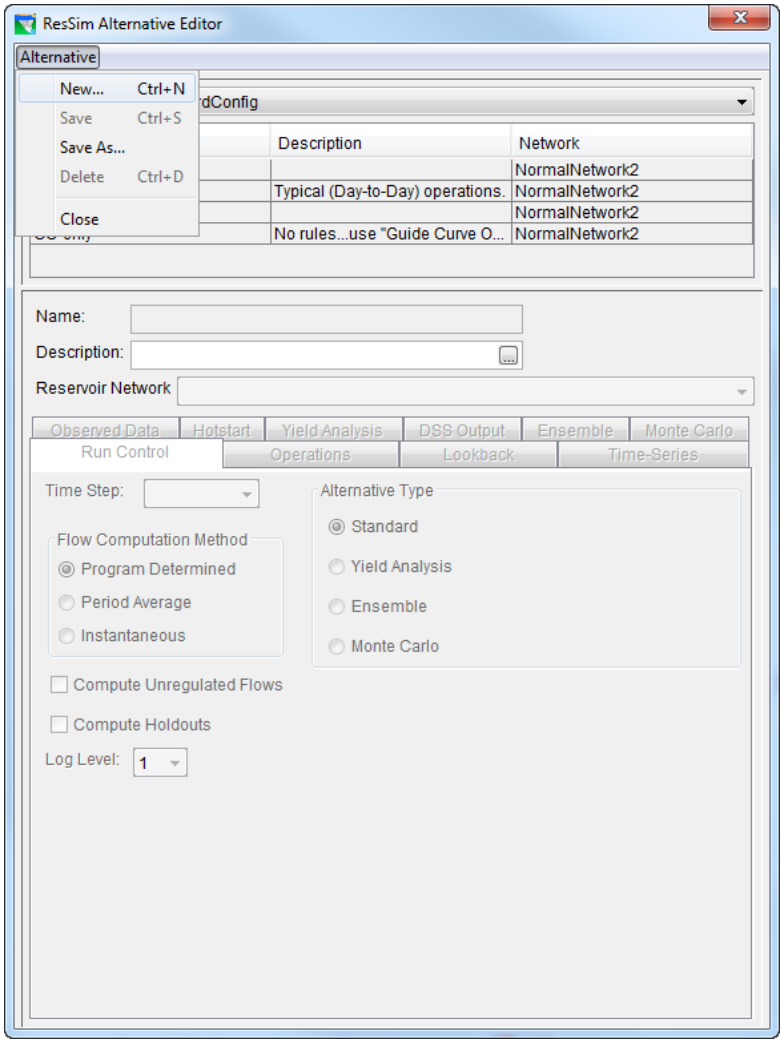


Figure 33: Alternative Editor, Create New Alternative

4. The **Run Control** tab (Figure 28) allows you to specify the **Time Step** and **Flow Computation Method** to be used in your simulation(s) *[Section 13.4]*.

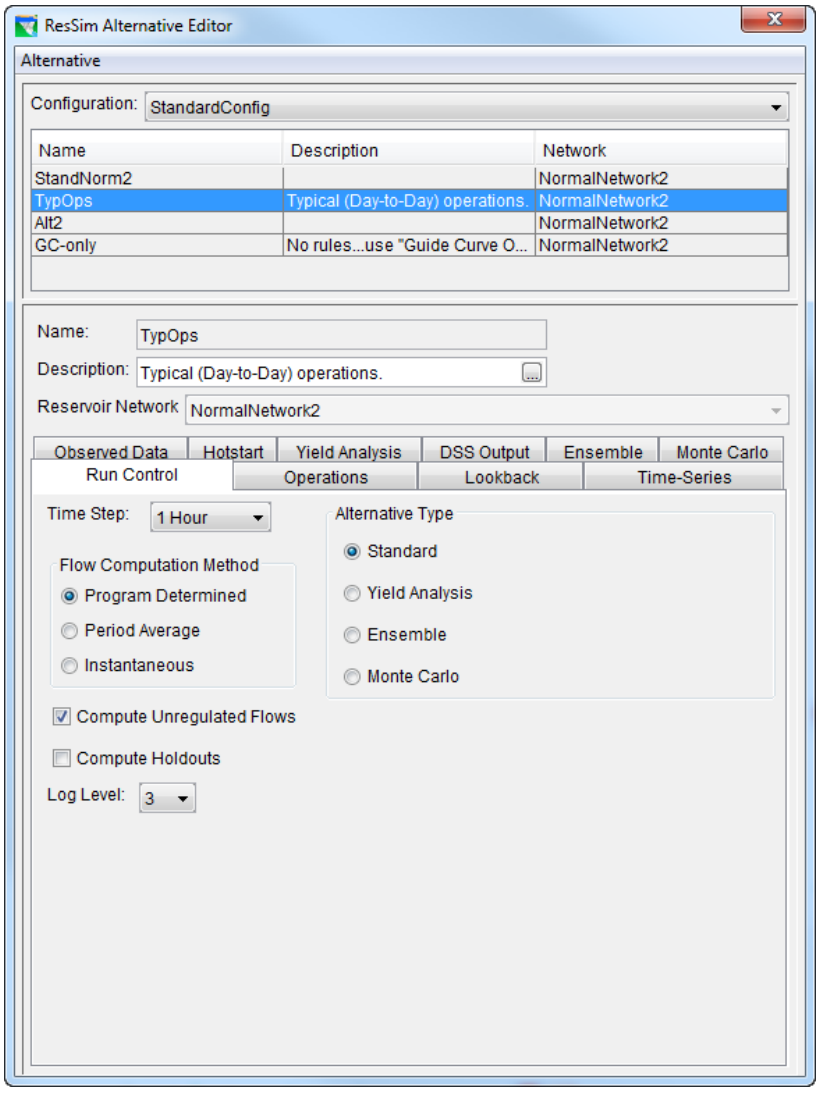


Figure 34: Alternative Editor, Run Control Tab

5. In the Alternative editor, select an Alternative. On the **Operations** tab, select an **Operation Set** for each reservoir *[Section 13.5]*. This can be accomplished in the Operation Set column by selecting (for each Reservoir) the operation set from the list, as shown in Figure 29.

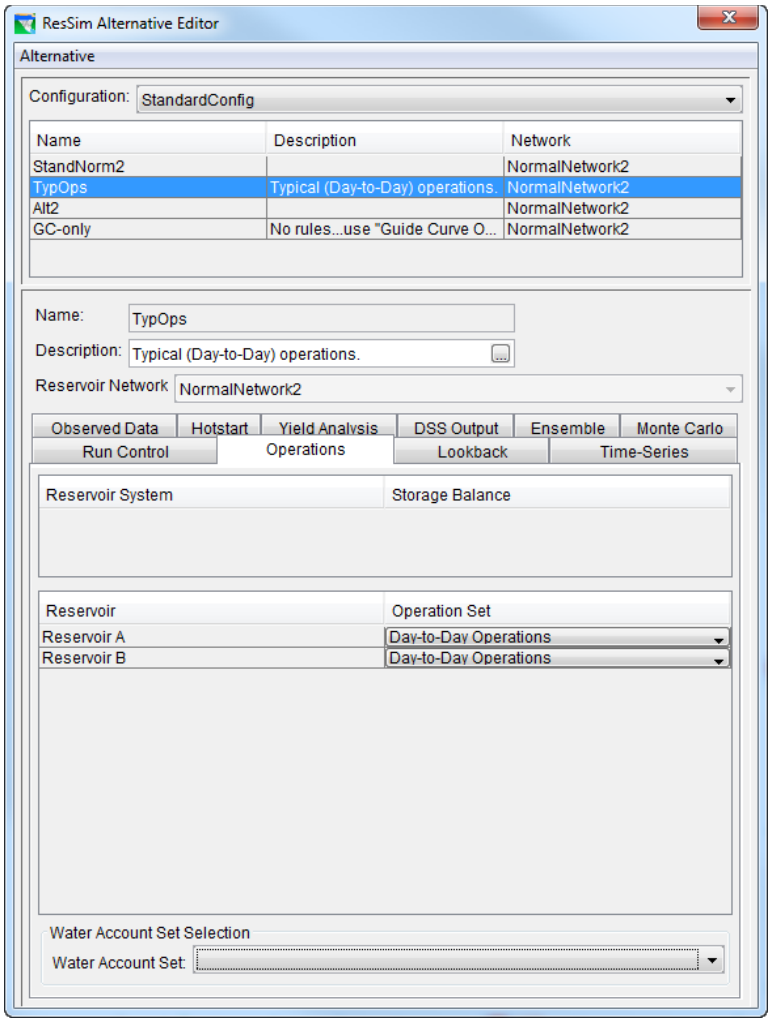


Figure 35: Alternative Editor, Operations Tab

6. On the **Lookback** tab *[Section 13.7]*, select the initial conditions **Type** (Times-Series or Constant) from the list for each Location listed, as shown in Figure 30. This is similar to selecting the operation set per reservoir. For **Variables** defined as Constants, enter a constant **Value** for the Elevation or Release.

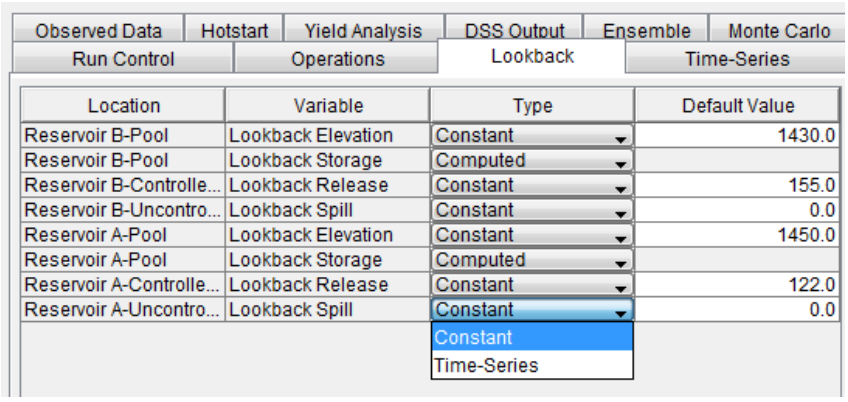


Figure 36: Alternative Editor, Lookback Tab

7. On the **Time-Series** tab *[Section 13.8]* select a row in the time-series mapping table (as shown in Figure 31). Each row in the table represents a *required input* for a local flow location (specified in the Local Flow tab within the Junction editor), an element/parameter you have defined on the Lookback tab as Time Series, or a Time Series that is referenced in the Operation rules. To attach a time-series record to the selected location, press the **Select DSS Path** button.

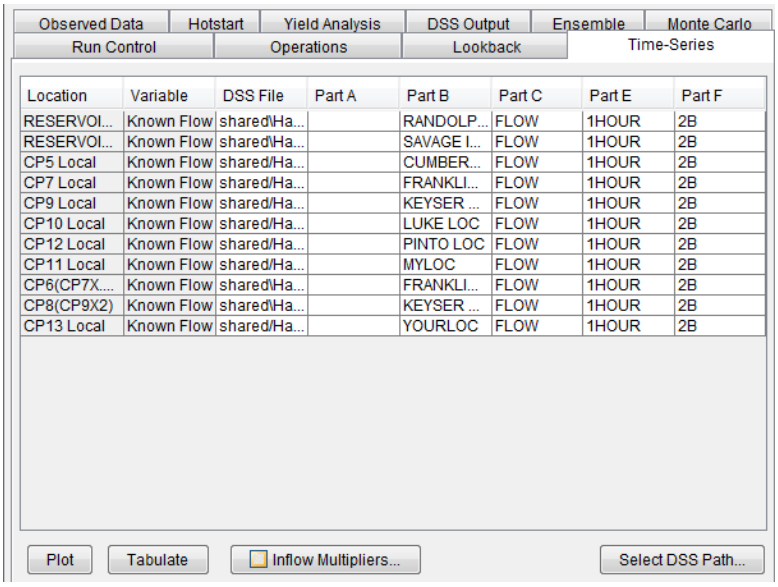


Figure 37: Alternative Editor, Time-Series Tab (for Mapping Data)

The DSS path selector window will appear as shown in Figure 32. Open a DSS file, use the parts filters to organize the catalog listing, and select a record. To get the selected record to map back to the alternative editor, press the button labeled **Set Pathname**.

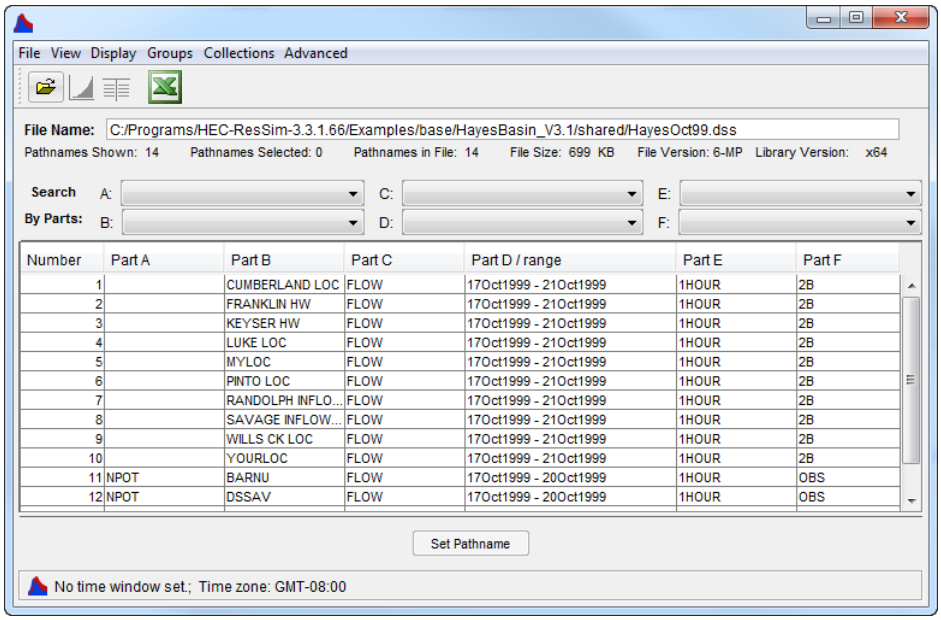


Figure 38: DSS Time-Series Record Selector

8. On the **Observed Data** tab *[Section 13.9]*, using the same procedure as the Time-Series mapping described above, select *optional* time-series records for use in comparing with computed results.

9. Typically, the Lookback period is used for *warming-up* the state (i.e., setting initial conditions) of the watershed for an alternative. However, instead of using the Lookback period and associated data for determining the system state, you can save (and subsequently use) the data from one simulation to another simulation by using the *optional* Hotstart options (Figure 33). Use the Hotstart as a way of assigning a previously simulated watershed state to another simulation. In order to use the Hotstart option, the user must first run a simulation that saves system state data to "Hotstart" files *[Section 13.10]*.

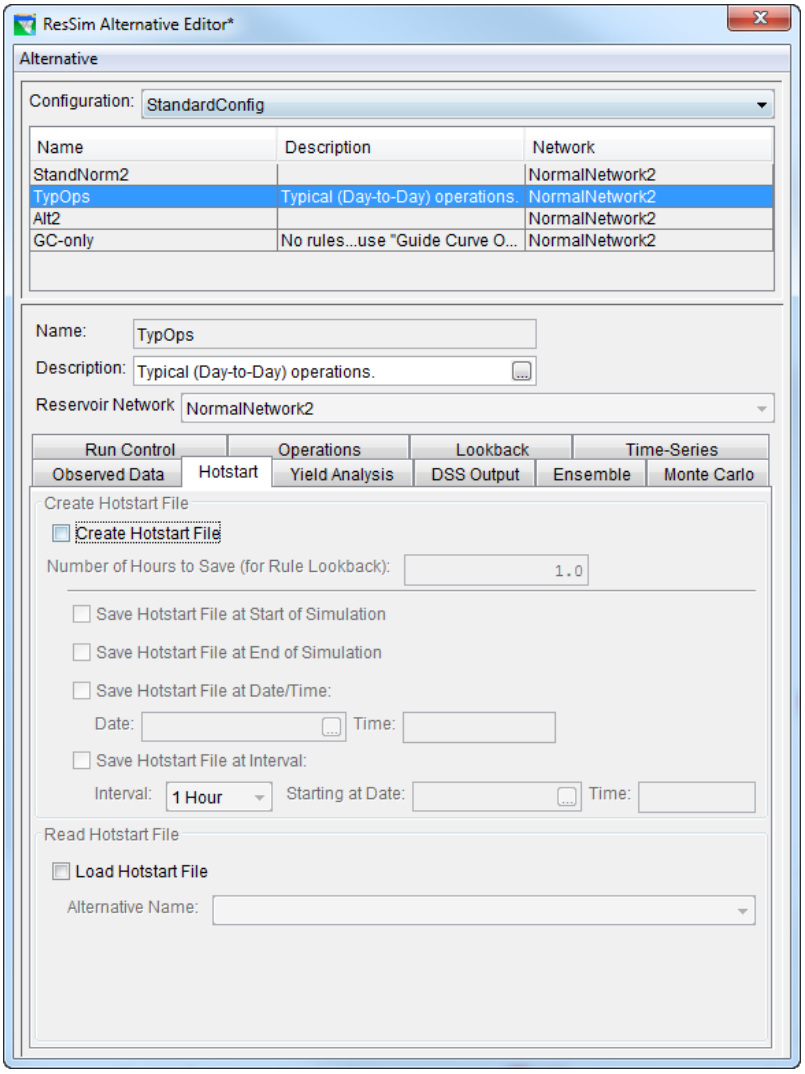


Figure 39: Alternative Editor, Hotstart Tab

Steps for Performing a Simulation

Once the reservoir system data are entered, the model can be configured for a Simulation. From the **Module** list, select **Simulation**. Model elements of a Reservoir Network will not display in the map region until a Simulation is created or opened and an Alternative has been made active.

A. Create/Open a Simulation.

To create a Simulation, from the **Simulation** menu select **New.** The window illustrated in Figure 34 will open *[Sections 14.2 and 14.3]*.

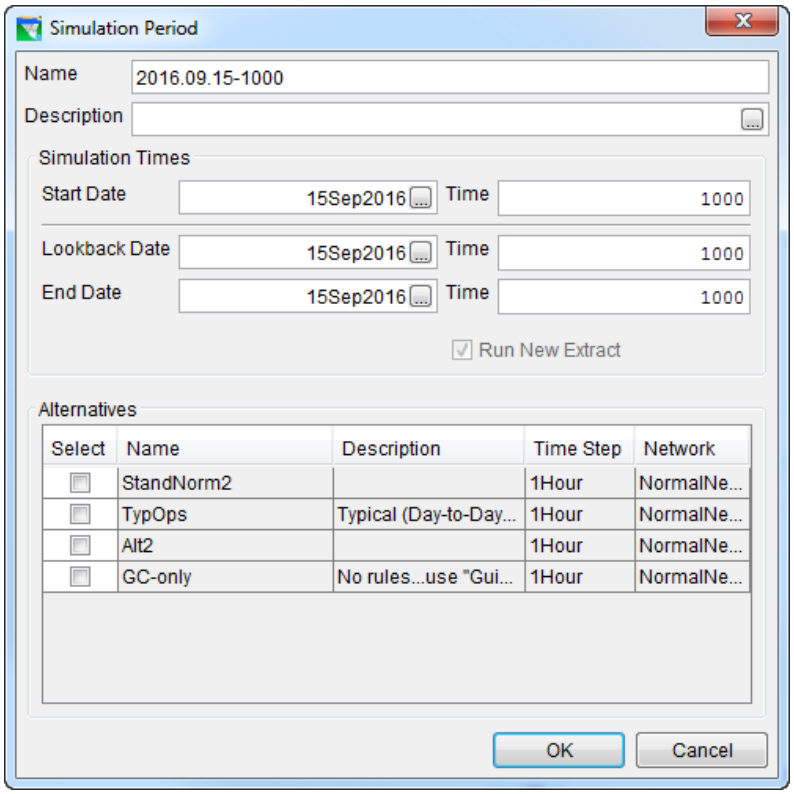


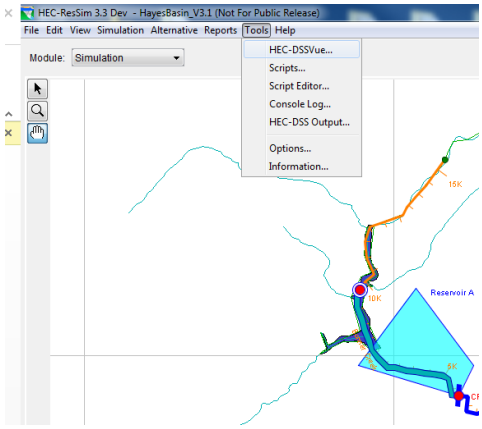
Figure 40: Simulation Period

- Enter a **Start Date** & **Time**, a **Lookback Date** & **Time**, and an **End Date** & **Time**.
- Enter a name for the Simulation in the **Name** box. A default name is generated based on the Start Date and Time. You can use the default name or you can change it to a more meaningful name. You can also enter a Simulation Description.
- Select a **Time Step** from the list.
- Select one or more applicable **Alternatives** for the Simulation and click **OK**.

B. Set Alternative to be Active.

In the "tree list" of alternatives for the simulation (on the right side of the screen), right-click on the desired alternative and select **Set as Active**. At this point, the model elements of the Reservoir Network will be displayed in the map region reflecting the active (notice **bolded name**) alternative *[Section 14.4.1]*.

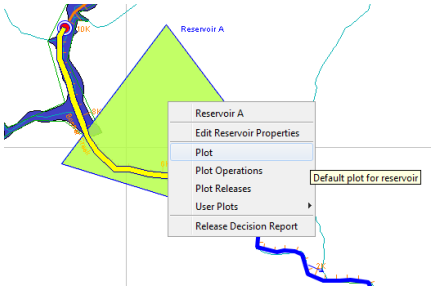
C. Compute.



After all data and parameters have been input as desired, click on the **Compute "Name of Alternative"** button in the control panel of the Simulation Module's main window to perform the computations *[Section 14.4.2]*.
A compute window will appear showing status messages and program progress. When the computation is finished, a "Compute Complete" message will appear and the status bar will read 100%. Click **Close** to close this window. *Note: You can stop the compute process before the compute is complete by clicking on* **Stop Compute**.

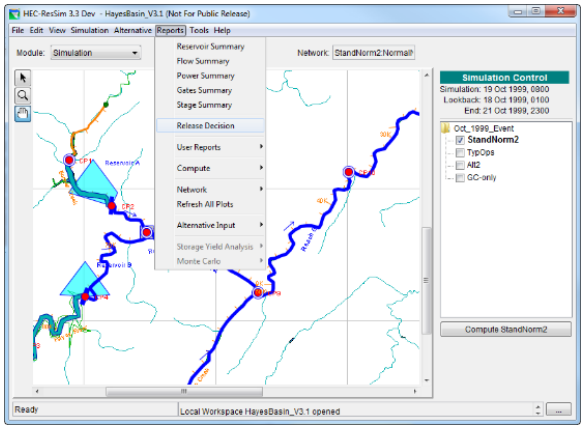
7. View Results.

Model results can be reviewed from three different approaches *[Section 14.5 and Appendices E & F]*.



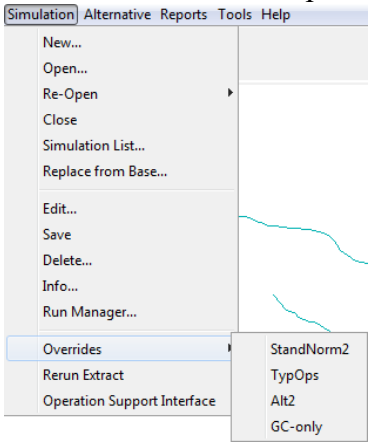
- **Plots.** On the model schematic you can right-click on a model element to get a menu list. Included on the list will be one or more plot options. Select the option by clicking on it. The default plot will display. The plotted results can be tabulated by selecting **Tabulate** from the plot's **File** menu *[Section 14.5.2]*.
- **Summary Reports.** From the **Reports** menu in the Simulation module, there is a list of Summary Reports for the model simulation *[Section 14.5.3]*. The **Reservoir**, **Flow**, **Power**, **Gates**, and **Stage** reports provide Average, Maximum and Minimum values for the simulation. The **Release Decision** summary report provides the current zone and limiting operating rule for each reservoir for all time periods of the simulation.

- **DSS Viewer.** When **HEC-DSSVue** is selected from the **Tools** menu, a DSS file is opened that contains the results of the simulation *[Section 14.8 and Appendix E]*. A list of pathnames is provided but a screened list can be obtained by selecting a pathname part from the lists in the **Search by Parts** section of the window. To select records to be displayed, highlight the pathnames and click on the **Select** button. After one or more records are selected, the buttons for plot and tabulate become active. Click either button to generate the associated output.



8. Specify Overrides.

Once you have completed a simulation, you can override model results by specifying either the reservoir Elevation Target, or Releases (per outlet), or Capacity Factors (per outlet), or Storage (or Elevation) values for some (or all) of the time steps *[Section 14.6.3]*. ResSim will use the values you specify for Elevation Target and Release Overrides, within the physical limits of the reservoir (i.e., the physical rate-of-change, the release capacity and the amount of water available in the reservoir may preclude ResSim from using your override values). To specify the override values, select **Overrides** from the **Simulation** menu in the Simulation module for the desired alternative; or, you can select the **Overrides** button by right-clicking on the "Active Alternative" and selecting **Edit → ResSim** from the shortcut menu and then click on the **Overrides** button. The Overrides Editor will open (Figure 35).



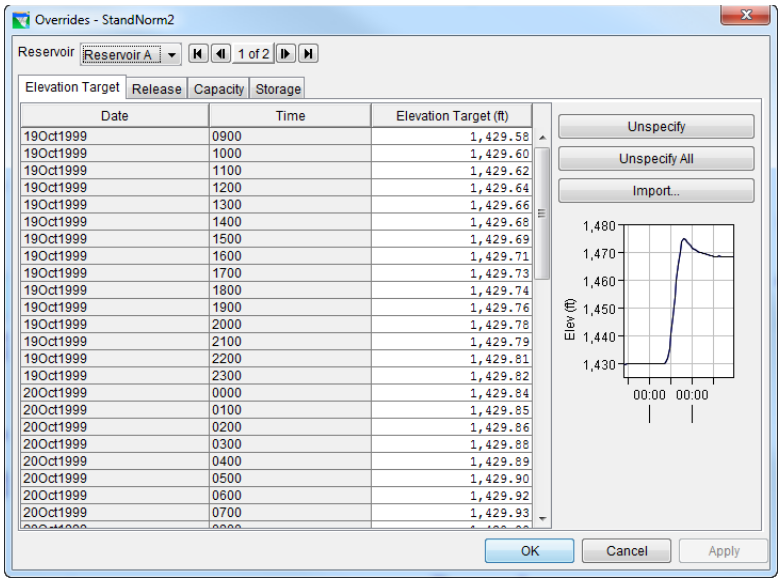
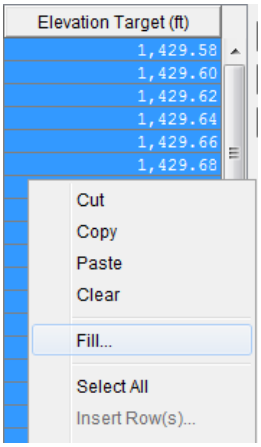


Figure 41: Overrides Editor

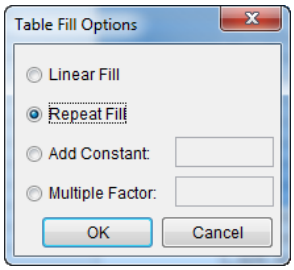
To specify override values for the "selected reservoir" (see Reservoir list at top of Overrides editor), do the following:

- Select either the **Elevation Target** or **Release** or **Capacity** or **Storage** overrides tab.
- To revise a *single* value, double-click in the cell you want to revise and enter the new value.
- To revise *multiple* adjacent values:

1. Point and click on the first cell, then drag your mouse or shift-click to highlight the cells to be revised.



2. Right-click on the highlighted cells, and select **Fill** from the shortcut menu. The Table Fill Options dialog box will open.



3. Select the appropriate fill option in the **Table Fill Options** dialog box, then click **OK**. The revised values will reflect the fill option you selected. Also, the color of the revised values changes from black to blue and the mini-plot shows the revised values as a red line.

    - For those time steps where you do not specify override values, ResSim uses the reservoir rules to determine the reservoir release values. After running a simulation using your override values, you may decide to no longer use some, or all, of your override values. If so, you can use the **Unspecify** and

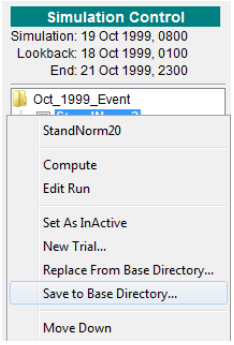
Unspecify All buttons to indicate that you want ResSim to determine the release values during the next compute.

- ❖ Use the **Unspecify** button when you want release values to be based on the reservoir rules for some of the override values you have specified. Highlight the cells where you have specified override values (they are green in the Release Overrides table), then press the **Unspecify** button.
- ❖ Use the **Unspecify All** button when you want all of the release values to be based on the reservoir rules and not have any overrides specified.

- Use the **Import** button when you want to use a Time Series of override values. The **Import Overrides Time Series** dialog box will open, which functions exactly like the **Select Time Series Path** dialog box used to select time-series records for an Alternative.

9. Manage Simulation Data.

There are two operations involved in managing Simulation data *[Section 14.7]*. You can ***save*** your data ***to*** the base directory to make it available for other Simulations, and you can ***replace*** data in a Simulation for a specific Alternative with data ***from*** the base directory. See Appendix A of the ResSim User's Manual for an overview of the ResSim directory structure, including a discussion of the base and simulation directories.



- **Save Data *to* the Base Directory**. When you edit model data from the Simulation Module, your changes apply only to an individual Alternative that is saved in your Simulation directory. If you want your changes to be available for subsequent Simulations, you will need to ***save*** the data ***back to*** the **Base** directory *[Section 14.7.1]*. This can be done by right-clicking on the **Alternative** in the **Simulation Control** menu, set the Alternative as **Active**, and then select **Save to Base Directory**. This will make your changes to physical and operational data available in the Reservoir Network Module.

- **Replace Data *from* the Base Directory**. If, while editing data in the Simulation Module, you need to revert to the original Alternative data from the Network Module, you can ***replace*** the changed data in your Simulation directory with data ***from*** the **Base** directory. *[Section 14.7.2]* This will make your *original data* (from the Reservoir Network Module) available in the Simulation Module, overwriting any changes you made to the selected alternative within the Simulation Module.